MATERIAL MODEL REFERENCE TABLES

The tables provided on the following pages list the material models, some of their attributes, and the general classes of physical materials to which the numerical models might be applied.

If a material model, without consideration of *MAT_ADD_EROSION, *MAT_ADD_-THERMAL_EXPANSION, *MAT_ADD_SOC_EXPANSION, *MAT_ADD_DAMAGE, *MAT_ADD_GENERALIZED_DAMAGE or *MAT_ADD_INELASTICITY, includes any of the following attributes, a “Y” will appear in the respective column of the table:

SRATE - Strain-rate effects
FAIL - Failure criteria
EOS - Equation-of-State required for 3D solids and 2D continuum elements
THERMAL - Thermal effects
ANISO - Anisotropic/orthotropic
DAM - Damage effects
TENS - Tension handled differently than compression in some manner

Potential applications of the material models, in terms of classes of physical materials, are abbreviated in the table as follows:

GN - General
CM - Composite
CR - Ceramic
FL - Fluid
FM - Foam
GL - Glass
HY - Hydrodynamic material
MT - Metal
PL - Plastic
RB - Rubber
SL - Soil, concrete, or rock
AD - Adhesive or Cohesive material
BIO - Biological material
CIV - Civil Engineering component
HT - Heat Transfer
F - Fabric

Material Number And Description		SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
1	Elastic								GN, FL
2	Orthotropic Elastic (Anisotropic-solids)					Y			CM, MT
3	Plastic Kinematic/Isotropic	Y	Y						CM, MT, PL
4	Elastic Plastic Thermal				Y				MT, PL
5	Soil and Foam							Y	FM, SL
6	Linear Viscoelastic	Y							RB
7	Blatz-Ko Rubber								RB
8	High Explosive Burn			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								MT
13	Isotropic Elastic 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	Y	SL
17	Oriented Crack (Elastoplastic w/ Fracture)		Y	Y		Y		Y	HY, MT, PL, CR
18	Power Law Plasticity (Isotropic)	Y							MT, PL
19	Strain Rate Dependent Plasticity	Y	Y						MT, PL
20	Rigid								
21	Orthotropic Thermal (Elastic)				Y	Y			GN
22	Composite Damage		Y			Y		Y	CM
23	Temperature Dependent Orthotropic				Y	Y			CM
24	Piecewise Linear Plasticity (Isotropic)	Y	Y						MT, PL
25	Inviscid Two Invariant Geologic Cap		Y					Y	SL
26	Honeycomb	Y	Y			Y		Y	CM, FM, SL
27	Mooney-Rivlin Rubber							Y	RB
28	Resultant Plasticity								MT
29	Force Limited Resultant Formulation							Y	

Material Number And Description		SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
30	Shape Memory								MT
31	Frazer-Nash Rubber							Y	RB
32	Laminated Glass (Composite)		Y						CM, GL
33	Barlat Anisotropic Plasticity					Y			CR, MT
33_96	Barlat YLD96	Y				Y			MT
34	Fabric					Y		Y	F
35	Plastic-Green Naghdi Rate	Y							MT
36	Three-Parameter Barlat Plasticity	Y			Y	Y			MT
37	Transversely Anisotropic Elastic Plastic					Y			MT
38	Blatz-Ko Foam								FM, PL
39	FLD Transversely Anisotropic					Y			MT
40	Nonlinear Orthotropic		Y		Y	Y		Y	CM
41-50	User Defined Materials	Y	Y	Y	Y	Y	Y	Y	GN
51	Bamman (Temp/Rate Dependent Plasticity)	Y			Y				GN
52	Bamman Damage	Y	Y		Y		Y		MT
53	Closed cell foam (Low density polyurethane)								FM
54	Composite Damage with Chang Failure		Y			Y	Y	Y	CM
55	Composite Damage with Tsai-Wu Failure		Y			Y	Y	Y	CM
57	Low Density Urethane Foam	Y	Y					Y	FM
58	Laminated Composite Fabric		Y			Y	Y	Y	CM, F
59	Composite Failure (Plasticity Based)		Y			Y		Y	CM, CR
60	Elastic with Viscosity (Viscous Glass)	Y			Y				GL
61	Kelvin-Maxwell Viscoelastic	Y							FM
62	Viscous Foam (Crash dummy Foam)	Y							FM
63	Isotropic Crushable Foam	Y						Y	FM
64	Rate Sensitive Powerlaw Plasticity	Y							MT
65	Zerilli-Armstrong (Rate/Temp Plasticity)	Y		Y	Y			Y	MT
66	Linear Elastic Discrete Beam	Y				Y			
67	Nonlinear Elastic Discrete Beam	Y				Y		Y	
68	Nonlinear Plastic Discrete Beam	Y	Y			Y			

Material Number	And Description	SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
69	SID Damper Discrete Beam	Y							
70	Hydraulic Gas Damper Discrete Beam	Y							
71	Cable Discrete Beam (Elastic)							Y	Cables
72	Concrete Damage (incl. Release III)	Y	Y	Y			Y	Y	SL
73	Low Density Viscous Foam	Y	Y					Y	FM
74	Elastic Spring Discrete Beam	Y	Y					Y	
75	Bilkhu/Dubois Foam							Y	FM
76	General Viscoelastic (Maxwell Model)	Y			Y			Y	RB
77	Hyperelastic and Ogden Rubber	Y						Y	RB
78	Soil Concrete		Y				Y	Y	SL
79	Hysteretic Soil (Elasto-Perfectly Plastic)		Y					Y	SL
80	Ramberg-Osgood								SL
81	Plasticity with Damage	Y	Y				Y		MT, PL
82	Plasticity with Damage Ortho	Y	Y			Y	Y		
83	Fu Chang Foam	Y	Y				Y	Y	FM
84	Winfrith Concrete	Y						Y	FM, SL
86	Orthotropic Viscoelastic	Y				Y			RB
87	Cellular Rubber	Y						Y	RB
88	MTS	Y		Y	Y				MT
89	Plasticity Polymer	Y						Y	PL
90	Acoustic							Y	FL
91	Soft Tissue	Y	Y			Y		Y	BIO
92	Soft Tissue (viscous)								
93	Elastic 6DOF Spring Discrete Beam	Y	Y			Y		Y	
94	Inelastic Spring Discrete Beam	Y	Y					Y	
95	Inelastic 6DOF Spring Discrete Beam	Y	Y			Y		Y	
96	Brittle Damage	Y	Y			Y	Y	Y	SL
97	General Joint Discrete Beam								
98	Simplified Johnson Cook	Y	Y						MT
99	Simpl. Johnson Cook Orthotropic Damage	Y	Y			Y	Y		MT
100	Spotweld	Y	Y				Y	Y	MT
101	GE Plastic Strain Rate	Y	Y					Y	PL

Material Number And Description		SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
102(_T)	Inv. Hyperbolic Sin (Thermal)	Y			Y				MT, PL
103	Anisotropic Viscoplastic	Y	Y			Y			MT
103P	Anisotropic Plastic					Y			MT
104	Damage 1	Y	Y			Y	Y		MT
105	Damage 2	Y	Y				Y		MT
106	Elastic Viscoplastic Thermal	Y			Y				PL
107	Modified Johnson Cook	Y	Y		Y		Y		MT
108	Ortho Elastic Plastic					Y			
110	Johnson Holmquist Ceramics	Y	Y				Y	Y	CR, GL
111	Johnson Holmquist Concrete	Y	Y				Y	Y	SL
112	Finite Elastic Strain Plasticity	Y							PL
113	Transformation Induced Plasticity (TRIP)				Y				MT
114	Layered Linear Plasticity	Y	Y						MT, PL, CM
115	Unified Creep								GN
115_O	Unified Creep Ortho					Y			GN
116	Composite Layup					Y			CM
117	Composite Matrix					Y			CM
118	Composite Direct					Y			CM
119	General Nonlinear 6DOF Discrete Beam	Y	Y			Y		Y	
120	Gurson	Y	Y				Y	Y	MT
121	General Nonlinear 1DOF Discrete Beam	Y	Y					Y	
122	Hill 3RC					Y			MT
122_3D	Hill 3R 3D					Y			MT, CM
122_TAB	Hill 3R Tabulated					Y			MT
123	Modified Piecewise Linear Plasticity	Y	Y						MT, PL
124	Plasticity Compression Tension	Y	Y					Y	MT, PL
125	Kinematic Hardening Transversely Aniso.					Y			MT
126	Modified Honeycomb	Y	Y			Y	Y	Y	CM, FM, SL
127	Arruda Boyce Rubber	Y							RB

Material Number And Description		SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
128	Heart Tissue					Y		Y	BIO
129	Lung Tissue	Y						Y	BIO
130	Special Orthotropic					Y			
131	Isotropic Smeared Crack		Y				Y	Y	MT, CM
132	Orthotropic Smeared Crack		Y			Y	Y		MT, CM
133	Barlat YLD2000	Y			Y	Y			MT
134	Viscoelastic Fabric								
135	Weak and Strong Texture Model	Y	Y			Y			MT
136	Vegter					Y			MT
136_STD	Vegter Standard Input	Y				Y			MT
136_2017	Vegter Simplified Input	Y				Y			MT
138	Cohesive Mixed Mode		Y			Y	Y	Y	AD
139	Modified Force Limited						Y	Y	
140	Vacuum								
141	Rate Sensitive Polymer	Y							PL
142	Transversely Isotropic Crushable Foam							Y	FM
143	Wood	Y	Y			Y	Y	Y	Wood
144	Pitzer Crushable Foam	Y						Y	FM
145	Schwer Murray Cap Model	Y	Y				Y	Y	SL
146	1DOF Generalized Spring	Y							
147	FWHA Soil	Y					Y	Y	SL
147N	FHWA Soil Nebraska	Y					Y	Y	SL
148	Gas Mixture				Y				FL
151	Evolving Microstructural Model of In-elast.	Y	Y		Y	Y	Y		MT
153	Damage 3	Y	Y				Y		MT, PL
154	Deshpande Fleck Foam		Y						FM
155	Plasticity Compression Tension EOS	Y	Y	Y				Y	Ice
156	Muscle	Y						Y	BIO
157	Anisotropic Elastic Plastic					Y			MT, CM
158	Rate-Sensitive Composite Fabric	Y	Y			Y	Y	Y	CM
159	CSCM	Y	Y				Y	Y	SL
160	ALE incompressible								

Material Number And Description		SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
161,162	Composite MSC (Dmg)	Y	Y			Y	Y	Y	CM
163	Modified Crushable Foam	Y						Y	FM
164	Brain Linear Viscoelastic	Y							BIO
165	Plastic Nonlinear Kinematic		Y						MT
166	Moment Curvature Beam	Y	Y					Y	CIV
167	McCormick	Y							MT
168	Polymer				Y			Y	PL
169	Arup Adhesive	Y	Y			Y		Y	AD
170	Resultant Anisotropic					Y			PL
171	Steel Concentric Brace						Y	Y	CIV
172	Concrete EC2		Y		Y			Y	SL, MT
173	Mohr Coulomb		Y			Y	Y	Y	SL
174	RC Beam						Y	Y	SL
175	Viscoelastic Thermal	Y			Y			Y	RB
176	Quasilinear Viscoelastic	Y	Y				Y	Y	BIO
177	Hill Foam							Y	FM
178	Viscoelastic Hill Foam (Ortho)	Y						Y	FM
179	Low Density Synthetic Foam	Y	Y			Y	Y	Y	FM
181	Simplified Rubber/Foam	Y	Y				Y	Y	RB, FM
183	Simplified Rubber with Damage	Y					Y	Y	RB
184	Cohesive Elastic		Y					Y	AD
185	Cohesive TH		Y			Y	Y	Y	AD
186	Cohesive General		Y			Y	Y	Y	AD
187	Semi-Analytical Model for Polymers – 1	Y	Y				Y	Y	PL
187L	SAMP light	Y						Y	PL
188	Thermo Elasto Viscoelastic Creep	Y			Y				MT
189	Anisotropic Thermoelastic				Y	Y			
190	Flow limit diagram 3-Parameter Barlat		Y			Y		Y	MT
191	Seismic Beam							Y	CIV
192	Soil Brick	Y				Y		Y	SL
193	Drucker Prager							Y	SL
194	RC Shear Wall		Y				Y	Y	CIV

Material Number And Description		SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
195	Concrete Beam	Y	Y				Y	Y	CIV
196	General Spring Discrete Beam	Y						Y	
197	Seismic Isolator	Y	Y			Y		Y	CIV
198	Jointed Rock		Y			Y		Y	SL
199	Barlat YLD2004	Y				Y			MT
199_27P	Barlat YLD2004 extended to 27 parameters by Aretz	Y				Y			MT
202	Steel EC3		Y		Y				CIV
203	Hysteretic Reinforcement		Y			Y	Y	Y	CIV
205	Discrete Beam Point Contact		Y					Y	GN, CIV
207	Simple ANIsotropic SAND (SANISAND)					Y		Y	SL
208	Bolt Beam		Y				Y	Y	MT
209	Hysteretic Beam		Y				Y	Y	CIV
211	SPR JLR	Y	Y						MT
213	Composite tabulated plasticity and damage	Y	Y		Y	Y	Y	Y	CM
214	Dry Fabric	Y	Y			Y	Y	Y	
215	4A Micromec	Y	Y			Y	Y		CM, PL
216	Elastic Phase Change								GN
217	Orthotropic Elastic Phase Change					Y			GN
218	Mooney Rivlin Rubber Phase Change							Y	RB
219	CODAM2		Y			Y	Y	Y	CM
220	Rigid Discrete								
221	Orthotropic Simplified Damage		Y			Y	Y	Y	CM
224	Tabulated Johnson Cook	Y	Y	Y	Y		Y	Y	HY, MT, PL
224_GYS	Tabulated Johnson Cook GYS	Y	Y	Y	Y		Y	Y	HY, MT, PL
225	Viscoplastic Mixed Hardening	Y	Y						MT, PL
226	Kinematic hardening Barlat 89					Y			MT
230	Elastic Perfectly Matched Layer (PML)	Y							SL
231	Acoustic PML								FL
232	Biot Linear Hysteretic Material	Y							SL

Material Number	And Description	SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
233	Cazacu Barlat					Y		Y	MT
234	Viscoelastic Loose Fabric	Y	Y			Y		Y	F
235	Micromechanic Dry Fabric					Y		Y	F
236	SCC_on_RCC		Y			Y		Y	CM, CR
237	Biot Hysteretic PML	Y							SL
238	Piecewise linear plasticity (PERT)	Y	Y						MT, PL
240	Cohesive mixed mode	Y	Y			Y	Y	Y	AD
241	Johnson Holmquist JH1	Y	Y				Y	Y	CR, GL
242	Kinematic hardening Barlat 2000					Y			MT
243	Hill 90	Y			Y	Y			MT
244	UHS Steel	Y			Y				MT
245	Orthotropic/anisotropic PML	Y							SL
246	Null material PML			Y					FL
248	PHS BMW	Y			Y	Y			MT
249	Reinforced Thermoplastic				Y	Y		Y	CM, F
249_ CRASH	Reinforced Thermoplastic Crash		Y			Y	Y	Y	CM, F
249_ UDfiber	Reinforced Thermoplastic UDfiber				Y	Y		Y	CM, F
251	Tailored Properties	Y	Y						MT, PL
252	Toughened Adhesive Polymer	Y	Y		Y	Y	Y	Y	AD
254	Generalized Phase Change	Y			Y				MT
255	Piecewise linear plastic thermal	Y	Y		Y			Y	MT
256	Amorphous solid (finite strain)	Y						Y	GL
258	Non-quadratic failure	Y	Y				Y		MT
260A	Stoughton non-associated flow	Y				Y			MT
260B	Mohr non-associated flow	Y	Y		Y	Y	Y		MT
261	Laminated Fracture Daimler Pinho	Y	Y			Y	Y	Y	CM
262	Laminated Fracture Daimler Camanho	Y	Y			Y	Y	Y	CM
263	Anisotropic plasticity					Y			MT
264	Tabulated Johnson Cook Orthotropic Plasticity	Y	Y	Y	Y	Y	Y	Y	HY, MT, PL
265	Constrained SPR2/SPR3		Y				Y		MT

Material Number	And Description	SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
266	Dispersed tissue					Y			BIO
267	Eight chain rubber	Y				Y			RB, PL
269	Bergström Boyce rubber	Y							RB
270	Welding material				Y				MT, PL
271	Powder compaction							Y	CR, SL
272	RHT concrete model	Y	Y				Y	Y	SL, CIV
273	Concrete damage plastic	Y	Y				Y	Y	SL
274	Paper					Y		Y	CM, PL
275	Smooth viscoelastic viscoplastic	Y							MT, PL
276	Chronological viscoelastic	Y			Y				RB
277	Adhesive curing viscoelastic	Y			Y				AD
278	CF Micromechanics	Y	Y		Y	Y			CM
279	Cohesive Paper		Y					Y	AD
280	Glass					Y	Y	Y	GL
291	Shape Memory Alloy				Y	Y		Y	MT
292	Isotropic Elastic for Peridynamic Solids		Y						GL, CR, PL, SL
292A	Elastic for Peridynamic Laminates		Y			Y			CM
293	COMPRF	Y				Y		Y	CM
295	Anisotropic hyperelastic					Y		Y	BIO, CM, RB
296	Soldering metal in semiconductor packaging	Y			Y				MT
303	Machine-learning base multiscale material model for fiber-reinforced composites					Y		Y	CM
305	Hot Plate Rolling	Y			Y				MT
307	Generalized Adhesive Curing	Y	Y		Y	Y	Y	Y	AD
317	RRR Polymer	Y							PL
318	TNM Polymer	Y			Y				PL
319	Incompressible Fluids with ISPG								FL
326	Gaskets							Y	AD
ALE_01	ALE Vacuum								FL
ALE_02	ALE Gas Mixture				Y				FL
ALE_03	ALE Viscous			Y				Y	FL

Material Number	And Description	SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
ALE_04	ALE Mixing Length								FL
ALE_05	ALE Incompressible								FL
ALE_06	ALE Herschel			Y				Y	FL
ISPG_01	Incompressible SPG Carreau model	Y			Y				FL
ISPG_02	Incompressible SPG Cross model	Y			Y				FL
ISPG_03	Incompressible SPG Netwonian flow behavior of an incompressible free surface flow				Y				FL
ISPG_04	Incompressible SPG Cross Castro Macosko model	Y			Y				FL
SPH_01	SPH Viscous			Y				Y	FL
SPH_02	SPH Incompressible Fluid							Y	FL
SPH_03	SPH Incompressible Structure								FL
S1	Spring Elastic (Linear)								
S2	Damper Viscous (Linear)	Y							
S3	Spring Elastoplastic (Isotropic)								
S4	Spring Nonlinear Elastic	Y						Y	
S5	Damper Nonlinear Viscous	Y						Y	
S6	Spring General Nonlinear							Y	
S7	Spring Maxwell (3-Parameter Viscoelastic)	Y							
S8	Spring Inelastic (Tension or Compression)							Y	
S13	Spring Trilinear Degrading		Y				Y		CIV
S14	Spring Squat Shearwall						Y		CIV
S15	Spring Muscle	Y						Y	BIO
B1	Seatbelt							Y	
T01	Thermal Isotropic				Y				HT
T02	Thermal Orthotropic				Y	Y			HT
T03	Thermal Isotropic (Temp Dependent)				Y				HT
T04	Thermal Orthotropic (Temp Dependent)				Y	Y			HT
T05	Thermal Discrete Beam				Y				HT
T06	Thermal Chemical Reaction				Y				HT
T07	Thermal CWM (Welding)				Y				HT

Material Number And Description		SRATE	FAIL	EOS	THERMAL	ANISO	DAM	TENS	APPS
T08	Thermal Orthotropic(Temp dep-load curve)				Y	Y			HT
T09	Thermal Isotropic (Phase Change)				Y				HT
T10	Thermal Isotropic (Temp dep-load curve)				Y				HT
T11	Thermal User Defined				Y				HT
T17	Thermal Chemical Reaction Orthotropic				Y	Y			HT
T18	Thermal ISPG				Y				HT

Alphabetized Materials List

Material Keyword	Number
*EOS_GASKET	*EOS_015
*EOS_GRUNEISEN	*EOS_004
*EOS_IDEAL_GAS	*EOS_012
*EOS_IGNITION_AND_GROWTH_OF_REACTION_IN_HE	*EOS_007
*EOS_JWL	*EOS_002
*EOS_JWLB	*EOS_014
*EOS_LINEAR_POLYNOMIAL	*EOS_001
*EOS_LINEAR_POLYNOMIAL_WITH_ENERGY_LEAK	*EOS_006
*EOS_MIE_GRUNEISEN	*EOS_016
*EOS_MURNAGHAN	*EOS_019
*EOS_PHASE_CHANGE	*EOS_013
*EOS_PROPELLANT_DEFLAGRATION	*EOS_010
*EOS_RATIO_OF_POLYNOMIALS	*EOS_005
*EOS_SACK_TUESDAY	*EOS_003
*EOS_TABULATED	*EOS_009
*EOS_TABULATED_COMPACTION	*EOS_008
*EOS_TENSOR_PORE_COLLAPSE	*EOS_011
*EOS_USER_DEFINED	*EOS_021-*EOS_030
*MAT_{OPTION}TROPIC_ELASTIC	*MAT_002
*MAT_1DOF_GENERALIZED_SPRING	*MAT_146
*MAT_3-PARAMETER_BARLAT	*MAT_036
*MAT_4A_MICROMEC	*MAT_215
*MAT_ACOUSTIC	*MAT_090
*MAT_ADD_AIRBAG_POROSITY_LEAKAGE	
*MAT_ADD_BASIC_INCREMENTAL_FAILURE	
*MAT_ADD_CHEM_SHRINKAGE	
*MAT_ADD_COHESIVE	

ALPHABETIZED MATERIALS LIST

Material Keyword	Number
*MAT_ADD_DAMAGE_DIEM	
*MAT_ADD_DAMAGE_GISSMO	
*MAT_ADD_EROSION	
*MAT_ADD_EXTVAR_EXPANSION	
*MAT_ADD_FATIGUE	
*MAT_ADD_GENERALIZED_DAMAGE	
*MAT_ADD_PERMEABILITY	
*MAT_ADD_PORE_AIR	
*MAT_ADD_SOC_EXPANSION	
*MAT_ADD_THERMAL_EXPANSION	
*MAT_ADHESIVE_CURING_VISCOELASTIC	*MAT_277
*MAT_ALE_GAS_MIXTURE	*MAT_ALE_02
*MAT_ALE_HERSCHEL	*MAT_ALE_06
*MAT_ALE_INCOMPRESSIBLE	*MAT_160
*MAT_ALE_MIXING_LENGTH	*MAT_ALE_04
*MAT_ALE_VACUUM	*MAT_ALE_01
*MAT_ALE_VISCOIS	*MAT_ALE_03
*MAT_AMORPHOUS_SOLIDS_FINITE_STRAIN	*MAT_256
*MAT_ANAND_VISCOPLASTICITY	*MAT_296
*MAT_ANISOTROPIC_ELASTIC	*MAT_002_ANISO
*MAT_ANISOTROPIC_ELASTIC_PLASTIC	*MAT_157
*MAT_ANISOTROPIC_HYPERELASTIC	*MAT_295
*MAT_ANISOTROPIC_PLASTIC	*MAT_103_P
*MAT_ANISOTROPIC_THERMOELASTIC	*MAT_189
*MAT_ANISOTROPIC_VISCOPLASTIC	*MAT_103
*MAT_ARRUDA_BOYCE_RUBBER	*MAT_127
*MAT_ARUP_ADHESIVE	*MAT_169
*MAT_BAMMAN	*MAT_051
*MAT_BAMMAN_DAMAGE	*MAT_052
*MAT_BARLAT_ANISOTROPIC_PLASTICITY	*MAT_033

Material Keyword	Number
*MAT_BARLAT_YLD2000	*MAT_133
*MAT_BARLAT_YLD2004	*MAT_199
*MAT_BARLAT_YLD2004_27P	*MAT_199_27P
*MAT_BARLAT_YLD96	*MAT_033_96
*MAT_BERGSTROM_BOYCE_RUBBER	*MAT_269
*MAT_BILKHU/DUBOIS_FOAM	*MAT_075
*MAT_BIOT_HYSTERETIC	*MAT_232
*MAT_BLATZ-KO_FOAM	*MAT_038
*MAT_BLATZ-KO_RUBBER	*MAT_007
*MAT_BOLT_BEAM	*MAT_208
*MAT_BRAIN_LINEAR_VISCOELASTIC	*MAT_164
*MAT_BRITTLE_DAMAGE	*MAT_096
*MAT_CABLE_DISCRETE_BEAM	*MAT_071
*MAT_CAZACU_BARLAT	*MAT_233
*MAT_CELLULAR_RUBBER	*MAT_087
*MAT_CF_MICROMECHANICS	*MAT_278
*MAT_CHRONOLOGICAL_VISCOELASTIC	*MAT_276
*MAT_CLOSED_CELL_FOAM	*MAT_053
*MAT_CODAM2	*MAT_219
*MAT_COHESIVE_ELASTIC	*MAT_184
*MAT_COHESIVE_GASKET	*MAT_326
*MAT_COHESIVE_GENERAL	*MAT_186
*MAT_COHESIVE_MIXED_MODE	*MAT_138
*MAT_COHESIVE_MIXED_MODE_ELASTOPLASTIC_RATE	*MAT_240
*MAT_COHESIVE_PAPER	*MAT_279
*MAT_COHESIVE_TH	*MAT_185
*MAT_COMPOSITE_DAMAGE	*MAT_022
*MAT_COMPOSITE_DIRECT	*MAT_118
*MAT_COMPOSITE_DMG_MSC	*MAT_162
*MAT_COMPOSITE_FAILURE_{OPTION}_MODEL	*MAT_059

ALPHABETIZED MATERIALS LIST

Material Keyword	Number
*MAT_COMPOSITE_LAYUP	*MAT_116
*MAT_COMPOSITE_MATRIX	*MAT_117
*MAT_COMPOSITE_MSC	*MAT_161
*MAT_COMPOSITE_TABULATED_PLASTICITY_DAMAGE	*MAT_213
*MAT_COMPRF	*MAT_293
*MAT_CONCRETE_BEAM	*MAT_195
*MAT_CONCRETE_DAMAGE	*MAT_072
*MAT_CONCRETE_DAMAGE_PLASTIC_MODEL	*MAT_273
*MAT_CONCRETE_DAMAGE_REL3	*MAT_072R3
*MAT_CONCRETE_EC2	*MAT_172
*MAT_CONSTRAINED	*MAT_265
*MAT_CRUSHABLE_FOAM	*MAT_063
*MAT_CSCM_{OPTION}	*MAT_159
*MAT_CWM	*MAT_270
*MAT_DAMAGE_1	*MAT_104
*MAT_DAMAGE_2	*MAT_105
*MAT_DAMAGE_3	*MAT_153
*MAT_DAMPER_NONLINEAR_VISCOUS	*MAT_S05
*MAT_DAMPER_VISCOUS	*MAT_S02
*MAT_DESHPANDE_FLECK_FOAM	*MAT_154
*MAT_DISCRETE_BEAM_POINT_CONTACT	*MAT_205
*MAT_DMN_COMPOSITE_FRC	*MAT_303
*MAT_DRUCKER_PRAGER	*MAT_193
*MAT_DRY_FABRIC	*MAT_214
*MAT_EIGHT_CHAIN_RUBBER	*MAT_267
*MAT_ELASTIC	*MAT_001
*MAT_ELASTIC_6DOF_SPRING_DISCRETE_BEAM	*MAT_093
*MAT_ELASTIC_FLUID	*MAT_001_FLUID
*MAT_ELASTIC_PERI	*MAT_292
*MAT_ELASTIC_PERI_LAMINATE	*MAT_292A

Material Keyword	Number
*MAT_ELASTIC_PHASE_CHANGE	*MAT_216
*MAT_ELASTIC_PLASTIC_HYDRO_{OPTION}	*MAT_010
*MAT_ELASTIC_PLASTIC_THERMAL	*MAT_004
*MAT_ELASTIC_SPRING_DISCRETE_BEAM	*MAT_074
*MAT_ELASTIC_VISCOPLASTIC_THERMAL	*MAT_106
*MAT_ELASTIC_WITH_VISCOSITY	*MAT_060
*MAT_ELASTIC_WITH_VISCOSITY_CURVE	*MAT_060C
*MAT_EMMI	*MAT_151
*MAT_ENHANCED_COMPOSITE_DAMAGE	*MAT_054-055
*MAT_EXTENDED_3-PARAMETER_BARLAT	*MAT_036E
*MAT_FABRIC	*MAT_034
*MAT_FABRIC_MAP	*MAT_034M
*MAT_FHWA_SOIL	*MAT_147
*MAT_FHWA_SOIL_NEBRASKA	*MAT_147_N
*MAT_FINITE_ELASTIC_STRAIN_PLASTICITY	*MAT_112
*MAT_FLD_3-PARAMETER_BARLAT	*MAT_190
*MAT_FLD_TRANSVERSELY_ANISOTROPIC	*MAT_039
*MAT_FORCE_LIMITED	*MAT_029
*MAT_FRAZER_NASH_RUBBER_MODEL	*MAT_031
*MAT_FU_CHANG_FOAM	*MAT_083
*MAT_GAS_MIXTURE	*MAT_148
*MAT_GENERAL_JOINT_DISCRETE_BEAM	*MAT_097
*MAT_GENERAL_NONLINEAR_1DOF_DISCRETE_BEAM	*MAT_121
*MAT_GENERAL_NONLINEAR_6DOF_DISCRETE_BEAM	*MAT_119
*MAT_GENERAL_SPRING_DISCRETE_BEAM	*MAT_196
*MAT_GENERAL_VISCOELASTIC	*MAT_076
*MAT_GENERALIZED_ADHESIVE_CURING	*MAT_307
*MAT_GENERALIZED_PHASE_CHANGE	*MAT_254
*MAT_GEOLOGIC_CAP_MODEL	*MAT_025
*MAT_GEPLASTIC_SRATE_2000a	*MAT_101

ALPHABETIZED MATERIALS LIST

Material Keyword	Number
*MAT_GLASS	*MAT_280
*MAT_GURSON	*MAT_120
*MAT_GURSON_JC	*MAT_120_JC
*MAT_GURSON_RCDC	*MAT_120_RCDC
*MAT_HEART_TISSUE	*MAT_128
*MAT_HIGH_EXPLOSIVE_BURN	*MAT_008
*MAT_HILL_3R	*MAT_122
*MAT_HILL_3R_3D	*MAT_122_3D
*MAT_HILL_3R_TABULATED	*MAT_122_TAB
*MAT_HILL_90	*MAT_243
*MAT_HILL_FOAM	*MAT_177
*MAT_HONEYCOMB	*MAT_026
*MAT_HOT_PLATE_ROLLING	*MAT_305
*MAT_HYDRAULIC_GAS_DAMPER_DISCRETE_BEAM	*MAT_070
*MAT_HYPERELASTIC_RUBBER	*MAT_077_H
*MAT_HYSTERETIC_BEAM	*MAT_209
*MAT_HYSTERETIC_REINFORCEMENT	*MAT_203
*MAT_HYSTERETIC_SOIL	*MAT_079
*MAT_IFPD	*MAT_319
*MAT_INELASTC_6DOF_SPRING_DISCRETE_BEAM	*MAT_095
*MAT_INELASTIC_6DOF_SPRING_DISCRETE_BEAM	*MAT_095
*MAT_INELASTIC_SPRING_DISCRETE_BEAM	*MAT_094
*MAT_INV_HYPERBOLIC_SIN(_THERMAL)	*MAT_102(_T)
*MAT_ISOTROPIC_ELASTIC_FAILURE	*MAT_013
*MAT_ISOTROPIC_ELASTIC_PLASTIC	*MAT_012
*MAT_ISOTROPIC_SMEARED_CRACK	*MAT_131
*MAT_ISPG_CARREAU	*MAT_ISPG_01
*MAT_ISPG_CROSS_CASTRO_MACOSKO	*MAT_ISPG_04
*MAT_ISPG_CROSSMODEL	*MAT_ISPG_02
*MAT_ISPG_ISO_NEWTONIAN	*MAT_ISPG_03

Material Keyword	Number
*MAT_JOHNSON_COOK	*MAT_015
*MAT_JOHNSON_HOLMQUIST_CERAMICS	*MAT_110
*MAT_JOHNSON_HOLMQUIST_CONCRETE	*MAT_111
*MAT_JOHNSON_HOLMQUIST_JH1	*MAT_241
*MAT_JOINTED_ROCK	*MAT_198
*MAT_KELVIN-MAXWELL_VISCOELASTIC	*MAT_061
*MAT_KINEMATIC_HARDENING_BARLAT2000	*MAT_242
*MAT_KINEMATIC_HARDENING_BARLAT89	*MAT_226
*MAT_KINEMATIC_HARDENING_TRANSVERSELY_ANISOTROPIC	*MAT_125
*MAT_LAMINATED_COMPOSITE_FABRIC	*MAT_058
*MAT_LAMINATED_FRACTURE_DAIMLER_CAMANHO	*MAT_262
*MAT_LAMINATED_FRACTURE_DAIMLER_PINHO	*MAT_261
*MAT_LAMINATED_GLASS	*MAT_032
*MAT_LAYERED_LINEAR_PLASTICITY	*MAT_114
*MAT_LINEAR_ELASTIC_DISCRETE_BEAM	*MAT_066
*MAT_LOU-YOON_ANISOTROPIC_PLASTICITY	*MAT_263
*MAT_LOW_DENSITY_FOAM	*MAT_057
*MAT_LOW_DENSITY_SYNTHETIC_FOAM_{OPTION}	*MAT_179
*MAT_LOW_DENSITY_VISCOUS_FOAM	*MAT_073
*MAT_LUNG_TISSUE	*MAT_129
*MAT_MCCORMICK	*MAT_167
*MAT_MICROMECHANICS_DRY_FABRIC	*MAT_235
*MAT_MODIFIED_CRUSHABLE_FOAM	*MAT_163
*MAT_MODIFIED_FORCE_LIMITED	*MAT_139
*MAT_MODIFIED_HONEYCOMB	*MAT_126
*MAT_MODIFIED_JOHNSON_COOK	*MAT_107
*MAT_MODIFIED_PIECEWISE_LINEAR_PLASTICITY	*MAT_123
*MAT_MODIFIED_ZERILLI_ARMSTRONG	*MAT_065
*MAT_MOHR_COULOMB	*MAT_173
*MAT_MOHR_NON_ASSOCIATED_FLOW	*MAT_260B

ALPHABETIZED MATERIALS LIST

Material Keyword	Number
*MAT_MOMENT_CURVATURE_BEAM	*MAT_166
*MAT_MOONEY-RIVLIN_RUBBER	*MAT_027
*MAT_MOONEY-RIVLIN_PHASE_CHANGE	*MAT_218
*MAT_MTS	*MAT_088
*MAT_MUSCLE	*MAT_156
*MAT_NON_QUADRATIC_FAILURE	*MAT_258
*MAT_NONLINEAR_ELASTIC_DISCRETE_BEAM	*MAT_067
*MAT_NONLINEAR_ORTHOTROPIC	*MAT_040
*MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM	*MAT_068
*MAT_NONLOCAL	
*MAT_NULL	*MAT_009
*MAT_OGDEN_RUBBER	*MAT_077_O
*MAT_OPTIONTROPIC_ELASTIC	*MAT_002
*MAT_ORIENTED_CRACK	*MAT_017
*MAT_ORTHO_ELASTIC_PLASTIC	*MAT_108
*MAT_ORTHOTROPIC_ELASTIC_PHASE_CHANGE	*MAT_217
*MAT_ORTHOTROPIC_SIMPLIFIED_DAMAGE	*MAT_221
*MAT_ORTHOTROPIC_SMEARED_CRACK	*MAT_132
*MAT_ORTHOTROPIC_THERMAL	*MAT_021
*MAT_ORTHOTROPIC_VISCOELASTIC	*MAT_086
*MAT_PAPER	*MAT_274
*MAT_PERT_PIECEWISE_LINEAR_PLASTICITY	*MAT_238
*MAT_PHS_BMW	*MAT_248
*MAT_PIECEWISE_LINEAR_PLASTIC_THERMAL	*MAT_255
*MAT_PIECEWISE_LINEAR_PLASTICITY	*MAT_024
*MAT_PITZER_CRUSHABL_EFOAM	*MAT_144
*MAT_PLASTIC_GREEN-NAGHDI_RATE	*MAT_035
*MAT_PLASTIC_KINEMATIC	*MAT_003
*MAT_PLASTIC_NONLINEAR_KINEMATIC	*MAT_165
*MAT_PLASTICITY_COMPRESSION_TENSION	*MAT_124

Material Keyword	Number
*MAT_PLASTICITY_COMPRESSION_TENSION_EOS	*MAT_155
*MAT_PLASTICITY_POLYMER	*MAT_089
*MAT_PLASTICITY_WITH_DAMAGE	*MAT_081
*MAT_PLASTICITY_WITH_DAMAGE_ORTHO(_RCDC)	*MAT_082(_RCDC)
*MAT_PML_{OPTION}TROPIC_ELASTIC	*MAT_245
*MAT_PML_ACOUSTIC	*MAT_231
*MAT_PML_ELASTIC	*MAT_230
*MAT_PML_ELASTIC_FLUID	*MAT_230
*MAT_PML_HYSTERETIC	*MAT_237
*MAT_PML_NULL	*MAT_246
*MAT_POLYMER	*MAT_168
*MAT_POWDER	*MAT_271
*MAT_POWER_LAW_PLASTICITY	*MAT_018
*MAT_PSEUDO_TENSOR	*MAT_016
*MAT_QUASILINEAR_VISCOELASTIC	*MAT_176
*MAT_RAMBERG-OSGOOD	*MAT_080
*MAT_RATE_SENSITIVE_COMPOSITE_FABRIC	*MAT_158
*MAT_RATE_SENSITIVE_POLYMER	*MAT_141
*MAT_RATE_SENSITIVE_POWERLAW_PLASTICITY	*MAT_064
*MAT_RC_BEAM	*MAT_174
*MAT_RC_SHEAR_WALL	*MAT_194
*MAT_REINFORCED_THERMOPLASTIC	*MAT_249
*MAT_REINFORCED_THERMOPLASTIC_UDFIBER	*MAT_249_UDFIBER
*MAT_RESULTANT_ANISOTROPIC	*MAT_170
*MAT_RESULTANT_PLASTICITY	*MAT_028
*MAT_RHT	*MAT_272
*MAT_RIGID	*MAT_020
*MAT_RIGID_DISCRETE	*MAT_220
*MAT_RRR_POLYMER	*MAT_317
*MAT_SAMP-1	*MAT_187

ALPHABETIZED MATERIALS LIST

Material Keyword	Number
*MAT_SAMP_LIGHT	*MAT_187L
*MAT_SCC_ON_RCC	*MAT_236
*MAT_SCHWER_MURRAY_CAP_MODEL	*MAT_145
*MAT_SEATBELT	*MAT_B01
*MAT_SEISMIC_BEAM	*MAT_191
*MAT_SEISMIC_ISOLATOR	*MAT_197
*MAT_SHAPE_MEMORY	*MAT_030
*MAT_SHAPE_MEMORY_ALLOY	*MAT_291
*MAT_SID_DAMPER_DISCRETE_BEAM	*MAT_069
*MAT_SIMPLIFIED_JOHNSON_COOK	*MAT_098
*MAT_SIMPLIFIED_JOHNSON_COOK_ORTHOTROPIC_DAMAGE	*MAT_099
*MAT_SIMPLIFIED_RUBBER/FOAM_{OPTION}	*MAT_181
*MAT_SIMPLIFIED_RUBBER_WITH_DAMAGE	*MAT_183
*MAT_SMOOTH_VISCOELASTIC_VISCOPLASTIC	*MAT_275
*MAT_SOFT_TISSUE	*MAT_091
*MAT_SOFT_TISSUE_VISCO	*MAT_092
*MAT_SOIL_AND_FOAM	*MAT_005
*MAT_SOIL_AND_FOAM_FAILURE	*MAT_014
*MAT_SOIL_BRICK	*MAT_192
*MAT_SOIL_CONCRETE	*MAT_078
*MAT_SOIL_SANISAND	*MAT_207
*MAT_SPECIAL_ORTHOTROPIC	*MAT_130
*MAT_SPH_INCOMPRESSIBLE_FLUID	*MAT_SPH_02
*MAT_SPH_INCOMPRESSIBLE_STRUCTURE	*MAT_SPH_03
*MAT_SPH_VISCOUS	*MAT_SPH_01
*MAT_SPOTWELD_{OPTION}	*MAT_100
*MAT_SPOTWELD_DAIMLERCHRYSLER	*MAT_100_DA
*MAT_SPR_JLR	*MAT_211
*MAT_SPRING_ELASTIC	*MAT_S01
*MAT_SPRING_ELASTOPLASTIC	*MAT_S03

Material Keyword	Number
*MAT_SPRING_GENERAL_NONLINEAR	*MAT_S06
*MAT_SPRING_INELASTIC	*MAT_S08
*MAT_SPRING_MAXWELL	*MAT_S07
*MAT_SPRING_MUSCLE	*MAT_S15
*MAT_SPRING_NONLINEAR_ELASTIC	*MAT_S04
*MAT_SPRING_SQUAT_SHEARWALL	*MAT_S14
*MAT_SPRING_TRILINEAR_DEGRADING	*MAT_S13
*MAT_STEEL_CONCENTRIC_BRACE	*MAT_171
*MAT_STEEL_EC3	*MAT_202
*MAT_STEINBERG	*MAT_011
*MAT_STEINBERG_LUND	*MAT_011_LUND
*MAT_STOUGHTON_NON_ASSOCIATED_FLOW	*MAT_260A
*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY	*MAT_019
*MAT_TABULATED_JOHNSON_COOK	*MAT_224
*MAT_TABULATED_JOHNSON_COOK_GYS	*MAT_224_GYS
*MAT_TABULATED_JOHNSON_COOK_ORTHO_PLASTICITY	*MAT_264
*MAT_TAILORED_PROPERTIES	*MAT_251
*MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC	*MAT_023
*MAT_THERMAL_CHEMICAL_REACTION	*MAT_T06
*MAT_THERMAL_CHEMICAL_REACTION_ORTHOTROPIC	*MAT_T17
*MAT_THERMAL_CWM	*MAT_T07
*MAT_THERMAL_DISCRETE_BEAM	*MAT_T05
*MAT_THERMAL_ISOTROPIC	*MAT_TO1
*MAT_THERMAL_ISOTROPIC_PHASE_CHANGE	*MAT_T09
*MAT_THERMAL_ISOTROPIC_TD	*MAT_T03
*MAT_THERMAL_ISOTROPIC_TD_LC	*MAT_T10
*MAT_THERMAL_ISPG	*MAT_T18
*MAT_THERMAL_ORTHOTROPIC	*MAT_T02
*MAT_THERMAL_ORTHOTROPIC_TD	*MAT_T04
*MAT_THERMAL_ORTHOTROPIC_TD_LC	*MAT_T08

ALPHABETIZED MATERIALS LIST

Material Keyword	Number
*MAT_THERMAL_USER_DEFINED	*MAT_T11
*MAT_THERMO_ELASTO_VISCOPLASTIC_CREEP	*MAT_188
*MAT_TISSUE_DISPERSED	*MAT_266
*MAT_TNM_POLYMER	*MAT_318
*MAT_TOUGHENED_ADHESIVE_POLYMER	*MAT_252
*MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC	*MAT_037
*MAT_TRANSVERSELY_ISOTROPIC_CRUSHABLE_FOAM	*MAT_142
*MAT_TRIP	*MAT_113
*MAT_UHS_STEEL	*MAT_244
*MAT_UNIFIED_CREEP	*MAT_115
*MAT_UNIFIED_CREEP_ORTHO	*MAT_115_O
*MAT_USER_DEFINED_MATERIAL_MODELS	*MAT_041-050
*MAT_VACUUM	*MAT_140
*MAT_VEGTER	*MAT_136
*MAT_VEGTER_STANDARD	*MAT_136_STD
*MAT_VEGTER_2017	*MAT_136_2017
*MAT_VISCOELASTIC	*MAT_006
*MAT_VISCOELASTIC_FABRIC	*MAT_134
*MAT_VISCOELASTIC_HILL_FOAM	*MAT_178
*MAT_VISCOELASTIC_LOOSE_FABRIC	*MAT_234
*MAT_VISCOELASTIC_THERMAL	*MAT_175
*MAT_VISCOPLASTIC_MIXED_HARDENING	*MAT_225
*MAT_VISCOUS_FOAM	*MAT_062
*MAT_WINFRITH_CONCRETE_REINFORCEMENT	*MAT_084_REINF
*MAT_WINFRITH_CONCRETE	*MAT_084
*MAT_WOOD_{OPTION}	*MAT_143
*MAT_WTM_STM	*MAT_135
*MAT_WTM_STM_PLC	*MAT_135_PLC

*MAT_ADD_AIRBAG_POROSITY_LEAKAGE

This command enables modeling porosity leakage through non-fabric material when such material is used as part of a control volume airbag. It applies to both *AIRBAG_-HYBRID and *AIRBAG_WANG_NEFSKE.

Card Summary:

Card 1a. Include this card if 0 < X0 < 1.

MID	X2	X3	ELA	FVOPT	X0	X1	

Card 1b. Include this card if X0 = 0 and FVOPT < 7.

MID	FLC	FAC	ELA	FVOPT	X0	X1	

Card 1c. Include this card if X0 = 0 and FVOPT ≥ 7.

MID	FLC	FAC	ELA	FVOPT	X0	X1	

Card 1d. Include this card if X0 = 1 and FVOPT < 7.

MID	FLC	FAC	ELA	FVOPT	X0	X1	

Card 1e. Include this card if X0 = 1 and FVOPT ≥ 7.

MID	FLC	FAC	ELA	FVOPT	X0	X1	

Data Card Definitions:

Include this card if 0 < X0 < 1.

Card 1a	1	2	3	4	5	6	7	8
Variable	MID	X2	X3	ELA	FVOPT	X0	X1	
Type	A	F	F	F	F	F	F	
Default	none	none	1.0	none	none	none	none	

VARIABLE	DESCRIPTION
MID	Material ID for which the porosity leakage property applies

VARIABLE	DESCRIPTION
X2	X2 is one of the coefficients of the porosity in the equation of Anagonye and Wang [1999]. (Defined below in description for X0/X1)
X3	X3 is one of the coefficients of the porosity in the equation of Anagonye and Wang [1999]. (Defined below in description for X0/X1)
ELA	Effective leakage area for blocked fabric, ELA LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of 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.
FVOPT	Fabric venting option. EQ.1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ.3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ.5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ.6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered. EQ.7: Leakage is based on gas volume outflow as a function of pressure load curve [Lian, 2000]. Blockage is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC. EQ.8: Leakage is based on gas volume outflow as a function of pressure load curve [Lian 2000]. Blockage of venting or porous area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC.

VARIABLE	DESCRIPTION
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $$A_{\mathrm{leak}} = A_0\left(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p\right)$$

This card is included if X0 = 0 and FVOPT < 7.

Card 1b	1	2	3	4	5	6	7	8
Variable	MID	FLC	FAC	ELA	FVOPT	X0	X1	
Type	A	F	F	F	F	F	F	
Default	none	opt	1.0	none	none	none	none	

VARIABLE	DESCRIPTION
MID	Material ID for which the porosity leakage property applies
FLC	Optional fabric porous leakage flow coefficient: GE.0.0: Fabric porous leakage flow coefficient LT.0.0: |FLC| is the load curve ID of the curve defining FLC as a function of time.
FAC	Optional fabric characteristic parameter: GE.0.0: Optional fabric characteristic parameter LT.0.0: |FAC| is the load curve ID of the curve defining FAC as a function of absolute pressure.
ELA	Effective leakage area for blocked fabric, ELA. LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of 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.
FVOPT	Fabric venting option. EQ.1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.2: Wang-Nefske formulas for venting through an orifice are

VARIABLE	DESCRIPTION
	used. Blockage of venting area due to contact is considered. EQ.3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ.5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ.6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $A_{\text{leak}} = A_0(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p)$

This card is included if X0 = 0 and FVOPT ≥ 7.

Card 1c	1	2	3	4	5	6	7	8
Variable	MID	FLC	FAC	ELA	FVOPT	X0	X1	
Type	A	F	F	F	F	F	F	
Default	none	opt	1.0	none	none	none	none	

VARIABLE	DESCRIPTION
MID	Material ID for which the porosity leakage property applies
FLC	Optional fabric porous leakage flow coefficient: GE.0.0: Fabric porous leakage flow coefficient LT.0.0: |FLC| is the load curve ID of the curve defining FLC as a function of time.
FAC	Optional fabric characteristic parameter: GE.0.0: optional fabric characteristic parameter

VARIABLE	DESCRIPTION
	LT.0.0: FAC defines leakage volume flux rate as a function of absolute pressure. The volume flux (per area) rate (per time) has the unit of velocity and it is equivalent to relative porous gas speed. $$\left[\frac{d(\mathrm{Vol}_{\mathrm{flux}})}{dt}\right] = \frac{[\mathrm{volume}]}{[\mathrm{area}]}\frac{1}{[\mathrm{time}]} = \frac{[\mathrm{length}]}{[\mathrm{time}]} = [\mathrm{velocity}]$$
ELA	Effective leakage area for blocked fabric, ELA. LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of 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.
FVOPT	Fabric venting option. EQ.7: Leakage is based on gas volume outflow as a function of pressure load curve [Lian, 2000]. Blockage is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC. EQ.8: Leakage is based on gas volume outflow as a function of pressure load curve [Lian 2000]. Blockage of venting or porous area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC.
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $$A_{\mathrm{leak}} = A_0(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p)$$

This card is included if X0 = 1 and FVOPT < 7.

Card 1d	1	2	3	4	5	6	7	8
Variable	MID	FLC	FAC	ELA	FVOPT	X0	X1	
Type	A	F	F	F	F	F	F	
Default	none	opt	1.0	none	none	none	none	

VARIABLE	DESCRIPTION
MID	Material ID for which the porosity leakage property applies
FLC	Optional fabric porous leakage flow coefficient: GE.0.0: fabric porous leakage flow coefficient LT.0.0: |FLC| is the load curve ID defining FLC as a function of the stretching ratio defined as $r_s = A/A_0$.
FAC	Optional fabric characteristic parameter: GE.0.0: optional fabric characteristic parameter LT.0.0: |FAC| is the load curve ID defining FAC as a function of the pressure ratio defined as $r_p = P_{\mathrm{air}}/P_{\mathrm{bag}}$. See Remark 2 of *MAT_FABRIC.
ELA	Effective leakage area for blocked fabric, ELA. LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of 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.
FVOPT	Fabric venting option. EQ.1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ.3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ.5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ.6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.

VARIABLE	DESCRIPTION
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $$A_{\mathrm{leak}} = A_0\left(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p\right)$$

This card is included if X0 = 1 and FVOPT ≥ 7.

Card 1e	1	2	3	4	5	6	7	8
Variable	MID	FLC	FAC	ELA	FVOPT	X0	X1	
Type	A	F	F	F	F	F	F	
Default	none	opt	1.0	none	none	none	none	

VARIABLE	DESCRIPTION
MID	Material ID for which the porosity leakage property applies
FLC	Optional fabric characteristic parameter: GE.0.0: Optional fabric characteristic parameter LT.0.0: |FAC| is the load curve ID defining FLC as a function of the stretching ratio defined as $r_s = A/A_0$.
FAC	Optional fabric characteristic parameter: GE.0.0: Optional fabric characteristic parameter LT.0.0: FAC defines leakage volume flux rate as a function of absolute pressure. The volume flux (per area) rate (per time) has the unit of velocity and it is equivalent to relative porous gas speed. $$\left[\frac{d(\mathrm{Vol_{flux}})}{dt}\right] = \frac{[\mathrm{volume}]}{[\mathrm{area}]}\frac{1}{[\mathrm{time}]} = \frac{[\mathrm{length}]}{[\mathrm{time}]} = [\mathrm{velocity}]$$
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
FVOPT	Fabric venting option. EQ.7: Leakage is based on gas volume outflow as a function of pressure load curve [Lian, 2000]. Blockage is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC. EQ.8: Leakage is based on gas volume outflow as a function of pressure load curve [Lian 2000]. Blockage of venting or porous area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC.
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $$A_{\text{leak}} = A_0\left(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p\right)$$

*MAT_ADD_BASIC_INCREMENTAL_FAILURE

Many of the implemented constitutive models do not support failure and erosion. *MAT_ADD_BASIC_INCREMENTAL_FAILURE provides a way to include load path-dependent and stress-state-dependent failure in these models. It applies to nonlinear element formulations including shells (including isogeometric shells) and solids (including isogeometric solids).

> **NOTE:** Use MAEF = 1 on *CONTROL_MAT to disable all *MAT_ADD_BASIC_INCREMENTAL_FAILURE commands in a model.

Card 1	1	2	3	4	5	6	7	8
Variable	MID		NUMFIP	VOLFRAC	NEROD			
Type	A		F	F	F			
Default	none		1.0	0.5	0.0			

Card 2	1	2	3	4	5	6	7	8
Variable	EPSF	LCSS	LCREGD	LCSRS	DMGEXP			
Type	F	F	F	F	F			
Default	0.0	0.0	0.0	0.0	1.0			

VARIABLE	DESCRIPTION
MID	Material identification for which this erosion definition applies
NUMFIP	Number or percentage of failed integration points prior to element deletion (default value is 1). NUMFIP does not apply to higher-order solid element types 24, 25, 26, 27, 28, and 29; rather, see the variable VOLFRAC. Also, when the material is a composite defined with *PART_COMPOSITE with different materials through-the-thickness, do not use NUMFIP; use *DEFINE_ELEMENT_-EROSION instead. GT.0.0: Number of integration points that must fail before an

VARIABLE	DESCRIPTION
	element is deleted. LT.0.0: Applies only to shells. |NUMFIP| is the percentage of layers that must fail before an element is deleted. For shell formulations with 4 integration points per layer, the layer is considered failed if any of the integration points in the layer fails.
VOLFRAC	Volume fraction required to fail before element deletion. The default is 0.5. It is used for higher-order solid element types 24, 25, 26, 27, 28, and 29, and all isogeometric solids and shell elements. See Remark 1.
NEROD	Option to turn off element erosion: EQ.0.0: Elements erode according to the definition of NUMFIP. EQ.1.0: Damage does not affect stresses, and elements do not erode. It could be used solely for post-processing damage.
EPSF	Plastic failure strain under uniaxial tension, meaning at a triaxiality of 1/3 and a Lode parameter of 1.0
LCSS	Load curve ID of the related material's stress-strain curve (hardening curve). If zero, then the appropriate curve of the associated material model is automatically picked (currently supported: 3, 24, 36, 81, 120, 123, 124, 133, 187, 224, 243, 251, 324).
LCREGD	Load curve ID defining the failure strain scaling factor as a function of element size
LCSRS	Load curve ID defining the failure strain scaling factor as a function of strain rate. If the first strain rate value in the curve is negative, it is assumed that all strain rate values are given as a natural logarithm of the strain rate. *The curve should not extrapolate to zero; otherwise, failure may occur at low strains.*
DMGEXP	Exponent for nonlinear damage accumulation

Basic Incremental Failure Model:

The Basic Incremental Failure Model is a phenomenological formulation that considers an incremental accumulation of a damage variable dependent on the plastic strain and the stress state through the triaxiality and the Lode parameter. The failure curve (plane

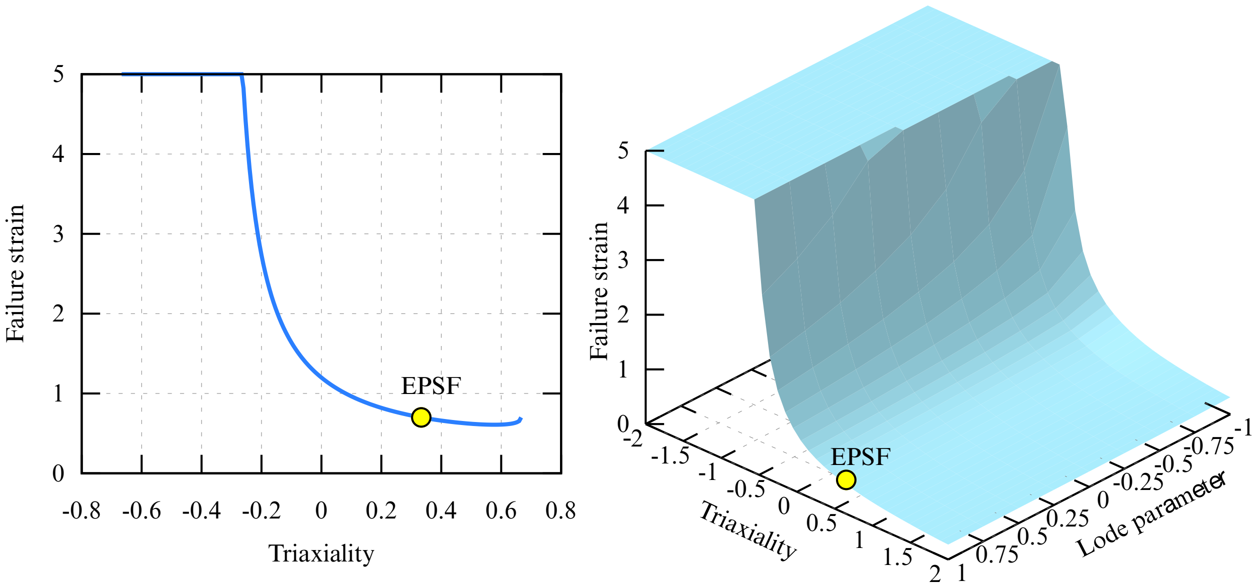


Figure 2-1. Examples of an internally generated failure curve (shells) and an internally generated failure surface (solids).

stress shells) or failure surface (solids) is internally generated based on the equation proposed by Cockcroft and Latham [1968].

The damage accumulation is given by:

$$\Delta D = \frac{\mathrm{DMGEXP} \times D^{\left(1-\frac{1}{\mathrm{DMGEXP}}\right)}}{\varepsilon_{\mathrm{f}}} \Delta\varepsilon_{\mathrm{p}}$$

where,

D	Damage value $(0 \leq D \leq 1)$. For numerical reasons, D is initialized to a value of 10^{-20} for all damage types in the first time step.
ε_{f}	Equivalent plastic strain to failure. It is determined from either a curve as a function of the current triaxiality value, η, (for plane stress shell elements) or a surface as a function of both the triaxiality and the Lode parameter, L, (for solids). The curve or surface is automatically generated as described below.
$\Delta\varepsilon_{\mathrm{p}}$	Equivalent plastic strain increment

The following equation gives the triaxiality, η, as a measure of the current stress state:

$$\eta = \frac{\sigma_{\mathrm{H}}}{\sigma_{\mathrm{M}}} \ ,$$

with hydrostatic stress, σ_{H}, and von Mises stress, σ_{M}. The Lode parameter, L, is defined as

$$L = \frac{27}{2}\frac{J_3}{{\sigma_M}^3} \ .$$

Here J_3 is the third invariant of the deviatoric stress.

The failure curve (shells) or failure surface (solids) is internally generated based on the equation proposed by Cockcroft & Latham [1968]:

$$\int_0^{\varepsilon_f} \max(\sigma_1, 0)\, d\varepsilon_p \le W_c$$

where σ_1 is the principal stress, ε_f is the strain at failure, and W_c the critical value at failure. The failure curve or failure surface is generated in such a way that EPSF is the plastic strain value at triaxiality 1/3 and Lode parameter 1.0 (see Figure 2-1). Generating the failure curve or surface requires a hardening curve. If specified, LCSS provides this hardening curve. Otherwise, it is determined based on the associated material model.

For shells, the failure curve is generated for the interval between $-{}^2\!/_3 \le \eta \le {}^2\!/_3$. For solids, the failure surface is generated for $-2 \le \eta \le 2$ and $-1 \le L \le 1$, and there is no extrapolation for triaxialities $\eta > 2$. In both cases, the plastic failure strain is bounded by $\varepsilon_f^p \le 5.0$. A file named mabif_crvtbl, generated at the beginning of the simulation, contains the internally generated curves/tables.

Providing optional load curve LCREGD activates an element size dependence. For this load curve, X-values are the element size and Y-values the scaling (regularization) factors. Full regularization is applied for $\eta \ge 1/3$, but no regularization for $\eta \le 0$. A linear interpolation is adopted for $0 < \eta < 1/3$. Figure 2-2 shows an example of the effect of the element size dependence for element sizes from 0.5 mm (reference) to 5.0 mm.

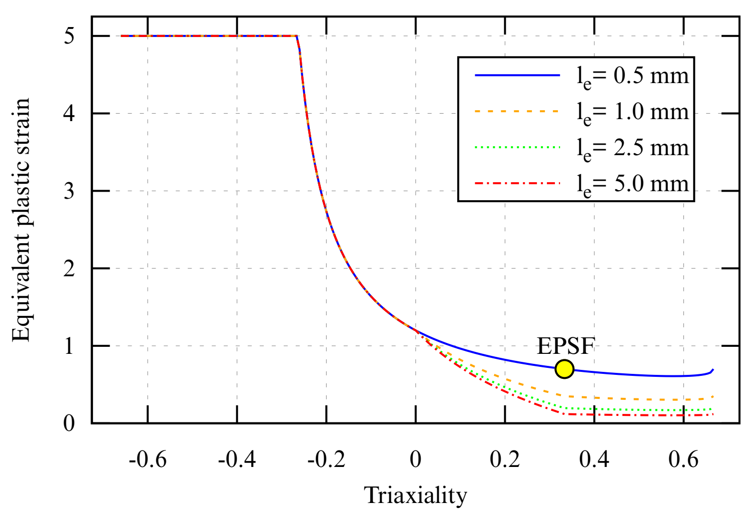


Figure 2-2. Example of the effect of the regularization on the failure curve

Remarks:

1. **VOLFRAC.** The volumes associated with individual integration points in higher-order finite elements and isogeometric elements vary widely. Thus, the number of failed integration points is not reliable for determining element failure. This failure model uses the volume fraction of the failed material for these types of elements to obtain a more stable and consistent response.

2. **History variables.** NEIPH and NEIPS must be set in *DATABASE_EXTENT_BINARY to output history data associated with this model. The damage history variables start at position ND, which is displayed in d3hsp file as, for example, "first damage history variable = 6" which means that ND = 6. For example, to view the damage parameter (first history variable) for a *MAT_024 shell element, set NEIPS = 6. In LS-PrePost, history variable #6 gives the damage parameter.

History Variable #	Description
ND	Damage parameter D ($10^{-20} < D \leq 1$)
ND + 1	Equivalent plastic strain
ND + 2	Regularization factor for failure strain (determined from LCREGD)
ND + 3	Calculated element size, l_e
ND + 4	Number of IPs/layers (NUMFIP > 0/< 0) that must fail before element deletion
ND + 5	Triaxiality variable, $\eta = \sigma_\mathrm{H}/\sigma_\mathrm{M}$
ND + 6	Lode parameter value L
ND + 7	Averaged triaxiality: $$\eta_{n+1}^{\mathrm{avg}} = \frac{1}{D_{n+1}}\left(D_n \times \eta_n^{\mathrm{avg}} + (D_{n+1} - D_n) \times \eta_{n+1}\right)$$
ND + 8	Averaged Lode parameter: $$L_{n+1}^{\mathrm{avg}} = \frac{1}{D_{n+1}}\left(D_n \times L_n^{\mathrm{avg}} + (D_{n+1} - D_n) \times L_{n+1}\right)$$
ND + 9	Alternative damage value: $D^{1/\mathrm{DMGEXP}}$

*MAT_ADD_CHEM_SHRINKAGE

The ADD_CHEM_SHRINKAGE option allows for adding the chemical shrinkage effect to a material model.

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

VARIABLE	DESCRIPTION
PID	Part ID for which the chemical shrinkage effect applies
LCID	Load curve ID (see *DEFINE_CURVE) defining the chemical shrinkage coefficient, β, or a proxy in experiments for the chemical shrinkage coefficient, α, as a function of temperature, T. If α as a function of T is defined, α is converted to the chemical shrinkage coefficient by LS-DYNA (see Remark 2).

Remarks:

1. **Chemical Shrinkage Effect on Strain.** If the chemical shrinkage effect is included, the strain rate tensor, $\dot{\boldsymbol{\varepsilon}}$, is given by

$$\dot{\boldsymbol{\varepsilon}} = \dot{\boldsymbol{\varepsilon}}^e + \dot{\boldsymbol{\varepsilon}}^p + \dot{\boldsymbol{\varepsilon}}^c \ .$$

Here, $\dot{\boldsymbol{\varepsilon}}^e$ is the elastic strain rate tensor, $\dot{\boldsymbol{\varepsilon}}^p$ is the plastic strain rate tensor, and $\dot{\boldsymbol{\varepsilon}}^c$ is the chemical shrinkage strain rate tensor. $\dot{\boldsymbol{\varepsilon}}^c$ is given by

$$\dot{\boldsymbol{\varepsilon}}^c = \beta \dot{T} \mathbf{I} \ .$$

Here β is the chemical shrinkage coefficient and $\dot{T}$ is the rate of temperature change.

2. **Chemical Shrinkage Coefficient.** The chemical shrinkage coefficient can be defined in two ways with LCID: either directly or through the proxy variable from experiments, α. If α is defined as the ordinate, LS-DYNA internally converts the ordinate of the load curve, LCID, to β:

$$\beta = (1 - \alpha)^3 - 1 \ .$$

Note that DATTYP on *DEFINE_CURVE *must be set to* -100 if LCID defines α as a function of temperature.

3. **Thermal Expansion with Shrinkage Effects.** If both thermal expansion and chemical shrinkage effects are modeled in one simulation, the thermal expansion should be defined with *MAT_THERMAL_ISOTROPIC_TITLE. The TITLE keyword option must be defined to distinguish between the thermal expansion and chemical shrinkage.

*MAT_ADD_COHESIVE

The ADD_COHESIVE option offers the possibility to use a selection of three-dimensional material models in LS-DYNA in conjunction with cohesive elements. See Remark 1.

Usually the cohesive elements (ELFORM = 19 and 20 of *SECTION_SOLID) can only be used with a small number of material models (41-50, 138, 184, 185, 186, 240). But with this additional keyword, a larger number of standard three-dimensional material models can be used that would only be available for solid elements in general. Currently the following material models are supported: 1, 3, 4, 6, 15, 24, 41-50, 57, 81, 82, 83, 89, 96, 98, 103, 104, 105, 106, 107, 115, 120, 123, 124, 141, 168, 173, 187, 188, 193, 224, 225, 251, 252, 255, 277, and 307.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	ROFLG	INTFAIL	THICK	UNIAX			
Type	I	F	F	F	F			
Default	none	0.0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
PID	Part ID for which the cohesive property applies.
ROFLG	Flag for whether density is specified per unit area or volume. EQ.0.0: Density specified per unit volume (default). EQ.1.0: Density specified per unit area for controlling the mass of cohesive elements with an initial volume of zero.
INTFAIL	The number of integration points required for the cohesive element to be deleted. The value of INTFAIL may range from 1 to 4 with 1 the recommended value. LT.0.0: Employs a Newton-Cotes integration scheme. The element will be deleted when |INTFAIL| integration points have failed. EQ.0.0: Employs a Newton-Cotes integration scheme. The element will *not* be deleted even if it satisfies the failure criterion. GT.0.0: Employs a Gauss integration scheme. The element will be deleted when INTFAIL integration points have

VARIABLE	DESCRIPTION
	failed.
THICK	Thickness of the adhesive layer (see Remark 3): EQ.0.0: The actual thickness of the cohesive element is used. GT.0.0: User specified thickness.
UNIAX	Flag for enforcing a uniaxial stress state (see Remark 2): EQ.0.0: No modification of the three-dimensional stress state (default). EQ.1.0: Stress components that are not used for the cohesive element are reset to 0.0 after each evaluation of the constitutive model.

Remarks:

1. **Three-dimensional constitutive laws with cohesive elements.** Cohesive elements possess 3 kinematic variables, namely, two relative displacements δ_1, δ_2 in tangential directions and one relative displacement δ_3 in normal direction. In a corresponding constitutive model, they are used to compute 3 associated traction stresses t_1, t_2, and t_3. For example, in the elastic case (*MAT_COHESIVE_-ELASTIC):

$$\begin{bmatrix} t_1 \\ t_2 \\ t_3 \end{bmatrix} = \begin{bmatrix} E_T & 0 & 0 \\ 0 & E_T & 0 \\ 0 & 0 & E_N \end{bmatrix} \begin{bmatrix} \delta_1 \\ \delta_2 \\ \delta_3 \end{bmatrix} .$$

Hypoelastic three-dimensional material models for standard solid elements, however, are formulated with respect to 6 independent strain rates and 6 associated stress rates. For isotropic elasticity (*MAT_ELASTIC):

$$\begin{bmatrix} \dot{\sigma}_{xx} \\ \dot{\sigma}_{yy} \\ \dot{\sigma}_{zz} \\ \dot{\sigma}_{xy} \\ \dot{\sigma}_{yz} \\ \dot{\sigma}_{zx} \end{bmatrix} = \frac{E}{(1+\nu)(1-2\nu)} \begin{bmatrix} 1-\nu & \nu & \nu & 0 & 0 & 0 \\ \nu & 1-\nu & \nu & 0 & 0 & 0 \\ \nu & \nu & 1-\nu & 0 & 0 & 0 \\ 0 & 0 & 0 & 1-2\nu & 0 & 0 \\ 0 & 0 & 0 & 0 & 1-2\nu & 0 \\ 0 & 0 & 0 & 0 & 0 & 1-2\nu \end{bmatrix} \begin{bmatrix} \dot{\varepsilon}_{xx} \\ \dot{\varepsilon}_{yy} \\ \dot{\varepsilon}_{zz} \\ \dot{\varepsilon}_{xy} \\ \dot{\varepsilon}_{yz} \\ \dot{\varepsilon}_{zx} \end{bmatrix} .$$

To be able to use such three-dimensional material models in a cohesive element environment, an assumption is necessary to transform 3 relative displacements to 6 strain rates. Therefore, we assume that neither lateral expansion nor in-plane shearing is possible. Thus,

$$\begin{bmatrix}\delta_1\\ \delta_2\\ \delta_3\end{bmatrix} \quad\rightarrow\quad \begin{bmatrix}\dot{\varepsilon}_{xx}\\ \dot{\varepsilon}_{yy}\\ \dot{\varepsilon}_{zz}\\ \dot{\varepsilon}_{xy}\\ \dot{\varepsilon}_{yz}\\ \dot{\varepsilon}_{zx}\end{bmatrix} = \begin{bmatrix}0\\ 0\\ \dot{\delta}_3/(t+\delta_3)\\ 0\\ \dot{\delta}_2/(t+\delta_3)\\ \dot{\delta}_1/(t+\delta_3)\end{bmatrix},$$

where t is the initial thickness of the adhesive layer; see parameter THICK. These strain rates are then used in a three-dimensional constitutive model to obtain new Cauchy stresses, where 3 components can finally be used for the cohesive element:

$$\begin{bmatrix}\sigma_{xx}\\ \sigma_{yy}\\ \sigma_{zz}\\ \sigma_{xy}\\ \sigma_{yz}\\ \sigma_{zx}\end{bmatrix} \quad\rightarrow\quad \begin{bmatrix}t_1\\ t_2\\ t_3\end{bmatrix} = \begin{bmatrix}\sigma_{zx}\\ \sigma_{yz}\\ \sigma_{zz}\end{bmatrix}.$$

For hyperelastic material models 57 and 83, the deformation gradient is obtained by an incremental update of the strain rates mentioned above.

2. **Forcing uniaxial stress state.** As stated in Remark 1, only three values from the six stress components are used for the cohesive element. By default, the remaining stress components are not modified. Consequently, transverse normal stresses, σ_{xx} and σ_{yy}, and in-plane shear stress, σ_{xy}, will in most cases build-up during the simulation of uniaxial loading of the cohesive zone due to Poisson's effect and the given reduced strain rate tensor. These stress components should not affect the response of the cohesive element for elastic or viscoelastic material models. They will, however, have a significant effect for most materials with plasticity If undesired, the effect can be reduced by setting the UNIAX flag which resets the unused stress components σ_{xx}, σ_{yy} and σ_{xy} to 0.0 after each evaluation of the three-dimensional constitutive model.

3. **Critical time step.** The critical time step size for cohesive elements depends on nodal masses (that is, element volume and density) and the stiffness of the material, $\max(E_T, E_N)$. Note that stiffness has units of stress per length^3, such as N/mm^3. With *MAT_ADD_COHESIVE, the elastic moduli (stress unit) from the corresponding 3-dimensional material model are taken and related to the thickness (length unit) of the cohesive element. Thus, the thickness of the cohesive element (either coming from geometry or THICK) changes the critical time step size. Therefore, we recommend using a reasonable value for THICK.

4. **Output to d3plot.** If this keyword is used with a three-dimensional material model, the output to d3plot or elout is organized as with other material models for cohesive elements; see for example *MAT_184. Instead of the usual six stress components, three traction stresses are written into those databases. The in-

plane shear traction along the 1-2 edge replaces the x component of stress, the orthogonal in-plane shear traction replaces the y component of stress, and the traction in the normal direction replaces the z component of stress.

*MAT_ADD_DAMAGE_DIEM

Many of the constitutive models in LS-DYNA do not allow failure and erosion. The ADD_DAMAGE_DIEM option provides a way of including damage and failure in these models. DIEM comprises various "damage initiation and evolution models." See remarks for details.

This keyword originates from a split out of *MAT_ADD_EROSION, where only "sudden" failure criteria without damage remain. It applies to nonlinear element formulations including 2D continuum elements, beam element formulation 1, 3D shell elements (including isogeometric shells), 3D solid elements (including isogeometric solids) and thick shells.

NOTE: All *MAT_ADD_DAMAGE_DIEM commands in a model can be disabled by using *CONTROL_MAT.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	NDIEMC	DINIT	DEPS	NUMFIP	VOLFRAC		
Type	A	F	F	F	F	F		
Default	none	0.0	0.0	0.0	1.0	0.5		

Data Card Pairs. Include NDIEMC pairs of Cards 2 and 3.

Card 2	1	2	3	4	5	6	7	8
Variable	DITYP	P1	P2	P3	P4	P5		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

Card 3	1	2	3	4	5	6	7	8
Variable	DETYP	DCTYP	Q1	Q2	Q3	Q4		
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 for which this erosion definition applies. A unique number or label must be specified (see *PART).
NDIEMC	Number of damage initiation and evolution model (DIEM) criteria to be applied, at most 5 is allowed.
DINIT	Damage initialization option: EQ.0: No action is taken. EQ.1: Damage history is initiated based on values of the initial plastic strains and the initial strain tensor. This is to be used in multistage analyses.
DEPS	Plastic strain increment between evaluation of damage instability and evolution criteria. See Remark 1. The default is zero.
NUMFIP	Number or percentage of failed integration points prior to element deletion (default value is 1). NUMFIP does not apply to higher order solid element types 24, 25, 26, 27, 28, and 29, rather see the variable VOLFRAC. Also, when the material is a composite defined with *PART_COMPOSITE with different materials through-the-thickness, do not use NUMFIP; use *DEFINE_ELEMENT_-EROSION instead. GT.0.0: Number of integration points which must fail before element is deleted. LT.0.0: Applies only to shells. |NUMFIP| is the percentage of layers which must fail before an element fails. For shell formulations with 4 integration points per layer, the layer is considered failed if any of the integration points in the layer fails.
VOLFRAC	Volume fraction required to fail before element deletion. The

VARIABLE	DESCRIPTION
	default is 0.5. It is used for higher-order solid element types 24, 25, 26, 27, 28, and 29, and all isogeometric solids and shell elements. See Remark 3.
DITYP	Damage initiation type (see Damage Initiation section): EQ.0.0: Ductile based on stress triaxiality EQ.1.0: Shear EQ.2.0: MSFLD EQ.3.0: FLD EQ.4.0: Ductile based on normalized principal stress
P1	Damage initiation parameter: DITYP.EQ.0.0: Load curve/table ID representing plastic strain at the onset of damage as a function of stress triaxiality (η) and optionally plastic strain rate, represented by ε_D^p in the theory below. If the first strain rate value in the table is negative, it is assumed to be given with respect to logarithmic strain rate. DITYP.EQ.1.0: Load curve/table ID representing plastic strain at onset of damage as a function of shear influence (θ) and optionally plastic strain rate, represented by ε_D^p in the theory below. If the first strain rate value in the table is negative, it is assumed to be given with respect to logarithmic strain rate. DITYP.EQ.2.0: Load curve/table ID representing plastic strain at onset of damage as a function of ratio of principal plastic strain rates (α) and optionally plastic strain rate, represented by ε_D^p in the theory below. If the first strain rate value in the table is negative, it is assumed to be given with respect to logarithmic strain rate. DITYP.EQ.3.0: Load curve/table ID representing plastic strain at onset of damage as a function of ratio of principal plastic strain rates (α) and optionally plastic strain rate, represented by ε_D^p in the theory below. If the first strain rate value in the table is negative, it is assumed to be given with respect

VARIABLE	DESCRIPTION
	to logarithmic strain rate.
	DITYP.EQ.4.0: Load curve/table ID representing plastic strain at onset of damage as a function of stress state parameter (β) and optionally plastic strain rate, represented by ε_D^p in the theory below. If the first strain rate value in the table is negative, it is assumed to be given with respect to logarithmic strain rate.
P2	Damage initiation parameter: DITYP.EQ.0.0: Not used DITYP.EQ.1.0: Pressure influence parameter, k_s DITYP.EQ.2.0: Layer specification: EQ.0: Mid layer EQ.1: Outer layer DITYP.EQ.3.0: Layer specification: EQ.0: Mid layer EQ.1: Outer layer DITYP.EQ.4.0: Triaxiality influence parameter, k_d
P3	Damage initiation parameter: DITYP.EQ.0.0: Not used DITYP.EQ.1.0: Computation of maximum shear stress for shells: EQ.0: 3-dimensional EQ.1: 2-dimensional DITYP.EQ.2.0: Initiation formulation: EQ.0: Direct EQ.1: Incremental DITYP.EQ.3.0: Initiation formulation: EQ.0: Direct EQ.1: Incremental DITYP.EQ.4.0: Not used
P4	Plane stress option for shell elements: EQ.0.0: Transverse shear stresses, σ_{yz} and σ_{zx}, are included in the computation of stress invariants, such as the

VARIABLE	DESCRIPTION
	triaxiality. EQ.1.0: Transverse shear stresses, σ_{yz} and σ_{zx}, are *not* included in the computation of stress invariants, such as the triaxiality. Useful in combination with "plane stress" material models, where the transverse shear stresses are also excluded from the yield condition, such as *MAT_-024_2D or *MAT_036.
P5	Load curve or table ID representing regularization factor as a function of the characteristic element size (curve) or regularization factor as a function of the characteristic element size and abscissa value of the criterion used (table). The criterion is the curve/table specified in P1. For example, for DITYP = 0.0, the regularization factor would depend on stress triaxiality. This factor scales the plastic strain at the onset of damage defined with P1.
DETYP	Damage evolution type: EQ.0.0: Linear softening. Evolution of damage is a function of the plastic displacement after the initiation of damage. EQ.1.0: Linear softening, Evolution of damage is a function of the fracture energy after the initiation of damage.
DCTYP	Damage composition option for multiple criteria: EQ.-1.0: Damage not coupled to stress EQ.0.0: Maximum EQ.1.0: Multiplicative
Q1	Damage evolution parameter: DETYP.EQ.0.0: Plastic displacement at failure, u_f^p. A negative value corresponds to a *table* ID for u_f^p as a function of triaxiality and damage. DETYP.EQ.1.0: Fracture energy at failure, G_f
Q2	Set to 1.0 to output information to log files (messag and d3hsp) when an integration point fails.
Q3	Damage evolution parameter: DETYP.EQ.0.0: Exponent, α, in nonlinear damage evolution

VARIABLE	DESCRIPTION
	law, activated when $u_f^p > 0$ and $\alpha > 0$. DETYP.EQ.1.0: Not used.
Q4	Load curve or table ID representing regularization factor as a function of the characteristic element size (curve) or regularization factor as a function of the characteristic element size and abscissa value of the criterion used (table). The criterion is the curve/table specified in P1. For example, for DITYP = 0.0, the regularization factor would depend on stress triaxiality. If Q4 is input with a negative sign, the second table input should be plastic strain rate instead of the abscissa value. This factor scales the damage evolution parameter Q1.

Remarks:

1. **DEPS.** In DIEM, you may invoke up to 5 damage initiation and evolution criteria. For the sake of efficiency, the parameter DEPS can be used to only check these criteria in quantified increments of plastic strain. In other words, the criteria are only checked when the effective plastic strain goes beyond DEPS, 2 × DEPS, 3 × DEPS, etc. For DEPS = 0 the checks are performed in each step there is plastic flow. A reasonable value of DEPS could, for instance, be DEPS = 0.0001.

2. **Damage initiation and evolution variables.** Assume that n initiation/evolution types have been specified in the input deck (n = NDIEMC). At each integration point a damage initiation variable, ω_D^i, and an evolution history variable D^i exist, such that,

$$\omega_D^i \in [0, \infty)$$

and

$$D^i \in [0,1] \ , \qquad i = 1, \ldots n \ .$$

These are initially set to zero and evolve with the deformation of the elements according to rules associated with the specific damage initiation and evolution type chosen, see below for details.

These quantities can be post-processed as ordinary material history variables and their positions in the history variables array is given in d3hsp, search for the string *Damage history listing*. The damage initiation variables do not influence the results but serve to indicate the onset of damage. As an alternative, the keyword *DEFINE_MATERIAL_HISTORIES can be used to output the instability and damage, following

*DEFINE_MATERIAL_HISTORIES Properties					
Label	**Attributes**				**Description**
Instability	-	-	-	-	Maximum initiation variable, $\max\limits_{i=1,\ldots,n} \omega_D^i$
Damage	-	-	-	-	Effective damage D, see below

The damage evolution variables govern the damage in the material and are used to form the global damage $D \in [0,1]$. Each criterion is of either of DCTYP set to maximum (DCTYP = 0) or multiplicative (DCTYP = 1), or one could choose to not couple damage to the stress by setting DCTYP = −1. This means that the damage value is calculated and stored, but it is not affecting the stress as for the other options, so if all DCTYP are set to −1 there will be no damage or failure. Letting I_{max} denote the set of evolution types with DCTYP set to maximum and I_{mult} denote the set of evolution types with DCTYP set to multiplicative the global damage, D, is defined as

$$D = \max(D_{\text{max}}, D_{\text{mult}}) \ ,$$

where

$$D_{\text{max}} = \max_{i \in I_{\text{max}}} D^i \ ,$$

and

$$D_{\text{mult}} = 1 - \prod_{i \in I_{\text{mult}}} \left(1 - D^i\right) \ .$$

The damage variable relates the macroscopic (damaged) to microscopic (true) stress by

$$\sigma = (1 - D)\tilde{\sigma} \ .$$

Once the damage has reached the level of D_{erode} (=0.99 by default) the stress is set to zero and the integration point is assumed failed and not processed thereafter. For NUMFIP > 0, a shell element is eroded and removed from the finite element model when NUMFIP integration points have failed. For NUMFIP < 0, a shell element is eroded and removed from the finite element model when -NUMFIP percent of the layers have failed.

3. **VOLFRAC.** The volumes associated with individual integration points in higher order finite elements and isogeometric elements varies widely. Thus, the number of failed integration points is not a reliable criterion for determining element failure. To obtain a more stable and consistent response, LS-DYNA uses the volume fraction of the failed material for these types of elements.

Damage Initiation, ω_D:

For each evolution type i, ω_D^i governs the onset of damage. For $i \neq j$, the evolution of ω_D^i is independent from the evolution of ω_D^j. The following list enumerates the algorithms for modelling damage initiation.

In this subsection we suppress the superscripted i indexing the evolution type.

1. **Ductility based on stress triaxiality (DITYP = 0).** For the ductile initiation option, a function $\varepsilon_D^p = \varepsilon_D^p(\eta, \dot{\varepsilon}^p)$ represents the plastic strain at onset of damage (P1). This is a function of stress triaxiality defined as

$$\eta = -\frac{p}{q} \ ,$$

with p being the pressure and q the von Mises equivalent stress. Optionally this can be defined as a table with the second dependency being on the effective plastic strain rate $\dot{\varepsilon}^p$, where a negative sign of the first strain rate in the table means that it is in logarithmic scale. The damage initiation history variable evolves according to

$$\omega_D = \int_0^{\varepsilon^p} \frac{d\varepsilon^p}{\varepsilon_D^p} \ .$$

2. **Shear (DITYP = 1).** For the shear initiation option, a function $\varepsilon_D^p = \varepsilon_D^p(\theta, \dot{\varepsilon}^p)$ represents the plastic strain at onset of damage (P1). This is a function of a shear stress function defined as

$$\theta = \frac{q + k_S p}{\tau} \ .$$

Here p is the pressure, q is the von Mises equivalent stress and τ is the maximum shear stress defined as a function of the principal stress values:

$$\tau = \frac{\left(\sigma_{\text{major}} - \sigma_{\text{minor}}\right)}{2} \ .$$

Parameter P3 allows you to select which principal stresses are used in this equation for shell elements. With P3 = 0, only the in-plane stresses are considered (2-dimensional approach), whereas with P3 = 1, they are computed from the full stress tensor (3-dimensional approach). The latter case leads to higher shear fracture risk for the range between uniaxial and biaxial loading. Introduced here is also the pressure influence parameter k_s (P2). Optionally this can be defined as a table with the second dependency being on the effective plastic strain rate $\dot{\varepsilon}^p$, where a negative sign of the first strain rate in the table means that it is in logarithmic scale. The damage initiation history variable evolves according to

$$\omega_D = \int_0^{\varepsilon^p} \frac{d\varepsilon^p}{\varepsilon_D^p}.$$

3. **MSFLD (DITYP = 2).** For the MSFLD initiation option, a function $\varepsilon_D^p = \varepsilon_D^p(\alpha, \dot{\varepsilon}^p)$ represents the plastic strain at onset of damage (P1). This is a function of the ratio of principal plastic strain rates defined as

$$\alpha = \frac{\dot{\varepsilon}^p_{\text{minor}}}{\dot{\varepsilon}^p_{\text{major}}}.$$

The MSFLD criterion is only relevant for shells and with restrictions (discussed in the section MSFLD and FLD with solid and thick shell elements) for hexa/penta solids/tshells. The principal strains should be interpreted as the in-plane principal strains. For simplicity the plastic strain evolution in this formula is assumed to stem from an associated von Mises flow rule. Hence,

$$\alpha = \frac{s_{\text{minor}}}{s_{\text{major}}}$$

with s being the deviatoric stress. This ensures that the calculation of α, is in a sense, robust at the expense of being slightly inaccurate for materials with anisotropic yield functions and/or non-associated flow rules. Optionally this can be defined as a table with the second dependency being on the effective plastic strain rate, $\dot{\varepsilon}^p$, where a negative sign of the first strain rate in the table means that it is in logarithmic scale. For $\dot{\varepsilon}^p = 0$, the value of ε_D^p is set to a large number to prevent the onset of damage for no plastic evolution. Furthermore, the plastic strain used in this failure criteria is a modified effective plastic strain that only evolves when the pressure is negative, meaning the material is not affected in compression.

This modified plastic strain can be monitored as the second history variable of the initiation history variables in the binary output database. For P3 = 0, the damage initiation history variable is calculated directly from the ratio of (modified) plastic strain and the critical plastic strain

$$\omega_D = \max_{t \le T} \frac{\varepsilon^p}{\varepsilon_D^p} \ .$$

This should be interpreted as the maximum value up to this point in time. If P3 = 1 the damage initiation history variable is instead incrementally updated from the ratio of (modified) plastic strain and the critical plastic strain

$$\omega_D = \int_0^{\varepsilon^p} \frac{d\varepsilon^p}{\varepsilon_D^p} \ .$$

For this initiation option with shells, P2 is used to determine the layer in the shell where the criterion is evaluated. If P2 = 0, the criterion is evaluated in the mid-layer only, whereas if P2 = 1, it is evaluated in the outer layers only (bottom and top). This can be used to distinguish between a membrane instability typically used for FLD evaluations (P2 = 0) and a bending instability (P2 = 1). For shells, as soon as ω_D reaches 1 in any of the integration points of interest, *all* integration points in the shell goes over in damage mode, meaning subsequent damage is applied to the entire element. For solids/tshells, only P2 = 0 is currently supported, and when ω_D reaches 1 in the center of the section then all elements in the section goes into damage mode. Again, more details for solids or thick shells are provided in the section titled MSFLD and FLD with solid and thick shell elements.

4. **FLD (DITYP = 3).** The FLD initiation criterion is identical to MSFLD with one subtle difference: the plastic strain used to evaluate the criteria does not account for the sign of the hydrostatic stress but is instead identical to the effective plastic strain directly from the underlying material model. In other words, it is not the modified plastic strain used in the MSFLD criterion, but apart from that it is an identical criterion.

5. **Ductile based on normalized principal stress (DITYP = 4).** For the ductile initiation option the plastic strain at the onset of damage (P1) is taken as a function of β and $\dot{\varepsilon}^p$, that is $\varepsilon_D^p = \varepsilon_D^p(\beta, \dot{\varepsilon}^p)$. Here β is the normalized principal stress

$$\beta = \frac{q + k_d p}{\sigma_{\text{major}}} \quad ,$$

where p is the pressure, q is the von Mises equivalent stress, σ_{major} is the major principal stress, and k_d is the pressure influence parameter specified in the P2 field. Optionally, this can be defined as a table with the second dependency being on the effective plastic strain rate $\dot{\varepsilon}^p$, where a negative sign of the first strain rate in the table means that it is in logarithmic scale. The damage initiation history variable evolves according to

$$\omega_D = \int_0^{\varepsilon^p} \frac{d\varepsilon^p}{\varepsilon_D^p} \quad .$$

MSFLD and FLD with solid and thick shell elements

When using MSFLD or FLD with solid or thick shell elements, the following restrictions apply:

- The part should be a thin walled section, with a well-defined “*thickness*” direction, $\boldsymbol{t}$, and associated “*plane*” indicated in blue in Figure 2-3.
- Only low order hexahedra or pentahedra may be used.
- The same number of elements in $\boldsymbol{t}$-direction must be used, essentially in the form of an extruded shell mesh. The stack of elements at any location, from bottom to top, comprises the “section”.
- The element numbering scheme must in itself indicate the thickness direction, $\boldsymbol{t}$, as illustrated in Figure 2-3, for each element in the part.

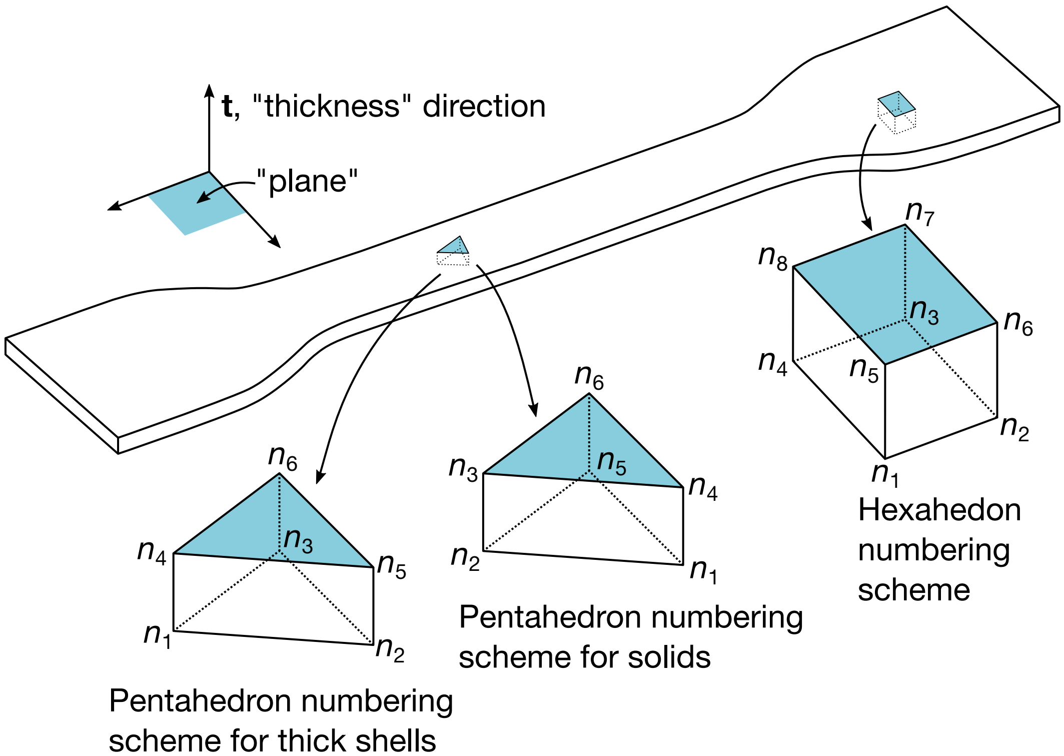


Figure 2-3. Solid and thick shell elements must be oriented in a part in a specific way when using MSFLD or FLD (DITYP = 2 or 3). This figure illustrates the required element numbering scheme and the thickness direction, **t**.

- The geometry may be curved, but the mesh topology must not change. Thus, for T-intersections and similar geometries, appropriate pre-processing measures must be undertaken.

Damage Evolution, D:

For the evolution of the associated damage variable, D, we introduce the plastic displacement, u^P, which evolves according to

$$\dot{u}^p = \begin{cases} 0 & \omega_D < 1 \\ l\dot{\varepsilon}^p & \omega_D \geq 1 \end{cases}$$

Here l is a characteristic length of the element. Fracture energy is related to plastic displacement as follows

$$G_f = \int_0^{u_f^p} \sigma_y du^p \quad ,$$

where σ_y is the yield stress. The following list enumerates the algorithms available for modelling damage.

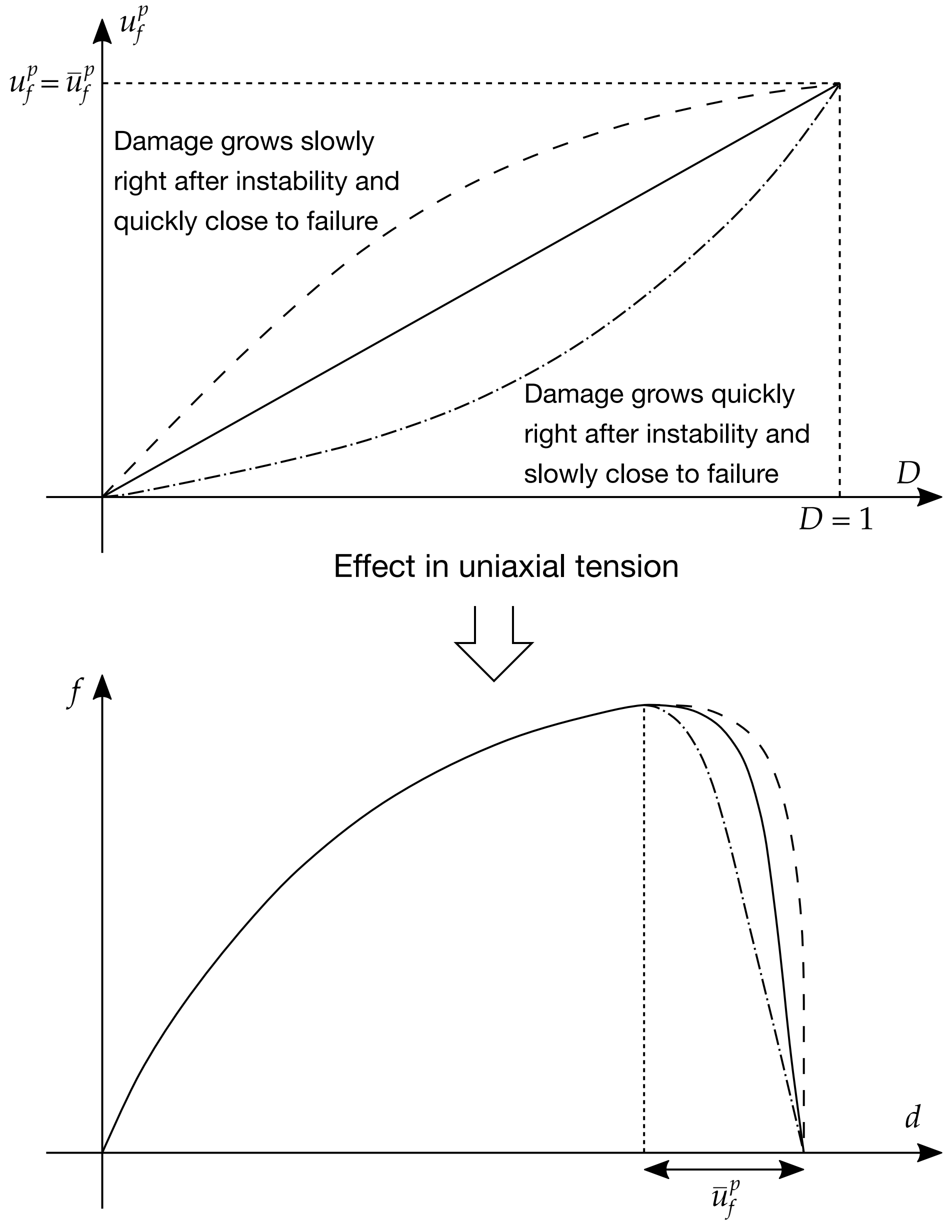


Figure 2-4. Top plot illustrates plastic displacement at failure as a function of damage for a given triaxiality for DETYP = 0 and $\alpha = 0$. The different curves illustrate different possible types of post-instability characteristics. The bottom plot illustrates qualitatively how these curves may interact in a tension test. d is the displacement and f is the force.

1. **Linear (DETYP = 0).** With this option, if $\alpha = 0$ (Q3) and Q1 is positive, the damage variable evolves linearly with the plastic displacement after onset of damage:

$$\dot{D} = \frac{\dot{u}^p}{u_f^p} \ .$$

Here u_f^p is the plastic displacement at failure (Q1).

If Q1 is negative, then –Q1 refers to a table that defines u_f^p as a function of triaxiality and damage, that is, $u_f^p = u_f^p(\eta, D)$. In this case with $\alpha = 0$, the damage evolution law generalizes to:

$$\dot{D} = \frac{\dot{u}^p}{\frac{\partial u_f^p}{\partial D}} .$$

For a fixed triaxiality, η, $\bar{u}_f^p := u_f^p(\eta, 1)$ defines the plastic displacement at failure, and the shape of $u_f^p(\eta, D)$ as a function of D determines the post-instability characteristics.

A linear curve, as illustrated by the solid line in Figure 2-4, corresponds exactly to a constant plastic displacement to failure equal to $\bar{u}_f^p$ and can be seen as a reference curve for this discussion. For simplicity assume uniaxial tension ($\eta = 1/3$). A curve with positive curvature, represented by the dash-dots in Figure 2-4, means that damage evolves quickly right after onset of instability and more slowly when approaching failure. In contrast, damage evolves slowly early and more quickly later on for a curve with negative curvature, represented by the dashes. The qualitative effect these curves have in a uniaxial tension test is also illustrated. The correlation between a damage curve and the actual behavior in tests is not straightforward, thus these curves need to be established on a trial-and-error basis.

For $\alpha > 0$ and $u_f^p > 0$, the damage evolution follows an exponential law given by

$$D = \frac{1 - e^{-\alpha \frac{u^p}{u_f^p}}}{1 - e^{-\alpha}},$$

where $u^p = \int \dot{u}^p$.

2. **Linear (DETYP = 1).** With this option the damage variable evolves linearly as follows

$$\dot{D} = \frac{\dot{u}^p}{u_f^p} \quad ,$$

where $u_f^p = 2G_f/\sigma_{y_0}$ and σ_{y_0} is the yield stress when failure criterion is reached.

*MAT_ADD_DAMAGE_GISSMO_{OPTION}

Available options include:

<BLANK>

STOCHASTIC

Many of the constitutive models in LS-DYNA do not allow failure and erosion. *MAT_-ADD_DAMAGE_GISSMO provides a way to include damage and failure in these models. GISSMO is the "generalized incremental stress-state dependent damage model." It applies to nonlinear element formulations including 2D continuum elements, beam element formulation 1, 3D shells (including isogeometric shells), 3D thick shells, 3D solids (including isogeometric solids), and SPH. See GISSMO Damage Model for details. The STOCHASTIC option allows spatially varying failure behavior. See *DEFINE_STOCHASTIC_VARIATION and *DEFINE_HAZ_PROPERTIES for additional information.

*MAT_ADD_DAMAGE_GISSMO originates from splitting *MAT_ADD_EROSION. Only "sudden" failure criteria without damage remain in *MAT_ADD_EROSION.

NOTE: Use *CONTROL_MAT to disable all *MAT_ADD_-DAMAGE_GISSMO commands in a model.

Card 1	1	2	3	4	5	6	7	8
Variable	MID		DTYP	REFSZ	NUMFIP	VOLFRAC		
Type	A		F	F	F	F		
Default	none		0.0	0.0	1.0	0.5		

Card 2	1	2	3	4	5	6	7	8
Variable	LCSDG	ECRIT	DMGEXP	DCRIT	FADEXP	LCREGD	INSTF	LCSOFT
Type	F	F	F	F	F	F	I	I
Default	0.0	0.0	1.0	0.0	1.0	0.0	0	0

Card 3	1	2	3	4	5	6	7	8
Variable	LCSRS	SHRF	BIAXF	LCDLIM	MIDFAIL	HISVN	SOFT	LP2BI
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

This card is optional.

Card 4	1	2	3	4	5	6	7	8
Variable	RGTR1	RGTR2						
Type	F	F						
Default	0.0	0.0						

VARIABLE	DESCRIPTION
MID	Material identification for which this erosion definition applies. A unique number or label must be specified (see *PART).
DTYP	DTYP is interpreted digit-wise as follows:

$$\text{DTYP} = [NM] = M + 10 \times N$$

M.EQ.0: Damage is accumulated, but there is no coupling to flow stress and no failure.

M.EQ.1: Damage is accumulated, and element failure occurs for $D = 1$. Coupling of damage to flow stress depending on parameters, see GISSMO Damage Model below.

N.EQ.0: Equivalent plastic strain is the driving quantity for the damage. (To be more precise, it's the history variable that LS-PrePost blindly labels as "plastic strain." What this history variable actually represents depends on the material model.)

N.GT.0: The Nth additional history variable is the driving quantity for damage. These additional history variables are the same ones flagged by the *DATABASE_EXTENT_-BINARY keyword's NEIPS and NEIPH fields. For example, for solid elements with *MAT_187, setting $N =$

VARIABLE	DESCRIPTION
	6 causes volumetric plastic strain to be the driving quantity for the GISSMO damage.
REFSZ	Reference element size for which an additional output of damage (and potentially plastic strain) will be generated. This is necessary to ensure the applicability of resulting damage quantities when transferred to different mesh sizes. GT.0: Reference size related damage values are written to history variables ND + 9 and ND + 10. These damage values are computed in the same fashion as the actual damage, just with the given reference element size. LT.0: The reference element size is |REFSZ|. In addition to the reference size related damage values, a corresponding plastic strain is computed and written to history variable ND + 17. See Remark 2.
NUMFIP	Number or percentage of failed integration points prior to element deletion (default value is 1). NUMFIP does not apply to higher order solid element types 24, 25, 26, 27, 28, and 29, rather see the variable VOLFRAC. Also, when the material is a composite defined with *PART_COMPOSITE with different materials through-the-thickness, do not use NUMFIP; use *DEFINE_ELEMENT_EROSION instead. GT.0.0: Number of integration points which must fail before element is deleted. LT.0.0: Applies only to shells. |NUMFIP| is the percentage of layers which must fail before an element is deleted. For shell formulations with 4 integration points per layer, the layer is considered failed if any of the integration points in the layer fails.
VOLFRAC	Volume fraction required to fail before element deletion. The default is 0.5. It is used for higher-order solid element types 24, 25, 26, 27, 28, and 29, and all isogeometric solids and shell elements. See Remark 3.
LCSDG	Failure strain curve/table or function: GT.0.0: Load curve ID or table ID. As a load curve, it defines equivalent plastic strain to failure as a function of triaxiality. As a table, it defines for each Lode parameter value (between -1 and 1) a load curve ID giving the equivalent plastic strain to failure as a function of

VARIABLE	DESCRIPTION
	triaxiality for that Lode parameter value. With HISVN ≠ 0, a 3D table can be used, where failure strain is a function of the history variable (TABLE_3D), Lode parameter (TABLE), and triaxiality (CURVE). With HISVN = 0, a 3D table introduces thermal effects, that is, failure strain is a function of temperature (TABLE_3D), Lode parameter (TABLE), and triaxiality (CURVE). As a 4D table, failure strain is a function of strain rate (TABLE_4D), temperature (TABLE_3D), Lode parameter (TABLE), and triaxiality (CURVE).
	LT.0.0: |LCSDG| is the ID of a function (*DEFINE_FUNCTION) with the arguments triaxiality η, Lode parameter L, plastic strain rate $\dot{\varepsilon}^p$, temperature T, history variable HISVN, and element size l_e: $f(\eta, L, \dot{\varepsilon}^p, T, \mathrm{HISVN}, l_e)$. Note that the sequence of the arguments is important, not their names.
ECRIT	Critical plastic strain (material instability); see below.
	LT.0.0: |ECRIT| is either a load curve ID defining critical equivalent plastic strain versus triaxiality or a table ID defining critical equivalent plastic strain as a function of triaxiality and Lode parameter (as in LCSDG). With HISVN ≠ 0, a 3D table can be used, where critical strain is a function of the history variable (TABLE_3D), Lode parameter (TABLE), and triaxiality (CURVE). With HISVN = 0, a 3D table introduces thermal effects, that is, critical strain is a function of temperature (TABLE_3D), Lode parameter (TABLE), and triaxiality (CURVE). As a 4D table, critical strain is a function of strain rate (TABLE_4D), temperature (TABLE_3D), Lode parameter (TABLE), and triaxiality (CURVE).
	EQ.0.0: Fixed value DCRIT defining critical damage is read (see below).
	GT.0.0: Fixed value for stress-state independent critical equivalent plastic strain
DMGEXP	Exponent for nonlinear damage accumulation; see GISSMO Damage Model and Remark 2.
DCRIT	Damage threshold value (critical damage). If a load curve of critical plastic strain or fixed value is given by ECRIT, input is ignored.

VARIABLE	DESCRIPTION
FADEXP	Exponent for damage-related stress fadeout: LT.0.0: |FADEXP| is a load curve ID or table ID. As a load curve it gives the fading exponent as a function of element size. As a table, it specifies the fading exponent as a function triaxiality (TABLE) and element size (CURVE). For 3D tables, it specifies the fading exponent as a function Lode parameter (TABLE_3D), triaxiality (TABLE), and element size (CURVE). GT.0.0: Constant fading exponent
LCREGD	Load curve ID or table ID defining element size dependent regularization factors for equivalent plastic strain to failure: GT.0.0: Load curve ID (regularization factor as a function of element size) or table ID (regularization factor vs. element size curves vs. effective rate) LT.0.0: |LCREGD| is a table ID (regularization factor vs. element size curves vs. triaxiality) or a 3D table ID (regularization factor as function of Lode parameter, triaxiality, and element size). This table provides an alternative to the use of SHRF and BIAXF for defining the effect of triaxiality on element size regularization of equivalent plastic strain to failure.
INSTF	Flag for governing the behavior of instability measure, F, and fading exponent, FADEXP (see GISSMO Damage Model): EQ.0: F is incrementally updated, and FADEXP, if from a table, is allowed to vary. EQ.1: F is incrementally updated, and FADEXP is kept constant after $F = 1$. EQ.2: F is only 0 or 1 (after ECRIT is reached), and FADEXP, if from a table, is allowed to vary. EQ.3: F is only 0 or 1 (after ECRIT is reached), and FADEXP is kept constant after $F = 1$.
LCSOFT	Load curve or table with ID |LCSOFT| giving the soft reduction factor for failure strain in crashfront elements. Crashfront elements are elements that are direct neighbors of failed (deleted) elements. A load curve specifies the soft reduction factor as a function of triaxiality. A table gives the soft reduction factor as a function of triaxiality (curve) and element size (table). The sign of LCSOFT

VARIABLE	DESCRIPTION
	determines which strains are scaled: EQ.0: Inactive GT.0: Plastic failure strain, ε_f (LCSDG), and critical plastic strain, $\varepsilon_{\mathrm{p,loc}}$ (ECRIT), will be scaled by LCSOFT. LT.0: Only plastic failure strain, ε_f (LCSDG), will be scaled by LCSOFT. SOFT is ignored when LCSOFT is active.
LCSRS	Load curve ID or table ID. Load curve ID defining failure strain scaling factor for LCSDG as a function of strain rate. If the first strain rate value in the curve is negative, it is assumed that all strain rate values are given as a natural logarithm of the strain rate. *The curve should not extrapolate to zero or failure may occur at low strain.* Table ID defining failure strain scaling factor as a function of strain rate (TABLE) and triaxiality (CURVE). GT.0: Scale ECRIT. LT.0: Do not scale ECRIT.
SHRF	Reduction factor for regularization for shear stress states. This parameter can be defined between -1.0 and +1.0. See Remark 1.
BIAXF	Reduction factor for regularization for biaxial stress states. This parameter can be defined between -1.0 and +1.0. See Remark 1.
LCDLIM	Load curve ID defining damage limit values as a function of triaxiality. Damage can be restricted to values less than 1.0 to prevent further stress reduction and failure for certain triaxialities.
MIDFAIL	Mid-plane failure option for shell elements. If active, then critical strain is only checked at the mid-plane integration point (IP), meaning an odd number for NIP should be used. The other integration points compute their damage, but no coupling to the stresses is done first. As soon as the mid-plane IP reaches ECRIT/DCRIT, then all the other IPs are also checked. EQ.0.0: Inactive. EQ.1.0: Active. Those of the non-mid-plane IPs that are already above their critical value immediately start to reduce the stresses. Those still below the critical value still do not couple, only if they reach their criterion. EQ.2.0: Active. All of the non-mid-plane IPs immediately start

VARIABLE	DESCRIPTION
	to reduce the stresses. NUMFIP is active. EQ.3.0: Active. Same as 2 but when D = 1 is reached in the middle integration point, the element is eroded instantaneously. NUMFIP is disregarded. EQ.4.0: Active. Damage and failure are applied only to the midpoint. When D = 1 at the midpoint, the element is eroded. NUMFIP is disregarded. Integration points away from the midplane see no stress reduction and no failure.
HISVN	History variable used to evaluate the 3D table LCSDG (and optionally 3D table ECRIT < 0): GT.0.0: Constant value LT.0.0: The constant value found at position |HISVN| where |HISVN| is the location in the history array of *INITIAL_STRESS_ SHELL/SOLID.
SOFT	Softening reduction factor for failure strain in crashfront elements. Crashfront elements are elements that are direct neighbors of failed (deleted) elements. EQ.0.0: Inactive GT.0.0: Plastic failure strain, ε_f (LCSDG), and critical plastic strain, $\varepsilon_{\mathrm{p,loc}}$ (ECRIT), will be scaled by SOFT. LT.0.0: Only plastic failure strain, ε_f (LCSDG), will be scaled by |SOFT|.
LP2BI	Option to use a bending indicator instead of the Lode parameter. If active (> 0), the expression “bending indicator” replaces the term “Lode parameter” everywhere in this manual page. We adopted the bending indicator from *MAT_258 (compare with variable Ω). LP2BI > 0 is only available for shell elements and requires NUMFIP = 1. EQ.0.0: Inactive. EQ.1.0: Active. Constant regularization (LCREGD) applied. EQ.2.0: Active. Regularization (LCRGED) fully applied under pure membrane loading (Ω = 0) but not at all under pure bending (Ω = 1). Linear interpolation in between.

VARIABLE	DESCRIPTION
RGTR1	First triaxiality value for optional "tub-shaped" regularization. See Remark 1.
RGTR2	Second triaxiality value for optional "tub-shaped" regularization. See Remark 1.

GISSMO Damage Model:

The GISSMO damage model is a phenomenological formulation that allows for an incremental description of damage accumulation, including softening and failure. It is intended to provide a maximum in variability for the description of damage for a variety of metallic materials, such as *MAT_024, *MAT_036, and *MAT_103. The input of parameters is based on tabulated data, allowing the user to directly convert test data to numerical input. The model was originally developed by Neukamm et al. [2008] and further investigated and enhanced by Effelsberg et al. [2012] and Andrade et al. [2014, 2016].

The model is based on an incremental formulation of damage accumulation:

$$\Delta D = \frac{\mathrm{DMGEXP} \times D^{\left(1-\frac{1}{\mathrm{DMGEXP}}\right)}}{\varepsilon_f} \Delta\varepsilon_p$$

where,

D Damage value $(0 \le D \le 1)$. For numerical reasons, D is initialized to a value of 10^{-20} for all damage types in the first time step.

ε_f Equivalent plastic strain to failure, determined from LCSDG as a function of the current triaxiality value η (and Lode parameter L as an option).

A typical failure curve LCSDG for metal sheet, modelled with shell elements is shown in Figure 2-5. Triaxiality should be monotonically increasing in this curve. A reasonable range for triaxiality is -2/3 to 2/3 if shell elements are used (plane stress).

For 3-dimensional stress states (solid elements), the possible range of triaxiality goes from $-\infty$ to $+\infty$, but to get a good resolution in the internal load curve discretization (depending on parameter LCINT of *CONTROL_SOLUTION) you should define lower limits, such as -1 to 1 if LCINT = 100 (default).

$\Delta\varepsilon_p$ Equivalent plastic strain increment

For constant values of failure strain, this damage rate can be integrated to get a relation of damage and actual equivalent plastic strain:

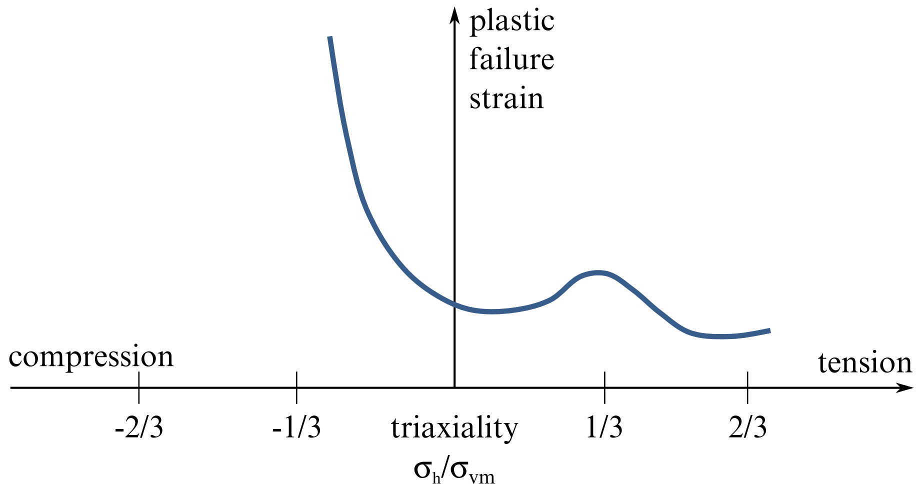


Figure 2-5. Typical failure curve for metal sheet, modeled with shell elements.

$$D = \left(\frac{\varepsilon_p}{\varepsilon_f}\right)^{\mathrm{DMGEXP}}, \qquad \text{for } \varepsilon_f = \text{constant}$$

Triaxiality, η, as a measure of the current stress state is defined as

$$\eta = \frac{\sigma_H}{\sigma_M} ,$$

with hydrostatic stress, σ_H, and von Mises stress, σ_M. Lode parameter L as an additional measure of the current stress state is defined as

$$L = \frac{27}{2}\frac{J_3}{{\sigma_M}^3} ,$$

with third invariant of the stress deviator, J_3.

For DTYP = 0, damage is accumulated according to the description above, yet no softening and failure is taken into account. Thus, parameters ECRIT, DCRIT and FADEXP will not have any influence. This option can be used to calculate pre-damage in multi-stage deformations without influencing the simulation results.

For DTYPE = 1, elements will be deleted if $D \geq 1$.

Depending on the set of parameters given by ECRIT (or DCRIT) and FADEXP, a Lemaitre-type coupling of damage and stress *(effective stress concept)* can be used.

To define damage, use one of the following three principal ways:

1. Input of a fixed value of critical plastic strain (ECRIT > 0.). As soon as the magnitude of plastic strain reaches this value, the current damage parameter D is stored as critical damage DCRIT and the damage coupling flag is set to unity, in order to facilitate an identification of critical elements in post-processing. From this point on, damage is coupled to the stress tensor using the following relation:

$$\sigma = \tilde{\sigma}\left[1 - \left(\frac{D - \mathrm{DCRIT}}{1 - \mathrm{DCRIT}}\right)^{\mathrm{FADEXP}}\right]$$

This leads to a continuous reduction of stress, up to the load-bearing capacrmarkrity completely vanishing as D reaches unity. The fading exponent FADEXP can be element size dependent to allow for the consideration of an element-size dependent amount of energy to be dissipated during element fade-out.

2. Input of a load curve defining critical plastic strain as a function of triaxiality (ECRIT < 0.), pointing to load curve ID |ECRIT|. This allows for a definition of triaxiality-dependent material instability, which takes account of instability and localization occurring depending on the actual load case. One possibility is the use of instability curves predicted by instability models (e.g., Swift, Hill, Marciniak-Kuczynski, etc.). Another possibility is the use of a transformed Forming Limit Diagram as an input for the expected onset of softening and localization. Using this load curve, the instability measure F is accumulated using the following relation, which is similar to the accumulation of damage D except for the instability curve is used as an input:

$$\Delta F = \frac{\mathrm{DMGEXP}}{\varepsilon_{p,loc}} F^{\left(1-\frac{1}{\mathrm{DMGEXP}}\right)} \Delta\varepsilon_p$$

with,

F	Instability measure ($0 \le F \le 1$).
$\varepsilon_{\mathrm{p,loc}}$	Equivalent plastic strain to instability, determined from ECRIT
$\Delta\varepsilon_p$	Equivalent plastic strain increment

As soon as the instability measure F reaches unity, the current value of damage D in the respective element is stored. Damage will from this point on be coupled to the flow stress using the relation described above.

3. If no input for ECRIT is made, parameter DCRIT will be considered. Coupling of damage to the stress tensor starts if this value *(damage threshold)* is exceeded ($0 \le \mathrm{DCRIT} \le 1$). Coupling of damage to stress is done using the relation described above.

This input allows for the use of extreme values also – for example, DCRIT = 1.0 would lead to no coupling at all, and element deletion under full load (brittle fracture).

Remarks:

1. **Regularization.** The values of SHRF and BIAXF generally lie between 0.0 and 1.0 where 0.0 means full regularization and 1.0 means no regularization under shear (triaxiality = 0.0 for SHRF = 1.0) or biaxial tension (triaxiality = 2/3 for BIAXF = 1.0). Any other intermediate triaxiality follows a linear interpolation between triaxiality 0.0 and 1/3 and also between triaxiality 1/3 and 2/3. Notice that a full regularization is always for a one-dimensional tensile stress state (triaxiality = 1/3) according to the factors defined under LCREGD (see the next paragraph for an exception to this restriction). For the sake of generalization, both SHRF and BIAXF can also assume negative values (e.g., SHRF=-1.0 and BIAXF=-1.0). In this case, regularization is affected not at triaxialities 0.0 and 2/3 but rather at the triaxialities where the failure curve (LCSDG) crosses the instability curve (-ECRIT). The use of a triaxiality-dependent regularization approach may be necessary because simple regularization only depending on the element size can be unrealistic for certain stress states.

   The restriction of a full regularization at triaxiality = 1/3 can be lifted with the optional parameters RGTR1 and RGTR2. As shown in Figure 2-6, full regularization starts at RGTR1 and ends at RGTR2. A linear interpolation is used between 0.0 and RGTR1 and between RGTR2 and 2/3. Together with SHRF = BIAXF = 1 this gives a trapezoidal-(or tub-)shaped regularization. This seems to be a reasonable approach in many cases and is therefore easily accessible now.

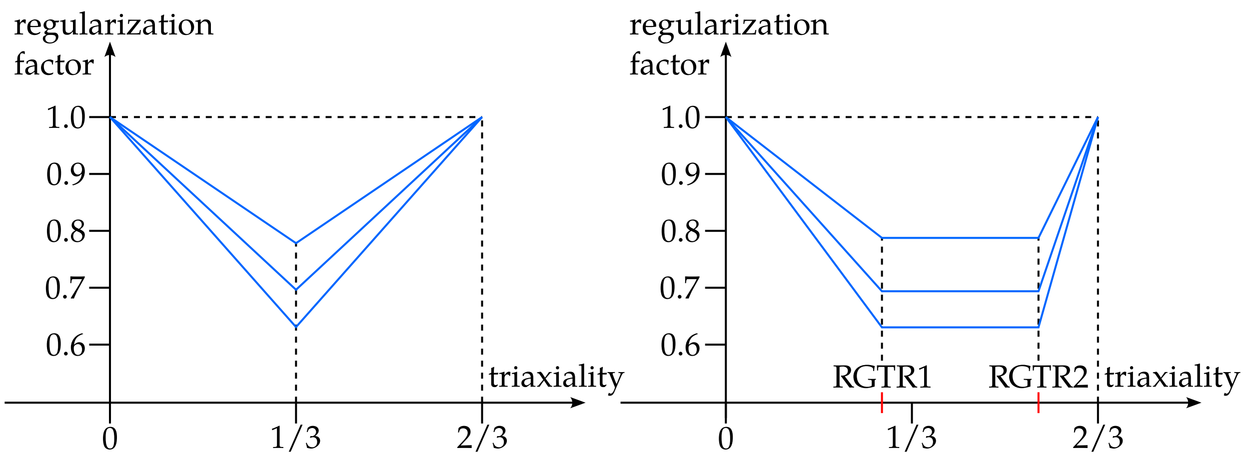


Figure 2-6. The left figure provides an example of the regularization curves produced with an LCREGD curve for three different element sizes. SHRF and BIAXF are both 1.0 in this case. The right figure illustrates how these curves become tub-shaped by additionally defining RGTR1 and RGTR2.

2. **Reference element size.** If the results of a first simulation should be transferred to a second computation with potentially modified mesh size, such as mapping from forming to crash, it might be necessary to alter damage values

(and maybe plastic strain) as well. For that purpose, reference element size REFSZ can be defined. With REFSZ > 0, corresponding damage is computed in the same fashion as the actual damage, just with the given reference element size instead, and written to history variable ND + 9. An alternative approach is available with the definition of REFSZ < 0. In that case, a plastic strain with regard to |REFSZ| is computed first:

$$\Delta\varepsilon_p^{|\mathrm{REFSZ}|} = \Delta\varepsilon_p \frac{\varepsilon_p^f(|\mathrm{REFSZ}|) - \varepsilon_p^{\mathrm{ECRIT}}}{\varepsilon_p^f(l_e) - \varepsilon_p^{\mathrm{ECRIT}}} \qquad (\text{if } F \geq 1)$$

The accumulated value of that is written to history variable ND + 17. Afterwards, damage with respect to the |REFSZ| is computed similarly to the standard damage accumulation, only using this new reference plastic strain:

$$\Delta D^{|\mathrm{REFSZ}|} = \frac{\mathrm{DMGEXP}\times\left(D^{|\mathrm{REFSZ}|}\right)^{\left(1-\frac{1}{\mathrm{DMGEXP}}\right)}}{\varepsilon_p^f(|\mathrm{REFSZ}|)} \Delta\varepsilon_p^{|\mathrm{REFSZ}|}$$

This “reference damage” is stored on history variable ND + 9.

3. **VOLFRAC.** The volumes associated with individual integration points in higher order finite elements and isogeometric elements varies widely. Thus, the number of failed integration points is not a reliable criterion for determining element failure. To obtain a more stable and consistent response, LS-DYNA uses the volume fraction of the failed material for these types of elements.

4. **History Variable.** History variables of the GISSMO damage model are written to the post-processing database. Therefore, NEIPH and NEIPS must be set in *DATABASE_EXTENT_-BINARY. The damage history variables start at position ND, which is displayed in d3hsp file as, for example, “first damage history variable = 6” which means that ND = 6. For example, if you wish to view the damage parameter (first GISSMO history variable) for a *MAT_024 shell element, you must set NEIPS = 6. In LS-PrePost, you access the damage parameter as history variable #6.

History Variable #	Description
ND	Damage parameter D ($10^{-20} < D \leq 1$)
ND + 1	Damage threshold DCRIT
ND + 2	Domain flag for damage coupling (0: no coupling, 1: coupling)
ND + 3	Triaxiality variable, $\eta = \sigma_H/\sigma_M$
ND + 4	Equivalent plastic strain

History Variable #	Description
ND + 5	Regularization factor for failure strain (determined from LCREGD)
ND + 6	Exponent for stress fading FADEXP
ND + 7	Calculated element size, l_e
ND + 8	Instability measure F
ND + 9	Resultant damage parameter D for element size REFSZ
ND + 10	Resultant damage threshold DCRIT for element size REFSZ
ND + 11	Averaged triaxiality: $$\eta_{n+1}^{\text{avg}} = \frac{1}{D_{n+1}} \left(D_n \times \eta_n^{\text{avg}} + (D_{n+1} - D_n) \times \eta_{n+1}\right)$$
ND + 12	Lode parameter value L (only calculated if LCSDG refers to a table)
ND + 13	Alternative damage value: $D^{1/\text{DMGEXP}}$
ND + 14	Averaged Lode parameter: $$L_{n+1}^{\text{avg}} = \frac{1}{D_{n+1}} \left(D_n \times L_n^{\text{avg}} + (D_{n+1} - D_n) \times L_{n+1}\right)$$
ND + 15	MIDFAIL control flag (set to -1 in case mid-plane IP reaches ECRIT/DCRIT)
ND + 16	Number of IPs/layers (NUMFIP > 0/< 0) that must fail before an element gets deleted
ND + 17	Plastic strain value related to reference element size (only if REFSZ < 0)
ND + 18	Effective damage value (stress scaling factor)
ND + 19	History variable for 3D table LCSDG (only if HISVN ≠ 0)
ND + 20	Random scale factor on failure strain (only if STOCHASTIC option is used)

*MAT_ADD_EROSION

Many of the constitutive models in LS-DYNA do not allow failure and erosion. The ADD_EROSION option provides a way of including failure in these models. This option can also be applied to constitutive models that already include other failure/erosion criteria.

LS-DYNA applies each of the failure criteria defined here independently. Upon satisfaction of a sufficient number of the specified criteria (see NCS on Card 1), LS-DYNA deletes the element from the calculation.

This keyword applies to nonlinear element formulations, including 2D continuum elements, beam formulations 1 and 11, 3D shell elements (including isogeometric shells), 3D thick shell elements, 3D solid elements (including isogeometric solids), and SPH.

Damage models GISSMO and DIEM are still available using IDAM on Card 3 for backward compatibility. The keywords *MAT_ADD_DAMAGE_DIEM and *MAT_ADD_-DAMAGE_GISSMO are preferable methods for adding damage. A combination of *MAT_ADD_EROSION failure criteria with damage from *MAT_ADD_DAMAGE_-DIEM/GISSMO is possible as long as IDAM = 0 is used.

> **NOTE:** To disable all *MAT_ADD_EROSION commands in a model, use *CONTROL_MAT.

Card Summary:

Card 1. This card is required.

MID	EXCL	MXPRES	MNEPS	EFFEPS	VOLEPS	NUMFIP	NCS

Card 2. This card is required.

MNPRES	SIGP1	SIGVM	MXEPS	EPSSH	SIGTH	IMPULSE	FAILTM

Card 3. This card is optional.

IDAM							LCREGD

Card 4. This card is optional.

LCFLD	NSFF	EPSTHIN	ENGCRT	RADCRT	LCEPS12	LCEPS13	LCEPSMX

Card 5. This card is optional.

DTEFLT	VOLFRAC	MXTMP	DTMIN				

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	EXCL	MXPRES	MNEPS	EFFEPS	VOLEPS	NUMFIP	NCS
Type	A	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	0.0	0.0	1.0	1.0/0.0

VARIABLE	DESCRIPTION
MID	Material identification for which this erosion definition applies. A unique number or label must be specified (see *PART).
EXCL	The exclusion number (default value of 0.0 is recommended). For any failure value in *MAT_ADD_EROSION which is set to this exclusion number, the associated failure criterion is not invoked. Or in other words, only the failure values which are not set to the exclusion number are invoked. The default value of EXCL (0.0) eliminates from consideration any failure criterion whose failure value is left blank or set to 0.0. As an example, to prevent a material from developing tensile pressure, you could specify an unusual value for the exclusion number, such as 1234, set MNPRES to 0.0, and set all the other failure values in *MAT_ADD_EROSION to 1234. However, use of an exclusion number in this way is nonessential since the same effect could be achieved without use of the exclusion number by setting MNPRES to a very small negative value and leaving all the other failure values blank (or set to zero).
MXPRES	Maximum pressure at failure, $P_{\max}$. If the value is exactly zero, it is automatically excluded to maintain compatibility with old input files.
MNEPS	Minimum principal strain at failure, $\varepsilon_{\min}$. If the value is exactly zero, it is automatically excluded to maintain compatibility with old input files.
EFFEPS	Maximum effective strain at failure: $$\varepsilon_{\mathrm{eff}} = \sum_{ij} \sqrt{\frac{2}{3}\varepsilon_{ij}^{\mathrm{dev}}\varepsilon_{ij}^{\mathrm{dev}}}.$$

VARIABLE	DESCRIPTION
	If the value is exactly zero, it is automatically excluded to maintain compatibility with old input files. If the value is negative, then |EFFEPS| is the effective plastic strain at failure. In combination with cohesive elements, EFFEPS is the maximum effective in-plane strain.
VOLEPS	Volumetric strain at failure, $$\varepsilon_{\mathrm{vol}} = \varepsilon_{11} + \varepsilon_{22} + \varepsilon_{33} \ ,$$ or $$\ln(\text{relative volume}) \ .$$ VOLEPS can be a positive or negative number depending on whether the failure is in tension or compression, respectively. If the value is exactly zero, it is automatically excluded to maintain compatibility with old input files.
NUMFIP	Number or percentage of failed integration points prior to element deletion (default is 1). See Remark 2. NUMFIP does not apply to higher order solid element types 24, 25, 26, 27, 28, and 29, rather see the variable VOLFRAC. Also, when the material is a composite defined with *PART_COMPOSITE with different materials through-the-thickness, this field should not be used; use *DEFINE_ELEMENT_EROSION instead. GT.0.0: Number of integration points which must fail before element is deleted LT.0.0: Applies only to shells. |NUMFIP| is the percentage of integration points which must exceed the failure criterion before the element fails. If NUMFIP < -100, then |NUMFIP|-100 is the number of failed integration points prior to element deletion.
NCS	Number of failure conditions to satisfy before failure occurs. For example, if SIGP1 and SIGVM are defined and if NCS = 2, both failure criteria must be met before element deletion can occur. The default is set to unity.

Card 2	1	2	3	4	5	6	7	8
Variable	MNPRES	SIGP1	SIGVM	MXEPS	EPSSH	SIGTH	IMPULSE	FAILTM
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
MNPRES	Minimum pressure at failure, P_{min}.
SIGP1	Maximum principal stress at failure, σ_{max} LT.0: -SIGP1 is a load curve ID giving the maximum principal stress at failure as a function of the effective strain rate (the curve should not extrapolate to zero or failure may occur at low strain). A filter can be applied to the effective strain rate according to DTEFLT (see Card 5).
SIGVM	Equivalent stress at failure, $\bar{\sigma}_{\mathrm{max}}$ LT.0: -SIGVM is a load curve ID giving the equivalent stress at failure as a function of the effective strain rate (the curve should not extrapolate to zero or failure may occur at low strain). A filter can be applied to the effective strain rate according to DTEFLT (see Card 5).
MXEPS	Variable to invoke a failure criterion based on maximum principal strain. GT.0.0: Maximum principal strain at failure, $\varepsilon_{\mathrm{max}}$ LT.0.0: -MXEPS is the ID of a curve giving maximum principal strain at failure as a function of effective strain rate (the curve should not extrapolate to zero or failure may occur at low strain). A filter is applied to the effective strain rate according to DTEFLT (see Card 5).
EPSSH	Tensorial shear strain at failure, $\gamma_{\mathrm{max}}/2$
SIGTH	Threshold stress, σ_0
IMPULSE	Stress impulse for failure, K_{f}

VARIABLE	DESCRIPTION
FAILTM	Failure time. When the problem time exceeds the failure time, the material is removed. GT.0: Failure time is active during any phase of the analysis. LT.0: Failure time is set to |FAILTM|. This criterion in inactive during the dynamic relaxation phase.

Damage Model Card. The following card is optional.

Card 3	1	2	3	4	5	6	7	8
Variable	IDAM							LCREGD
Type	A8							F
Default	0.0							0.0

VARIABLE	DESCRIPTION
IDAM	Flag for damage model. EQ.0: No damage model is used. NE.0: Damage models GISSMO or DIEM, see manuals of R10 and before. Still available here for backward compatibility (see preferred keywords *MAT_ADD_DAMAGE_-DIEM/GISSMO as of R11).
LCREGD	Load curve ID defining element size dependent regularization factors. This feature can be used with the standard failure criteria of Cards 1 (MXPRES, MNEPS, EFFEPS, VOLEPS), 2 (MNPRES, SIGP1, SIGVM, MXEPS, EPSSH, IMPULSE) and 4 (LCFLD, EPSTHIN).

Additional Failure Criteria Card. This card is optional.

Card 4	1	2	3	4	5	6	7	8
Variable	LCFLD	NSFF	EPSTHIN	ENGCRT	RADCRT	LCEPS12	LCEPS13	LCEPSMX
Type	F	F	F	F	F	I	I	I
Default	0.0	0.0	0.0	0.0	0.0	0	0	0

VARIABLE	DESCRIPTION
LCFLD	Load curve ID or table ID. Load curve defines the Forming Limit Diagram, where minor engineering strains in percent are defined as abscissa values and major engineering strains in percent are defined as ordinate values. Table defines for each strain rate (LCFLD > 0) or for each shell thickness (LCFLD < 0) an associated FLD curve. The forming limit diagram is shown in Figure M39-1. When defining the curve, list pairs of minor and major strains starting with the left most point and ending with the right most point. This criterion is only available for shell elements.
NSFF	Number of explicit time step cycles for stress fade-out used in the LCFLD criterion. Default is 10.
EPSTHIN	Thinning strain at failure for thin and thick shells. GT.0.0: Individual thinning for each integration point from z-strain LT.0.0: Averaged thinning strain from element thickness change
ENGCRT	Critical energy for nonlocal failure criterion; see Remark 1i below.
RADCRT	Critical radius for nonlocal failure criterion; see Remark 1i below.
LCEPS12	Load curve ID defining in-plane shear strain limit γ_{12}^c as a function of element size. See Remark 1j.
LCEPS13	Load curve ID defining through-thickness shear strain limit γ_{13}^c as a function of element size. See Remark 1j.
LCEPSMX	Load curve ID defining in-plane major strain limit ε_1^c as a function of element size. See Remark 1j.

Additional Failure Criteria Card. This card is optional.

Card 5	1	2	3	4	5	6	7	8
Variable	DTEFLT	VOLFRAC	MXTMP	DTMIN				
Type	F	F	F	F				
Default	↓	0.5	none	none				

VARIABLE	DESCRIPTION
DTEFLT	The time period (or inverse of the cutoff frequency) for the low-pass filter applied to the effective strain rate when SIGP1, SIGVM, or MXEPS is negative. If DTEFLT is set to zero or left blank, no filtering of the effective strain rate is performed.
VOLFRAC	The volume fraction required to fail before the element is deleted. The default is 0.5. It is used for higher order solid element types 24, 25, 26, 27, 28, and 29, and all isogeometric solids and shell elements. See Remark 4.
MXTMP	Maximum temperature at failure
DTMIN	Minimum time step size at failure

Remarks:

1. **Failure Criteria.** In addition to failure time, supported criteria for failure are:

   a) $P \geq P_{max}$, where P is the pressure (positive in compression), and P_{max} is the maximum pressure at failure

   b) $\varepsilon_3 \leq \varepsilon_{min}$, where ε_3 is the minimum principal strain, and ε_{min} is the minimum principal strain at failure

   c) $P \leq P_{min}$, where P is the pressure (positive in compression), and P_{min} is the minimum pressure at failure

   d) $\sigma_1 \geq \sigma_{max}$, where σ_1 is the maximum principal stress, and σ_{max} is the maximum principal stress at failure

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

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

g) $\gamma_1 \geq \gamma_{\max}/2$, where γ_1 is the maximum tensorial shear strain = $(\varepsilon_1 - \varepsilon_3)/2$, and $\gamma_{\max}$ is the engineering shear strain at failure

h) The Tuler-Butcher criterion,

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

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

i) A nonlocal failure criterion which is mainly intended for windshield impact can be defined using ENGCRT, RADCRT, and one additional "main" failure criterion (only SIGP1 is available at the moment). All three parameters should be defined for one part, namely, the windshield glass, and the glass should be discretized with shell elements. The course of events of this nonlocal failure model is as follows: If the main failure criterion SIGP1 is fulfilled, the corresponding element is flagged as the center of impact, but no element erosion takes place yet. Then, the internal energy of shells inside a circle, defined by RADCRT, around the center of impact is tested against the product of the given critical energy ENGCRT and the "area factor". The area factor is defined as,

$$\text{Area Factor} = \frac{\text{total area of shell elements found inside the circle}}{2\pi \times \text{RADCRT}^2}$$

The reason for having two times the circle area in the denominator is that we expect two layers of shell elements, as would typically be the case for laminated windshield glass. If this energy criterion is exceeded, all elements of the part are now allowed to be eroded by the main failure criterion.

Up through version R14.0, this nonlocal energy criterion could only be used once in an LS-DYNA model. This was based on the assumption that one calculates the head impact on a glass pane, where both pane layers (inner and outer) were united in one part. The factor "2" in the above formula comes from this assumption.

In subsequent versions (R14.1, R15, ...). it is possible to define the energy criterion for each part separately, meaning as often as desired in a model. The criterion can be defined with either *MAT_ADD_EROSION or *MAT_-

280. This could be used, for example, to assign different values for ENGCRT and RADCRT to the inner and outer glass layers or in even more general cases. We kept the factor “2” in the formula for the Area Factor to not falsify old results.

j) An element size dependent mixed-mode fracture criterion (MMFC) can be defined for shell elements using load curves LCEPS12, LCEPS23, and LCEPSMX. Failure happens if NCS (see Card 1) of these three criteria are met

LCEPS12:	$\gamma_{12} = \frac{1}{2}(\varepsilon_1 - \varepsilon_2) \geq \gamma_{12}^c(l_e)$	if	$-2.0 \leq \varepsilon_2/\varepsilon_1 \leq -0.5$
LCEPS13:	$\gamma_{13} = \frac{1}{2}(\varepsilon_1 - \varepsilon_3) \geq \gamma_{13}^c(l_e)$	if	$-0.5 \leq \varepsilon_2/\varepsilon_1 \leq 1.0$
LCEPSMX:	$\varepsilon_1 \geq \varepsilon_1^c(l_e)$	if	$-0.5 \leq \varepsilon_2/\varepsilon_1 \leq 1.0$

where γ_{12} and γ_{13} are in-plane and through-thickness shear strains, ε_1 and ε_2 are in-plane major and minor strains, and ε_3 is the through-thickness strain. The characteristic element size is l_e and it is computed as the square root of the shell element area. More details can be found in Zhu & Zhu (2011).

2. **NUMFIP.** Element erosion depends on the type of element and the value of NUMFIP.

   a) When NUMFIP > 0, elements erode when NUMFIP points fail.

   b) For shells only, when -100 ≤ NUMFIP < 0, elements erode when |NUMFIP| percent of the integration points fail.

   c) For shells only, when NUMFIP < -100, elements erode when |NUMFIP| – 100 integration points fail.

   For NUMFIP > 0 and -100 ≤ NUMFIP < 0, layers retain full strength until the element is eroded. For NUMFIP < -100, the stress at an integration point immediately drops to zero when failure is detected at that integration point.

3. **Instability.** If the keyword *DEFINE_MATERIAL_HISTORIES is used to output the instability, the following table gives a summary of the output properties. Currently only failure values based on the first two cards of this keyword are supported but others can be added on request; for the unsupported options the output will be zero. The instability value is defined as the quantity of interest divided by its corresponding upper limit (restricted to be positive).

*DEFINE_MATERIAL_HISTORIES Properties					
Label	Attributes				Description
Instability	-	-	-	-	Maximum of the ones listed below
Instability	-1	-	-	-	P/MXPRES
Instability	-2	-	-	-	ε_3/MNEPS
Instability	-3	-	-	-	ε_p/EFFEPS
Instability	-4	-	-	-	$\varepsilon_{\mathrm{vol}}$/VOLEPS
Instability	-5	-	-	-	P/MNPRES
Instability	-6	-	-	-	σ_1/SIGP1
Instability	-7	-	-	-	$\sqrt{\frac{3}{2}\sigma'_{ij}\sigma'_{ij}}$/SIGVM
Instability	-8	-	-	-	ε_1/MXEPS
Instability	-9	-	-	-	γ_1/EPSSH
Instability	-10	-	-	-	$\int_0^t [\max(0, \sigma_1 - \sigma_0)]^2 dt$/IMPULSE
Instability	-12	-	-	-	t/FAILTM

4. **VOLFRAC.** The volumes associated with individual integration points in higher order finite elements and isogeometric elements varies widely. Thus, the number of failed integration points is not a reliable criterion for determining element failure. To obtain a more stable and consistent response, LS-DYNA uses the volume fraction of the failed material for these types of elements.

*MAT_ADD_EXTVAR_EXPANSION

The ADD_EXTVAR_EXPANSION option adds an expansion property to an (arbitrary) material model in LS-DYNA. The expansion is controlled by the state of an external variable defined with *LOAD_EXTERNAL_VARIABLE. This option currently applies to hypoelastic material models. It is supported for solid element types -2, -1, 1, 2, and 10 and shell element types -16, 2, and 16.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCID	MULT	LCIDY	MULTY	LCIDZ	MULTZ	IDEV
Type	I	I	F	I	F	I	F	I
Default	none	0	1.0	LCID	MULT	LCID	MULT	0

VARIABLE	DESCRIPTION
PID	Part ID for which the expansion property applies
LCID	For isotropic material models, LCID is the load curve ID defining the expansion coefficient, $\gamma(\alpha)$, as a function of the external variable, α. In this case, LCIDY, MULTY, LCIDZ, and MULTZ are ignored. For anisotropic material models, LCID and MULT define the expansion coefficient in the local material a-direction. In either case, if LCID is zero, the expansion coefficient is constant and equal to MULT.
MULT	Scale factor scaling load curve given by LCID
LCIDY	Load curve ID defining the expansion coefficient in the local material b-direction as a function of the external variable. If zero, the expansion coefficient in the local material b-direction is constant and equal to MULTY. If MULTY = 0.0 as well, LCID and MULT specify the expansion coefficient in the local material b-direction.
MULTY	Scale factor scaling load curve given by LCIDY
LCIDZ	Load curve ID defining the expansion coefficient in the local material c-direction as a function of the external variable If zero, the expansion coefficient in the local material c-direction is constant and equal to MULTZ. If MULTZ = 0.0 as well, LCID and MULT specify the expansion coefficient in the local material c-direction.

VARIABLE	DESCRIPTION
MULTZ	Scale factor scaling load curve given by LCIDZ
IDEV	External variable ID

Remarks:

When invoking the isotropic external variable expansion property (no local y and z parameters) for a material, the stress update is based on the elastic strain rates $\dot{\varepsilon}_{ij}^{e}$. Those are calculated based on the total strain rate $\dot{\varepsilon}_{ij}$, the value α of the external variable and its rate $\dot{\alpha}$:

$$\dot{\varepsilon}_{ij}^{e} = \dot{\varepsilon}_{ij} - \gamma(\alpha)\dot{\alpha} \times \delta_{ij}$$

with expansion coefficient γ.

For orthotropic properties, which apply only to materials with anisotropy, this equation is generalized to

$$\dot{\varepsilon}_{ij}^{e} = \dot{\varepsilon}_{ij} - \gamma_k(\alpha)\dot{\alpha}\, q_{ik}q_{jk} \quad .$$

Here q_{ij} represents the matrix with material directions with respect to the current configuration.

MAT_ADD_FATIGUE_*{OPTION}

Available options include:

<BLANK>

EN

The ADD_FATIGUE option defines the S-N or the E-N (with option EN) fatigue property of a material model.

Card Summary:

Card 1a. This card is included if and only if no keyword option (<BLANK>) is used and LCID > 0.

MID	LCID	LTYPE				SNLIMT	SNTYPE

Card 1b. This card is included if and only if no keyword option (<BLANK>) is used and LCID < 0.

MID	LCID	LTYPE	A	B	STHRES	SNLIMT	SNTYPE

Card 1c. This card is included if and only if the keyword option EN is used.

MID	KP	NP	SIGMAF	EPSP	BP	CP	

Card 2a. This card is read if no keyword option (<BLANK>) is used and LCID < 0. Include one card for each additional S-N curve segment. Between zero and seven of these cards may be included in the deck. This input ends at the next keyword ("*") card.

			A*i*	B*i*	STHRES*i*		

Card 2b. This card is read if the keyword option EN is used. Card 2b is not needed if E and PR have been defined in the original material card.

E	PR						

Data Card Definitions:

Card 1a	1	2	3	4	5	6	7	8
Variable	MID	LCID	LTYPE				SNLIMT	SNTYPE
Type	A	I	I				I	I
Default	none	-1	0				0	0

VARIABLE	DESCRIPTION
MID	Material ID for which the fatigue property applies
LCID	S-N fatigue curve ID: GT.0: S-N fatigue curve ID
LTYPE	Type of S-N curve: EQ.0: Semi-log interpolation (default) EQ.1: Log-log interpolation EQ.2: Linear-linear interpolation
SNLIMT	SNLIMNT determines the algorithm used when stress is lower than the lowest stress on S-N curve. EQ.0: Use the life at the last point on S-N curve EQ.1: Extrapolation from the last two points on S-N curve EQ.2: Infinity
SNTYPE	Stress type of S-N curve: EQ.0: Stress range (default) EQ.1: Stress amplitude

Card 1b	1	2	3	4	5	6	7	8
Variable	MID	LCID	LTYPE	A	B	STHRES	SNLIMT	SNTYPE
Type	A	I	I	F	F	F	I	I
Default	none	-1	0	0.0	0.0	none	0	0

VARIABLE	DESCRIPTION
MID	Material ID for which the fatigue property applies
LCID	S-N fatigue curve ID: EQ.-1: S-N fatigue curve uses equation $NS^b = a$ EQ.-2: S-N fatigue curve uses equation $\log(S) = a - b\log(N)$ EQ.-3: S-N fatigue curve uses equation $S = a\,N^b$ EQ.-4: S-N fatigue curve uses equation $S = a - b\log(N)$
LTYPE	Type of S-N curve: EQ.0: Semi-log interpolation (default) EQ.1: Log-log interpolation EQ.2: Linear-linear interpolation
A	Material parameter a in S-N fatigue equation
B	Material parameter b in S-N fatigue equation
STHRES	Fatigue threshold stress
SNLIMT	SNLIMIT determines the algorithm used when stress is lower than STHRES. EQ.0: Use the life at STHRES EQ.1: *Ignored* EQ.2: Infinity
SNTYPE	Stress type of S-N curve. EQ.0: Stress range (default) EQ.1: Stress amplitude

Card 1c	1	2	3	4	5	6	7	8
Variable	MID	KP	NP	SIGMAF	EPSP	BP	CP	
Type	A	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
MID	Material identification for which the fatigue property applies
KP	K', the cyclic strength coefficient
NP	N', the cyclic strain hardening exponent
SIGMAP	σ_f', the fatigue strength coefficient
EPSP	ε_f', the fatigue ductility coefficient
BP	b', the fatigue strength exponent (Basquin's exponent)
CP	c', the fatigue ductility exponent (Coffin-Manson exponent)

S-N Curve Segment Cards. Include one card for each additional S-N curve segment. Between zero and seven of these cards may be included in the deck. This input ends at the next keyword ("*") card.

Card 2a	1	2	3	4	5	6	7	8
Variable				A*i*	B*i*	STHRES*i*		
Type				F	F	F		
Default				0.0	0.0	none		

VARIABLE	DESCRIPTION
A*i*	Material parameter a in S-N fatigue equation for the i^{th} segment
B*i*	Material parameter b in S-N fatigue equation for the i^{th} segment

VARIABLE	DESCRIPTION
STHRES*i*	Fatigue threshold stress for the i^{th} segment which acts as the lower stress limit of that segment

Card 2b	1	2	3	4	5	6	7	8
Variable	E	PR						
Type	I	F						
Default	none	none						

VARIABLE	DESCRIPTION
E	Young's modulus
PR	Poisson's ratio

Remarks:

1. **S-N Curves.** For fatigue analysis based on stress (OPTION = <BLANK>), S-N curves can be defined by *DEFINE_CURVE or by a predefined equation. When they are defined by curves, the abscissa values (the first column under *DEFINE_CURVE) represent N (number of cycles to failure) and the ordinate values (2nd column under *DEFINE_CURVE) represent S (stress). There are 4 different predefined equations:

   a) *LCID* = *-1*:

   $$NS^b = a$$

   b) *LCID* = *-2*:

   $$\log(S) = a - b\log(N)$$

   c) *LCID* = *-3*:

   $$S = a\,N^b$$

   d) *LCID* = *-4*:

   $$S = a - b\log(N)$$

   Here N is the number of cycles for fatigue failure and S is the stress amplitude. Note that the two equations can be converted to each other, with some minor algebraic manipulation on the constants a and b.

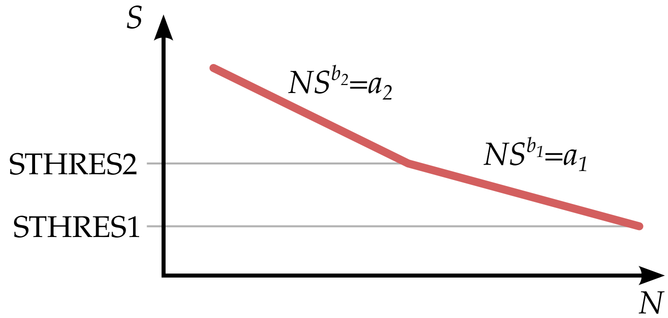


Figure 2-7. S-N Curve having multiple slopes

To define an S-N curve with multiple slopes, the S-N curve can be split into multiple segments with each segment defined by a set of parameters A*i*, B*i* and STHRES*i*. Up to 8 sets of the parameters (A*i*, B*i* and STHRES*i*) can be defined. The lower limit of the i^{th} segment is represented by the threshold stress STHRES*i*, as shown in Figure 2-7. This only applies to the case where LCID < 0.

2. **Related Keywords.** This model is applicable to frequency domain fatigue analysis, defined by the keywords: *FREQUENCY_DOMAIN_RANDOM_VIBRATION_FATIGUE and *FREQUENCY_DOMAIN_SSD_FATIGUE. It also applies to time domain fatigue analysis, defined by the keyword *FATIGUE (see these keywords for further details).

3. **Strain-Based Fatigue.** For fatigue analysis based on strain (OPTION = EN), the cyclic stress-strain curve is defined by

$$\varepsilon = \frac{\sigma}{E} + \left(\frac{\sigma}{K'}\right)^{\frac{1}{n'}} \quad .$$

The relationship between true local strain amplitude and endurance is

$$\frac{\Delta\varepsilon}{2} = \frac{\sigma_f'}{E}(2N)^{b'} + \varepsilon_f'(2N)^{c'} \quad .$$

*MAT_ADD_GENERALIZED_DAMAGE

This option provides a way of including generalized (tensor type) damage and failure in standard LS-DYNA material models. The basic idea is to apply a general damage model (e.g. GISSMO) using several history variables as damage driving quantities at the same time. With this feature it may be possible to obtain, for example, anisotropic damage behavior or separate stress degradation for volumetric and deviatoric deformations. A maximum of three simultaneous damage evolutions (meaning definition of 3 history variables) is possible. A detailed description of this model can be found in Erhart et al. [2017].

This option currently applies to shell element types 1, 2, 3, 4, 16, and 17 and solid element types -2, -1, 1, 2, 3, 4, 10, 13, 15, 16, and 17.

Card Summary:

Card 1. This card is required.

MID	IDAM	DTYP	REFSZ	NUMFIP	LP2BI	PDDT	NHIS

Card 2. This card is required.

HIS1	HIS2	HIS3	IFLG1	IFLG2	IFLG3	IFLG4	

Card 3. This card is required.

D11	D22	D33	D44	D55	D66		

Card 4a. Include this card for shell elements

D12	D21	D24	D42	D14	D41		

Card 4b. Include this card for solid elements.

D12	D21	D23	D32	D13	D31		

Card 5.1. Define NHIS sets of Cards 5.1 and 5.2 (total of 2 × NHIS cards) for each history variable (HIS*n*).

LCSDG	ECRIT	DMGEXP	DCRIT	FADEXP	LCREG		

Card 5.2. Define NHIS sets of Cards 5.1 and 5.2 (total of 2 × NHIS cards) for each history variable (HIS*n*).

LCSRS	SHRF	BIAXF	LCDLIM	MIDFAIL	NFLOC		

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	IDAM	DTYP	REFSZ	NUMFIP	LP2BI	PDDT	NHIS
Type	A	I	I	F	F	F	I	I
Default	none	0	0	0.0	1.0	0.0	0	1

VARIABLE	DESCRIPTION
MID	Material ID for which this generalized damage definition applies
IDAM	Flag for damage model: EQ.0: No damage model is used. EQ.1: GISSMO damage model
DTYP	Flag for damage behavior: EQ.0: Damage is accumulated; no coupling to flow stress, no failure. EQ.1: Damage is accumulated; element failure occurs for D = 1.
REFSZ	Reference element size, for which an additional output of damage will be generated. This is necessary to ensure the applicability of resulting damage quantities when transferred to different mesh sizes.
NUMFIP	Number of failed integration points prior to element deletion. The default is unity. LT.0: |NUMFIP| is the percentage of layers which must fail before the element fails.
LP2BI	Option to use a bending indicator instead of the Lode parameter. If active (> 0), the expression “bending indicator” replaces the term “Lode parameter” everywhere in this manual page. We adopted the bending indicator from *MAT_258 (compare with variable Ω). LP2BI > 0 is only available for shell elements and requires NUMFIP = 1. EQ.0.0: Inactive.

VARIABLE	DESCRIPTION
	EQ.1.0: Active. Constant regularization (LCREG) applied. EQ.2.0: Active. Regularization (LCREG) fully applied under pure membrane loading ($\Omega = 0$) but not at all under pure bending ($\Omega = 1$). Linear interpolation in between.
PDDT	Pre-defined damage tensors. If non-zero, damage tensor coefficients D11 to D66 on Cards 3 and 4 will be ignored. See Remark 2. EQ.0: No pre-defined damage tensor is used. EQ.1: Isotropic damage tensor EQ.2: 2-parameter isotropic damage tensor for volumetric-deviatoric split EQ.3: Anisotropic damage tensor as in MAT_104 (FLAG = -1) EQ.4: 3-parameter damage tensor associated with IFLG1 = 2
NHIS	Number of history variables as driving quantities ($1 \leq \text{NHIS} \leq 3$)

Card 2	1	2	3	4	5	6	7	8
Variable	HIS1	HIS2	HIS3	IFLG1	IFLG2	IFLG3	IFLG4	
Type	I	I	I	I	I	I	I	
Default	0	optional	optional	0	0	0	0	

VARIABLE	DESCRIPTION
HIS*n*	Choice of variable as driving quantity for damage, called "history value" in the following: EQ.0: Equivalent plastic strain rate is the driving quantity for the damage if IFLG1 = 0. Alternatively, if IFLG1 = 1, components of the plastic strain rate tensor are driving quantities for damage (see Remarks 2 and 3). GT.0: The rate of the additional history variable HIS*n* is the driving quantity for damage. IFLG1 should be set to 0. LT.0: *DEFINE_FUNCTION IDs defining the damage driving quantities as a function of the components of the plastic strain rate tensor; IFLG1 should be set to 1.

VARIABLE	DESCRIPTION
IFLG1	Damage driving quantities: EQ.0: Rates of history variables HIS*n* EQ.1: Specific components of the plastic strain rate tensor; see Remarks 2 and 3. EQ.2: Predefined functions of plastic strain rate components for orthotropic damage model. HIS*n* inputs will be ignored, and IFLG2 should be set to 1. This option is available for shell elements only. EQ.3: Specific components of the total strain rate tensor; see Remarks 2 and 3.
IFLG2	Damage strain coordinate system: EQ.0: Local element system (shells) or global system (solids) EQ.1: Material system, only applicable for non-isotropic material models. Supported models for shell elements: all materials with AOPT feature. Supported models for solid elements: 22, 33, 41-50, 58, 103, 122, 126, 133, 157, 199, 233. EQ.2: Principal strain system (rotating) EQ.3: Principal strain system (fixed when instability/coupling starts)
IFLG3	Erosion criteria and damage coupling system: EQ.0: Erosion occurs when one of the damage parameters computed reaches unity; the damage tensor components are based on the individual damage parameters d1 to d3. EQ.1: Erosion occurs when a single damage parameter *D* reaches unity; the damage tensor components are based on this single damage parameter. Results in the isotropic limit case will only be correct if DMGEXP is set to 1.0 for all history variables. EQ.2: Activation of the Domain of Shell-to-Solid Equivalence (DSSE) for shell elements, cf. Pack and Mohr (2017). Two damage variables are necessary for this model (a fracture initiation variable D1 and a localization initiation variable D2). If D1 reaches 1.0, stresses are set to zero and the integration point is no longer able to sustain any load. If D2 = 1.0, no action is taken, and the integration point is still mechanically active. Erosion occurs when at least one

VARIABLE	DESCRIPTION
	of the two damage variables (D1 or D2) reaches unity for all integration points. Additional required settings for this model: NUMFIP = -100, DCRIT = 1, PDDT = 1, and NFLOC = 0.
IFLG4	Damage drivers' evolution flag. This option is relevant for cyclic loading when IFLG1 is set to 1 or 3. Damage cannot increase with decreasing strain or history variable, but as soon as the strain/history increase again after unloading (i.e., below the previously reached maximum), the damage also increases again (behavior with IFLG4 = 0). This can be prevented with IFLG4 = 1, where the last maximum strain/history is saved.

Card 3	1	2	3	4	5	6	7	8
Variable	D11	D22	D33	D44	D55	D66		
Type	I	I	I	I	I	I		

Damage for Shell Elements Card. This card is included for shell elements.

Card 4a	1	2	3	4	5	6	7	8
Variable	D12	D21	D24	D42	D14	D41		
Type	I	I	I	I	I	I		

Damage for Solid Elements. This card is included for solid elements.

Card 4b	1	2	3	4	5	6	7	8
Variable	D12	D21	D23	D32	D13	D31		
Type	I	I	I	I	I	I		

VARIABLE	DESCRIPTION
D*ij*	DEFINE_FUNCTION IDs for damage tensor coefficients; see Remark 2.

Damage Definition Cards for IDAM = 1 (GISSMO). NHIS sets of Cards 5.1 and 5.2 (total of 2 × NHIS cards) must be defined for each history variable (HIS*n*).

Card 5.1	1	2	3	4	5	6	7	8
Variable	LCSDG	ECRIT	DMGEXP	DCRIT	FADEXP	LCREG		
Type	I	F	F	F	F	I		
Default	0	0.0	1.0	0.0	1.0	0		

VARIABLE	DESCRIPTION
LCSDG	Load curve ID defining corresponding history value to failure as a function of triaxiality.
ECRIT	Critical history value (material instability): LT.0.0: |ECRIT| is load curve ID defining critical history value as a function of triaxiality. EQ.0.0: Fixed value DCRIT defining critical damage is read. GT.0.0: Fixed value for stress-state independent critical history value
DMGEXP	Exponent for nonlinear damage accumulation
DCRIT	Damage threshold value (critical damage). If a load curve of critical history value or fixed value is given by ECRIT, input is ignored.
FADEXP	Exponent for damage-related stress fadeout. LT.0.0: |FADEXP| is load curve ID defining element-size dependent fading exponent GT.0.0: Constant fading exponent
LCREG	Load curve ID defining element size dependent regularization factors for history value to failure

Damage Definition Cards for IDAM = 1 (GISSMO). NHIS sets of Cards 5.1 and 5.2 (total of 2 × NHIS cards) must be defined for each history variable (HIS*n*).

Card 5.2	1	2	3	4	5	6	7	8
Variable	LCSRS	SHRF	BIAXF	LCDLIM	MIDFAIL	NFLOC		
Type	I	F	F	I	F	F		
Default	0	0.0	0.0	0	0.0	0.0		

VARIABLE	DESCRIPTION
LCSRS	Load curve ID defining failure history value scaling factor for LCSDG as a function of history value rate. If the first rate value in the curve is negative, it is assumed that all rate values are given as natural logarithm of the history rate. GT.0: Scale ECRIT as well. LT.0: Do not scale ECRIT.
SHRF	Reduction factors for regularization at triaxiality = 0 (shear)
BIAXF	Reduction factors for regularization at triaxiality = 2/3 (biaxial)
LCDLIM	Load curve ID defining damage limit values as a function of triaxiality. Damage can be restricted to values less than 1.0 to prevent further stress reduction and failure for certain triaxialities.
MIDFAIL	Mid-plane failure option for shell elements. If active, then critical strain is only checked at the mid-plane integration point, meaning an odd number for NIP should be used. Damage is computed at the other integration points, but no coupling to the stresses is done first. As soon as the mid-plane IP reaches ECRIT/DCRIT, then all the other IPs are also checked (exception: MIDFAIL = 4). EQ.0.0: Inactive EQ.1.0: Active. The stresses immediately begin to reduce for non-mid-plane IPs that are already above their critical value. Coupling only occurs for IPs that reach their criterion. EQ.2.0: Active. The stresses immediately begin to reduce for all the non-mid-plane IPs. NUMFIP is active.

VARIABLE	DESCRIPTION
	EQ.3.0: Active. Same as 2, but when D = 1 is reached in the middle integration point, the element is eroded instantaneously. NUMFIP is disregarded.
	EQ.4.0: Active. Damage and failure is applied only on the midpoint. When D = 1 on the midpoint, the element is eroded. NUMFIP is disregarded. Integration points away from the midplane see no stress reduction and no failure.
NFLOC	Optional "local" number of failed integration points prior to element deletion. Overwrites the definition of NUMFIP for history variable HIS*n*.

Remarks:

1. **Comparison to GISSMO Damage Model.** The GISSMO damage model is described in detail in the remarks of *MAT_ADD_DAMAGE_GISSMO. If NHIS = 1 and HIS1 = 0 is used, this keyword behaves the same as GISSMO. The main difference between this keyword and GISSMO is that up to 3 independent but simultaneous damage evolutions are possible. Therefore, parameters LCSDG, ECRIT, DMGEXP, DCRIT, FADEXP, LCREGD, LCSRS, SHRF, BIAXF, and LCDLIM can be defined separately for each history variable.

2. **Damage Tensor.** The relation between nominal (damaged) stresses σ_{ij} and effective (undamaged) stresses $\tilde{\sigma}_{ij}$ is now expressed as

$$\begin{bmatrix}\sigma_{11}\\\sigma_{22}\\\sigma_{33}\\\sigma_{12}\\\sigma_{23}\\\sigma_{31}\end{bmatrix}=\begin{bmatrix}D_{11}&D_{12}&D_{13}&0&0&0\\D_{21}&D_{22}&D_{23}&0&0&0\\D_{31}&D_{32}&D_{33}&0&0&0\\0&0&0&D_{44}&0&0\\0&0&0&0&D_{55}&0\\0&0&0&0&0&D_{66}\end{bmatrix}\begin{bmatrix}\tilde{\sigma}_{11}\\\tilde{\sigma}_{22}\\\tilde{\sigma}_{33}\\\tilde{\sigma}_{12}\\\tilde{\sigma}_{23}\\\tilde{\sigma}_{31}\end{bmatrix}$$

with damage tensor **D**. Each damage tensor coefficient D_{ij} can be defined using *DEFINE_FUNCTION as a function of damage parameters d_1 to d_3. For simple isotropic damage driven by plastic strain (NHIS = 1, HIS1 = 0, IFLG1 = IFLG2 = IFLG3 = 0) that would be

$$\begin{bmatrix}\sigma_{11}\\\sigma_{22}\\\sigma_{33}\\\sigma_{12}\\\sigma_{23}\\\sigma_{31}\end{bmatrix}=(1-d_1)\begin{bmatrix}1&0&0&0&0&0\\0&1&0&0&0&0\\0&0&1&0&0&0\\0&0&0&1&0&0\\0&0&0&0&1&0\\0&0&0&0&0&1\end{bmatrix}\begin{bmatrix}\tilde{\sigma}_{11}\\\tilde{\sigma}_{22}\\\tilde{\sigma}_{33}\\\tilde{\sigma}_{12}\\\tilde{\sigma}_{23}\\\tilde{\sigma}_{31}\end{bmatrix}$$

That means the following function should be defined for D11 to D66 (Card 3):

```
*DEFINE_FUNCTION
1,D11toD66
func1(d1,d2,d3)=(1.0-d1)
```

and all entries in Card 4 can be left empty or equal to zero in that case.

If GISSMO (IDAM = 1) is used, the damage parameters used in those functions are internally replaced by

$$d_i \quad \rightarrow \quad \left(\frac{d_i - \mathrm{DCRIT}_i}{1 - \mathrm{DCRIT}_i}\right)^{\mathrm{FADEXP}_i}$$

In the case of plane stress (shell) elements, coupling between normal stresses and shear stresses is implemented and the damage tensor is defined as below:

$$\begin{bmatrix} \sigma_{11} \\ \sigma_{22} \\ 0 \\ \sigma_{12} \\ \sigma_{23} \\ \sigma_{31} \end{bmatrix} = \begin{bmatrix} D_{11} & D_{12} & 0 & D_{14} & 0 & 0 \\ D_{21} & D_{22} & 0 & D_{24} & 0 & 0 \\ 0 & 0 & D_{33} & 0 & 0 & 0 \\ D_{41} & D_{42} & 0 & D_{44} & 0 & 0 \\ 0 & 0 & 0 & 0 & D_{55} & 0 \\ 0 & 0 & 0 & 0 & 0 & D_{66} \end{bmatrix} \begin{bmatrix} \tilde{\sigma}_{11} \\ \tilde{\sigma}_{22} \\ 0 \\ \tilde{\sigma}_{12} \\ \tilde{\sigma}_{23} \\ \tilde{\sigma}_{31} \end{bmatrix}$$

Since the evaluation of *DEFINE_FUNCTION for variables D11 to D66 is relatively time consuming, pre-defined damage tensors (PDDT) can be used. Currently the following options are available for shell elements:

PDDT	Damage Tensor
1	$(1-D_1)\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$
2	$\begin{bmatrix} 1-\frac{2}{3}D_1-\frac{1}{3}D_2 & \frac{1}{3}D_1-\frac{1}{3}D_2 & 0 & 0 & 0 & 0 \\ \frac{1}{3}D_1-\frac{1}{3}D_2 & 1-\frac{2}{3}D_1-\frac{1}{3}D_2 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1-D_1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$
3	$\begin{bmatrix} 1-D_1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1-D_2 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1-\frac{1}{2}(D_1+D_2) & 0 & 0 \\ 0 & 0 & 0 & 0 & 1-\frac{1}{2}D_2 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1-\frac{1}{2}D_1 \end{bmatrix}$

PDDT	Damage Tensor
4	$\begin{bmatrix} 1-D_1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1-D_2 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1-D_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$

and the following ones for solid elements:

PDDT	Damage Tensor
1	$(1-D_1)\begin{bmatrix} 1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1 \end{bmatrix}$
2	$\begin{bmatrix} 1-\frac{2}{3}D_1-\frac{1}{3}D_2 & \frac{1}{3}D_1-\frac{1}{3}D_2 & \frac{1}{3}D_1-\frac{1}{3}D_2 & 0 & 0 & 0 \\ \frac{1}{3}D_1-\frac{1}{3}D_2 & 1-\frac{2}{3}D_1-\frac{1}{3}D_2 & \frac{1}{3}D_1-\frac{1}{3}D_2 & 0 & 0 & 0 \\ \frac{1}{3}D_1-\frac{1}{3}D_2 & \frac{1}{3}D_1-\frac{1}{3}D_2 & 1-\frac{2}{3}D_1-\frac{1}{3}D_2 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1-D_1 & 0 & 0 \\ 0 & 0 & 0 & 0 & 1-D_1 & 0 \\ 0 & 0 & 0 & 0 & 0 & 1-D_1 \end{bmatrix}$
3	$\begin{bmatrix} 1-D_1 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1-D_2 & 0 & 0 & 0 & 0 \\ 0 & 0 & 1-D_3 & 0 & 0 & 0 \\ 0 & 0 & 0 & 1-\frac{1}{2}(D_1+D_2) & 0 & 0 \\ 0 & 0 & 0 & 0 & 1-\frac{1}{2}(D_2+D_3) & 0 \\ 0 & 0 & 0 & 0 & 0 & 1-\frac{1}{2}(D_3+D_1) \end{bmatrix}$

3. **History Variables.** The increment of the damage parameter is computed in GISSMO based on a driving quantity that has the dimension of a strain rate:

$$\dot{d} = n d^{1-1/n} \frac{\dot{\mathrm{HIS}}_i}{\mathrm{epf}}$$

The history variables defined by the user through HIS*i* should thus have the dimension of a strain as the rate is computed internally by MAT_ADD_GENERALIZED_DAMAGE:

$$\dot{\mathrm{HIS}}_i = \frac{\mathrm{HIS}_i(t^{n+1}) - \mathrm{HIS}_i(t^n)}{t^{n+1} - t^n}$$

History variables can either come directly from associated material models (IFLG1 = 0 and HIS*i* > 0), or they can be equivalent to plastic strain rate tensor components (IFLG1 = 1 and HIS*i* = 0):

$$\dot{\mathrm{HIS}}_1 = \dot{\varepsilon}^p_{xx}, \quad \dot{\mathrm{HIS}}_2 = \dot{\varepsilon}^p_{yy}, \quad \dot{\mathrm{HIS}}_3 = \dot{\varepsilon}^p_{xy} \quad \text{for IFLG2} = 0$$
$$\dot{\mathrm{HIS}}_1 = \dot{\varepsilon}^p_{aa}, \quad \dot{\mathrm{HIS}}_2 = \dot{\varepsilon}^p_{bb}, \quad \dot{\mathrm{HIS}}_3 = \dot{\varepsilon}^p_{ab} \quad \text{for IFLG2} = 1$$
$$\dot{\mathrm{HIS}}_1 = \dot{\varepsilon}^p_{1}, \quad \dot{\mathrm{HIS}}_2 = \dot{\varepsilon}^p_{2}, \quad \dot{\mathrm{HIS}}_3 = 0 \quad \text{for IFLG2} = 2$$

or they can be provided via *DEFINE_FUNCTIONs by the user (IFLG1 = 1 and HISi < 0):

$$\dot{\mathrm{HIS}}_i = f_i\left(\dot{\varepsilon}^p_{xx}, \dot{\varepsilon}^p_{yy}, \dot{\varepsilon}^p_{zz}, \dot{\varepsilon}^p_{xy}, \dot{\varepsilon}^p_{yz}, \dot{\varepsilon}^p_{zx}\right) \quad \text{for IFLG2} = 0$$
$$\dot{\mathrm{HIS}}_i = f_i\left(\dot{\varepsilon}^p_{aa}, \dot{\varepsilon}^p_{bb}, \dot{\varepsilon}^p_{zz}, \dot{\varepsilon}^p_{ab}, \dot{\varepsilon}^p_{bz}, \dot{\varepsilon}^p_{za}\right) \quad \text{for IFLG2} = 1$$
$$\dot{\mathrm{HIS}}_i = f_i\left(\dot{\varepsilon}^p_{1}, \dot{\varepsilon}^p_{2}\right) \quad \text{for IFLG2} = 2$$

The following example defines a history variable (HISi = -1234) as function of the transverse shear strains in material coordinate system (a, b, z) for shells:

```
*DEFINE_FUNCTION
      1234
fhis1(eaa,ebb,ezz,eab,ebz,eza)=1.1547*sqrt(ebz**2+eza**2)
```

The plastic strain rate tensor is not always available in the material law and is estimated as:

$$\dot{\boldsymbol{\varepsilon}}^p = \frac{\dot{\varepsilon}^p_{eff}}{\dot{\varepsilon}_{eff}}\left[\dot{\boldsymbol{\varepsilon}} - \frac{\dot{\varepsilon}_{vol}}{3}\delta\right]$$

This is a good approximation for isochoric materials with small elastic strains (such as metals) and correct for J2 plasticity.

You can also use the *total* strain rate components $\dot{\varepsilon}_{ij}$ instead of the *plastic* strain rate components $\dot{\varepsilon}^p_{ij}$ by changing IFLG1 = 1 to IFLG1 = 3. Setting IFLG4 = 1 should be considered in that case (see description for IFLG4).

The following table gives an overview of the driving quantities used for incrementing the damage in function of the input parameters (strain superscript "p" for "plastic" is omitted for convenience):

IFLG1	IFLG2	HISi > 0	HISi = 0	HISi < 0
0	0	$\dot{\mathrm{HIS}}_i$	$\dot{\varepsilon}$	–
0	1	$\dot{\mathrm{HIS}}_i$	–	–
0	2	$\dot{\mathrm{HIS}}_i$	–	–
1/3	0	–	$\dot{\varepsilon}_{ij}$	$f(\dot{\varepsilon}_{ij})$
1/3	1	–	$\dot{\varepsilon}^{mat}_{ij}$	$f(\dot{\varepsilon}^{mat}_{ij})$
1/3	2	–	$\dot{\varepsilon}_i$	$f(\dot{\varepsilon}_i)$

IFLG1	IFLG2	HISi > 0	HISi = 0	HISi < 0
2	0	–	–	–
2	1	Preprogrammed functions of plastic strain rate		
2	2	–	–	–

4. **Post-Processing History Variables.** History variables of the GENERALIZED_DAMAGE model are written to the post-processing database behind those already occupied by the material model which is used in combination:

History Variable #	Description
ND	Triaxiality variable σ_H/σ_M
ND + 1	Lode parameter value
ND + 2	Single damage parameter D, $\left(10^{-20} < D \leq 1\right)$, only for IFLG3 = 1
ND + 3	Damage parameter d_1
ND + 4	Damage parameter d_2
ND + 5	Damage parameter d_3
ND + 6	Damage threshold DCRIT_1
ND + 7	Damage threshold DCRIT_2
ND + 8	Damage threshold DCRIT_3
ND + 12	History variable HIS_1
ND + 13	History variable HIS_2
ND + 14	History variable HIS_3
ND + 15	Angle between principal and material axes
ND + 21	Characteristic element size (used in LCREG)

For instance, ND = 6 for *MAT_024, ND = 9 for *MAT_036, and ND = 23 for *MAT_187. Exact information of the variable locations can be found in the d3hsp section "MAGD damage history listing."

*MAT_ADD_INELASTICITY

The purpose of this card is to add inelasticity features to an arbitrary standard material model. It may either be used as a modular concept on top of a simple elastic model or patching a more complex material model with a missing inelastic feature.

This keyword is under development and currently only applies to shell types 2, 4 and 16, and solid types -18, -2, -1, 1, 2, 10, 15, 16 and 17. Implicit as well as explicit analyses are supported, and the user should be aware of an extra cost associated with using this feature.

Card Summary:

Card 1. This card is required. NIELINKS groups of Cards 4 through 6 should follow this card, possibly after input of anisotropy information in Cards 2 and 3.

MID	NIELINKS		G	K	AOPT	MACF	BETA

Card 2. For AOPT > 0, define Cards 2 and 3.

XP	YP	ZP	A1	A2	A3		

Card 3. For AOPT > 0, define Cards 2 and 3.

V1	V2	V3	D1	D2	D3		

Card 4. For each link, Cards 4 through 6 are required. NIELAWS groups of Cards 5 and 6 should follow immediately after each Card 4.

NIELAWS	WEIGHT						

Card 5. NIELAWS sets of Cards 5 and 6 are required after each Card 4.

LAW	MODEL						

Card 6a. This card is required for LAW = 3 and MODEL = 1.

P1	P2						

Card 6b. This card is required for LAW = 3 and MODEL = 2.

P1							

Card 6c. This card is included for LAW = 5 and MODEL ≤ 2.

P1	P2	P3					

Card 6d. This card is required for LAW = 5 and MODEL = 3, and LAW = 6 and MODEL = 4.

P1	P2	P3	P4	P5	P6	P7	

Card 6e. This card is required for LAW = 5 and MODEL = 4, and LAW = 6 and MODEL = 5.

P1	P2	P3	P4				

Card 6f. This card is required for LAW = 6 and MODEL ≤ 3

P1	P2	P3	P4	P5	P6	P7	P8

Data Card Definitions:

Main Card. Only one instance of this card is needed. NIELINKS groups of Cards 4 through 6 should follow this card, possibly after input of anisotropy information in Cards 2 and 3.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	NIELINKS		G	K	AOPT	MACF	BETA
Type	A	I		F	F	F	F	F
Default	none	1		0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification for which this inelasticity definition applies. A unique number or label must be specified (see *PART).
NIELINKS	Number of links/networks/phases specified by the user. An additional link may be added internally if the weights below do not sum up to unity.
G	Characteristic shear modulus used for some of the inelasticity models. This should reflect the elastic stiffness for the material without any inelasticity effects. For instance, if *MAT_ELASTIC is used, set $G = E/(2(1+\nu))$.
K	Characteristic bulk modulus used for some of the inelasticity models. This should reflect the elastic stiffness for the material without

VARIABLE	DESCRIPTION
	any inelasticity effects. For instance, if *MAT_ELASTIC is used, set $K = E/(3(1-2\nu))$.
AOPT	Material axes option (see *MAT_002 for a detailed description): EQ.0.0: Locally orthotropic with material axes determined by element nodes. For shells only, the material axes are then rotated about the normal vector to the surface of the shell by the angle BETA. EQ.1.0: Locally orthotropic with material axes determined by a point, P, in space and the global location of the element center. This option is for solid elements only. EQ.2.0: Globally orthotropic with material axes determined by vectors defined below EQ.3.0: Locally orthotropic material axes determined by a vector **v** and the normal vector to the plane of the element. 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then **a** and **b** are rotated about **c** by an angle BETA. BETA may be set in the keyword input for the element or in the input for this keyword. Note that for solids, the material axes may be switched depending on the choice of MACF. The switch may occur before or after applying BETA depending on the value of MACF. EQ.4.0: Locally orthotropic in cylindrical coordinate system with the material axes determined by a vector, **v**, and an originating point, P, defining the centerline axis. This option is for solid elements only. LT.0.0: The absolute value of AOPT is a coordinate system ID.
MACF	Material axes change flag for solid elements: EQ.-4: Switch material axes b and c before BETA rotation EQ.-3: Switch material axes a and c before BETA rotation EQ.-2: Switch material axes a and b before BETA rotation

VARIABLE	DESCRIPTION
	EQ.1: No change, default EQ.2: Switch material axes *a* and *b* after BETA rotation EQ.3: Switch material axes *a* and *c* after BETA rotation EQ.4: Switch material axes *b* and *c* after BETA rotation Figure M2-2 indicates when LS-DYNA applies MACF during the process to obtain the final material axes. If BETA on *ELEMENT_-SOLID_*{OPTION}* is defined, then that BETA is used for the rotation for all AOPT options. Otherwise, if AOPT = 3, the BETA input on Card 3 rotates the axes. For all other values of AOPT, the material axes will be switched as specified by MACF, but no BETA rotation will be performed.
BETA	Material angle in degrees for AOPT = 0 (shells only) and AOPT = 3 (all element types). This angle may be overriden on the element card; see *ELEMENT_SHELL_BETA and *ELEMENT_SOLID_ORTHO.

Anisotropy cards. Include Cards 2 and 3 if AOPT > 0.

Card 2	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 3	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
XP, YP, ZP	Coordinates of point, *p*, for AOPT = 1 and 4; see *MAT_002.

VARIABLE	DESCRIPTION
A1, A2, A3	Components of vector, **a**, for AOPT = 2; see *MAT_002.
V1, V2, V3	Components of vector, **v**, for AOPT = 3 and 4; see *MAT_002.
D1, D2, D3	Components of vector, **d**, for AOPT = 2; see *MAT_002.

Link/network/phase Cards. Include NIELINKS sets of all cards that follow; NIELAWS groups of Cards 5 and 6 should follow immediately after each Card 4.

Card 4	1	2	3	4	5	6	7	8
Variable	NIELAWS	WEIGHT						
Type	I	F						
Default	none	0 or 1						

VARIABLE	DESCRIPTION
NIELAWS	Number of inelasticity laws that apply to this material model at this link, each contributing in its own way to the total inelastic strain (rate)
WEIGHT	Weight of this link/network/phase used when computing total stress.

Inelasticity model cards. Include NIELAWS sets of Cards 5 and 6; the Card 6 determined by the law and model selected should follow immediately after Card 5.

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

VARIABLE	DESCRIPTION
LAW	Inelasticity law. One of the laws listed below must be chosen:

VARIABLE	DESCRIPTION
	EQ.3: Isotropic hardening plasticity EQ.5: Creep EQ.6: Viscoelasticity
MODEL	Model definition with choice dependent on the specified law above. A valid combination of law and model must be chosen. For isotropic hardening plasticity (LAW = 3), choices are EQ.1: Linear hardening EQ.2: Hardening from curve/table For creep (LAW = 5), choices are EQ.1: Norton incremental formulation EQ.2: Norton total formulation EQ.3: Norton-Bailey formulation EQ.4: Bergström-Boyce formulation For viscoelasticity (LAW = 6), choices are EQ.1: Bulk and shear decay, with optional temperature shifts, hypoelastic version EQ.2: Bulk and shear decay, with optional temperature shifts, hyperelastic version #1 EQ.3: Bulk and shear decay, with optional temperature shifts, hyperelastic version #2 EQ.4: Norton-Bailey formulation EQ.5: Bergström-Boyce formulation

Inelasticity Parameters. This card is included for LAW = 3 and MODEL = 1.

Card 6a	1	2	3	4	5	6	7	8
Variable	P1	P2						
Type	F	F						
Default	0.0	0.0						

VARIABLE	DESCRIPTION
P1	Virgin yield stress, σ_0
P2	Hardening, H

Inelasticity Parameters. This card is included for LAW = 3 and MODEL = 2.

Card 6b	1	2	3	4	5	6	7	8
Variable	P1							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
P1	Curve or table ID that defines the hardening

Inelasticity Parameters. This card is included for LAW = 5 and MODEL ≤ 2.

Card 6c	1	2	3	4	5	6	7	8
Variable	P1	P2	P3					
Type	F	F	F					
Default	0.0	0.0	0.0					

VARIABLE	DESCRIPTION
P1	Norton creep parameter, A
P2	Norton creep parameter, m
P3	Norton creep parameter, n

Inelasticity Parameters. This card is included for LAW = 5 with MODEL = 3 and for LAW = 6 with MODEL = 4.

Card 6d	1	2	3	4	5	6	7	8
Variable	P1	P2	P3	P4	P5	P6	P7	
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
P1	Norton-Bailey creep parameter, A
P2	Norton-Bailey creep parameter, σ_0
P3	Norton-Bailey creep parameter, n
P4	Norton-Bailey creep parameter, T_0
P5	Norton-Bailey creep parameter, p
P6	Norton-Bailey creep parameter, m
P7	Norton-Bailey creep parameter, ε_0

Inelasticity Parameters. This card is included for LAW = 5 with MODEL = 4 and for LAW = 6 with MODEL = 5.

Card 6e	1	2	3	4	5	6	7	8
Variable	P1	P2	P3	P4				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
P1	Bergström-Boyce creep parameter, A
P2	Bergström-Boyce creep parameter, m

VARIABLE	DESCRIPTION
P3	Bergström-Boyce creep parameter, C
P4	Bergström-Boyce creep parameter, E

Inelasticity Parameters. This card is included for LAW = 6 and MODEL = 1, 2, or 3.

Card 6f	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
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
P1	Shear decay coefficient, β_G
P2	Bulk decay coefficient, β_K
P3	Shear reference temperature, T_G
P4	Shear shift coefficient, A_G
P5	Shear shift coefficient, B_G
P6	Bulk reference temperature, T_K
P7	Bulk shift coefficient, A_K
P8	Bulk shift coefficient, B_K

Remarks:

General

The resulting stress from an integration point with inelasticities is the sum of the stress σ_I from each link, weighed by its weight, w_I (see WEIGHT above). A link in this context can also be referred to as a network or a phase, depending on the physical interpretation, and we use the subscript I to refer to a specific one. So, the stress, σ, is in the end given by

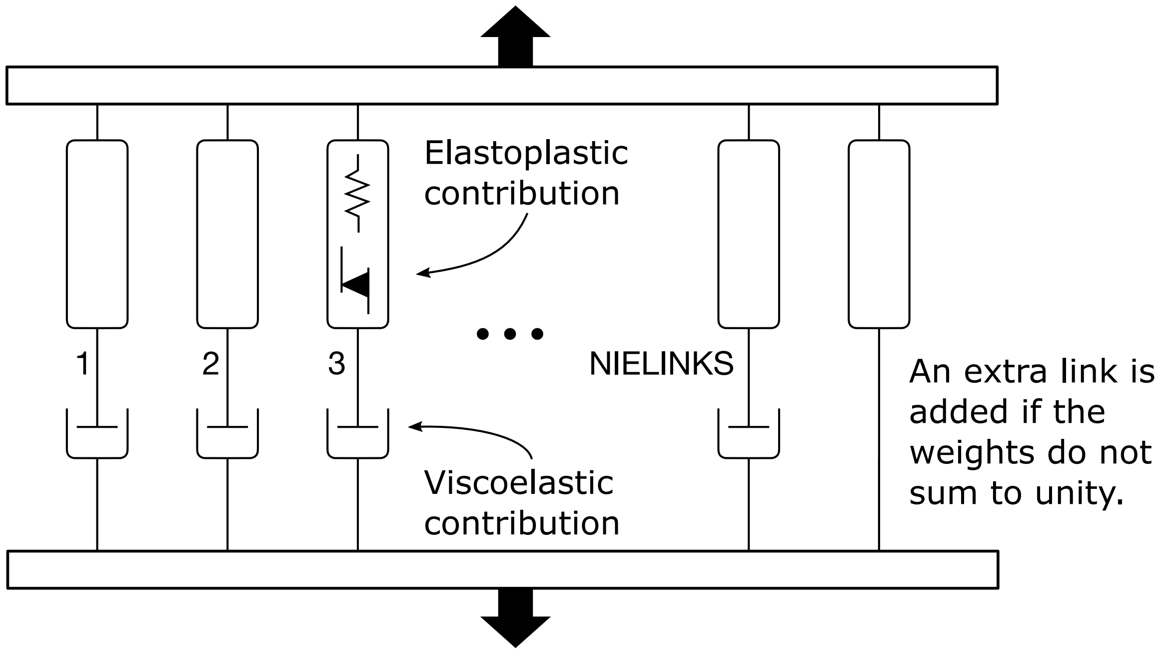


Figure 2-8. Schematic view of how inelasticity is added to the model.

$$\sigma = \sum_{I=1}^{\mathrm{NIELINKS}(+1)} w_I \sigma_I \ .$$

The data for the links are specified by the user, except for a possible last one which is internally created if the weights do not sum to unity (whence the +1 in the number of terms in the sum above). This last link will get its stress $\sigma_{\mathrm{NIELINKS+1}}$ only from the material model without any inelasticities, and its weight will be

$$w_{\mathrm{NIELINKS+1}} = 1 - \sum_{I=1}^{\mathrm{NIELINKS}} w_I \ ,$$

that is, just enough for the total weight to sum to 1. The stress for each link will be treated next, for which we drop the subscript I for the sake of clarity, and emphasize that this first part will only treat creep and plasticity, since viscoelasticity is somewhat different and explained on its own at the end of this section.

A single link/network/phase

Infinitesimal description

The inelasticity feature assumes that the strain or strain rate is somehow decomposed into an elastic and inelastic part. This decomposition is in general not trivial and depends upon the underlying material model, but to make things simple we can begin by restricting ourselves to a small deformation context. In this case the decomposition is *additive*, so

$$\varepsilon = \varepsilon_e + \varepsilon_i \ ,$$

where $\boldsymbol{\varepsilon}$ is the *total* (given) strain, $\boldsymbol{\varepsilon}_e$ is the *elastic* strain, and $\boldsymbol{\varepsilon}_i$ is the *inelastic* strain. A material model then amounts to determining the stress for the elastic strain, which can be written as

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}(\boldsymbol{\varepsilon}_e) = \boldsymbol{\sigma}(\boldsymbol{\varepsilon} - \boldsymbol{\varepsilon}_i).$$

The material model used as a basis for this feature, that is, the model indicated by parameter MID above, here acts as the function $\boldsymbol{\sigma}(*)$. If no inelasticity is added to the model, $\boldsymbol{\varepsilon}_i = \mathbf{0}$ and the stress will be given by $\boldsymbol{\sigma}(\boldsymbol{\varepsilon})$. It is simply a plain evaluation of the material model in the absence of this keyword. For linear elasticity, for instance, the function would be given by Hooke's law

$$\boldsymbol{\sigma}(\boldsymbol{\varepsilon}) = \boldsymbol{C}\boldsymbol{\varepsilon} \ ,$$

where $\boldsymbol{C}$ is the Hooke elasticity tensor. Needless to say, the material model itself can deal with inelasticities of various kinds, such as plasticity, creep, thermal expansion and viscoelasticity, so the variable $\boldsymbol{\varepsilon}_i$ is restricted to the inelasticities specifically defined here and thus added to whatever is used in the material model. For the sake of generality, we allow the inelastic strain to come from many sources and be combined:

$$\boldsymbol{\varepsilon}_i = \boldsymbol{\varepsilon}_i^1 + \boldsymbol{\varepsilon}_i^2 + \boldsymbol{\varepsilon}_i^3 + \cdots \ .$$

Here each superscript on the right-hand side refers to a specific combination of LAW and MODEL (excluding viscoelastic laws).

Large strain formulation

For incrementally updated material models, using hypoelasticity with an objective rate of stress, the exposition above is generalized to large deformations by applying the appropriate time derivative to strains and stresses:

$$\boldsymbol{\varepsilon} \rightarrow \boldsymbol{D}, \qquad \boldsymbol{\varepsilon}_e \rightarrow \boldsymbol{D}_e, \qquad \boldsymbol{\varepsilon}_i \rightarrow \boldsymbol{D}_i, \qquad \boldsymbol{\sigma} \rightarrow \boldsymbol{\sigma}^{\nabla}, \ldots$$

Here $\boldsymbol{D}$ is the rate of deformation tensor, and ∇ indicates an objective time derivative[3]. For now, we restrict the evolution of inelastic strain to be based on a von Mises stress potential:

$$\boldsymbol{D}_i^j = \dot{\varepsilon}_i^j \frac{\partial \overline{\sigma}}{\partial \boldsymbol{\sigma}} \ ,$$

where

$$\overline{\sigma} = \sqrt{\frac{3}{2}\boldsymbol{s}:\boldsymbol{s}} \quad \left(\boldsymbol{s} = \boldsymbol{\sigma} - \frac{1}{3}\boldsymbol{\sigma}:\boldsymbol{I}\right)$$

is the von Mises effective stress, and $\dot{\varepsilon}_i^j$ is the rate of effective inelastic strain for the MODEL and LAW corresponding to superscript j. The constitutive law is thus written as

$$\boldsymbol{\sigma}^{\nabla} = \boldsymbol{\sigma}^{\nabla}(\boldsymbol{D}_e) = \boldsymbol{\sigma}^{\nabla}(\boldsymbol{D} - \boldsymbol{D}_i) \ .$$

[3] In LS-DYNA the objective rate is to be understood as the Jaumann rate for solid elements and the rate resulting from the specific co-rotational formulation for shell elements.

For hyperelastic materials the role of $\boldsymbol{D}_e$ is replaced by the *elastic deformation gradient*, $\boldsymbol{F}_e$, and instead of a constitutive law for the rate of stress, the total stress is given as

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}(\boldsymbol{F}_e) \ .$$

The evolution of the elastic deformation gradient is taken as

$$\dot{\boldsymbol{F}}_e = (\boldsymbol{L} - \boldsymbol{L}_i)\boldsymbol{F}_e \ ,$$

where $\boldsymbol{L} = \frac{\partial v}{\partial x}$ is the *spatial velocity gradient* and $\boldsymbol{L}_i$ is the inelastic part. For simplicity, we assume zero plastic spin for all involved features, thus $\boldsymbol{W}_i = \boldsymbol{0}$ and $\boldsymbol{L}_i = \boldsymbol{D}_i$.

From here, we will give the evolution law of the effective inelastic strain for the available contributions.

Isotropic hardening (LAW = 3)

The current yield stress is defined as

$$\sigma_Y = \begin{cases} \sigma_0 + H\varepsilon_p & \text{MODEL = 1} \\ c(\varepsilon_p, \dot{\varepsilon}_p) & \text{MODEL = 2} \end{cases} ,$$

where the inelastic strain is represented by the plastic strain, ε_p, and c is the curve or table used to evaluate the yield stress. The evolution of plastic strain is given by the KKT condition

$$\overline{\sigma} - \sigma_Y \leq 0, \qquad \dot{\varepsilon}_p \geq 0, \qquad (\overline{\sigma} - \sigma_Y)\dot{\varepsilon}_p = 0 \ .$$

In other words, it is the classical von Mises plasticity available in many standard plasticity models; see, for instance, *MAT_PIECEWISE_LINEAR_PLASTICITY (*MAT_024). As an example, materials 1 and 2 below are equivalent.

```
*MAT_ELASTIC
$      mid        ro         e        pr
         1    7.8e-9  210000.0       0.3
*MAT_ADD_INELASTICITY
$      mid
         1
$  nielaws
         1
$      law     model
         3         2
$      cid
         1
*MAT_PIECEWISE_LINEAR_PLASTICITY
$      mid        ro         e        pr
         1    7.8e-9  210000.0       0.3
$                          lcss
                              1
```

CID/LCSS can be either a curve or table defining effective stress as a function of effective plastic strain.

Creep (LAW = 5)

For creep, the inelastic strain is represented by the creep strain, ε_c. The evolution depends on the model specified.

a) *Norton incremental formulation (MODEL = 1)*

$$\dot{\varepsilon}_c = A\bar{\sigma}^m t^n .$$

This is essentially the creep law available in *MAT_THERMO_ELASTO_-VISCOPLASTIC_CREEP (*MAT_188).

b) *Norton total formulation (MODEL = 2).*

$$\dot{\varepsilon}_c = \frac{d}{dt}(A\bar{\sigma}^m t^n) \quad .$$

This is essentially the creep law available in *MAT_UNIFIED_CREEP (*MAT_115), with some slight modifications.

c) *Norton-Bailey formulation (MODEL = 3).*

$$\dot{\varepsilon}_c = \left(A \left(\frac{\bar{\sigma}}{\sigma_0}\right)^n \left(\frac{T}{T_0}\right)^p \left((m+1)(\varepsilon_0 + \varepsilon_c)\right)^m \right)^{\frac{1}{m+1}} \quad .$$

Here T is the current temperature.

d) *Bergström-Boyce formulation (MODEL = 4).*

$$\dot{\varepsilon}_c = A(\lambda_c - 1 + E)^C \bar{\sigma}^m \quad ,$$

where $\lambda_c = \sqrt{\frac{1}{3}\boldsymbol{I}:\boldsymbol{B}_c} \geq 1$ and $\boldsymbol{B}_c = \exp\{2\boldsymbol{\varepsilon}_c\}$

Viscoelasticity (LAW = 6)

In the absence of viscoelasticity, we are now done with the description of the stress update, and we simply set

$$\boldsymbol{s}_I = \boldsymbol{s}$$

$$p_I = p$$

where we use $\boldsymbol{s}_I$ and p_I to denote the final deviatoric stress and pressure in link I that is used in the weighted sum at the beginning of this section. The $\boldsymbol{s}$ and p are to be understood as the deviatoric stress and pressure resulting from treatment of creep and plasticity that we just covered, so $\boldsymbol{\sigma} = \boldsymbol{s} - p\boldsymbol{I}$. For viscoelasticity the stress in link I will be subject to stress decay (relaxation and creep), in that it evolves according to the specified viscoelastic law. For deviatoric and volumetric decay coefficients β_s and β_p, we have for the hypoelastic laws (MODEL = 1 and MODEL = 4):

$$\begin{aligned} \boldsymbol{s}_I^\nabla &= \boldsymbol{s}^\nabla - \beta_s \boldsymbol{s}_I \\ \dot{p}_I &= \dot{p} - \beta_p p_I \end{aligned}$$

The hyperelastic laws are formulated directly in terms of the Kirchhoff stress $\boldsymbol{\tau} = J\boldsymbol{\sigma}$, where $J = \det\boldsymbol{F}$. More specifically, using the notation $q = -\frac{1}{3}\boldsymbol{\tau}:\boldsymbol{I}$ and $\boldsymbol{t} = \boldsymbol{\tau} + q\boldsymbol{I}$, we have for hyperelastic law #1 (MODEL = 2)

$$\begin{aligned} \boldsymbol{t}_I &= \boldsymbol{t} - \operatorname{dev}\left[\beta_s \int_0^t e^{-\beta_s(t-s)} \bar{\boldsymbol{F}}_{s\to t} \boldsymbol{t}(s) \bar{\boldsymbol{F}}_{s\to t}^T ds\right] \\ q_I &= q - \beta_p \int_0^t e^{-\beta_p(t-s)} q(s) ds \end{aligned}$$

and for hyperelastic law #2 (MODEL = 3)

$$\begin{aligned} \boldsymbol{t}_I &= \boldsymbol{t} - \operatorname{sym}\left[\beta_s \int_0^t e^{-\beta_s(t-s)} \boldsymbol{F}_{s\to t} \boldsymbol{t}(s) \boldsymbol{F}_{s\to t}^{-1} ds\right] \\ q_I &= q - \beta_p \int_0^t e^{-\beta_p(t-s)} q(s) ds \end{aligned}$$

The Kirchhoff stress for link I is obtained as $\boldsymbol{\tau}_I = \boldsymbol{t}_I - q_I\boldsymbol{I}$. Here we use $\bar{\boldsymbol{F}}_{s\to t} = J_{s\to t}^{-1/3}\boldsymbol{F}_{s\to t}$, where $J_{s\to t} = \det\boldsymbol{F}_{s\to t}$, and $\boldsymbol{F}_{s\to t}$ is the deformation gradient between the configuration at time s and time t. For law #1, $\bar{\boldsymbol{F}}_{s\to t}$ is used to push the stress forward from time s to time t, while for law #2, $\boldsymbol{F}_{s\to t}$ is used to transform the stress from time s to time t, both essential to preserve frame invariance.

The decay coefficients can be constants but can also dependent on the state of the system (stress, internal variables, temperature, etc.). Note that if the decay coefficients are equal to zero ($\beta_s = \beta_p = 0$), this is equivalent to not having viscoelasticity. Currently, we can specify temperature dependent decay coefficients to affect both the deviatoric and volumetric stress, formalized in the following.

Linear viscoelasticity (MODEL = 1)

For viscoelasticity, the decay of stress is governed by the decay coefficients β_s and β_p, optionally incorporating shift functions depending on the temperature T. In this implementation, the shear and bulk decay are given as

$$\begin{aligned} \beta_s &= \beta_G \phi_G(T) \\ \beta_p &= \beta_K \phi_K(T) \end{aligned}$$

where ϕ_* ($*$ being G or K) are shift functions given by

$$\phi_*(T) = \begin{cases} e^{-A_*\left(\frac{1}{T}-\frac{1}{T_*}\right)} & \text{if } B_* = 0 \text{ (Arrhenius)} \\ e^{-A_*\left(\frac{T-T_*}{B_*+T-T_*}\right)} & \text{if } B_* \neq 0 \text{ (Williams} - \text{Landel} - \text{Ferry)} \end{cases}$$

This is essentially the viscoelastic law available in *MAT_GENERAL_VISCOELASTIC (*MAT_076), except that the driving mechanism for the stress is here $\boldsymbol{s}^{\nabla}$ and $\dot{p}$ rather than $2G_I\boldsymbol{D}_{\text{dev}}$ and $K_I D_{\text{vol}}$. Note also, that in contrast to *MAT_076, the shift coefficients are to be given for each link and for both the shear and bulk decay. This allows for using independent shifts for each link, and if the traditional usage of the shift functions is desired one needs to put the same triplet (i.e., the values of T_*, A_* and B_*) on all links (parameters P3-P5 for shear, and P6-P8 for bulk). If *MAT_ELASTIC is used in combination with viscoelasticity here, the two formulations can be made (almost) equivalent after proper transformation of input data. For instance, the following two material definitions (1 and 2) are equivalent;

```
*MAT_ELASTIC
$      mid        ro         e        pr
         1    7.8e-9  210000.0       0.2
*MAT_ADD_INELASTICITY
$      mid  nielinks
         12
$  nielaws    weight
         10.3
$      law     model
         6         1
$    betag     betak
      0.05
$  nielaws    weight
         10.5
$      law     model
         6         1
$    betag     betak
     0.005
*MAT_GENERAL_VISCOELASTIC
$      mid        ro      bulk
         2    7.8e-9 116666.67
$ blank card

$        g     betag
   43750.0     0.005
   26250.0      0.05
   17500.0
```

In general, with

$$G = \frac{E}{2(1+\nu)}, \quad K = \frac{E}{3(1-2\nu)}$$

in *MAT_ELASTIC, and G_I and K_I in *MAT_GENERAL_VISCOELASTIC, we have

$$G = \sum G_I\,, \qquad K = \sum K_I$$

while the weights are given as

$$w_I = \frac{G_I}{G} = \frac{K_I}{K}$$

which implies that the add inelasticity approach is somewhat more restrictive than the general approach. However, an almost pure shear/bulk link can be created by setting the bulk/shear decay coefficient to a very large number compared to the simulation time. To be specific, to get a shear link set $\beta_K \gg 1/T$ and to get a bulk link set $\beta_G \gg 1/T$, where T is the termination time. See also VFLAG in *MAT_GENERAL_HYPERELASTIC_RUBBER for the difference of the two approaches.

Nonlinear viscoelasticity

The nonlinear creep laws (LAW = 5) can be formulated as nonlinear viscoelastic laws (LAW = 6) by setting

$$\beta_s = \frac{G}{\sigma_I}\dot{\varepsilon}_c$$
$$\beta_p = 0$$

where G is an estimated elastic stiffness of the base material, σ_I is the von Mises effective stress of $\boldsymbol{s}_I$ and $\dot{\varepsilon}_c$ is the creep law of interest. Currently the following models are supported

e) *Norton-Bailey formulation (MODEL = 4).*

$$\dot{\varepsilon}_c = \left(A \left(\frac{\sigma_I}{\sigma_0}\right)^n \left(\frac{T}{T_0}\right)^p \left((m+1)(\varepsilon_0 + \varepsilon_c)\right)^m \right)^{\frac{1}{m+1}}\,,$$

where T is the current temperature.

f) Bergström-Boyce formulation (MODEL = 5).

$$\dot{\varepsilon}_c = A(\lambda_c - 1 + E)^C \sigma_I^m\,,$$

where $\lambda_c = \sqrt{\frac{1}{3}\mathbf{I}\colon\mathbf{B}_c} \geq 1$ and $\mathbf{B}_c = \exp\{2\boldsymbol{\varepsilon}_c\}$

For linear elasticity the creep and nonlinear viscoelastic laws are equivalent. For other models they are similar assuming that a reasonable value of G is used (see input field 4 on Card 1).

History

With *DEFINE_MATERIAL_HISTORIES you can output the effective plastic and creep strains for plastic and creep models, respectively. The presence of this keyword in the input deck will automatically move the total plastic strain to the appropriate location in the d3plot database. Its value will be

$$\varepsilon_p = \sum_{I=1}^{\text{NIELINKS}} w_I \varepsilon_p^I.$$

The creep strain can also be retrieved similarly as shown in the following table.

**DEFINE_MATERIAL_HISTORIES Properties*					
Label	**Attributes**				**Description**
Effective Creep Strain	-	-	-	-	$\varepsilon_c = \sum_{I=1}^{\text{NIELINKS}} w_I \varepsilon_c^I$

*MAT_ADD_PERMEABILITY

Add permeability to material model for consolidation calculations. See *CONTROL_-PORE_FLUID.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	PERM	PERMY	PERMZ	THEXP	LCKZ	PMTYP	
Type	A	F/I	F/I	F/I	F	I	I	
Default	none	none	PERM	PERM	0.0	none	0	

VARIABLE	DESCRIPTION
MID	Material identification – must be same as the structural material.
PERM	Permeability or load curve ID defining permeability, depending on the definition of PMTYP below. If PERMY and PERMZ are non-zero, then PERM gives the permeability in the global X direction. See Remark 3.
PERMY	Optional permeability or load curve ID defining permeability in the global Y direction, depending on the definition of PMTYP below
PERMZ	Optional permeability or load curve ID defining permeability in the global Z direction, depending on the definition of PMTYP below
THEXP	Undrained volumetric thermal expansion coefficient (see Remark 2): GE.0.0: Constant undrained volumetric thermal expansion coefficient LT.0.0: |THEXP| is the ID of a load curve giving the thermal expansion coefficient (y-axis) as a function of temperature (x-axis).
LCKZ	Load curve giving factor on PERM as a function of z-coordinate
PMTYP	Permeability definition type: EQ.0: PERM is a constant.

VARIABLE	DESCRIPTION
	EQ.1: PERM is a load curve ID giving permeability (y-axis) as a function of the volume ratio of current volume to volume in the stress-free state (x-axis).
	EQ.2: PERM is a load curve ID giving permeability (y-axis) as a function of effective plastic strain (x-axis) of materials other than MAT_072R3. For MAT_072R3, the x-axis is the output selector specified by NOUT; see *MAT_072R3.
	EQ.3: PERM is a load curve ID giving permeability (y-axis) as a function of effective pressure (x-axis) which is positive when in compression.

Remarks:

1. **Permeability Units.** The units of PERM are length/time (volume flow rate of water per unit area per gradient of pore pressure head).

2. **Thermal Expansion.** THEXP represents the thermal expansion of the material caused by the pore fluid (units: 1/temperature). It should be set equal to $n\alpha_w$, where n is the porosity of the soil and α_w is the volumetric thermal expansion coefficient of the pore fluid. If the pore fluid is water, the thermal expansion coefficient varies strongly with temperature; a curve of coefficient as a function of temperature may be input instead of a constant value. Note that this property is for *volumetric* strain increase, whereas regular thermal expansion coefficients (e.g. on *MAT or *MAT_ADD_THERMAL_EXPANSION) are linear, meaning they describe thermal expansion in one direction. The volumetric expansion coefficient is three times the linear thermal expansion coefficient. The regular thermal expansion coefficients apply to the soil skeleton and to drained parts. Pore pressure can be generated due to the difference of expansion coefficients of the soil skeleton and pore fluid, that is, if THEXP is not equal to three times the regular thermal expansion coefficient for the part.

3. **Isotropic/Orthotropic Permeability.** If only PERM is defined and PERMY and PERMZ are left blank or zero, the permeability is isotropic. To obtain orthotropic permeability, define values for PERM, PERMY and PERMZ, giving the permeability in the global X, Y and Z directions respectively.

*MAT_ADD_PORE_AIR

For pore air pressure calculations.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	PA_RHO	PA_PRE	PORE				
Type	A	I	F	F				
Default	none	AIR_RO	AIR_RO	1.				
Remarks	1			1, 2				

Card 2	1	2	3	4	5	6	7	8
Variable	PERM1	PERM2	PERM3	CDARCY	CDF	LCPGD1	LCPGD2	LCPGD3
Type	F	F	F	F	F	I	I	I
Default	0.	PERM1	PERM1	1.	0.	none	LCPGD1	LCPGD1
Remarks	2, 3, 4, 5	2, 3, 4, 5	2, 3, 4, 5	1	1, 5	6	6	6

VARIABLE	DESCRIPTION
MID	Material identification which must be same as the structural material
PA_RHO	Initial density of pore air. The default is the atmospheric air density, AIR_RO, defined in *CONTROL_PORE_AIR
PA_PRE	Initial pressure of pore air. The default is the atmospheric air pressure, AIR_P, defined in *CONTROL_PORE_AIR
PORE	Porosity, meaning the ratio of pores to total volume (default = 1)
PERM*i*	Permeability of pore air along x, y, and z-directions. If less than 0, PERM*i* is taken to be the curve ID defining the permeability coefficient as a function of volume ratio of current volume to volume in

VARIABLE	DESCRIPTION
	the stress free state.
CDARCY	Coefficient of Darcy's law
CDF	Coefficient of Dupuit-Forchheimer law
LCPGD*i*	Curves defining non-linear Darcy's laws along x, y and z-directions

Remarks:

1. **Card 1.** This card must be defined for all materials requiring consideration of pore air pressure. The pressure contribution of pore air is $(\rho - \rho_{\text{atm}})RT\times$ PORE, where ρ and ρ_{atm} are the current and atmospheric air densities, R is air's gas constant, T is atmospheric air temperature and PORE is the porosity. The values for R, T and PORE are assumed to be constant during simulation.

2. **Permeability Model.** The unit of PERM*i* is $[\text{Length}]^3[\text{time}]/[\text{mass}]$, (air flow velocity per gradient of excess pore pressure), i.e.

$$(\text{CDARCY} + \text{CDF} \times |v_i|) \times \text{PORE} \times v_i = \text{PERM}i \times \frac{\partial P_a}{\partial x_i}, \qquad i = 1,2,3$$

   where v_i is the pore air flow velocity along the i^{th} direction, $\partial P_a/\partial x_i$ is the pore air pressure gradient along the i^{th} direction, and $x_1 = x$, $x_2 = y$, $x_3 = z$.

3. **Default Values for PERM2 and PERM3.** PERM2 and PERM3 are assumed to be equal to PERM1 when they are not defined. A definition of "0" means no permeability.

4. **Local Coordinate Systems.** When MID is an orthotropic material, such as *MAT_002 or *MAT_142, (x, y, z), or $(1,2,3)$, refers to the local material coordinate system (a, b, c); otherwise it refers to the global coordinate system.

5. **CDF for Viscosity.** CDF can be used to consider the viscosity effect for high speed air flow.

6. **Nonlinearity.** LCPGD*i* can be used to define a non-linear Darcy's law as follows:

$$(\text{CDARCY} + \text{CDF} \times |v_i|) \times \text{PORE} \times v_i = \text{PERM}i \times f_i \frac{\partial P_a}{\partial x_i}, \qquad i = 1,2,3$$

   where f_i is the value of the function defined by the LCPGD*i* field. The linear version of Darcy's law (see Remark 2) can be recovered when the LCPGD*i* curves are defined as straight lines of slope of 1.

*MAT_ADD_PROPERTY_DEPENDENCE_{OPTION}

Available options include:

FREQ

TIME

The ADD_PROPERTY_DEPENDENCE option defines dependence of a material property on frequency or time.

Card 1	1	2	3					
Variable	MID	PROP	LCID					
Type	A	C	I					
Default	none	none	0					

VARIABLE	DESCRIPTION
MID	Material identification for which the property dependence applies
PROP	Name of the property (same as the variable for a material model in keyword card). For example, "E" is used for Young's modulus in *MAT_ELASTIC. See Remark 4.
LCID	Curve ID to define the property dependence. For the FREQ keyword option, the abscissa values define frequency; for the TIME keyword option, the abscissa values define time. The ordinate values define the property at each frequency or each time

Remarks:

1. **Overview.** This keyword defines how a property (for example, the Young's modulus) of a material changes with frequency (for FREQ option) or with time (for TIME option). Particularly, *MAT_ADD_PROPERTY_DEPENDENCE_-FREQ can be used in direct SSD analysis (*FREQUENCY_DOMAIN_SSD_DIRECT_FREQUENCY_DEPENDENT).

2. **Properties without Frequency/Time Dependence.** Some properties of a material model have no frequency or time dependence. A warning message will be

issued if a dependence curve is defined on a property of a material, which has no frequency or time dependence.

3. **Initial Property Values.** The original property value defined in a material card will be overridden by the property values defined at frequency or time 0 in this keyword. If the starting frequency or time of LCID in this keyword is larger than 0, then the original property value defined in the material card is used until the starting frequency or time of LCID is reached.

4. **Supported Material Models and Properties.** So far, only the Young's modulus (E) of *MAT_ELASTIC is supported by this keyword. More material models (and properties) will be supported in the future.

*MAT_ADD_PZELECTRIC

The ADD_PZELECTRIC option is used to occupy an arbitrary material model in LS-DYNA with a piezoelectric property. This option applies to 4-node solids, 6-node solids, 8-node solids, thick shells, 2D plane strain elements and axisymmetric solids. Orthotropic properties are assumed. This feature is available in SMP since 115324/dev and MPP since 126577/dev. We recommend a double precision executable.

Card Summary:

Card 1. This card is required.

MID	DTYPE	GPT	AOPT				

Card 2. This card is required.

DXX	DYY	DZZ	DXY	DXZ	DYZ		

Card 3. This card is required.

PX11	PX22	PX33	PX12	PX13	PX23	PY11	PY22

Card 4. This card is required.

PY33	PY12	PY13	PY23	PZ11	PZ22	PZ33	PZ12

Card 5. This card is required.

PZ13	PZ23						

Card 6. This card is required.

XP	YP	ZP	A1	A2	A3		

Card 7. This card is required.

			D1	D2	D3		

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	DTYPE	GPT	AOPT				
Type	A	A	F	I				
Default	none	S	8	0				

VARIABLE	DESCRIPTION
MID	Material ID for which the piezoelectric properties apply
DTYPE	Type of piezoelectric property definition (see remarks below) EQ.S: Stress based definition EQ.E: Strain based definition
GPT	Number of Gauss points used for integration: EQ.0: Default value 8, full integration EQ.1: Reduced integration
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.

Card 2	1	2	3	4	5	6	7	8
Variable	DXX	DYY	DZZ	DXY	DXZ	DYZ		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
D$\alpha\beta$	Dielectric permittivity matrix, $d_{\alpha\beta}$. α, $\beta = x$, y, z (see remarks below).

Card 3	1	2	3	4	5	6	7	8
Variable	PX11	PX22	PX33	PX12	PX13	PX23	PY11	PY22
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	PY33	PY12	PY13	PY23	PZ11	PZ22	PZ33	PZ12
Type	F	F	F	F	F	F	F	F

Card 5	1	2	3	4	5	6	7	8
Variable	PZ13	PZ23						
Type	F	F						

VARIABLE	DESCRIPTION
Pαij	Piezoelectric matrix which depends on DTYPE (see remarks below). $\alpha = x$, y, z and i,j = 1, 2, 3.

Card 6	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
XP, YP, ZP	Coordinates of point p for AOPT = 1
A1, A2, A3	Components of vector **a** for AOPT = 2

Card 7	1	2	3	4	5	6	7	8
Variable				D1	D2	D3		
Type				F	F	F		

VARIABLE	DESCRIPTION
D1, D2, D3	Components of vector **d** for AOPT = 2

Remarks:

The stress-based definition for piezoelectric effects is:

$$\sigma_{ij} = k_{ijkl}\varepsilon_{kl} - p_{\alpha ij}E_\alpha$$
$$\Delta_\alpha = p_{\alpha kl}\varepsilon_{kl} + d_{\alpha\beta}E_\beta$$

Here σ_{ij} are the mechanical stresses, k_{ijkl} are the material stiffness constants, ε_{kl} are the material strains, $p_{\alpha ij}$ are the stress-based piezoelectric coefficients, Δ_α are the electric displacements, E_α are the electronic fields, and $d_{\alpha\beta}$ are the dielectric permittivity constants.

The strain-based definition for piezoelectric effects is:

$$\varepsilon_{ij} = f_{ijkl}\sigma_{kl} + P_{\alpha ij}E_\alpha$$
$$\Delta_\alpha = P_{\alpha kl}\sigma_{kl} + d_{\alpha\beta}E_\beta$$

Here f_{ijkl} are the material flexibility parameters and $P_{\alpha ij}$ are the strain-based piezoelectric coefficients.

*MAT_ADD_SOC_EXPANSION

The ADD_SOC_EXPANSION option adds a state of charge (SOC) expansion property to an (arbitrary) material model in LS-DYNA. The state of charge comes from the EM module during a coupled simulation. This option currently only applies to solid elements type -2, -1, 1, 2, and 10 and to hypoelastic material models.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCID	MULT	LCIDY	MULTY	LCIDZ	MULTZ	
Type	I	I	F	I	F	I	F	
Default	none	0	1.0	LCID	MULT	LCID	MULT	

VARIABLE	DESCRIPTION
PID	Part ID for which the SOC expansion property applies
LCID	For isotropic material models, LCID is the load curve ID defining the SOC expansion coefficient as a function of state of charge. In this case, LCIDY, MULTY, LCIDZ, and MULTZ are ignored. For anisotropic material models, LCID and MULT define the SOC expansion coefficient in the local material *a*-direction. In either case, if LCID is zero, the SOC expansion coefficient is constant and equal to MULT.
MULT	Scale factor scaling load curve given by LCID
LCIDY	Load curve ID defining the SOC expansion coefficient in the local material *b*-direction as a function of state of charge. If zero, the SOC expansion coefficient in the local material *b*-direction is constant and equal to MULTY. If MULTY = 0.0 as well, LCID and MULT specify the SOC expansion coefficient in the local material *b*-direction.
MULTY	Scale factor scaling load curve given by LCIDY
LCIDZ	Load curve ID defining the SOC expansion coefficient in the local material *c*-direction as a function of state of charge. If zero, the SOC expansion coefficient in the local material *c*-direction is constant and equal to MULTZ. If MULTZ = 0.0 as well, LCID and MULT specify the SOC expansion coefficient in the local material *c*-

VARIABLE	DESCRIPTION
	direction.
MULTZ	Scale factor scaling load curve given by LCIDZ

Remarks:

When invoking the isotropic SOC expansion property (no local y and z parameters) for a material, the stress update is based on the elastic strain rates given by

$$\dot{\varepsilon}_{ij}^{e} = \dot{\varepsilon}_{ij} - \gamma(\mathrm{SOC})\dot{\mathrm{SOC}} \times \delta_{ij}$$

rather than on the total strain rates, $\dot{\varepsilon}_{ij}$. For orthotropic properties, which apply only to materials with anisotropy, this equation is generalized to

$$\dot{\varepsilon}_{ij}^{e} = \dot{\varepsilon}_{ij} - \gamma_k(\mathrm{SOC})\dot{\mathrm{SOC}}\ q_{ik}q_{jk} \quad .$$

Here q_{ij} represents the matrix with material directions with respect to the current configuration.

*MAT_ADD_THERMAL_EXPANSION

The ADD_THERMAL_EXPANSION option adds a thermal expansion property to an arbitrary material model in LS-DYNA. This option applies to all nonlinear solid, shell, thick shell and beam elements and to all material models except those models which use resultant formulations, such as *MAT_RESULTANT_PLASTICITY and *MAT_SPECIAL_-ORTHOTROPIC. Orthotropic expansion effects are supported for anisotropic materials.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCID	MULT	LCIDY	MULTY	LCIDZ	MULTZ	
Type	I	I	F	I	F	I	F	
Default	none	none	1.0	LCID	MULT	LCID	MULT	

VARIABLE	DESCRIPTION
PID	Part ID for which the thermal expansion property applies
LCID	For isotropic material models, LCID is the load curve ID defining the thermal expansion coefficient as a function of temperature. In this case, LCIDY, MULTY, LCIDZ, and MULTZ are ignored. For anisotropic material models, LCID and MULT define the thermal expansion coefficient in the local material *a*-direction. In either case, if LCID is zero, the thermal expansion coefficient is constant and equal to MULT.
MULT	Scale factor scaling load curve given by LCID
LCIDY	Load curve ID defining the thermal expansion coefficient in the local material *b*-direction as a function of temperature. If zero, the thermal expansion coefficient in the local material *b*-direction is constant and equal to MULTY. If MULTY = 0 as well, LCID and MULT define the thermal expansion coefficient in the local material *b*-direction.
MULTY	Scale factor scaling load curve given by LCIDY
LCIDZ	Load curve ID defining the thermal expansion coefficient in the local material *c*-direction as a function of temperature. If zero, the thermal expansion coefficient in the local material *c*-direction is constant and equal to MULTZ. If MULTZ = 0 as well, LCID and MULT define the thermal expansion coefficient in the local material

VARIABLE	DESCRIPTION
	c-direction.
MULTZ	Scale factor scaling load curve given by LCIDZ

Remarks:

When invoking the isotropic thermal expansion property (no local y and z parameters) for a material, the stress update is based on the elastic strain rates given by

$$\dot{\varepsilon}^e_{ij} = \dot{\varepsilon}_{ij} - \alpha(T)\dot{T}\delta_{ij}$$

rather than on the total strain rates $\dot{\varepsilon}_{ij}$. For a material with the stress based on the deformation gradient, F_{ij}, the elastic part of the deformation gradient is used for the stress computations:

$$F^e_{ij} = J_T^{-1/3} F_{ij} \ ,$$

where J_T is the thermal Jacobian. The thermal Jacobian is updated using the rate given by

$$\dot{J}_T = 3\alpha(T)\dot{T}J_T \ .$$

For orthotropic properties, which apply only to materials with anisotropy, these equations are generalized to

$$\dot{\varepsilon}^e_{ij} = \dot{\varepsilon}_{ij} - \alpha_k(T)\dot{T}q_{ik}q_{jk}$$

and

$$F^e_{ij} = F_{ik}\beta_l^{-1}Q_{kl}Q_{jl} \ ,$$

where the β_i are updated as

$$\dot{\beta}_i = \alpha_i(T)\dot{T}\beta_i \ .$$

Here q_{ij} represents the matrix with material directions with respect to the current configuration whereas Q_{ij} are the corresponding directions with respect to the initial configuration. For (shell) materials with multiple layers of different anisotropy directions, the mid surface layer determines the orthotropy for the thermal expansion.

*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, three-dimensional shell elements, and thick shell elements. This option applies to a subset of elasto-plastic materials that include a damage-based failure criterion.

Card 1	1	2	3	4	5	6	7	8
Variable	IDNL	PID	P	Q	L	NFREQ	NHV	NHVT
Type	I	I	F	F	F	I	I	I
Default	none	none	none	none	none	none	none	none

History Cards. Include as many cards as needed to identify the NHV and NHVT history variables. *One card 2 will be read, even if both NHV and NHVT are zero.* If only NHV > 0, then NL*i* are assumed to be incremental variables. If only NHVT > 0, then NL*i* are assumed to be non-incremental variables. If both NHV and NHVT are nonzero, then NHV variables will be read starting at Card 2, and NHVT variables will be read starting on a new line.

Card 2	1	2	3	4	5	6	7	8
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

Symmetry Plane Cards. Define one card for each symmetry plane. Up to six symmetry planes can be defined. The next keyword ("*") card terminates this input.

Cards 3	1	2	3	4	5	6	7	8
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 Remark 4.
Q	Exponent of weighting function. A typical value might be 2. See Remark 4.
L	Characteristic length. This length should span a few elements. See Remark 4.
NFREQ	Number of time steps between searching for integration points that lie in the neighborhood. Nonlocal smoothing will be done each cycle using these neighbors until the next search is done. The neighbor search can add significant computational time, so it is suggested that NFREQ be set to a value between 10 and 100 depending on the problem. This parameter may be somewhat problem dependent. If NFREQ = 0, a single search will be done at the start of the calculation.
NHV	Number of variables with nonlocal treatment of increments. See Remark 1
NHVT	Number of variables with nonlocal treatment of total values. See Remark 1.
NL1, …, NL8	Identifies the history variable(s) for nonlocal treatment. Define NHV + NHVT values (maximum of 8 values per line). See Remark 2.

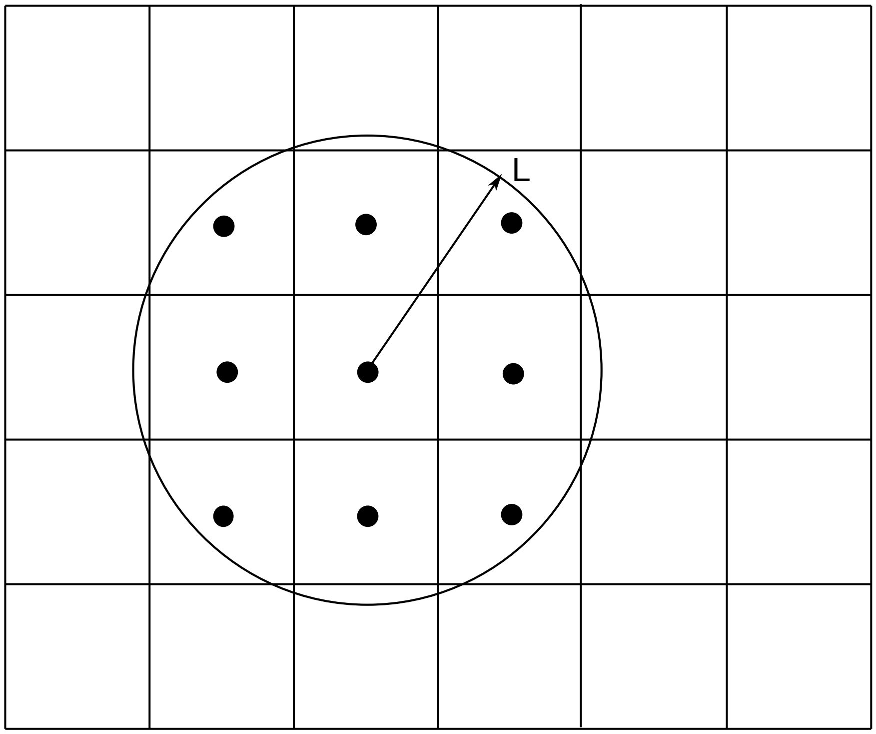


Figure 2-9. Here $\dot{f}_r$ and x_r are respectively the nonlocal rate of increase of damage and the center of the element e_r, and $\dot{f}^i_{\text{local}}$, V_i and y_i are respectively the local rate of increase of damage, the volume and the center of element e_i.

VARIABLE	DESCRIPTION
XC1, YC1, ZC1	Coordinate of point on symmetry plane
XC2, YC2, ZC2	Coordinate of a point along the normal vector

Remarks:

1. **NHV and NHVT.** NHV is a count of the number of variables for which increments of the variable are used in the nonlocal function. NHVT is a count of the number of variables for which the whole value of the variable is used in the nonlocal function. NHVT type variables would be used only if the variable is itself an increment of some value which is rare. Many history variables are calculated by a sum of increments, but since the variable is the sum, one would include this variable in the NHV type variables for nonlocal treatment so that only the increments are modified.

2. **History Variables.** For elastoplastic material models in LS-DYNA which use the plastic strain as a failure criterion, setting the variable NL1 to 1 would flag plastic strain for nonlocal treatment. A sampling of other history variables that can be flagged for nonlocal treatment are listed in the table below. The value in

the third column in the table below corresponds to the history variable number as tabulated at http://www.dynasupport.com/howtos/material/history-variables. Note that the NL*n* value is the history variable number plus 1.

Material Model Name		*MAT_NONLOCAL NL*n* Value	History Variable Number
JOHNSON_COOK	15	5 (shells); 7 (solids)	4 (shells); 6 (solids)
PLASTICITY_WITH_DAMAGE	81	2	1
DAMAGE_1	104	4	3
DAMAGE_2	105	2	1
JOHNSON_HOLMQUIST_CONCRETE	111	2	1
GURSON	120	2	1

3. **Integration Points and Nonlocal Equations.** When applying the nonlocal equations to shell and thick 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, so that, for example, the outer integration points lie on the same surface.

4. **Nonlocal Equations.** The equations and our implementation are based on the implementation by Worswick and Lalbin [1999] of the nonlocal theory to Pijaudier-Cabot and Bazant [1987]. Let Ω_r be the neighborhood of radius, L, of element e_r and $\{e_i\}_{i=1,\dots,N_r}$ the list of elements included in Ω_r, then

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

where

$$W_r = W(x_r) = \int w(x_r - y)\, dy \approx \frac{1}{W_r}\sum_{i=1}^{N_r} w_{ri} V_i$$

$$w_{ri} = w(x_r - y_i) = \frac{1}{\left[1 + \left(\frac{\|x_r - y_i\|}{L}\right)^p\right]^q}$$

*MAT_ELASTIC_{OPTION}

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

Available options include:

<BLANK>

FLUID

such that the keyword cards appear as:

*MAT_ELASTIC or MAT_001

*MAT_ELASTIC_FLUID or MAT_001_FLUID

The fluid option is valid for solid elements only.

Card Summary:

Card 1. This card is required.

MID	RO	E	PR	DA	DB	K	

Card 1.1. Include this card when E < 0.0.

EFUNC	CNVT	ITERLM					

Card 2. Include this card when using the FLUID keyword option.

VC	CP						

Data Card Defintions:

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Definition of Young's modulus GT.0: E is the Young's modulus. LT.0: |E| is the ID of a curve defining Young's modulus as a function of elemental variables, EFUNC. It is supported for explicit simulation only.
PR	Poisson's ratio
DA	Axial damping factor (used for Belytschko-Schwer beam, type 2, only).
DB	Bending damping factor (used for Belytschko-Schwer beam, type 2, only).
K	Bulk modulus (define for fluid option only).

Additional card for E < 0.

Card 1.1	1	2	3	4	5	6	7	8
Variable	EFUNC	CNVT	ITERLM					
Type	A	F	I					
Default	P	10^{-3}	3					

VARIABLE	DESCRIPTION
EFUNC	Element variable used as the independent variable of curve |E|: EQ.P: Elemental pressure
CNVT	Convergence tolerance. It is needed when EFUNC is a variant of stress.
ITERLM	Iteration limit. It is needed when EFUNC is a variant of stress.

Additional card for FLUID keyword option.

Card 2	1	2	3	4	5	6	7	8
Variable	VC	CP						
Type	F	F						
Default	none	10^{20}						

VARIABLE	DESCRIPTION
VC	Tensor viscosity coefficient. Values between .1 and .5 should be okay.
CP	Cavitation pressure (default = 10^{20}).

Remarks:

1. **Finite strains.** This hypoelastic material model may not be stable for finite (large) strains. If large strains are expected, a hyperelastic material model, such as *MAT_002, would be more appropriate.

2. **Damping factors.** 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:

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

3. **Effective plastic strain.** The history variable labeled as “effective plastic strain” by LS-PrePost is volumetric strain in the case of *MAT_ELASTIC.

4. **Truss elements and damping stress.** Truss elements include a damping stress given by

$$\sigma = 0.05\rho cL/\Delta t$$

where ρ is the mass density, c is the material wave speed, L is the element length, and Δt is the computation time step.

If the damping stress is undesired, it can be switched off with IRATE = 2 on *CONTROL_IMPLICIT_DYNAMICS.

5. **FLUID keyword option.** For the FLUID keyword option, the bulk modulus field, K, must be defined, and both the Young's modulus and Poisson's ratio fields are ignored. 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:

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

where ΔL 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. For orthotropic solids an isotropic frictional damping is available.

Depending on the element type and solver, the implementation of this material model changes. See the theory manual for more details than the overview provided here. In the case of solids with an explicit solver or nonlinear implicit solver (meaning NSOLVR ≠ 1 on *CONTROL_IMPLICIT_SOLUTION), the model is the (hyperelastic) St. Venant-Kirchhoff model. The stress update is performed using the second Piola-Kirchhoff tensor. It is then transformed into the Cauchy stress for output. For shells (and this includes the 2D continuum elements, that is, shell types 13, 14, and 15), the model is implemented in the local coordinates of the shell as linear elasticity for explicit and nonlinear implicit. While the material response is linear, the shells themselves can undergo finite rotations consistent with applied forces. For the linear implicit solver, this material model is a linear elasticity model.

> **NOTE:** This material does not support specification of a material angle, β_i, for each through-thickness integration point of a shell.

Available options include:

ORTHO

ANISO

such that the keyword cards appear:

*MAT_ORTHOTROPIC_ELASTIC or MAT_002	(4 cards follow)
*MAT_ANISOTROPIC_ELASTIC or MAT_002_ANIS	(5 cards follow)

Card Summary:

Card 1a.1. This card is required for the ORTHO keyword option.

MID	RO	EA	EB	EC	PRBA	PRCA	PRCB

Card 1a.2. This card is required for the ORTHO keyword option.

GAB	GBC	GCA	AOPT	G	SIGF		

Card 1b.1. This card is required for the ANISO keyword option.

MID	RO	C11	C12	C22	C13	C23	C33

Card 1b.2. This card is required for the ANISO keyword option.

C14	C24	C34	C44	C15	C25	C35	C45

Card 1b.3. This card is required for the ANISO keyword option.

C55	C16	C26	C36	C46	C56	C66	AOPT

Card 2. This card is required.

XP	YP	ZP	A1	A2	A3	MACF	IHIS

Card 3. This card is required.

V1	V2	V3	D1	D2	D3	BETA	REF

Data Card Definitions:

Orthotropic Card 1. Included for ORTHO keyword option.

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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 (nonzero value required but not used for shells)
PRBA	ν_{ba}, Poisson's ratio in the ba direction

VARIABLE	DESCRIPTION
PRCA	ν_{ca}, Poisson's ratio in the *ca* direction
PRCB	ν_{cb}, Poisson's ratio in the *cb* direction

Orthotropic Card 2. Included for ORTHO keyword option.

Card 1a.2	1	2	3	4	5	6	7	8
Variable	GAB	GBC	GCA	AOPT	G	SIGF		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
GAB	G_{ab}, shear modulus in the *ab* direction
GBC	G_{bc}, shear modulus in the *bc* direction
GCA	G_{ca}, shear modulus in the *ca* direction
AOPT	Material axes option (see Figure M2-1 and the Material Directions section):

EQ.0.0: Locally orthotropic with material axes determined by element nodes as shown in Figure M2-1. The **a**-direction is from node 1 to node 2 of the element. The **b**-direction is orthogonal to the **a**-direction and is in the plane formed by nodes 1, 2, and 4. When this option is used in two-dimensional planar and axisymmetric analysis, it is critical that the nodes in the element definition be numbered counterclockwise for this option to work correctly. For shells only, the material axes are then rotated about the normal vector to the surface of the shell by the angle BETA.

EQ.1.0: Locally orthotropic with material axes determined by a point, P, 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 **a** and **d** input below, as with *DEFINE_COORDINATE_VECTOR

EQ.3.0: Locally orthotropic material axes determined by a

VARIABLE	DESCRIPTION
	vector **v** and the normal vector to the plane of the element (see Figure M2-1). 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then **a** and **b** are rotated about **c** by an angle BETA. BETA may be set in the keyword input for the element or in the input for this keyword. Note that for solids, the material axes may be switched depending on the choice of MACF. The switch may occur before or after applying BETA depending on the value of MACF. EQ.4.0: Locally orthotropic in a 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. LT.0.0: |AOPT| is a coordinate system ID (see *DEFINE_COORDINATE_*OPTION*).
G	Shear modulus for frequency independent damping. Frequency independent damping is based on a spring and slider in series. The critical stress for the slider mechanism is SIGF. For the best results, the value of G should be 250-1000 times greater than SIGF. This option applies only to solid elements.
SIGF	Limit stress for frequency independent, frictional, damping

Anisotropic Card 1. Included for ANISO keyword option.

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

Anisotropic Card 2. Included for ANISO keyword option.

Card 1b.2	1	2	3	4	5	6	7	8
Variable	C14	C24	C34	C44	C15	C25	C35	C45
Type	F	F	F	F	F	F	F	F

Anisotropic Card 3. Included for ANISO keyword option.

Card 1b.3	1	2	3	4	5	6	7	8
Variable	C55	C16	C26	C36	C46	C56	C66	AOPT
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
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.
AOPT	Material axes option (see Figure M2-1 and the Material Directions section): EQ.0.0: Locally orthotropic with material axes determined by element nodes as shown in Figure M2-1. The **a**-direction is from node 1 to node 2 of the element. The **b**-direction is orthogonal to the **a**-direction and is in the plane formed by nodes 1, 2, and 4. When this option is used in two-dimensional planar and axisymmetric analysis, it is critical that the nodes in the element definition be numbered counterclockwise for this option to work correctly. For shells only, the material axes are then rotated

VARIABLE	DESCRIPTION
	about the normal vector to the surface of the shell by the angle BETA.
	EQ.1.0: Locally orthotropic with material axes determined by a point, P, 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 **a** and **d** input below, as with *DEFINE_COORDINATE_VECTOR
	EQ.3.0: Locally orthotropic material axes determined by a vector **v** and the normal vector to the plane of the element (see Figure M2-1). 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then **a** and **b** are rotated about **c** by an angle BETA. BETA may be set in the keyword input for the element or in the input for this keyword. Note that for solids, the material axes may be switched depending on the choice of MACF. The switch may occur before or after applying BETA depending on the value of MACF.
	EQ.4.0: Locally orthotropic in a 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.
	LT.0.0: |AOPT| is a coordinate system ID (see *DEFINE_COORDINATE_*OPTION*).

Local Coordinate System Card 1. Required for all keyword options.

Card 2	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3	MACF	IHIS
Type	F	F	F	F	F	F	I	I

VARIABLE	DESCRIPTION
XP, YP, ZP	Coordinates of point *P* for AOPT = 1 and 4
A1, A2, A3	Components of vector **a** for AOPT = 2
MACF	Material axes change flag for solid elements: EQ.-4: Switch material axes *b* and *c* before BETA rotation EQ.-3: Switch material axes *a* and *c* before BETA rotation EQ.-2: Switch material axes *a* and *b* before BETA rotation EQ.1: No change, default EQ.2: Switch material axes *a* and *b* after BETA rotation EQ.3: Switch material axes *a* and *c* after BETA rotation EQ.4: Switch material axes *b* and *c* after BETA rotation Figure M2-2 indicates when LS-DYNA applies MACF when finding the final material axes. If BETA on *ELEMENT_SOLID_*{OPTION}* is defined, then that BETA rotates the axes for all AOPT options. Otherwise, unless AOPT = 3, the material axes will be switched as specified by MACF, but no BETA rotation will occur. If AOPT = 3, then BETA input on Card 3 rotates the axes.
IHIS	Flag for anisotropic stiffness terms initialization (for solid elements only): EQ.0: C11, C12, … from Cards 1b.1, 1b.2, and 1.b3 are used. EQ.1: C11, C12, … are initialized with history data from *INITIAL_STRESS_SOLID.

Local Coordinate System Card 2. Required for all keyword options.

Card 3	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	REF
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
V1, V2, V3	Components of vector **v** for AOPT = 3 and 4.
D1, D2, D3	Components of vector **d** for AOPT = 2.

VARIABLE	DESCRIPTION
BETA	Angle in degrees of a material rotation about the c-axis, available for AOPT = 0 (shells and tshells only) and AOPT = 3 (all element types). This angle may be overridden on the element card; see *ELEMENT_SHELL_BETA, *ELEMENT_TSHELL_BETA, and *ELEMENT_SOLID_ORTHO.
REF	Flag to use reference geometry specified with *INITIAL_FOAM_REFERENCE_GEOMETRY to initialize the stress tensor. EQ.0.0: Off EQ.1.0: On

Remarks:

1. **Stress-strain material law.** The material law that relates stresses to strains is defined as:

$$\mathbf{C} = \mathbf{T}^{\mathrm{T}}\mathbf{C}_L\mathbf{T}$$

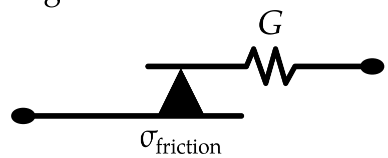
where $\mathbf{T}$ is a transformation matrix, and $\mathbf{C}_L$ is the constitutive matrix defined in terms of the material constants of the orthogonal material axes, $\{\mathbf{a}, \mathbf{b}, \mathbf{c}\}$. The inverse of $\mathbf{C}_L$ for the orthotropic case is defined as:

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

where,

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

2. **Frequency-independent damping.** Frequency independent damping is obtained by having a spring and slider in series as shown in the following sketch:



This option applies only to orthotropic solid elements and affects only the deviatoric stresses.

3. **Poisson's ratio.** PRBA is the minor Poisson's ratio if EA > EB, and the major Poisson's ratio will be equal to PRBA(EA/EB). If EB > EA, then PRBA is the major Poisson's ratio. PRCA and PRCB are similarly defined. They are the minor Poisson's ratio if EA > EC or EB > EC, and the major Poisson's ratio if the EC > EA or EC > EB.

   Care should be taken when using material parameters from third party products regarding the directional indices *a*, *b* and *c*, as they may differ from the definition used in LS-DYNA.

4. **History variables.** This material has the following history variables. Note that for shells and thick shells with element formulations 1, 2, and 6 the history variable labeled as "effective plastic strain" by LS-PrePost is stiffness component C_{11}.

History Variable #	Description (solids and thick shells 3, 5, and 7)	Description (shells and thick shells 1, 2 and 6)
1	Deformation gradient component F_{11}	Stiffness component C_{12}
2	Deformation gradient component F_{12}	Stiffness component C_{13}
3	Deformation gradient component F_{13}	Stiffness component C_{14}
4	Deformation gradient component F_{21}	Stiffness component C_{22}
5	Deformation gradient component F_{22}	Stiffness component C_{23}
6	Deformation gradient component F_{23}	Stiffness component C_{24}
7	Deformation gradient component F_{31}	Stiffness component C_{33}
8	Deformation gradient component F_{32}	Stiffness component C_{34}
9	Deformation gradient component F_{33}	Stiffness component C_{44}

History Variable #	Description (solids and thick shells 3, 5, and 7)	Description (shells and thick shells 1, 2 and 6)
10		Stiffness component C_{55}
11		Stiffness component C_{56}
12		Stiffness component C_{66}

Material Directions:

We will give an overview of how LS-DYNA finds the principal material directions for solid and shell elements for this material and other anisotropic materials based on the input. We will call the material coordinate system the $\{\mathbf{a}, \mathbf{b}, \mathbf{c}\}$ coordinate system. The AOPT options illustrated in Figure M2-1 define the preliminary $\{\mathbf{a}, \mathbf{b}, \mathbf{c}\}$ system for all elements of the parts that use the material, but this is not the final material direction. The $\{\mathbf{a}, \mathbf{b}, \mathbf{c}\}$ system defined by the AOPT options may be offset by a final rotation about the **c**-axis. The offset angle we call BETA. Note that *ELEMENT_SOLID_ORTHO and *ELEMENT_SHELL_MCID allow you to set the preliminary $\{\mathbf{a}, \mathbf{b}, \mathbf{c}\}$ coordinate system for individual solid and shell elements, instead of using the preliminary system created with AOPT. Figures M2-2 and M2-3 give the flow chart for finding the final material direction based on the input to the *MAT keyword and the *ELEMENT keyword. As indicated in the figures, the orientation of the final material axes is updated continuously through the simulation as the element moves and deforms, in accordance with the invariant numbering scheme (INN in *CONTROL_ACCURACY).

For solid elements, the BETA angle is specified in one of two ways. When using AOPT = 3, the BETA parameter defines the offset angle for all elements that use the material. The BETA parameter on *MAT has no meaning for the other AOPT options. Alternatively, a BETA angle can be defined for individual solid elements as described in Remark 5 for *ELEMENT_SOLID_ORTHO. The BETA angle set using the ORTHO option is available for all values of AOPT, and it overrides the BETA angle on the *MAT card for AOPT = 3 (see Figure M2-2). Note that when the BETA angle is applied in either case depends on the value of MACF (if available) of the *MAT keyword. With MACF two of the material directions may be switched.

There are two fundamental differences between shell and solid element orthotropic materials. (In the following discussion, tshell elements fall into the “shell” category.) First, the **c**-direction is always normal to a shell element such that the **a** and **b**-directions are within the plane of the element. Second, for some anisotropic materials, shell elements may have unique fiber directions within each layer through the thickness of the element so that a layered composite can be modeled with a single element.

As a result of the **c**-direction being defined by the element normal, MACF does not apply to shells. Similarly, AOPT = 1 and AOPT = 4 are not available for shells. Also, AOPT = 2 requires only the vector **a** be specified since **d** is not used. The shell procedure projects the inputted **a**-direction onto each element surface.

Note that when AOPT = 0 is used in two-dimensional planar and axisymmetric analysis, it is critical that the nodes in the element definition be numbered counterclockwise for this option to work correctly.

Similar to solid elements, the $\{\mathbf{a}, \mathbf{b}, \mathbf{c}\}$ coordinate system determined by AOPT is then modified by a rotation about the **c**-axis which we will call ϕ. For those materials that allow a unique rotation angle for each integration point through the element thickness, the rotation angle is calculated by

$$\phi_i = \beta + \beta_i \ ,$$

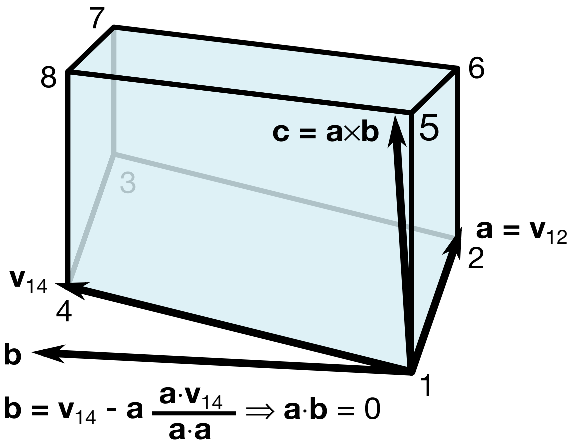
where β is a rotation for the element, and β_i is the rotation for the i^{th} layer of the element. The β angle can be input using the BETA parameter on the *MAT data but will be overridden for individual elements if *ELEMENT_SHELL_BETA (*ELEMENT_TSHELL_BETA) is used. The β_i angles are input using the ICOMP = 1 option of *SECTION_SHELL (*SECTION_TSHELL) or with *PART_COMPOSITE (*PART_COMPOSITE_TSHELL). If β or β_i is omitted, they are assumed to be zero.

All anisotropic shell materials have the BETA parameter on the *MAT card available for both AOPT = 0 and AOPT = 3, except for materials 91 and 92 which have it available (but called FANG instead of BETA) for AOPT = 0, 2, and 3.

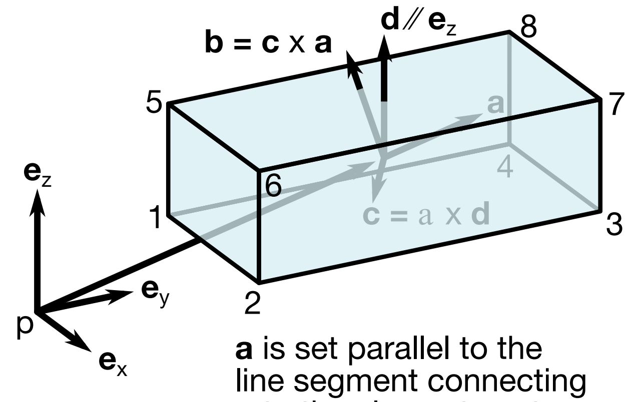
All anisotropic shell materials allow an angle for each integration point through the thickness, β_i, *except for* materials 2, 86, 91, 92, 117, 130, 170, 172, and 194.

Illustration of AOPT: Figure M2-1

AOPT = 0.0



AOPT = 1.0 (solid only)

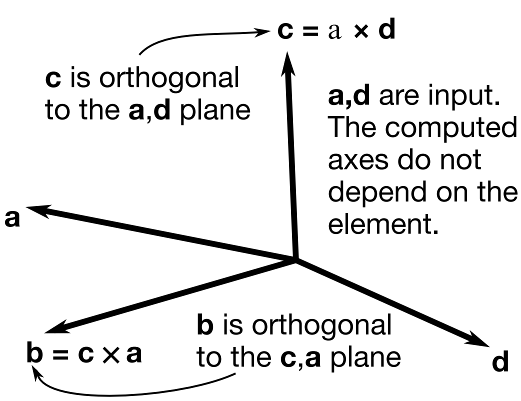


a is set parallel to the line segment connecting p to the element center.

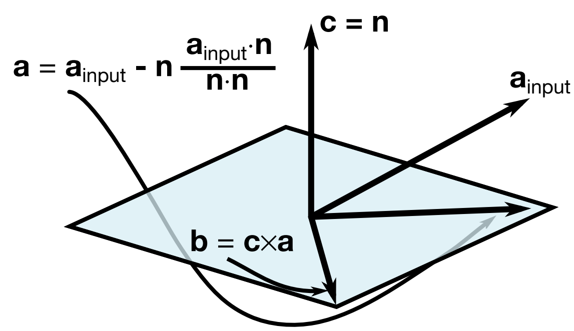
d is set parallel to $\mathbf{e}_z$.

input(p) → {**a**} → {**c**} → {**b**}

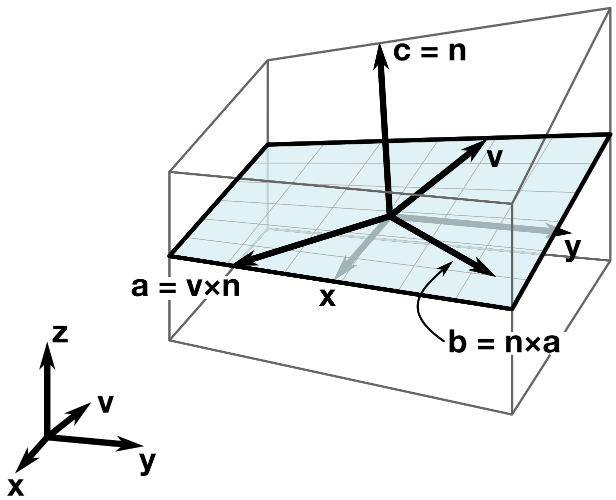
AOPT = 2.0 (solid)



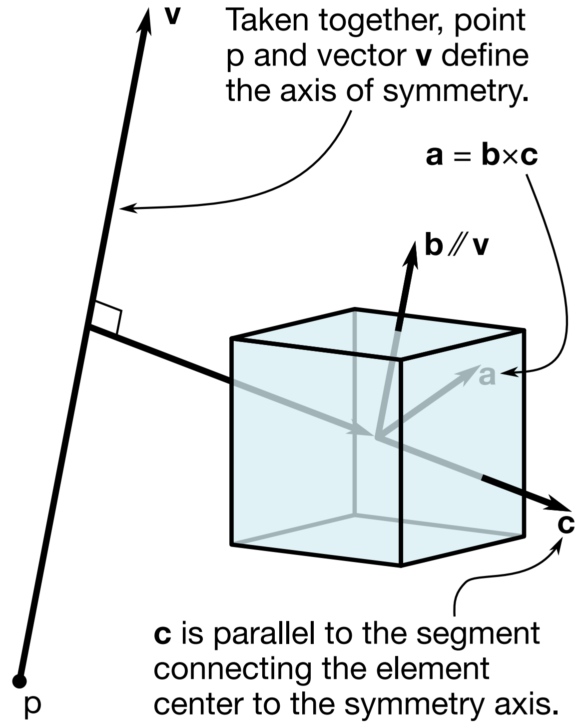
AOPT = 2.0 (shell)



AOPT = 3.0



AOPT = 4.0 (solid only)



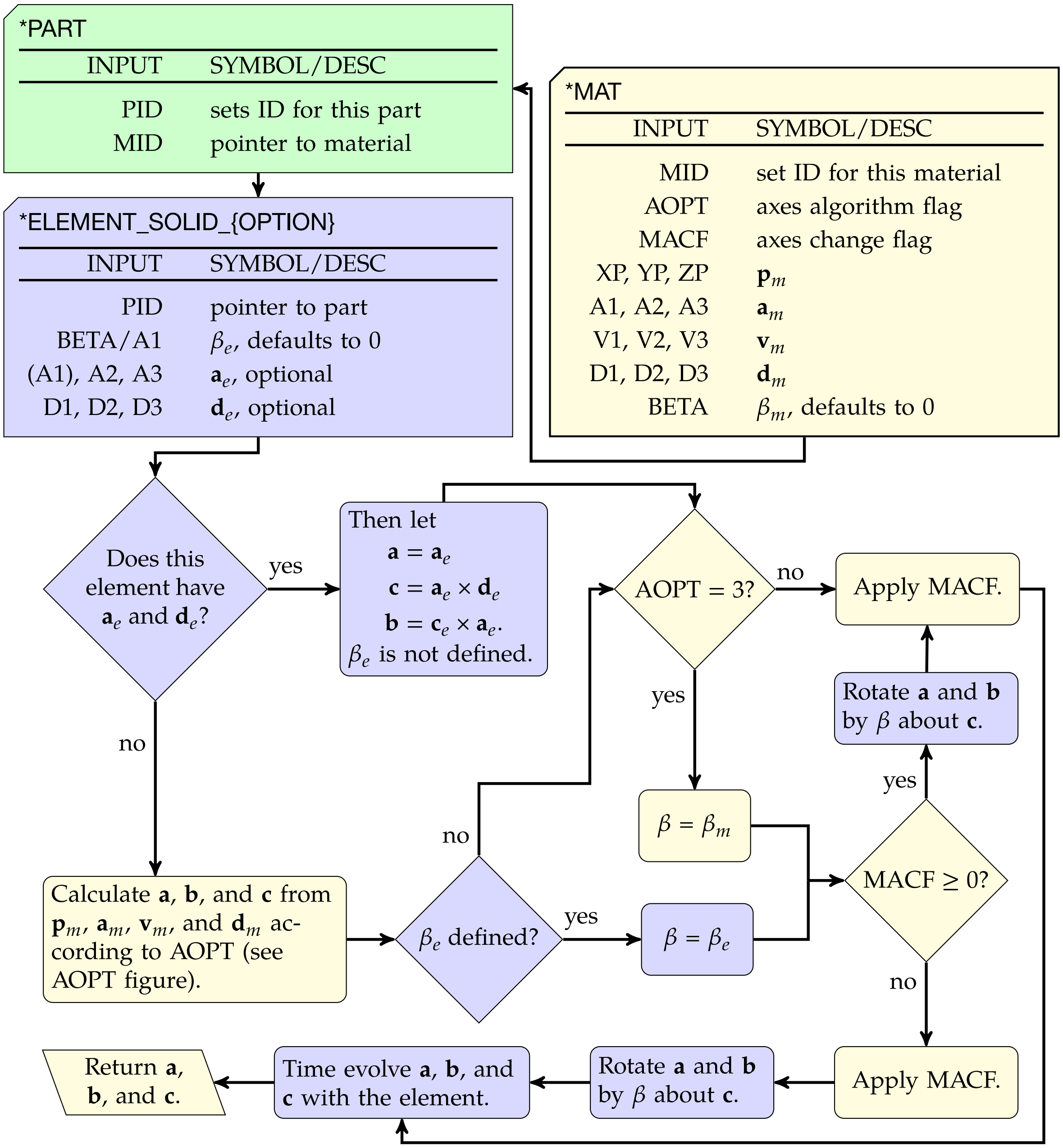


Figure M2-2. Flowchart showing how for each solid element LS-DYNA determines the vectors $\{\mathbf{a}, \mathbf{b}, \mathbf{c}\}$ from the input.

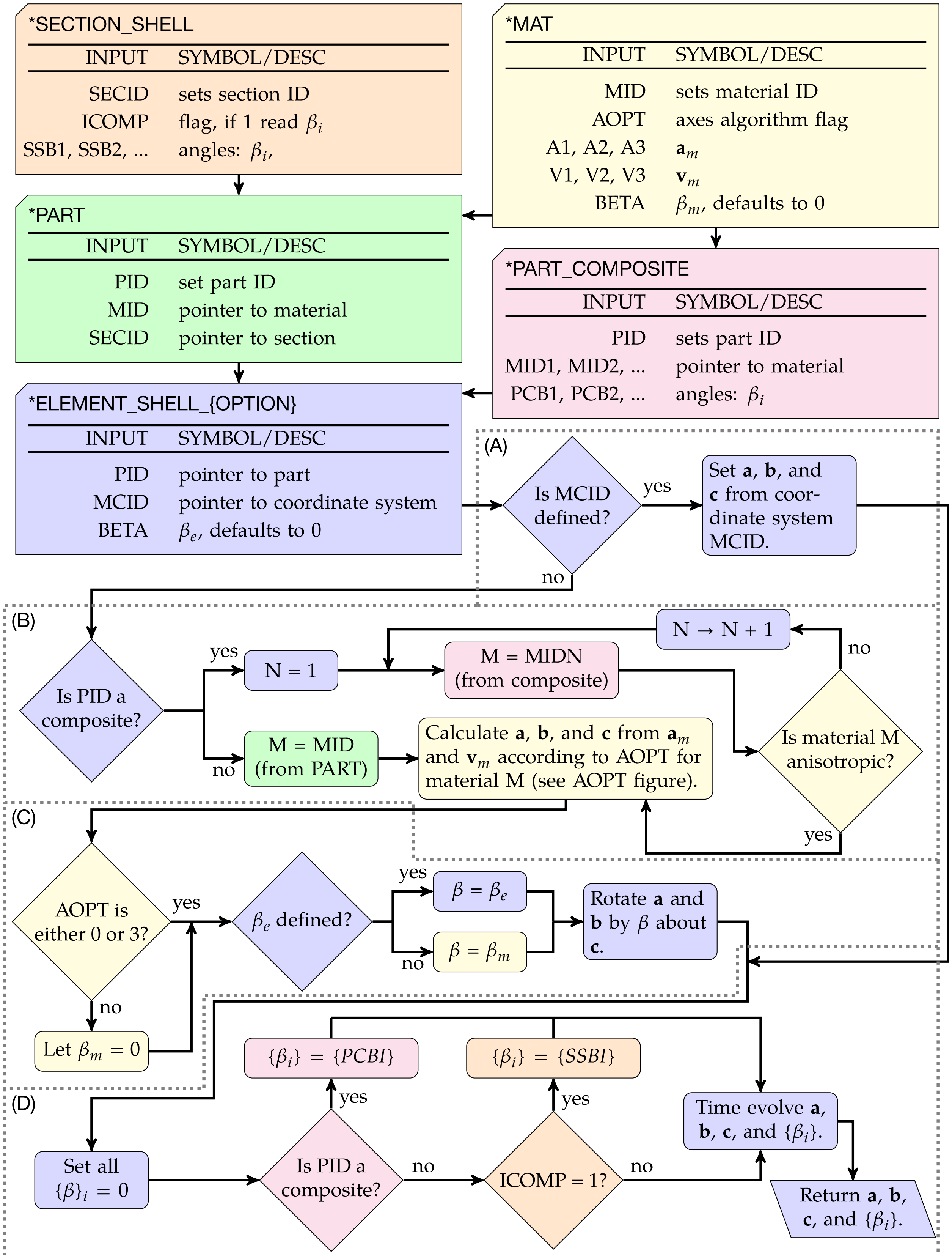


Figure M2-3. Flowchart for shells: (a) check for coordinate system ID; (b) process AOPT; (c) determine β; and (d) for each layer determine β_i.

*MAT_PLASTIC_KINEMATIC

This is Material Type 3. This model is suited for modelling 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 and Truss), shell, and solid elements.

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

Card 2	1	2	3	4	5	6	7	8
Variable	SRC	SRP	FS	VP				
Type	F	F	F	F				
Default	0.0	0.0	10^{20}	0.0				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Tangent modulus; see Figure M3-1.
BETA	Hardening parameter, $0 < \beta' < 1$. See Remark 2.
SRC	Strain rate parameter, C, for the Cowper Symonds strain rate model; see Remark 1. If zero, rate effects are not considered.

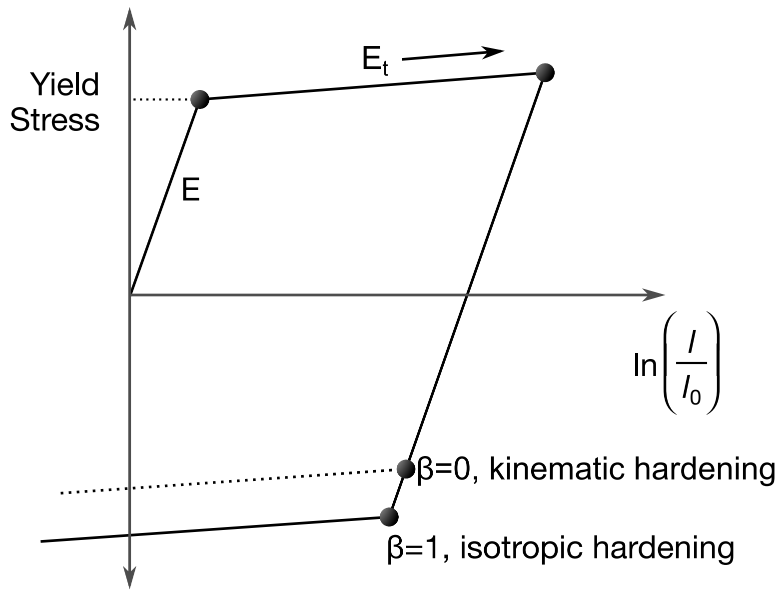


Figure M3-1. Elastic-plastic behavior with kinematic and isotropic hardening where l_0 and l are undeformed and deformed lengths of uniaxial tension specimen. E_t is the slope of the bilinear stress strain curve.

VARIABLE	DESCRIPTION
SRP	Strain rate parameter, p, for Cowper Symonds strain rate model; see Remark 1. If zero, rate effects are not considered.
FS	Effective plastic strain for eroding elements
VP	Formulation for rate effects: EQ.0.0: scale yield stress (default) EQ.1.0: viscoplastic formulation (recommended)

Remarks:

1. **Cowper Symonds Strain Rate Model.** 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. To ignore strain rate effects set both SRC and SRP to zero.

2. **Hardening Parameter.** 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 M3-1. For isotropic hardening, $\beta' = 1$, Material Model 12, *MAT_ISOTROPIC_ELASTIC_PLASTIC, requires less storage and is more efficient. Whenever possible, Material 12 is recommended for solid elements, but for shell elements, it is less accurate and thus Material 12 is not recommended in this case.

3. **History Variables.** This material has the following extra history variables.

History Variable #	Description
1	Back stress component *xx*
2	Back stress component *yy*
3	Back stress component *xy*
4	Back stress component *yz*
5	Back stress component *zx*

*MAT_ELASTIC_PLASTIC_THERMAL

This is Material Type 4. Temperature dependent material coefficients can be defined with this material. A maximum of eight temperatures with the corresponding data can be defined, but a minimum of two points is required. When this material type is used, it is necessary to define nodal temperatures by activating a coupled analysis or by using another option to define the temperatures, such as *LOAD_THERMAL_LOAD_CURVE, or *LOAD_THERMAL_VARIABLE.

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

Card 2	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 3	1	2	3	4	5	6	7	8
Variable	E1	E2	E3	E4	E5	E6	E7	E8
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	PR1	PR2	PR3	PR4	PR5	PR6	PR7	PR8
Type	F	F	F	F	F	F	F	F

Card 5	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 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	1	2	3	4	5	6	7	8
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 or label must be specified (see *PART).
RO	Mass density
T*i*	Temperatures. The minimum required is 2 while the maximum allowed is 8.
E*i*	Corresponding Young's modulus at temperature T*i*
PR*i*	Corresponding Poisson's ratio at temperature T*i*
ALPHA*i*	Corresponding coefficient of thermal expansion at temperature T*i*
SIGY*i*	Corresponding yield stress at temperature T*i*
ETAN*i*	Corresponding plastic hardening modulus at temperature T*i*

Remarks:

1. **Material Model.** The stress update for this material follows the standard approach for hypo-elastoplasticity, using the Jaumann rate for objectivity. The rate of Cauchy stress $\boldsymbol{\sigma}$ can be expressed as

$$\dot{\boldsymbol{\sigma}} = \mathbf{C}\left(\dot{\boldsymbol{\varepsilon}} - \dot{\boldsymbol{\varepsilon}}_T - \dot{\boldsymbol{\varepsilon}}_p\right) + \dot{\mathbf{C}}\mathbf{C}^{-1}\boldsymbol{\sigma} \quad ,$$

where $\mathbf{C}$ is the temperature dependent isotropic elasticity tensor, $\dot{\boldsymbol{\varepsilon}}$ is the rate-of-deformation, $\dot{\boldsymbol{\varepsilon}}_T$ is the thermal strain rate, and $\dot{\boldsymbol{\varepsilon}}_p$ is the plastic strain rate. The coefficient of thermal expansion is defined as the instantaneous value. Thus, the thermal strain rate becomes

$$\dot{\boldsymbol{\varepsilon}}_T = \alpha \dot{T}\mathbf{I}$$

where α is the temperature dependent thermal expansion coefficient, $\dot{T}$ is the rate of temperature and $\mathbf{I}$ is the identity tensor. Associated von Mises plasticity is adopted, resulting in

$$\dot{\boldsymbol{\varepsilon}}_p = \dot{\varepsilon}_p \frac{3\mathbf{s}}{2\bar{\sigma}}$$

where $\dot{\varepsilon}_p$ is the effective plastic strain rate, $\mathbf{s}$ is the deviatoric stress tensor, and $\bar{\sigma}$ is the von Mises effective stress. The last term accounts for stress changes due to change in stiffness with respect to temperature, using the total elastic strain defined as $\boldsymbol{\varepsilon}_e = \mathbf{C}^{-1}\boldsymbol{\sigma}$. As a way to intuitively understand this contribution, the special case of small displacement elasticity neglecting everything but the temperature dependent elasticity parameters gives

$$\dot{\boldsymbol{\sigma}} = \frac{d}{dt}(\mathbf{C}\boldsymbol{\varepsilon}) \quad ,$$

showing that the stress may change without any change in strain.

2. **Model Requirements and Restrictions.** 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 thermo-elastic material is considered, do not define SIGY and ETAN.

3. **History Variables.** This material has the following extra history history variables.

History Variable #	Description
1	Reference temperature
3	Current temperature

*MAT_SOIL_AND_FOAM

This is Material Type 5. It is a relatively simple material model for representing soil, concrete, or crushable foam. A table can be defined if thermal effects are considered in the pressure as a function of volumetric strain behavior.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	KUN	A0	A1	A2	PC
Type	A	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	VCR	REF	LCID					
Type	F	F	F					

Card 3	1	2	3	4	5	6	7	8
Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	EPS9	EPS10						
Type	F	F						

Card 5	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

Card 6	1	2	3	4	5	6	7	8
Variable	P9	P10						
Type	F	F						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
G	Shear modulus
KUN	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 (< 0)
VCR	Volumetric crushing option: EQ.0.0: on EQ.1.0: loading and unloading paths are the same
REF	Use reference geometry to initialize the pressure. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY. This option does not initialize the deviatoric stress state. EQ.0.0: off EQ.1.0: on
LCID	Load curve ID for compressive pressure (ordinate) as a function of volumetric strain (abscissa). If LCID is defined, then the curve is used instead of the input for EPS*i* and P*i*. It makes no difference whether the values of volumetric strain in the curve are input as positive or negative since internally, a negative sign is applied to the absolute value of each abscissa entry. If LCID refers to a table, the response is extended to being temperature dependent.

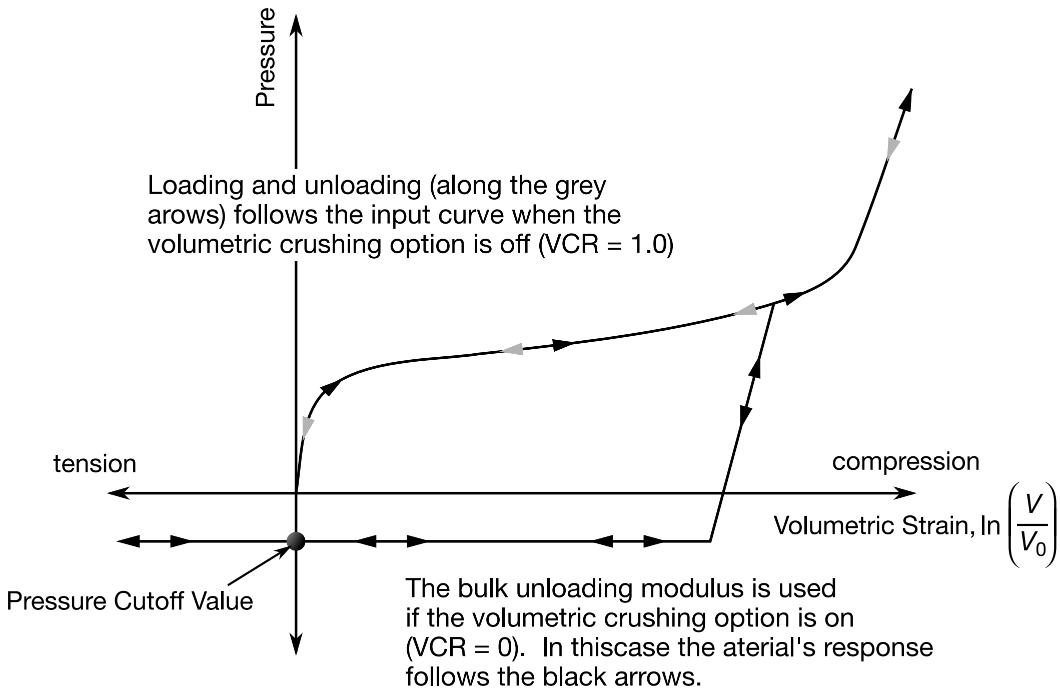


Figure M5-1. Pressure as a function of volumetric strain curve for soil and crushable foam model. The volumetric strain is given by the natural logarithm of the relative volume, V.

VARIABLE	DESCRIPTION
EPS1, …	Volumetric strain values in pressure as a function of volumetric strain curve (see Remarks below). A maximum of 10 values including 0.0 are allowed and a minimum of 2 values are necessary. If EPS1 is not 0.0, then a point $(0.0, 0.0)$ will be automatically generated and a maximum of nine values may be input.
P1, P2, …, PN	Pressures corresponding to volumetric strain values given on Cards 3 and 4.

Remarks:

1. **Variable Definitions.** 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 a 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].

2. **Yield Strength.** The deviatoric perfectly plastic yield function, ϕ, is described in terms of the second invariant J_2,

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

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

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

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

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

There is no strain hardening on this surface.

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

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

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

3. **Equivalent Plastic Strain.** The history variable labeled as "plastic strain" by LS-PrePost is actually plastic volumetric strain. Note that when VCR = 1.0, plastic volumetric strain is zero.

*MAT_VISCOELASTIC

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

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
BULK	Elastic bulk modulus. LT.0.0: |BULK| is a load curve of bulk modulus as a function of temperature.
G0	Short-time shear modulus, G_0; see the Remarks below. LT.0.0: |G0| is a load curve of short-time shear modulus as a function of temperature.
GI	Long-time (infinite) shear modulus, G_∞. LT.0.0: |GI| is a load curve of long-time shear modulus as a function of temperature.
BETA	Decay constant. LT.0.0: |BETA| is a load curve of decay constant as a function of temperature.

Remarks:

The shear relaxation behavior is described by [Hermann and Peterson, 1968]:

$$G(t) = G_\infty + (G_0 - G_\infty)\exp(-\beta t) \ .$$

A Jaumann rate formulation is used:

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

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 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	REF				
Type	A	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
G	Shear modulus
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: off EQ.1.0: on

Remarks:

1. **Stress-Strain Relationship.** The strain energy density potential for the Blatz-Ko rubber is

$$W(\mathbf{C}) = \frac{G}{2}\left[I_1 - 3 + \frac{1}{\beta}\left(I_3^{-\beta} - 1\right)\right]$$

where G is the shear modulus, I_1 and I_3 are the first and third invariants of the right Cauchy-Green tensor $\mathbf{C} = \mathbf{F}^\mathrm{T}\mathbf{F}$, and

$$\beta = \frac{v}{1 - 2v} \ .$$

The second Piola-Kirchhoff stress is computed as

$$\mathbf{S} = 2\frac{\partial W}{\partial \mathbf{C}} = G\left[\mathbf{I} - I_3^{-\beta}\mathbf{C}^{-1}\right]$$

from which the Cauchy stress is obtained by a push-forward from the reference to current configuration divided by the relative volume $J = \det(\mathbf{F})$,

$$\sigma = \frac{1}{J}\mathbf{F}\mathbf{S}\mathbf{F}^{\mathrm{T}} = \frac{G}{J}\left[\mathbf{B} - I_3^{-\beta}\mathbf{I}\right].$$

Here we use $\mathbf{B} = \mathbf{F}\mathbf{F}^{\mathrm{T}}$ to denote the left Cauchy-Green tensor, and Poisson's ratio, v, above is set internally to $v = 0.463$; also see Blatz and Ko [1962].

2. **History Variables.** For solids, the 9 history variables store the deformation gradient, whereas for shells, the gradient is stored in the slot for effective plastic strain along with the first 8 history variables (the 9th stores in internal flag). If a dynain file is created using INTERFACE_SPRINGBACK_LSDYNA, then NSHV should be set to 9 so that the *INITIAL_STRESS_SHELL cards have the correct deformation gradient from which the stresses are to be calculated.

*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 [1973].

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	D	PCJ	BETA	K	G	SIGY
Type	A	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
D	Detonation velocity
PCJ	Chapman-Jouget pressure
BETA	Beta burn flag (see remarks below): EQ.0.0: beta and 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:

$$p = F p_{\text{eos}}(V, E) \ ,$$

where p_{eos} is the pressure from the equation of state (either types 2, 3, or 14), 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

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

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

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

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

$$F = F_2 \ .$$

F_1 is not computed. BETA = 1 does not allow for the initialization of the lighting time.

If the programmed burn is used, BETA = 2.0, the undetonated high explosive material 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. The location and time of detonation is controlled by *INITIAL_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, namely, $\phi > 0$, the deviatoric trial stress is scaled to obtain the final deviatoric stress at time $n + 1$:

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

If $\phi \leq 0$, then

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

Before detonation, pressure is given by the expression

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

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.

In the case of solids and thick shells, 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.

Beams and shells that use this material type are completely bypassed in the element processing; however, the mass of the null beam or shell elements is computed and added to the nodal points which define the connectivity. The mass of null beams is ignored if the value of the density is less than 10^{-11}. The Young's modulus and Poisson's ratio are used only for setting the contact stiffness, and it is recommended that reasonable values be input. The variables PC, MU, TEROD, and CEROD do not apply to beams and shells. Historically, null beams and/or null shells have been used as an aid in modeling of contact, but this practice is now seldom needed.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PC	MU	TEROD	CEROD	YM	PR
Type	A	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 or label must be specified (see *PART).
RO	Mass density
PC	Pressure cutoff (≤ 0.0). See Remark 4.
MU	Dynamic viscosity, μ (optional). See Remark 1.
TEROD	Relative volume. V/V_0, for erosion in tension. Typically, use values greater than unity. If zero, erosion in tension is inactive.
CEROD	Relative volume, 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)

VARIABLE	DESCRIPTION
PR	Poisson’s ratio (used for null beams and shells only)

Remarks:

These remarks apply to solids and thick shells only.

1. **Material Model.** When used with solids or thick shells, this material must be used with an equation-of-state. Pressure cutoff is negative in tension. A (deviatoric) viscous stress of the form,

$$\sigma'_{ij} = 2\mu\dot{\varepsilon}'_{ij} \ ,$$

is computed for nonzero μ, where $\dot{\varepsilon}'_{ij}$ is the deviatoric strain rate and μ is the dynamic viscosity. Analyzing the dimensions of the above equation gives units of the components in SI of

$$\left[\frac{N}{m^2}\right] \sim \left[\frac{N}{m^2}s\right]\left[\frac{1}{s}\right] \ .$$

Therefore, μ may have units of [Pa × s].

2. **Hourglass Control.** Null materials have no shear stiffness (except from viscosity) and hourglass control must be used with great care. In some applications, the default hourglass coefficient may lead to significant energy losses. In general, for fluids the hourglass coefficient QM should be small (in the range 10^{-6} to 10^{-4}), and the hourglass type IHQ should be set to 1 (default).

3. **Yield Strength.** The Null material has no yield strength and behaves in a fluid-like manner.

4. **Cut-off Pressure.** The cut-off pressure, PC, must be defined to allow for a material to “numerically” cavitate. In other words, when a material undergoes dilatation above certain magnitude, it should no longer be able to resist this dilatation. Since dilatation stress or pressure is negative, setting the PC limit to a very small negative number would allow for the material to cavitate once the pressure in the material goes below this negative value.

MAT_ELASTIC_PLASTIC_HYDRO_*{OPTION}

This is Material Type 10. This material allows the modeling of an elastic-plastic hydrodynamic material and requires an equation-of-state (*EOS).

Available options include:

<BLANK>

SPALL

STOCHASTIC

The keyword card can appear in three ways:

*MAT_ELASTIC_PLASTIC_HYDRO or MAT_010

*MAT_ELASTIC_PLASTIC_HYDRO_SPALL or MAT_010_SPALL

*MAT_ELASTIC_PLASTIC_HYDRO_STOCHASTIC or MAT_010_STOCHASTIC

Card Summary:

Card 1. This card is required.

MID	RO	G	SIG0	EH	PC	FS	CHARL

Card 2. This card is included if and only if the SPALL keyword option is used.

A1	A2	SPALL					

Card 3. This card is required.

EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8

Card 4. This card is required.

EPS9	EPS10	EPS11	EPS12	EPS13	EPS14	EPS15	EPS16

Card 5. This card is required.

ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8

Card 6. This card is required.

ES9	ES10	ES11	ES12	ES13	ES14	ES15	ES16

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	SIG0	EH	PC	FS	CHARL
Type	A	F	F	F	F	F	F	F
Default	none	none	none	0.0	0.0	-∞	0.0	0.0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
G	Shear modulus
SIG0	Yield stress; see Remark 2.
EH	Plastic hardening modulus; see Remark 2.
PC	Pressure cutoff (≤ 0.0). If zero, a cutoff of -∞ is assumed.
FS	Effective plastic strain at which erosion occurs.
CHARL	Characteristic element thickness for deletion. This applies to 2D solid elements that lie on a boundary of a part. If the boundary element thins down due to stretching or compression, and if it thins to a value less than CHARL, the element will be deleted. The primary application of this option is to predict the break-up of axisymmetric shaped charge jets.

Spall Card. Additional card for SPALL keyword option.

Card 2	1	2	3	4	5	6	7	8
Variable	A1	A2	SPALL					
Type	F	F	F					

VARIABLE	DESCRIPTION
A1	Linear pressure hardening coefficient
A2	Quadratic pressure hardening coefficient
SPALL	Spall type (see Remark 3): EQ.0.0: Default set to "1.0" EQ.1.0: Tensile pressure is limited by PC, that is, p is always ≥ PC. EQ.2.0: If $\sigma_{\max} \geq -\mathrm{PC}$ element spalls and tension, $p < 0$, is never allowed. EQ.3.0: $p < \mathrm{PC}$ element spalls and tension, $p < 0$, is never allowed.

Card 3	1	2	3	4	5	6	7	8
Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F

Card 4	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

VARIABLE	DESCRIPTION
EPS*i*	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. See Remark 2.

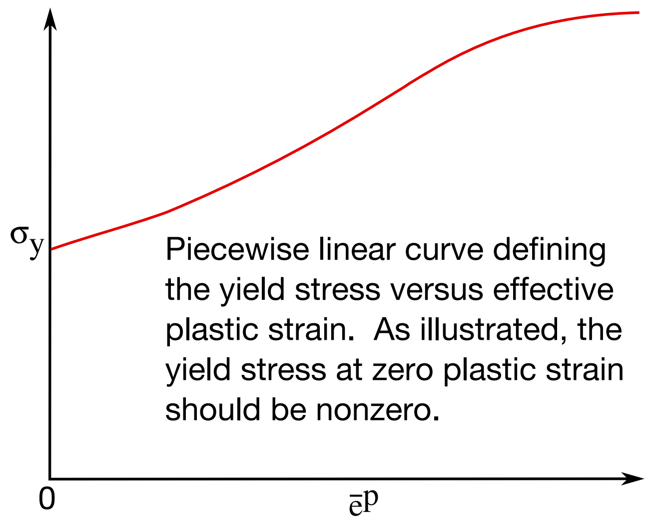


Figure M10-1. Effective stress as a function of effective plastic strain curve. See EPS and ES input.

Card 5	1	2	3	4	5	6	7	8
Variable	ES1	ES2	ES3	ES4	ES5	ES6	ES7	ES8
Type	F	F	F	F	F	F	F	F

Card 6	1	2	3	4	5	6	7	8
Variable	ES9	ES10	ES11	ES12	ES13	ES14	ES15	ES16
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
ES*i*	Effective stress. Define up to 16 values. See Remark 2.

Remarks:

1. **Model Overview.** 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 as a function of 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.

The STOCHASTIC option allows spatially varying yield and failure behavior. See *DEFINE_STOCHASTIC_VARIATION for additional information.

2. **Yield Stress and Plastic Hardening Modulus.** If ES and EPS values are undefined, the yield stress and plastic hardening modulus are taken from SIG0 and EH. In this case, the bilinear stress-strain curve shown in M10-1 is obtained with hardening parameter, $\beta = 1$. The yield strength is calculated as

$$\sigma_y = \sigma_0 + E_h \bar{\varepsilon}^p + (a_1 + p a_2)\max[p, 0] \ .$$

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

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

The pressure, p, is taken as positive in compression.

If ES and EPS are specified, a curve like that shown in Figure M10-1 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

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

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

3. **Spall Models.** A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads.

   a) 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.

b) 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.

c) 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.

*MAT_STEINBERG

This is Material Type 11. This material is available for modeling materials deforming at very high strain rates (> 10^5 per second) and can be used with solid elements. The yield strength is a function of temperature and pressure. An equation of state determines the pressure.

This model applies to a wide range of materials, including those with pressure-dependent yield behavior. In addition, the incorporation of an equation of state permits accurate modeling of a variety of different materials. The spall model options permit the incorporation of material failure, fracture, and disintegration effects under tensile loads.

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

Card 2	1	2	3	4	5	6	7	8
Variable	B	BP	H	F	A	TMO	GAMO	SA
Type	F	F	F	F	F	F	F	F

Card 3	1	2	3	4	5	6	7	8
Variable	PC	SPALL	RP	FLAG	MMN	MMX	EC0	EC1
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
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 or label must be specified (see *PART).
RO	Mass density
G0	Basic shear modulus. See Remark 2.
SIGO	σ_o; see Remark 3 below.
BETA	β; see Remark 3 below.
N	n; see Remark 3 below.
GAMA	γ_i, initial plastic strain; see Remark 3 below.
SIGM	σ_m; see Remark 3.
B	b; see Remark 2.
BP	b'; see Remark 3.
H	h; see Remarks 2 and 3.
F	f; see Remark 3.
A	Atomic weight (if = 0.0, RP must be defined). See Remark 2.
TMO	T_{mo}; see Remark 2.
GAMO	γ_o; see Remark 2.
SA	a; see Remark 2.
PC	Pressure cutoff (default = -10^{30}). See Remark 5.
SPALL	Spall type (see Remark 5): EQ.0.0: Default, set to “2.0” EQ.1.0: $p \geq$ PC EQ.2.0: If $\sigma_{\mathrm{max}} \geq$ -PC, element spalls, and tension, $p < 0$, is never allowed. EQ.3.0: If $p <$ PC element spalls, and tension, $p < 0$, is never allowed.
RP	R'. If $R' \neq 0.0$, A is not defined. See Remark 2.

VARIABLE	DESCRIPTION
FLAG	Flag for cold compression energy fit (see Remarks 2 and 4): EQ.0.0: η coefficients (default) EQ.1.0: μ coefficients
MMN	Optional μ or η minimum value (μ_{min} or η_{min}), depending on FLAG.
MMX	Optional μ or η maximum value (μ_{max} or η_{max}), depending on FLAG.
EC0, …, EC9	Cold compression energy coefficients (optional). See Remark 2.

Remarks:

1. **References.** 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.

2. **Shear Modulus.** In terms of the foregoing input parameters, we define the shear modulus, G, before the material melts as:

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

where p is the pressure, V is the relative volume, E_c (see Remark 4) is the cold compression energy, and E_m is the melting energy. E_c is given by:

$$E_c(x) = \int_0^x p(X)dX - \frac{900R'\exp(ax)}{(1-x)^{(\gamma_o - a)}}$$

with

$$x = 1 - V\ .$$

E_m is defined as:

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

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

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

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

In the above equations 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.

3. **Yield Strength.** The yield strength, σ_y, is given by:

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

if E_m exceeds E_i (see Remark 2). Here, σ_o' is:

$$\sigma_o' = \sigma_o[1 + \beta(\gamma_i + \bar{\varepsilon}^p)]^n$$

where σ_o is the initial yield stress and γ_i is the initial plastic strain. If the work-hardened yield stress σ_o' exceeds σ_m, σ_o' is set equal to σ_m. After the materials melt, σ_y and G are set to one half their initial value.

4. **Cold Compression Energy.** 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 field FLAG as:

$$E_c(\eta^i) = \sum_{i=0}^{9} \mathrm{EC}_i \eta^i$$

$$E_c(\mu^i) = \sum_{i=0}^{9} \mathrm{EC}_i \mu^i$$

where EC_i is the i^{th} coefficient and:

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

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

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

5. **Spall Models.** A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads.

   a) *Pressure Limit Model.* The pressure limit model, SPALL = 1, limits the hydrostatic tension to the specified value, p_{cut}. If a pressure more tensile than this limit is 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.

   b) *Maximum Principal Stress Spall Model.* The maximum principal stress spall model, SPALL = 2, detects spall if the maximum principal stress, $\sigma_{\max}$, exceeds the limiting value $-p_{\mathrm{cut}}$. Note that the negative sign is required because p_{cut} is measured positive in compression, while $\sigma_{\max}$ is positive in tension. Once spall is detected with this model, the deviatoric stresses are reset to zero, and no hydrostatic tension ($p < 0$) is permitted. If tensile pressures are calculated, they are reset to 0 in the spalled material. Thus, the spalled material behaves as a rubble or incohesive material.

c) *Hydrostatic Tension Spall Model.* 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.

*MAT_STEINBERG_LUND

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

The keyword cards can appear in two ways:

*MAT_STEINBERG_LUND or MAT_011_LUND

Card Summary:

Card 1. This card is required.

MID	RO	G0	SIG0	BETA	N	GAMA	SIGM

Card 2. This card is required.

B	BP	H	F	A	TMO	GAMO	SA

Card 3. This card is required.

PC	SPALL	RP	FLAG	MMN	MMX	EC0	EC1

Card 4. This card is required.

EC2	EC3	EC4	EC5	EC6	EC7	EC8	EC9

Card 5. This card is required.

UK	C1	C2	YP	YA	YM		

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G0	SIG0	BETA	N	GAMA	SIGM
Type	A	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
G0	Basic shear modulus
SIGO	σ_o; see Remark 3 of *MAT_011.
BETA	β; see Remark 3 of *MAT_011.
N	n; see Remark 3 of *MAT_011.
GAMA	γ_i, initial plastic strain; see Remark 3 of *MAT_011.
SIGM	σ_m; see Remark 3 of *MAT_011.

Card 2	1	2	3	4	5	6	7	8
Variable	B	BP	H	F	A	TMO	GAMO	SA
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
B	b; see Remark 2 of *MAT_011
BP	b'; see Remark 3 of *MAT_011.
H	h; see Remarks 2 and 3 of *MAT_011.
F	f; see Remark 3 of *MAT_011.
A	Atomic weight (if = 0.0, RP must be defined). See Remark 2 of *MAT_011.
TMO	T_{mo}; see Remark 2 of *MAT_011.
GAMO	γ_o; see Remark 2 of *MAT_011.
SA	a; see Remark 2 of *MAT_011.

Card 3	1	2	3	4	5	6	7	8
Variable	PC	SPALL	RP	FLAG	MMN	MMX	EC0	EC1
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	EC2	EC3	EC4	EC5	EC6	EC7	EC8	EC9
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
PC	Pressure cutoff (p_{cut}) or -σ_f (default = -10^{30})
SPALL	Spall type (see Remark 5 of *MAT_011): EQ.0.0: Default, set to "2.0" EQ.1.0: $p \geq$ PC EQ.2.0: If $\sigma_{\max} \geq$ -PC, element spalls, and tension, $p < 0$, is never allowed. EQ.3.0: If $p <$ PC element spalls, and tension, $p < 0$, is never allowed.
RP	R'. If $R' \neq 0.0$, A is not defined. See Remark 2 of *MAT_011.
FLAG	Flag for cold compression energy fit (see Remarks 2 and 4 of *MAT_011): EQ.0.0: η coefficients (default) EQ.1.0: μ coefficients
MMN	Optional μ or η minimum value ($\mu_{\min}$ or $\eta_{\min}$), depending on FLAG.
MMX	Optional μ or η maximum value ($\mu_{\max}$ or $\eta_{\max}$), depending on FLAG.
EC0, …, EC9	Cold compression energy coefficients (optional). See Remark 2 of *MAT_011.

Card 5	1	2	3	4	5	6	7	8
Variable	UK	C1	C2	YP	YA	YM		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
UK	Activation energy for rate dependent model
C1	Exponent pre-factor in rate dependent model
C2	Coefficient of drag term in rate dependent model
YP	Peierls stress for rate dependent model
YA	A thermal 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:

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

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

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

and comes from the limit of the first term in σ_y being small. In this case $Y_A f(\varepsilon_p)$ reduces to σ_0' from the *MAT_011 material model (see Remark 3 of *MAT_011). The second limit is on the thermal part:

$$Y_T \le Y_P \ .$$

*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 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	1	2	3	4	5	6	7	8
Variable	MID	RO	G	SIGY	ETAN	BULK		
Type	A	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
G	Shear modulus
SIGY	Yield stress
ETAN	Plastic hardening modulus
BULK	Bulk modulus, K

Remarks:

The pressure is integrated in time from

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

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

Card 2	1	2	3	4	5	6	7	8
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 or label must be specified (see *PART).
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, causing the material to behave 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 reference 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 tensile 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 are otherwise confined by nodal boundary conditions.

MAT_JOHNSON_COOK_*{OPTION}

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, we recommend the much less expensive *MAT_SIMPLIFIED_-JOHNSON_COOK model. The simplified model can be used with beam elements.

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.

Available options include:

<BLANK>

STOCHASTIC

The STOCHASTIC option enables spatially varying yield and failure behavior. See *DEFINE_STOCHASTIC_VARIATION for additional information.

Card Summary:

Card 1. This card is required.

MID	RO	G	E	PR	DTF	VP	RATEOP

Card 2. This card is required.

A	B	N	C	M	TM	TR	EPS0

Card 3. This card is required.

CP	PC	SPALL	IT	D1	D2	D3	D4

Card 4a. This card is included for RATEOP = 0.0 or 2.0 or for VP = 0.0.

D5		EROD	EFMIN	NUMINT			

Card 4b. This card is included for RATEOP = 1.0, 3.0, or 4.0.

D5	C2/P/XNP	EROD	EFMIN	NUMINT			

Card 4c. This card is included for RATEOP = 5.0.

D5	D	EROD	EFMIN	NUMINT	K	EPS1	

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	E	PR	DTF	VP	RATEOP
Type	A	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 or label must be specified (see *PART).
RO	Mass density
G	Shear modulus. G and an equation-of-state are required for element types that use a 3D stress update, such as solids, 2D shell forms 13-15, and tshell forms 3, 5, and 7. For other element types, G is ignored, and E and PR must be provided.
E	Young's Modulus (see note above pertaining to G)
PR	Poisson's ratio (see note above pertaining to G)
DTF	Minimum time step size for automatic element deletion (shell elements). The element will be deleted when the solution time step size drops below DTF × TSSFAC where TSSFAC is the time step scale factor defined by *CONTROL_TIMESTEP. See Remark 4.
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation
RATEOP	Form of strain rate term. RATEOP is ignored if VP = 0. See Remark 5.

VARIABLE	DESCRIPTION
	EQ.0.0: Log-linear Johnson-Cook (default) EQ.1.0: Log-quadratic Huh-Kang (2 parameters) EQ.2.0: Exponential Allen-Rule-Jones EQ.3.0: Exponential Cowper-Symonds (2 parameters) EQ.4.0: Nonlinear rate coefficient (2 parameters) EQ.5.0: Log-exponential Couque (4 parameters)

Card 2	1	2	3	4	5	6	7	8
Variable	A	B	N	C	M	TM	TR	EPS0
Type	F	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	none	none	none	none

VARIABLE	DESCRIPTION
A	Constant A in the flow stress. See equations in Remark 1.
B	Constant B in the flow stress. See equations in Remark 1.
N	Constant n in the flow stress. See equations in Remark 1.
C	Constant C in the flow stress. See equations in Remarks 1 and 5.
M	Constant m in the flow stress. See equations in Remark 1.
TM	Melt temperature
TR	Room temperature
EPS0	Quasi-static threshold strain rate (see Remark 1). Ideally, this value represents the highest strain rate for which no rate adjustment to the flow stress is needed and is input in units of $[\text{time}]^{-1}$. For example, if strain rate effects on the flow stress first become apparent at strain rates greater than $10^{-2}\ \text{s}^{-1}$, and the system of units for the model input is {kg, mm, ms}, then EPSO should be set to 10^{-5}.

Card 3	1	2	3	4	5	6	7	8
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

VARIABLE	DESCRIPTION
CP	Specific heat (superseded by heat capacity in *MAT_THERMAL_*OPTION* if a coupled thermal/structural analysis)
PC	Tensile failure stress or tensile pressure cutoff (PC < 0.0)
SPALL	Spall type (see Remark 3): EQ.0.0: Set to “2.0” (default). EQ.1.0: Tensile pressure is limited by PC, that is, p is always ≥ PC. *Shell Element* Specific Behavior: EQ.2.0: Shell elements are deleted when $\sigma_{max} \geq -\mathrm{PC}$. EQ.3.0: Shell elements are deleted when $p < \mathrm{PC}$. *Solid Element* Specific Behavior EQ.2.0: For solid elements $\sigma_{max} \geq -\mathrm{PC}$ resets tensile stresses to zero. Compressive stress are still allowed. EQ.3.0: For solid elements $p < \mathrm{PC}$ resets the pressure to zero thereby disallowing tensile pressure.
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 – D4	Failure parameters; see Remark 2. If D3 < 0.0, it will be converted to its absolute value.

This card is included for RATEOP = 0.0 or 2.0 or for VP = 0.0.

Card 4a	1	2	3	4	5	6	7	8
Variable	D5		EROD	EFMIN	NUMINT			
Type	F		F	F	I			
Default	0.0		0.0	10^{-6}	0			

VARIABLE	DESCRIPTION
D5	Failure parameter; see Remark 2.
EROD	Erosion flag: EQ.0.0: Element erosion allowed (default). NE.0.0: Element does not erode; deviatoric stresses set to zero when element fails.
EFMIN	Lower bound for calculated strain at fracture (see Remark 2)
NUMINT	Number of through thickness integration points which must fail before the shell element is deleted. If zero, all integration points must fail (the default). Since nodal fiber rotations limit strains at active integration points, we do not recommend the default because elements undergoing large strain are often not deleted using this criterion. Better results may be obtained when NUMINT is set to 1 or a number less than one half of the number of through thickness points. For example, if four through thickness points are used, NUMINT should not exceed 2, even for fully integrated shells which have 16 integration points.

This card is included for RATEOP = 1.0, 3.0, or 4.0.

Card 4b	1	2	3	4	5	6	7	8
Variable	D5	C2/P/XNP	EROD	EFMIN	NUMINT			
Type	F	F	F	F	I			
Default	0.0	0.0	0.0	10^{-6}	0			

VARIABLE	DESCRIPTION
D5	Failure parameter; see Remark 2.
C2/P/XNP	Strain rate parameter.

Field	Var	Model
C2	C_2	Huh-Kang
P	P	Cowper-Symonds
XNP	n'	Nonlinear Rate Coefficient

See Remark 5 for a description of these models.

VARIABLE	DESCRIPTION
EROD	Erosion flag: EQ.0.0: Element erosion allowed (default). NE.0.0: Element does not erode; deviatoric stresses set to zero when element fails.
EFMIN	Lower bound for calculated strain at fracture (see Remark 2)
NUMINT	Number of through thickness integration points which must fail before the shell element is deleted. If zero, all integration points must fail (the default). Since nodal fiber rotations limit strains at active integration points, we do not recommend the default because elements undergoing large strain are often not deleted using this criterion. Better results may be obtained when NUMINT is set to 1 or a number less than one half of the number of through thickness points. For example, if four through thickness points are used, NUMINT should not exceed 2, even for fully integrated shells which have 16 integration points.

This card is included for RATEOP = 5.0.

Card 4c	1	2	3	4	5	6	7	8
Variable	D5	D	EROD	EFMIN	NUMINT	K	EPS1	
Type	F	F	F	F	I	F	F	
Default	0.0	0.0	0.0	10^{-6}	0	0.0	none	

VARIABLE	DESCRIPTION
D5	Failure parameter; see Remark 2.
D	Strain rate parameter D for Couque term. See Remark 5.
EROD	Erosion flag: EQ.0.0: Element erosion allowed (default). NE.0.0: Element does not erode; deviatoric stresses set to zero when element fails.
EFMIN	Lower bound for calculated strain at fracture (see Remark 2)
NUMINT	Number of through thickness integration points which must fail before the shell element is deleted. If zero, all integration points must fail (the default). Since nodal fiber rotations limit strains at active integration points, we do not recommend the default because elements undergoing large strain are often not deleted using this criterion. Better results may be obtained when NUMINT is set to 1 or a number less than one half of the number of through thickness points. For example, if four through thickness points are used, NUMINT should not exceed 2, even for fully integrated shells which have 16 integration points.
K	Strain rate parameter for Couque term. See Remark 5.
EPS1	Reference strain rate for Couque term, characterizing the transition between the thermally activated regime and the viscous regime. Input in units of $[\text{time}]^{-1}$. See Remark 5.

Remarks:

1. **Flow Stress.** Johnson and Cook express the flow stress as

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

where

$$\begin{aligned} A, B, C, n, \text{and } m &= \text{input constants} \\ \bar{\varepsilon}^p &= \text{effective plastic strain} \\ \dot{\varepsilon}^* &= \begin{cases} \dfrac{\dot{\bar{\varepsilon}}}{\text{EPS0}} & \text{for VP} = 0 \quad \text{(normalized effective total strain rate)} \\ \dfrac{\dot{\bar{\varepsilon}}^p}{\text{EPS0}} & \text{for VP} = 1 \quad \text{(normalized effective plastic strain rate)} \end{cases} \end{aligned}$$

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

The quantity $T - T_{\text{room}}$ is stored as extra history variable 5. In the case of a mechanical-only analysis, this is the adiabatic temperature increase calculated as

$$T - T_{\text{room}} = \frac{\text{internal energy}}{(C_p \times \rho \times Vo)} ,$$

where

$$C_p \text{ and } \rho = \text{input constants}$$
$$Vo = \text{initial volume}$$

In a coupled thermal/mechanical analysis, $T - T_{\text{room}}$ includes heating/cooling from all sources, not just adiabatic heating from the internal energy.

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

2. **Strain at Fracture.** The strain at fracture is given by

$$\varepsilon^f = \max([D_1 + D_2 \exp D_3 \sigma^*][1 + D_4 \ln \dot{\varepsilon}^*][1 + D_5 T^*], \text{EFMIN}) ,$$

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

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

Fracture occurs when the damage parameter,

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

reaches the value of 1. D is stored as extra history variable 4 in shell elements and extra history variable 6 in solid elements.

3. **Spall Models.** A choice of three spall models is offered to represent material splitting, cracking, and failure under tensile loads:

   a) *Pressure Limit Model.* The pressure limit model limits the minimum hydrostatic pressure to the specified value, $p \geq p_{\text{min}}$. If the calculated pressure is more tensile than this limit, 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.

b) *Maximum Principal Stress Model.* The maximum principal stress spall model detects spall if the maximum principal stress, σ_{max}, exceeds the limiting value σ_p. Once spall in solids 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.

c) *Hydrostatic Tension Model.* The hydrostatic tension spall model detects spall if the pressure becomes more tensile than the specified limit, p_{min}. Once spall in solids is detected with this model, 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.

4. **Shell Element Deletion Based on Time Step.** This material model also supports a shell element deletion criterion based on the maximum stable time step size for the element, Δt_{max} (see DTF on Card 1). 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.

5. **Optional Strain Rate Forms.** The standard Johnson-Cook strain rate term is linear in the logarithm of the strain rate (see Remark 1):

$$1 + C \ln \dot{\varepsilon}^*$$

You can replace this term by setting RATEOP > 0. These additional rate forms are currently available for solid and shell elements but only when the viscoplastic rate option is active (VP = 1). If VP is set to zero, RATEOP is ignored.

The first additional available rate form enables some additional data fitting by using the quadratic form proposed by Huh & Kang [2002]:

$$1 + C \ln \dot{\varepsilon}^* + C_2 (\ln \dot{\varepsilon}^*)^2$$

Four additional exponential forms are available, one due to Allen, Rule & Jones [1997]:

$$(\dot{\varepsilon}^*)^C \quad ,$$

the Cowper-Symonds-like [1958] form:

$$1 + \left(\frac{\dot{\varepsilon}^p_{\text{eff}}}{C}\right)^{\frac{1}{P}} \quad ,$$

the nonlinear rate coefficient:

$$1 + C\left(\varepsilon^p_{\text{eff}}\right)^{n'} \ln \dot{\varepsilon}^* .$$

and the Couque [2014] form,

$$1 + C \ln \dot{\varepsilon}^* + D\left(\frac{\dot{\varepsilon}^p_{\text{eff}}}{\text{EPS1}}\right)^k .$$

See Huh and Kang [2002], Allen, Rule, and Jones [1997], Cowper and Symonds [1958], and Couque [2014].

6. **History Variables.** The following extra history variables may be output to the d3plot file (see *DATABASE_EXTENT_BINARY).

History Variable #	Description for Shell Elements	Description for Solid Elements
1	Failure value	
3	Current pressure cutoff	
4	Damage parameter, D	
5	Temperature change, $T - T_{\text{room}}$	Temperature change, $T - T_{\text{room}}$
6	Failure strain	Damage parameter, D

*MAT_PSEUDO_TENSOR

This is Material Type 16. This model has been used to analyze buried steel-reinforced concrete structures subjected to impulsive loadings.

This model can be used in two major modes - a simple tabular pressure-dependent yield surface and a potentially complex model featuring two yield-as-a-function-of-pressure functions with the means of migrating from one curve to the other. The Remarks section discusses these modes in detail. For both modes, load curve LCP 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, 9, or 11. If no EOS is set, the material model uses type 8 for a simple "generic" concrete model. See Remarks.

Card Summary:

Card 1. This card is required.

MID	RO	G	PR				

Card 2. This card is required.

SIGF	A0	A1	A2	A0F	A1F	B1	PER

Card 3. This card is required.

ER	PRR	SIGY	ETAN	LCP	LCR		

Card 4. This card is required.

X1	X2	X3	X4	X5	X6	X7	X8

Card 5. This card is required.

X9	X10	X11	X12	X13	X14	X15	X16

Card 6. This card is required.

YS1	YS2	YS3	YS4	YS5	YS6	YS7	YS8

Card 7. This card is required.

YS9	YS10	YS11	YS12	YS13	YS14	YS15	YS16

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	PR				
Type	A	F	F	F				
Default	none	none	none	optional				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
G	Shear modulus. If PR is set, this field is ignored. The shear modulus in this case is derived from PR and the bulk modulus of the EOS.
PR	Poisson's ratio

Card 2	1	2	3	4	5	6	7	8
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

VARIABLE	DESCRIPTION
SIGF	Tensile cutoff (maximum principal stress for failure)
A0	Cohesion
A1	Pressure hardening coefficient
A2	Pressure hardening coefficient
A0F	Cohesion for failed material

VARIABLE	DESCRIPTION
A1F	Pressure hardening coefficient for failed material
B1	Damage scaling factor (or exponent in Mode II.C)
PER	Percent reinforcement

Card 3	1	2	3	4	5	6	7	8
Variable	ER	PRR	SIGY	ETAN	LCP	LCR		
Type	F	F	F	F	F	F		
Default	0.0	0.0	none	0.0	none	none		

VARIABLE	DESCRIPTION
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.

Card 4	1	2	3	4	5	6	7	8
Variable	X1	X2	X3	X4	X5	X6	X7	X8
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 5	1	2	3	4	5	6	7	8
Variable	X9	X10	X11	X12	X13	X14	X15	X16
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
X*n*	Effective plastic strain, damage, or pressure. See Remarks below.

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	none	none	none	none	none	none	none	none

Card 7	1	2	3	4	5	6	7	8
Variable	YS9	YS10	YS11	YS12	YS13	YS14	YS15	YS16
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
YS*n*	Yield stress (Mode I) or scale factor (Mode II.B or II.C)

Remarks:

1. **Response Mode I (tabulated yield stress as a function of 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 M16-1. Examples

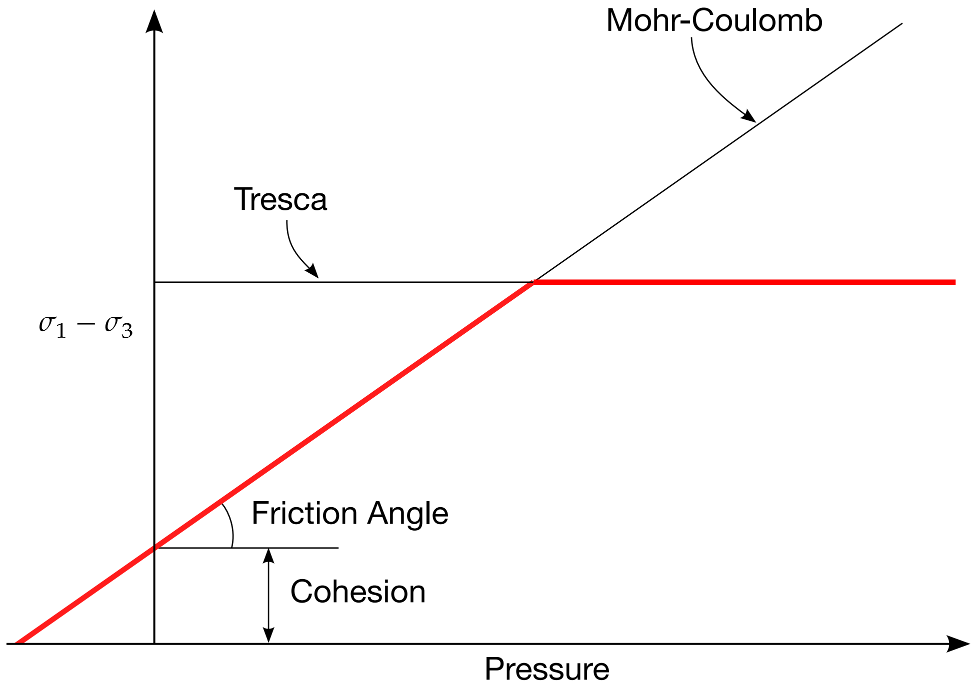


Figure M16-1. Mohr-Coulomb surface with a Tresca Limit.

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 $(\sigma_1 - \sigma_3)/2$, where σ_1 is the maximum principal stress and $\sigma_3$is the minimum principal stress.

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

To invoke Mode I of this model, set a_0, a_1, a_2, b_1, a_{0f}, and a_{1f} to zero. The tabulated values of pressure should then be specified on Cards 4 and 5, and the corresponding values of yield stress should be specified on Cards 6 and 7. The parameters relating to reinforcement properties, initial yield stress, and tangent modulus are not used in this response mode and should be set to zero.

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

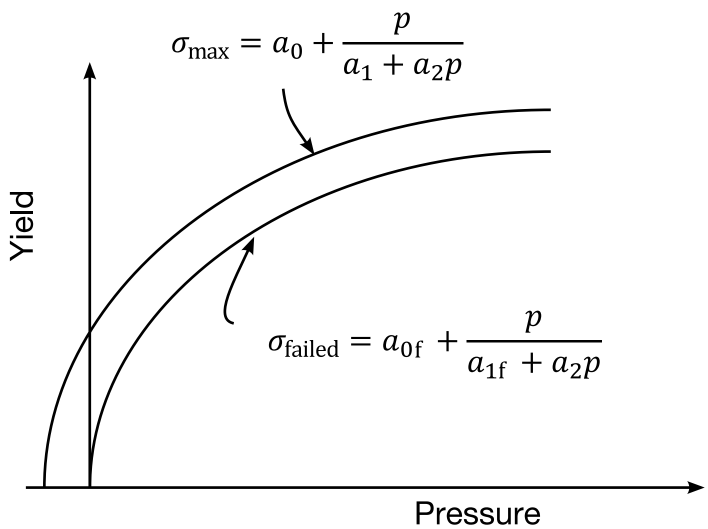


Figure M16-2. Two-curve concrete model with damage and failure

2. **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.

a) *Mode II.A (Simple Tensile Failure).* To use this mode, define a_0, a_1, a_2, a_{0f}, and a_{1f}, set b_1 to zero, and leave Cards 4 through 7 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 cutoff (σ_{cut}). When $\sigma_1 > \sigma_{\text{cut}}$ is detected, the yield strength is scaled back by a fraction of the distance between the two curves in each of the next 20 time steps so that after those 20 time steps, the yield strength is defined by the failure curve.

b) *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 use Cards 4 through 7 to define a scale factor, η, (Cards 6 and 7) as a function of effective plastic strain (Cards 4 and 5). LS-DYNA evaluates η at the current effective plastic strain and then calculates the yield stress as

$$\sigma_{\text{yield}} = \sigma_{\text{failed}} + \eta(\sigma_{\max} - \sigma_{\text{failed}}) ,$$

where $\sigma_{\max}$ and σ_{failed} are found as shown in Figure M16-2. This yield strength is then subject to scaling for tensile failure as described above. This type of model describes a strain hardening or softening material, such as concrete.

c) *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:

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

To use this model, define a_0, a_1, a_2, $a_{0\mathrm{f}}$, $a_{1\mathrm{f}}$, and b_1. Cards 4 through 7 now give η as a function of λ. η scales the yield stress as

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

before applying any tensile failure criteria.

3. **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 SIGF is assumed to be the unconfined concrete compressive strength, f_c', and –A0 is assumed to be a conversion factor from LS-DYNA pressure units to psi. (For example, if the model stress units are MPa, A0 should be set to –145.) In this case, the parameter values generated internally are

$$f_c' = \mathrm{SIGF} \qquad a_1 = \frac{1}{3} \qquad a_{0\mathrm{f}} = 0$$

$$\sigma_{\mathrm{cut}} = 1.7\left(\frac{f_c'^2}{-\mathrm{A0}}\right)^{\frac{1}{3}} \qquad a_2 = \frac{1}{3f_c'} \qquad a_{1\mathrm{f}} = 0.385$$

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

Note that these $a_{0\mathrm{f}}$ and $a_{1\mathrm{f}}$ defaults will be overridden by non-zero entries on Card 3. If plastic strain or damage scaling is desired, Cards 5 through 8 as well as b_1 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 as a function of volumetric strain model with no internal energy terms, and should give reasonable results for pressures up to 5 kbar (approximately 75,000 psi).

4. **Mixture model.** A reinforcement fraction, f_r, can be defined (indirectly as PER/100) along with properties of the reinforcement material. The bulk

modulus, shear modulus, and yield strength are then calculated from a simple mixture rule. For example, 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 moduli for the geologic material and the reinforcement material, respectively. This feature should be used with caution. It gives an isotropic effect in the material instead of the true anisotropic material behavior. A reasonable approach would be to use the mixture elements only where the reinforcing exists and plain elements elsewhere. When the mixture model is being used, the strain rate multiplier for the principal material is taken from load curve N1 and the multiplier for the reinforcement is taken from load curve N2.

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

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

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

Crack Propagation Card. Optional card for crack propagation to adjacent elements (see remarks).

Card 2	1	2	3	4	5	6	7	8
Variable	SOFT	CVELO						
Type	F	F						
Default	1.0	0.0						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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)

VARIABLE	DESCRIPTION
SOFT	Factor by which the fracture stress is reduced when a crack is coming from failed neighboring element. See remarks.
CVELO	Crack propagation velocity. See remarks.

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, $\varepsilon_{\mathrm{eff}}^p$, and the plastic hardening modulus, E_p:

$$\sigma_y = \sigma_0 + E_p \varepsilon_{\mathrm{eff}}^p \ .$$

The effective plastic strain is defined as:

$$\varepsilon_{\mathrm{eff}}^p = \int_0^t d\varepsilon_{\mathrm{eff}}^p \ ,$$

where

$$d\varepsilon_{\mathrm{eff}}^p = \sqrt{\frac{2}{3} d\varepsilon_{ij}^p d\varepsilon_{ij}^p}$$

and the plastic tangent modulus is defined in terms of the input tangent modulus, E_t, as

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

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

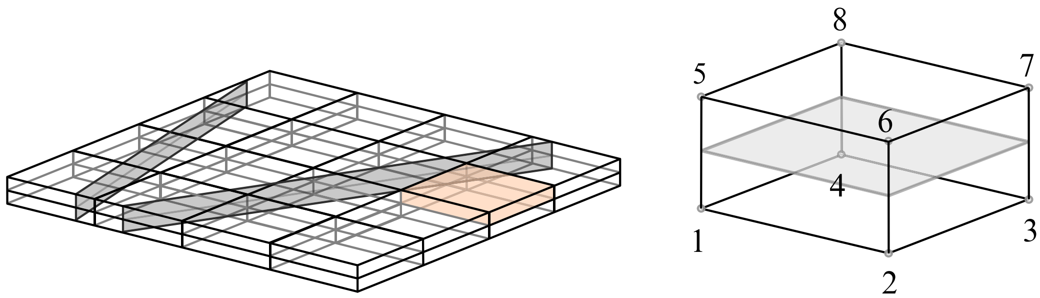


Figure M17-1. Thin structure (2 elements over thickness) with cracks and necessary element numbering.

that may otherwise be introduced into the system by the sudden fracture. The number of steps for stress reduction is 20 by default (CVELO = 0.0) or it is internally computed if CVELO > 0.0 is given, that is:

$$n_{\text{steps}} = \text{int}\left[\frac{L_e}{\text{CVELO} \times \Delta t}\right] ,$$

where L_e is the characteristic element length and Δt is the time step size.

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.

Crack propagation behavior to adjacent elements can be controlled using parameter SOFT for thin, shell-like structures (for example, only 2 or 3 solids over thickness). Additionally, LS-DYNA must know where the plane or solid element midplane is at each integration point for projection of crack plane on this element midplane. Therefore, element numbering must be as shown in Figure M17-1. Currently, only solid element type 1 is supported with that option.

*MAT_POWER_LAW_PLASTICITY

This is Material Type 18. This isotropic plasticity model with rate effects uses a power law hardening rule.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	K	N	SRC	SRP
Type	A	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	EPSF					
Type	F	F	F					
Default	0.0	0.0	0.0					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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, $\sigma_{y,0}$. Generally, this parameter is not necessary and the elastic strain to initial yield, ε_0, is calculated as described in the remarks section below. EQ.0.0: ε_0 is internally calculated. See Remarks. GT.0.0.and.LT.0.02: ε_0 is SIGY. GE.0.02: ε_0 is internally calculated with $\sigma_{y,0}$ = SIGY. See Remarks.
EPSF	Plastic failure strain for element deletion
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation

Remarks:

This model provides elastoplastic behavior with isotropic hardening. The yield stress, σ_y, is a function of plastic strain and obeys the equation:

$$\sigma_y = k\varepsilon^n = k(\varepsilon_0 + \bar{\varepsilon}^p)^n \ ,$$

where ε_0 is the elastic strain to initial yield and $\bar{\varepsilon}^p$is the effective plastic strain (logarithmic). If SIGY is set to zero, ε_0 is found by solving for the intersection of the linearly elastic loading equation with the strain hardening equation:

$$\begin{aligned}\sigma &= E\varepsilon \\ \sigma &= k\varepsilon^n\end{aligned}$$

Thus:

$$\varepsilon_0 = \left(\frac{E}{k}\right)^{\left[\frac{1}{n-1}\right]} \ .$$

If SIGY is nonzero but less than 0.02, ε_0 = SIGY. If SIGY is nonzero and greater than 0.02, the following equation gives ε_0:

$$\varepsilon_0 = \left(\frac{\sigma_{y,0}}{k}\right)^{\left[\frac{1}{n}\right]} \ ,$$

where the initial yield stress, $\sigma_{y,0}$, is SIGY.

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 incorporating the Cowper and Symonds formulation within the yield surface is optional. An additional cost is incurred, but the improvement in results can be dramatic.

*MAT_STRAIN_RATE_DEPENDENT_PLASTICITY

This is Material Type 19. A strain rate dependent material can be defined. For an alternative, see Material Type 24. A curve for the yield strength as a function of the effective strain rate must be defined. Optionally, Young's modulus and the tangent modulus can also be defined as a function of 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 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	VP			
Type	A	F	F	F	F			
Default	none	none	none	none	0.0			

Card 2	1	2	3	4	5	6	7	8
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 or label must be specified (see *PART).
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
VP	Formulation for rate effects: EQ.0.0: scale yield stress (default) EQ.1.0: viscoplastic formulation

VARIABLE	DESCRIPTION
LC1	Load curve ID defining the yield strength σ_0 as a function of the effective strain rate.
ETAN	Tangent modulus, E_t
LC2	Optional load curve ID defining Young's modulus as a function of the effective strain rate (available only when VP = 0; not recommended).
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 and absolute value of minimum principal stress EQ.3.0: maximum principal stress (R5 of version 971)

Remarks:

1. **Yield Stress.** 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 strain rate is available for post-processing as the first stored history variable. If the viscoplastic option is active, the plastic strain rate is output; otherwise, the effective strain rate defined above is output.

The yield stress is defined as

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

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

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

Both the 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 IDs are input as 0, then the constant values specified in the input are used.

2. **Load Curves.** 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.

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

   In addition to the above failure criterion, this material model also supports a shell element deletion criterion based on the maximum stable time step size for the element, $\Delta t_{\max}$. Generally, $\Delta t_{\max}$ goes down as the element becomes more distorted. To assure stability of time integration, the global LS-DYNA time step is the minimum of the $\Delta t_{\max}$ values calculated for all elements in the model. Using this option allows the selective deletion of elements whose time step $\Delta t_{\max}$ has fallen below the specified minimum time step, 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.

4. **Viscoplastic Formulation.** 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 Type 20. Parts made from this material are considered to belong to a rigid body (for each part ID). The coupling of a rigid body with MADYMO and CAL3D can also be defined using this material. Alternatively, a VDA surface can be attached as surface to model the geometry, such as for the tooling in metal forming applications. Optional global and local constraints on the mass center can be defined. A local consideration for output and user-defined airbag sensors may also optionally be chosen.

Card Summary:

Card 1. This card is required.

MID	RO	E	PR	N	COUPLE	M	ALIAS or RE

Card 2. This card must be included but may be left blank.

CMO	CON1	CON2	SPCNID	XSPC	YSPC	ZSPC	

Card 3. This card must be included but may be left blank.

LCO or A1	A2	A3	V1	V2	V3		

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	N	COUPLE	M	ALIAS or RE
Type	A	F	F	F	F	F	F	C/F
Default	none	none	none	none	0	0	0	opt / none

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density

VARIABLE	DESCRIPTION
E	Young's modulus. Reasonable values must be chosen for contact analysis (choice of penalty); see Remarks below.
PR	Poisson's ratio. Reasonable values must be chosen for contact analysis (choice of penalty); see Remarks below.
N	MADYMO3D 5.4 coupling flag, n: EQ.0: Use normal LS-DYNA rigid body updates. GT.0: The rigid body is coupled to the MADYMO 5.4 ellipsoid number n. LT.0: The rigid body is coupled to MADYMO 5.4 plane number, $\|n\|$.
COUPLE	Coupling option if applicable: EQ.-1: Attach VDA surface in ALIAS (defined in the eighth field) and automatically generate a mesh for viewing the surface in LS-PREPOST. MADYMO 5.4 / CAL3D coupling option: EQ.0: The undeformed geometry input to LS-DYNA corresponds to the local system for MADYMO 5.4 / CAL3D. The finite element mesh is input. EQ.1: The undeformed geometry input to LS-DYNA corresponds to the global system for MADYMO 5.4 / CAL3D. EQ.2: Generate a mesh for the ellipsoids and planes internally in LS-DYNA.
M	MADYMO3D 5.4 coupling flag, m: EQ.0: Use normal LS-DYNA rigid body updates, EQ.m: This rigid body corresponds to the MADYMO rigid body number, m. Rigid body updates are performed by MADYMO.
ALIAS	VDA surface alias name; see Appendix L.
RE	MADYMO 6.0.1 External Reference Number

Constraint Card. Must be included but may be left blank.

Card 2	1	2	3	4	5	6	7	8
Variable	CMO	CON1	CON2	SPCNID	XSPC	YSPC	ZSPC	
Type	F	I	I	I	F	F	F	
Default	0.0	0	0	0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
CMO	Constraint option, CMO (see Remark 5): EQ.+2.0: Constraints applied in global directions at the coordinates given by XSPC, YSPC, and ZSPC or the initial coordinates of node SPCNID. Unless prescribed motion is applied to the rigid body, the constraint coordinates are fixed in time. EQ.+1.0: Constraints applied in global directions, EQ.0.0: No constraints, EQ.-1.0: Constraints applied in local directions (SPC constraint). EQ.-2.0: Constraints applied in local directions (SPC constraint) at coordinates given by XSPC, YSPC, and ZSPC or the initial coordinates of node SPCNID. Unless prescribed motion is applied to the rigid body, the constraint coordinates are fixed in time.
CON1	First constraint parameter. If CMO > 0.0, then specify the 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,

VARIABLE	DESCRIPTION
	EQ.7: Constrained x, y, and z displacements.
	If CMO < 0.0, then specify the local coordinate system ID. See *DEFINE_COORDINATE_OPTION. This coordinate system is fixed in time.
CON2	Second constraint parameter:
	If CMO > 0.0, then specify the 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 CMO < 0.0, then specify the 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.
	To specify a combination of local constraints, input the sum of the desired constraints.
SPCNID	For $\lvert CMO \rvert = 2.0$, the constraint coordinates (see below) are the (initial) coordinates of the node with this ID.
XSPC,YSPC,ZSPC	Coordinates where the constraints act. Superseded by SPC-NID.

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

Card 3	1	2	3	4	5	6	7	8
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
LCO	Local coordinate system number for local output to rbdout. LCO also specifies the coordinate system used for *BOUNDARY_PRESCRIBED_MOTION_RIGID_LOCAL. Defaults to the principal coordinate system of the rigid body.
A1 - V3	Alternative method for specifying local system below: Define two vectors **a** and **v**, fixed to the rigid body which are used for output and the user defined airbag sensor subroutines. The output parameters are in the directions **a**, **b**, and **c** where the latter are given by the cross products $\mathbf{c} = \mathbf{a} \times \mathbf{v}$ and $\mathbf{b} = \mathbf{c} \times \mathbf{a}$. This input is optional.

Remarks:

1. **Rigid material.** 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.

2. **Parts.** Two unique rigid part IDs may not share common nodes unless they are merged together using the rigid body merge option. A rigid body, however, may be made up of disjoint finite element meshes. 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 IDs 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.

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

4. **Contact and material constants.** 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 problems with contact.

5. **Constraints.** Constraint directions for rigid bodies (CMO $\neq$ 0) 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. The constraint defined therein refers to the local system CID, which is updated with time.

   We strongly advise not applying nodal constraints, for instance, by *BOUNDARY_SPC_*OPTION*, to nodes of a rigid body as doing so may compromise the intended constraints in the case of an explicit simulation. Such SPCs will be skipped in an implicit simulation and a warning issued.

   If the intended constraints are not with respect to the calculated center of mass of the rigid body, the following alternative approaches can be used:

   a) Set $|\text{CMO}| = 2$ and choose the point at which the constraints shall act. These coordinates are referenced both by the constraints given on this card and those on *BOUNDARY_PRESCRIBED_MOTION_*OPTION*.

   b) *CONSTRAINED_JOINT_*OPTION* may often be used to obtain the desired effect. This approach typically entails defining a second rigid body that is fully constrained and then defining a joint between the two rigid bodies.

   c) Another alternative for defining rigid body constraints that are not with respect to the calculated center of mass of the rigid body is to manually specify the initial center of mass location using *PART_INERTIA. When using *PART_INERTIA, a full set of mass properties must be specified. Note that changing its mass properties will affect the rigid body's dynamic behavior.

Setting |CMO| = 2 not only allows for a constraint point other than the center of mass. The motion prescribed by *BOUNDARY_PRESCRIBED_MOTION also acts on this point. In addition, setting |CMO| = 2 treats the constraints (including those from *BOUNDARY_PRESCRIBED_MOTION) differently from |CMO| = 1. To allow for an arbitrary constraint point, the constraints are applied and solved during the kinematic update of the rigid body. Since the inertia does not need to be modified and no joints are involved, setting |CMO| = 2 is more accurate compared to options b and c above. No time penalty is to be expected.

To obtain reaction forces from constraints, see the SPC2BND flag of *CONTROL_OUTPUT.

6. **Coupling with MADYMO.** Only basic coupling is available for coupling with MADYMO 5.4.1. The coupling flags (N and M) must match with SYSTEM and ELLIPSOID/PLANE in the MADYMO input file and the coupling option (COUPLE) must be defined.

   Both basic and extended coupling are available for coupling with MADYMO 6.0.1:

   a) *Basic Coupling.* The external reference number (RE) must match the external reference number in the MADYMO XML input file. The coupling option (COUPLE) must be defined.

   b) *Extended Coupling.* Under this option MADYMO will handle the contact between the MADYMO and LS-DYNA models. The external reference number (RE) and the coupling option (COUPLE) are not needed. All coupling surfaces that interface with the MADYMO models need to be defined in *CONTACT_COUPLING.

*MAT_ORTHOTROPIC_THERMAL_{OPTION}

This is Material Type 21. It is a linearly elastic, orthotropic material with orthotropic thermal expansion. It is available for solids, shells, and thick shells.

Available options include:

<BLANK>

FAILURE

CURING

Card Summary:

Card 1. This card is required.

MID	RO	EA	EB	EC	PRBA	PRCA	PRCB

Card 2. This card is required.

GAB	GBC	GCA	AA	AB	AC	AOPT	MACF

Card 3. This card is required.

XP	YP	ZP	A1	A2	A3		

Card 4. This card is required.

V1	V2	V3	D1	D2	D3	BETA	REF

Card 5a. This card is included if and only if the keyword option FAILURE is used.

A1	A11	A2	A5	A55	A4	NIP	

Card 5b.1. This card is included if and only if the keyword option CURING is used.

K1	K2	C1	C2	M	N	R	

Card 5b.2. This card is included if and only if the keyword option CURING is used.

LCCHA	LCCHB	LCCHC	LCAA	LCAB	LCAC		

Data Card Definitions:

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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

Card 2	1	2	3	4	5	6	7	8
Variable	GAB	GBC	GCA	AA	AB	AC	AOPT	MACF
Type	F	F	F	F	F	F	F	I

VARIABLE	DESCRIPTION
GAB	G_{ab}, Shear modulus, ab
GBC	G_{bc}, Shear modulus, bc
GCA	G_{ca}, Shear modulus, ca
AA	α_a, coefficient of thermal expansion in the a-direction

VARIABLE	DESCRIPTION
AB	α_b, coefficient of thermal expansion in the *b*-direction
AC	α_c, coefficient of thermal expansion in the *c*-direction
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC, particularly the Material Directions section, for details): EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. For shells only, the material axes are then rotated about the normal vector to the surface of the shell by the angle BETA. EQ.1.0: Locally orthotropic with material axes determined by a point, *P*, 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 a vector **v** and the normal vector to the plane of the element. 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then **a** and **b** are rotated about **c** by an angle BETA. BETA may be set in the keyword input for the element or in the input for this keyword. Note that for solids, the material axes may be switched depending on the choice of MACF. The switch may occur before or after applying BETA depending on the value of MACF. EQ.4.0: Locally orthotropic in a 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. LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_*OPTION*).

VARIABLE	DESCRIPTION
MACF	Material axes change flag for solid elements: EQ.-4: Switch material axes *b* and *c* before BETA rotation EQ.-3: Switch material axes *a* and *c* before BETA rotation EQ.-2: Switch material axes *a* and *b* before BETA rotation EQ.1: No change, default EQ.2: Switch material axes *a* and *b* after BETA rotation EQ.3: Switch material axes *a* and *c* after BETA rotation EQ.4: Switch material axes *b* and *c* after BETA rotation Figure M2-2 indicates when LS-DYNA applies MACF during the process to obtain the final material axes. If BETA on *ELEMENT_SOLID_{OPTION} is defined, then that BETA is used for the rotation for all AOPT options. Otherwise, if AOPT = 3, the BETA input on Card 4 rotates the axes. For all other values of AOPT, the material axes will be switched as specified by MACF, but no BETA rotation will be performed.

Card 3	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
XP, YP, ZP	Coordinates of point *p* for AOPT = 1 and 4
A1, A2, A3	Components of vector **a** for AOPT = 2

Card 4	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	REF
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
V1, V2, V3	Components of vector **v** for AOPT = 3 and 4
D1, D2, D3	Components of vector **d** for AOPT = 2
BETA	Material angle in degrees for AOPT = 0 (shells and tshells only) and AOPT = 3 (all element types). It may be overridden on the element card; see *ELEMENT_SHELL_BETA, *ELEMENT_TSHELL_BETA, or *ELEMENT_SOLID_ORTHO.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: Off EQ.1.0: On

Failure Card. This card is only included if the FAILURE keyword option is used.

Card 5a	1	2	3	4	5	6	7	8
Variable	A1	A11	A2	A5	A55	A4	NIP	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
A1, A11, A2	Coefficients for the matrix dominated failure criterion
A5, A55, A4	Coefficients for the fiber dominated failure criterion
NIP	Number of integration points that must fail in an element before an element fails and is deleted

Curing Card. This card is included if and only if the CURING keyword option is used.

Card 5b.1	1	2	3	4	5	6	7	8
Variable	K1	K2	C1	C2	M	N	R	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
K1	Parameter k_1 for Kamal model. For details see remarks below.
K2	Parameter k_2 for Kamal model
C1	Parameter c_1 for Kamal model
C2	Parameter c_2 for Kamal model
M	Exponent m for Kamal model
N	Exponent n for Kamal model
R	Gas constant for Kamal model

Curing Card. This card is included if and only if the CURING keyword option is used.

Card 5b.2	1	2	3	4	5	6	7	8
Variable	LCCHA	LCCHB	LCCHC	LCAA	LCAB	LCAC		
Type	I	I	I	I	I	I		

VARIABLE	DESCRIPTION
LCCHA	Load curve for γ_a, coefficient of chemical shrinkage in the a-direction. Input γ_a as function of state of cure β.
LCCHB	Load curve for γ_b, coefficient of chemical shrinkage in the b-direction. Input γ_b as function of state of cure β.
LCCHC	Load curve for γ_c, coefficient of chemical shrinkage in the c-direction. Input γ_c as function of state of cure β.
LCAA	Load curve or table ID for α_a. If defined, parameter AA is ignored. If a load curve, then α_a is a function of temperature. If a table ID, the α_a is a function of the state of cure (table values) and temperature (see*DEFINE_TABLE).
LCAB	Load curve ID for α_b. If defined parameter, AB is ignored. See LCAA for further details.
LCAC	Load curve ID for α_c. If defined parameter, AC is ignored. See LCAA for further details.

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

$$\mathbf{S} = \mathbf{C} : \mathbf{E} = \mathbf{T}^{\mathrm{T}} \mathbf{C}_l \mathbf{T} : \mathbf{E}$$

where **T** is the transformation matrix [Cook 1974].

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

l_i, m_i, n_i are the direction cosines

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

and x_i' denotes the material axes. The constitutive matrix $\mathbf{C}_l$ is defined in terms of the material axes as

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

where the subscripts denote the material axes, meaning

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

Since $\mathbf{C}_l$ is symmetric

$$\frac{\upsilon_{12}}{E_{11}} = \frac{\upsilon_{21}}{E_{22}}, \ldots$$

The vector of Green-St. Venant strain components is

$$\mathbf{E}^{\mathrm{T}} = [E_{11}, E_{22}, E_{33}, E_{12}, E_{23}, E_{31}]$$

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

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

where T is temperature. After computing S_{ij} we then obtain the Cauchy stress:

$$\sigma_{ij} = \frac{\rho}{\rho_0}\frac{\partial x_i}{\partial X_k}\frac{\partial x_j}{\partial X_l}S_{kl}$$

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.

The failure models were derived by William Feng. The first one defines the matrix dominated failure mode,

$$F_m = A_1(I_1 - 3) + A_{11}(I_1 - 3)^2 + A_2(I_2 - 3) - 1 \ ,$$

and the second defines the fiber dominated failure mode,

$$F_f = A_5(I_5 - 1) + A_{55}(I_5 - 1)^2 + A_4(I_4 - 1) - 1 \ .$$

When either is greater than zero, the integration point fails, and the element is deleted after NIP integration points fail.

The coefficients A_i are defined in the input and the invariants I_i are the strain invariants

$$\begin{aligned}
I_1 &= \sum_{\alpha=1,3} C_{\alpha\alpha} \\
I_2 &= \frac{1}{2}[I_1^2 - \sum_{\alpha,\beta=1,3} C_{\alpha\beta}^2] \\
I_3 &= \det(\mathbf{C}) \\
I_4 &= \sum_{\alpha,\beta,\gamma=1,3} V_\alpha C_{\alpha\gamma} C_{\gamma\beta} V_\beta \\
I_5 &= \sum_{\alpha,\beta=1,3} V_\alpha C_{\alpha\beta} V_\beta
\end{aligned}$$

and $\mathbf{C}$ is the Cauchy strain tensor and $\mathbf{V}$ is the fiber direction in the undeformed state. By convention in this material model, the fiber direction is aligned with the a direction of the local orthotropic coordinate system.

The curing option implies that orthotropic chemical shrinkage is to be considered, resulting from a curing process in the material. The state of cure β is an internal material variable that follows the Kamal model

$$\frac{d\beta}{dt} = (K_1 + K_2\beta^m)(1-\beta)^n \quad \text{with} \qquad K_1 = k_1 e^{-\frac{c_1}{RT}},\ K_2 = k_2 e^{-\frac{c_2}{RT}}$$

Chemical strains are introduced as:

$$\begin{aligned}
\varepsilon_{aa}^{n+1} &= \varepsilon_{aa}^{n} + \gamma_a\left(\beta^{n+1} - \beta^{n}\right) \\
\varepsilon_{bb}^{n+1} &= \varepsilon_{bb}^{n} + \gamma_b\left(\beta^{n+1} - \beta^{n}\right) \\
\varepsilon_{cc}^{n+1} &= \varepsilon_{cc}^{n} + \gamma_c\left(\beta^{n+1} - \beta^{n}\right)
\end{aligned}$$

The coefficients, γ_a, γ_b, and γ_c, can be defined as functions of the state of cure β. Furthermore, the coefficients of thermal expansion, α_a, α_b, and α_c, can also be defined as functions of the state of cure, β, and the temperature, T, if the curing option is used.

The current degree of cure as well as the chemical shrinkage in the different directions is output in the history variables. For solid elements it can be found at positions 30 to 33 and for shell elements at positions 22 to 25.

*MAT_COMPOSITE_DAMAGE

This is Material Type 22. With this model, an orthotropic material with optional brittle failure for composites can be defined following the suggestion of [Chang and Chang 1987a, 1987b]. Failure can be modeled with three criteria; see the LS-DYNA Theory 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.

This material is available for shells, solids, thick shells, and SPH elements.

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

Card 2	1	2	3	4	5	6	7	8
Variable	GAB	GBC	GCA	KFAIL	AOPT	MACF	ATRACK	
Type	F	F	F	F	F	I	I	
Default	none	none	none	0.0	0.0	0	0	

Card 3	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 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	
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	

Card 5	1	2	3	4	5	6	7	8
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 or label must be specified (see *PART).
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

VARIABLE	DESCRIPTION
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.
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC, particularly the Material Directions section, for details): EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. For shells only, the material axes are then rotated about the normal vector to the surface of the shell by the angle BETA. EQ.1.0: Locally orthotropic with material axes determined by a point, *P*, 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 a vector **v** and the normal vector to the plane of the element. 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then **a** and **b** are rotated about **c** by an angle BETA. BETA may be set in the keyword input for the element or in the input for this keyword. Note that for solids, the material axes may be switched depending on the choice of MACF. The switch may occur before or after applying

VARIABLE	DESCRIPTION
	BETA depending on the value of MACF.
	EQ.4.0: Locally orthotropic in a 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.
	LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_*OPTION*).
MACF	Material axes change flag for solid elements:
	EQ.-4: Switch material axes *b* and *c* before BETA rotation
	EQ.-3: Switch material axes *a* and *c* before BETA rotation
	EQ.-2: Switch material axes *a* and *b* before BETA rotation
	EQ.1: No change, default
	EQ.2: Switch material axes *a* and *b* after BETA rotation
	EQ.3: Switch material axes *a* and *c* after BETA rotation
	EQ.4: Switch material axes *b* and *c* after BETA rotation
	Figure M2-2 indicates when LS-DYNA applies MACF during the process to obtain the final material axes. If BETA on *ELEMENT_-SOLID_*{OPTION}* is defined, then that BETA is used for the rotation for all AOPT options. Otherwise, if AOPT = 3, the BETA input on Card 4 rotates the axes. For all other values of AOPT, the material axes will be switched as specified by MACF, but no BETA rotation will be performed.
ATRACK	Material *a*-axis tracking flag (shell elements only):
	EQ.0: *a*-axis rotates with element (default).
	EQ.1: *a*-axis also tracks deformation (see Remark 2).
XP, YP, ZP	Coordinates of point *p* for AOPT = 1 and 4
A1, A2, A3	Components of vector **a** for AOPT = 2
V1, V2, V3	Components of vector **v** for AOPT = 3 and 4
D1, D2, D3	Components of vector **d** for AOPT = 2

VARIABLE	DESCRIPTION
BETA	Material angle in degrees for AOPT = 0 (shells and tshells only) and AOPT = 3. It may be overridden on the element card; see *ELEMENT_SHELL_BETA, *ELEMENT_TSHELL_BETA, or *ELEMENT_SOLID_ORTHO.
SC	Shear strength, *ab*-plane; see the LS-DYNA Theory Manual.
XT	Longitudinal tensile strength, *a*-axis; see the LS-DYNA Theory Manual.
YT	Transverse tensile strength, *b*-axis
YC	Transverse compressive strength, *b*-axis (positive value)
ALPH	Shear stress parameter for the nonlinear term in units of [stress^{-3}]; see the LS-DYNA Theory Manual.
SN	Normal tensile strength *(solid elements only)*
SYZ	Transverse shear strength *(solid elements only)*
SZX	Transverse shear strength *(solid elements only)*

Remarks:

1. **History data.** The number of additional integration point variables for shells written to the d3plot database is specified using the *DATABASE_EXTENT_BINARY keyword on the NEIPS field. These additional history variables are enumerated below:

History Variable[4]	Description	Value	LS-PrePost History Variable
ef(i)	tensile fiber mode	1 - elastic 0 - failed	See table below
cm(i)	tensile matrix mode		1
ed(i)	compressive matrix mode		2

The following components are stored as element component 7 instead of the effective plastic strain. Note that ef(i) for $i = 1,2,3$ is not retrievable.

[4] i ranges over the shell integration points.

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

2. **The ATRACK field.** The initial material directions are set using AOPT and the related data. By default, the material directions in shell elements are updated each cycle based on the rotation of the 1-2 edge, or else the rotation of all edges if the invariant node numbering option is set on *CONTROL_ACCURACY. When ATRACK = 1, an optional scheme is used in which the *a*-direction of the material tracks element deformation as well as rotation.

   At the start of the calculation, a line is passed through each element center in the direction of the material *a*-axis. This line will intersect the edges of the element at two points. The referential coordinates of these two points are stored and then used throughout the calculation to locate these points in the deformed geometry. The material *a*-axis is assumed to be in the direction of the line that passes through both points. If ATRACK = 0, the layers of a layered composite will always rotate together. However, if ATRACK = 1, the layers can rotate independently which may be more accurate, particularly for shear deformation. This option is available only for shell elements.

3. **Poisson's ratio.** If EA > EB, PRBA is the minor Poisson's ratio if EA > EB, and the major Poisson's ratio will be equal to PRBA × (EA/EB). If EB > EA, then PRBA is the major Poisson's ratio. PRCA and PRCB are similarly defined. They are the minor Poisson's ratio if EA>EC or EB>EC, and the major Poisson's ratio if the EC > EA or EC > EB.

   Care should be taken when using material parameters from third party products regarding the directional indices *a*, *b* and *c*, as they may differ from the definition used in LS-DYNA. For the direction indices used in LS-DYNA, see the remarks section of *MAT_002 / *MAT_*OPTION*TROPIC_ELASTIC.

*MAT_TEMPERATURE_DEPENDENT_ORTHOTROPIC

This is Material Type 23. It models an orthotropic elastic material with arbitrary temperature dependency. It is available for solids, shells, and thick shells.

Card Summary:

Card 1. This card is required.

MID	RO	AOPT	REF	MACF	IHYPO		

Card 2. This card is required.

XP	YP	ZP	A1	A2	A3		

Card 3. This card is required.

V1	V2	V3	D1	D2	D3	BETA	

Card 4.1. Define one set of constants on two cards using Cards 4.1 and 4.2 for each temperature point. Up to 48 points (96 cards) can be defined. The next keyword ("*") card terminates the input.

EA*i*	EB*i*	EC*i*	PRBA*i*	PRCA*i*	PRCB*i*		

Card 4.2. Define one set of constants on two cards using Cards 4.1 and 4.2 for each temperature point. Up to 48 points (96 cards) can be defined. The next keyword ("*") card terminates the input.

AA*i*	AB*i*	AC*i*	GAB*i*	GBC*i*	GCA*i*	T*i*	

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	AOPT	REF	MACF	IHYPO		
Type	A	F	F	F	I	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).

VARIABLE	DESCRIPTION
RO	Mass density
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC, particularly the Material Directions section, for details): EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. For shells only, the material axes are then rotated about the normal vector to the surface of the shell by the angle BETA. EQ.1.0: Locally orthotropic with material axes determined by a point, P, 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 a vector **v** and the normal vector to the plane of the element. 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then **a** and **b** are rotated about **c** by an angle BETA. BETA may be set in the keyword input for the element or in the input for this keyword. Note that for solids, the material axes may be switched depending on the choice of MACF. The switch may occur before or after applying BETA depending on the value of MACF. EQ.4.0: Locally orthotropic in a 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. LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_*OPTION*).

VARIABLE	DESCRIPTION
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY (see description of this keyword for more details). EQ.0.0: Off EQ.1.0: On
MACF	Material axes change flag for solid elements: EQ.-4: Switch material axes *b* and *c* before BETA rotation EQ.-3: Switch material axes *a* and *c* before BETA rotation EQ.-2: Switch material axes *a* and *b* before BETA rotation EQ.1: No change, default EQ.2: Switch material axes *a* and *b* after BETA rotation EQ.3: Switch material axes *a* and *c* after BETA rotation EQ.4: Switch material axes *b* and *c* after BETA rotation Figure M2-2 indicates when LS-DYNA applies MACF during the process to obtain the final material axes. If BETA on *ELEMENT_SOLID_*{OPTION}* is defined, then that BETA is used for the rotation for all AOPT options. Otherwise, if AOPT = 3, the BETA input on Card 3 rotates the axes. For all other values of AOPT, the material axes will be switched as specified by MACF, but no BETA rotation will be performed.
IHYPO	Option to switch between two different elastic approaches (only available for solid elements): EQ.0.0: Hyperelastic formulation, default EQ.1.0: Hypoelastic formulation (allows stress initialization through *INITIAL_STRESS_SOLID)

Card 2	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
XP, YP, ZP	Coordinates of point p for AOPT = 1 and 4
A1, A2, A3	Components of vector **a** for AOPT = 2

Card 3	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
V1, V2, V3	Components of vector **v** for AOPT = 3 and 4
D1, D2, D3	Components of vector **d** for AOPT = 2
BETA	Material angle in degrees for AOPT = 0 (shells and tshells only) and AOPT = 3. It may be overridden on the element card; see *ELEMENT_SHELL_BETA, *ELEMENT_TSHELL_BETA, or *ELEMENT_SOLID_ORTHO.

First Temperature Card. Define one set of constants on two cards using Cards 4.1 and 4.2 for each temperature point. Up to 48 points (96 cards) can be defined. The next keyword ("*") card terminates the input.

Card 4.1	1	2	3	4	5	6	7	8
Variable	EA*i*	EB*i*	EC*i*	PRBA*i*	PRCA*i*	PRCB*i*		
Type	F	F	F	F	F	F		

Second Temperature Card

Card 4.2	1	2	3	4	5	6	7	8
Variable	AA*i*	AB*i*	AC*i*	GAB*i*	GBC*i*	GCA*i*	T*i*	
Type	F	F	F	F	F	F	F	

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	i^{th} 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

$$\mathbf{S} = \mathbf{C}\colon \mathbf{E} = \mathbf{T}^{\mathrm{T}}\mathbf{C}_l\mathbf{T}\colon \mathbf{E}$$

where **T** is the transformation matrix [Cook 1974].

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

l_i, m_i, n_i are the direction cosines

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

and x_i' denotes the material axes. The temperature dependent constitutive matrix $\mathbf{C}_l$ is defined in terms of the material axes as

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

where the subscripts denote the material axes,

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

Since $\mathbf{C}_l$ is symmetric

$$\frac{v_{12}}{E_{11}} = \frac{v_{21}}{E_{22}}, \dots$$

The vector of Green-St. Venant strain components is

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

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

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

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

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

where T is temperature. After computing S_{ij} we then obtain the Cauchy stress:

$$\sigma_{ij} = \frac{\rho}{\rho_0} \frac{\partial x_i}{\partial X_k} \frac{\partial x_j}{\partial X_l} S_{kl}$$

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_{OPTION}

Available options include:

<BLANK>

LOG_INTERPOLATION

STOCHASTIC

MIDFAIL

2D

This is Material Type 24. It is an elasto-plastic material with an arbitrary stress as a function of strain curve that can also have an arbitrary strain rate dependency (see Remarks below). 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 non-uniform 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.

The LOG_INTERPOLATION keyword option interpolates the strain rates in a table LCSS with logarithmic interpolation.

The STOCHASTIC keyword option allows spatially varying yield and failure behavior. See *DEFINE_STOCHASTIC_VARIATION for additional information.

The MIDFAIL keyword option is available for thin shell elements and thick shell formulations which use thin shell material models. When included on the keyword line, this option causes failure to be checked only at the mid-plane of the element. If an element has an even number of layers, failure is checked in the two layers closest to the mid-plane.

The 2D keyword option is available only for shell elements. It invokes actual plane stress treatment, meaning transverse shear stresses are not part of the yield condition but are updated elastically.

All four keyword options can be combined with each other. The order of the options is arbitrary. Before R16, the combination of STOCHASTIC and 2D was not available. The shell-related keyword options, MIDFAIL and 2D, are ignored if used on solid elements.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	FAIL	TDEL
Type	A	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10^{21}	0.0

Card 2	1	2	3	4	5	6	7	8
Variable	C	P	LCSS	LCSR	VP			
Type	F	F	I	I	F			
Default	0.0	0.0	0	0	0.0			

Card 3	1	2	3	4	5	6	7	8
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	0.0	0.0	0.0	0.0

Card 4	1	2	3	4	5	6	7	8
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	0.0	0.0	0.0	0.0

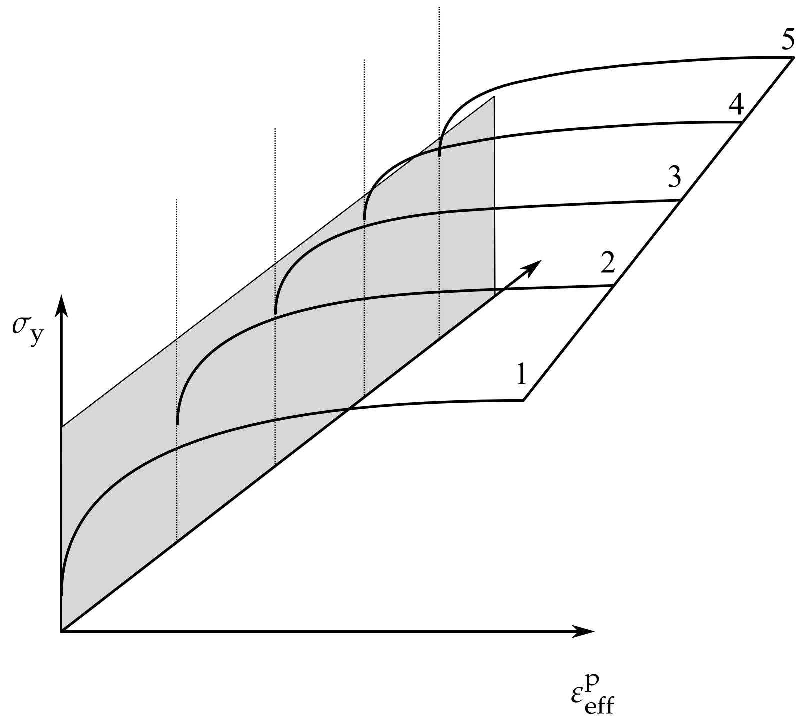


Figure M24-1. Rate effects may be accounted for by defining a table of curves. If a table ID is specified, a curve ID is given for each strain rate; see *DEFINE_TABLE. Intermediate values are found by interpolating between curves. Effective plastic strain as a function of yield stress is expected. If 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.

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress; ignored if LCSS > 0 except as described in Remark 1a.
ETAN	Tangent modulus; ignored if LCSS > 0 is defined.
FAIL	Failure flag: LT.0.0: User defined failure subroutine, `matusr_24` in dyn21.F, is called to determine failure.

VARIABLE	DESCRIPTION
	EQ.0.0: Failure is not considered. This option is recommended if failure is not of interest since many calculations will be saved. GT.0.0: Effective 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 Remarks 1 and 3.
P	Strain rate parameter, p; see Remark 1.
LCSS	Load curve ID or Table ID **Load Curve.** When LCSS is a load curve ID, it is taken as defining effective stress as a function of effective plastic strain. If defined, EPS1 - EPS8 and ES1 - ES8 are ignored. See Remark 7 for load curve rediscretization behavior. **Tabular Data.** The table ID defines for each strain rate value a load curve ID giving the stress as a function effective plastic strain for that rate; see Figure M24-1. When the strain rate falls below the minimum value, the stress as a function of effective plastic strain curve for the lowest value of strain rate is used. Likewise, when the strain rate exceeds the maximum value the stress as a function of effective plastic strain curve for the highest value of strain rate is used. C, P, LCSR, EPS1 - EPS8, and ES1 - ES8 are ignored if a table ID is defined. Linear interpolation between the discrete strain rates is used by default; logarithmic interpolation is used when the LOG_INTERPOLATION option is invoked. **Logarithmically Defined Tables.** Logarithmic interpolation between discrete strain rates is also assumed if the *first* value in the table is negative, in which case LS-DYNA assumes that all the table values represent the natural logarithm of a strain rate. Since the tables are internally discretized to equally space the table values, it makes good sense from an accuracy standpoint that the table values represent the natural log of strain rate when the lowest strain rate and highest strain rate differ by several orders of magnitude. Note that this option works only when the lowest strain rate has value less than 1.0. For values greater than or equal to 1.0, use the LOG_INTERPOLATION option. There is some additional computational cost associated with invoking logarithmic interpolation.

VARIABLE	DESCRIPTION
	Multi-Dimensional Tables. With VP = 3, yield stress can be a function of plastic strain, strain rate, and up to seven history variables (see Remark 4). That means LCSS can refer to *DEFINE_TABLE_XD or *DEFINE_TABLE_COMPACT up to a level of 9.
LCSR	Load curve ID defining strain rate scaling effect on yield stress. If LCSR is negative, the load curve is evaluated using a binary search for the correct interval for the strain rate. The binary search is slower than the default incremental search, but in cases where large changes in the strain rate may occur over a single time step, it is more robust. This option is not necessary for the viscoplastic formulation.
VP	Formulation for rate effects: EQ.-1.0: Cowper-Symonds with effective deviatoric strain rate rather than total EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation EQ.3.0: Same as VP = 0, but with filtered effective total strain rates (see Remark 3)
EPS1 - EPS8	Effective plastic strain values (optional). If used, 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:

1. **Stress-Strain Behavior.** The stress-strain behavior may be treated by a bilinear stress strain curve by defining the tangent modulus, ETAN. Alternately, a curve of effective stress as a function of effective plastic strain similar to that shown in Figure M10-1 may be defined by (EPS1, ES1) - (EPS8, ES8); however, a curve ID (LCSS) may be referenced instead if eight points are insufficient. The cost is roughly the same for either approach. Note that in the special case of uniaxial stress, true stress as a function of true plastic strain is equivalent to effective stress as a function effective plastic strain. The most general approach is to use the table definition (LCSS) discussed below.

Three options to account for strain rate effects are possible:

a) 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, $\sigma_y^s(\varepsilon_{\text{eff}}^p)$, which is typically given by a load curve ID and the initial yield stress, SIGY, multiplied by the Cowper-Symonds rate term as follows:

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

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

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

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

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

c) If different stress as a function of 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 must be used; see Figure M24-1.

2. **Viscoplastic Formulation.** A fully viscoplastic formulation is optional (variable VP = 1) which incorporates the different options above within the yield surface. An additional cost is incurred over the simple scaling, but the improvement in results can be dramatic.

3. **Filtered Strain Rates.** With the option VP = 3 it is possible to use filtered strain rates. This means that the total strain rate is used as with VP = 0, but this can now be filtered with the help of field C (not Cowper-Symonds in this case) and the following exponential moving average equation:

$$\dot{\varepsilon}_n^{\text{avg}} = \mathrm{C} \times \dot{\varepsilon}_{n-1}^{\text{avg}} + (1 - \mathrm{C}) \times \dot{\varepsilon}_n$$

This might be helpful if a table LCSS with crossing yield curves is used.

4. **Yield Stress Depending on History Variables.** When VP = 3, the yield stress defined with LCSS can depend on up to seven history variables through a multi-dimensional table. These seven history variables are history variables 6 through 12 which you will have to set using *INITIAL_HISTORY_NODE or *INITIAL_-STRESS_SOLID/SHELL and whose meanings are, therefore, determined by you. For instance, you can set the values of history variables 6, 9, and 10 for certain nodes and have the value of yield stress depend upon history variables 6, 9, and 10. Note that these history variables are only initialized and do not evolve in time. See *DEFINE_TABLE_XD or *DEFINE_TABLE_COMPACT for more details.

5. **Implicit Calculations.** For implicit calculations with this material involving severe nonlinear hardening, the radial return method may result in inaccurate stress-strain response. Setting IACC = 1 on *CONTROL_ACCURACY activates a fully iterative plasticity algorithm, which will remedy this. This is not to be confused with the MITER flag on *CONTROL_SHELL which governs the treatment of the plane stress assumption for shell elements. If failure is applied with this option, incident failure will initiate damage, and the stress will continuously degrade to zero before erosion for a deformation of 1% plastic strain. So for instance, if the failure strain is FAIL = 0.05, then the element is eroded when $\bar{\varepsilon}^p = 0.06$ and the material goes from intact to completely damaged between $\bar{\varepsilon}^p = 0.05$ and $\bar{\varepsilon}^p = 0.06$. The reason is to enhance implicit performance by maintaining continuity in the internal forces.

6. **Failure Output.** For a nonzero failure strain, *DEFINE_MATERIAL_HISTORIES can be used to output the failure indicator.

**DEFINE_MATERIAL_HISTORIES Properties*

Label	Attributes				Description
Instability	-	-	-	-	Failure indicator $\varepsilon_{\text{eff}}^p / \varepsilon_{\text{fail}}^p$, see FAIL
Plastic Strain Rate	-	-	-	-	Effective plastic strain rate $\dot{\varepsilon}_{\text{eff}}^p$

7. **LCSS Rediscretization.** In the special case where LCSS is a *DEFINE_CURVE, LCSS is not rediscretized (see LCINT in *DEFINE_CURVE).

*MAT_GEOLOGIC_CAP_MODEL

This is Material Type 25. This is an inviscid two-invariant geologic cap model. This material model can be used for geomechanical problems or for materials such as concrete; see references cited below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	G	ALPHA	THETA	GAMMA	BETA
Type	A	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	R	D	W	X0	C	N		
Type	F	F	F	F	F	F		

Card 3	1	2	3	4	5	6	7	8
Variable	PLOT	FTYPE	VEC	TOFF				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
BULK	Initial bulk modulus, K
G	Initial shear modulus
ALPHA	Failure envelope parameter, α
THETA	Failure envelope linear coefficient, θ
GAMMA	Failure envelope exponential coefficient, γ

VARIABLE	DESCRIPTION
BETA	Failure envelope exponent, β
R	Cap, surface axis ratio
D	Hardening law exponent
W	Hardening law coefficient
X0	Initial intersection of the cap surface with the J_1 axis, X_0
C	Kinematic hardening coefficient, $\bar{c}$
N	Kinematic hardening parameter
PLOT	Save the following variable for plotting in LS-PrePost, where it will be labeled as "effective plastic strain:" EQ.1: hardening parameter, κ EQ.2: cap -J1 axis intercept, $X(\kappa)$ EQ.3: volumetric plastic strain ε_v^p EQ.4: first stress invariant, J_1 EQ.5: second stress invariant, $\sqrt{J_2}$ EQ.6: not used EQ.7: not used EQ.8: response mode number EQ.9: number of iterations
FTYPE	Formulation flag: EQ.1: soils (cap surface may contract) EQ.2: concrete and 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).

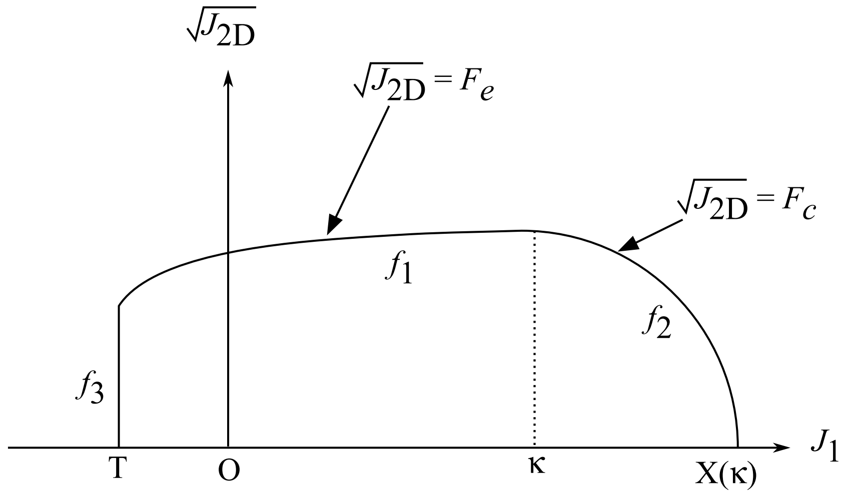


Figure M25-1. The yield surface of the two-invariant cap model in pressure $\sqrt{J_{2D}} - J_1$ space. Surface f_1 is the failure envelope, f_2 is the cap surface, and f_3 is the tension cutoff.

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, Vaughan, and Sandler [1978]. A brief discussion of the extended cap model and its parameters is given below.

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 M25-1 First, there is a failure envelope surface, denoted f_1 in the figure. The functional form of f_1 is

$$f_1 = \sqrt{J_{2D}} - \min[F_e(J_1), T_{\text{mises}}],$$

where F_e is given by

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

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

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

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

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

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

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

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

$$f_3 \equiv T - J_1$$

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

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

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

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

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

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

The yield condition may be written

$$\begin{aligned} f_1(s) &\le 0 \\ f_2(s, \kappa) &\le 0 \\ f_3(s) &\le 0 \end{aligned}$$

and the plastic consistency condition requires that

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

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

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

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

One of the major advantages of the cap model over other classical pressure-dependent plasticity models is the ability to control the amount of dilatancy produced under shear loading. Dilatancy 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 M25-1). In models such as the Drucker-Prager and Mohr-Coulomb, this dilatancy continues as long as shear loads are applied, and in many cases produces far more dilatancy than is experimentally observed in material tests. In the cap model, when the failure surface is active, dilatancy is produced just as with the Drucker-Prager and Mohr-Coulumb 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 (dilatancy) is created. Adjustment of the parameters that control the rate of cap contractions permits experimentally observed amounts of dilatancy 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 dilatancy, 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 $\bar{c}$ 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 equations. The history tensor $\boldsymbol{\alpha}$ is assumed deviatoric and therefore has only 5 unique components. The evolution of the back stress tensor is governed by the nonlinear hardening law

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

where $\bar{c}$ is a constant, $\bar{F}$ is a scalar function of $\boldsymbol{\sigma}$ and $\boldsymbol{\alpha}$ and $\dot{e}^p$ is the rate of deviatoric 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

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

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

The cap model contains a number of parameters which must be chosen to represent a particular material and are generally based on experimental data. The parameters α, β, θ, and γ are usually evaluated by fitting a curve through failure data taken from a set of triaxial compression tests. The parameters W, D, and X_0 define the cap hardening law. The value W represents 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 ratio 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. This material is available for solid elements and for thick shell formulations 3, 5, and 7.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	VF	MU	BULK
Type	A	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	.05	0.0

Card 2	1	2	3	4	5	6	7	8
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	MACF
Type	F	F	F	F	F	F		I

Card 4	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	D1	D2	D3	TSEF	SSEF	V1	V2	V3
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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. The default, 0.05, is recommended.
BULK	Bulk viscosity flag: EQ.0.0: Bulk viscosity is not used. This is recommended. EQ.1.0: Bulk viscosity is active and $\mu = 0$. This will give results identical to previous versions of LS-DYNA.
LCA	Load curve ID (see *DEFINE_CURVE) for σ_{aa} as a function of either relative volume or volumetric strain. See Remarks 1 and 3.
LCB	Load curve ID (see *DEFINE_CURVE) for σ_{bb} as a function of either relative volume or volumetric strain. By default, LCB = LCA. See Remarks 1 and 3.
LCC	Load curve ID (see *DEFINE_CURVE) for σ_{cc} as a function of either relative volume or volumetric strain. By default, LCC = LCA. See Remarks 1 and 3.
LCS	Load curve ID (see *DEFINE_CURVE) for shear stress as a function of either relative volume or volumetric strain. By default, LCS = LCA. Each component of shear stress may have its own load curve. See Remarks 1 and 3.

VARIABLE	DESCRIPTION
LCAB	Load curve ID (see *DEFINE_CURVE) for σ_{ab} as a function of either relative volume or volumetric strain. By default, LCAB = LCS. See Remarks 1 and 3.
LCBC	Load curve ID (see *DEFINE_CURVE) for σ_{bc} as a function of either relative volume or volumetric strain. By default, LCBC = LCS. See Remarks 1 and 3.
LCCA	Load curve ID (see *DEFINE_CURVE) for σ_{ca} as a function of either relative volume or volumetric strain. Default LCCA = LCS. See Remarks 1 and 3.
LCSR	Load curve ID (see *DEFINE_CURVE) for strain-rate effects defining the scale factor as a function of strain rate. This is optional. The curves defined above are scaled using this curve. See Remark 3.
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, particularly the Material Directions section, for details): EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. For shells only, the material axes are then rotated about the normal vector to the surface of the shell by the angle BETA. EQ.1.0: Locally orthotropic with material axes determined by a point, P, 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

VARIABLE	DESCRIPTION
	EQ.3.0: Locally orthotropic material axes determined by a vector **v** and the normal vector to the plane of the element. 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then **a** and **b** are rotated about **c** by an angle BETA. BETA may be set in the keyword input for the element. Note that for solids, the material axes may be switched depending on the choice of MACF. The switch may occur before or after applying BETA depending on the value of MACF.
	EQ.4.0: Locally orthotropic in a 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.
	LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_*OPTION*).
MACF	Material axes change flag for solid elements:
	EQ.-4: Switch material axes *b* and *c* before BETA rotation
	EQ.-3: Switch material axes *a* and *c* before BETA rotation
	EQ.-2: Switch material axes *a* and *b* before BETA rotation
	EQ.1: No change, default
	EQ.2: Switch material axes *a* and *b* after BETA rotation
	EQ.3: Switch material axes *a* and *c* after BETA rotation
	EQ.4: Switch material axes *b* and *c* after BETA rotation
	Figure M2-2 indicates when LS-DYNA applies MACF during the process to obtain the final material axes. BETA, if needed, is specified on *ELEMENT_SOLID_*{OPTION}*.
XP YP ZP	Coordinates of point *p* for AOPT = 1 and 4.
A1 A2 A3	Components of vector **a** for AOPT = 2.

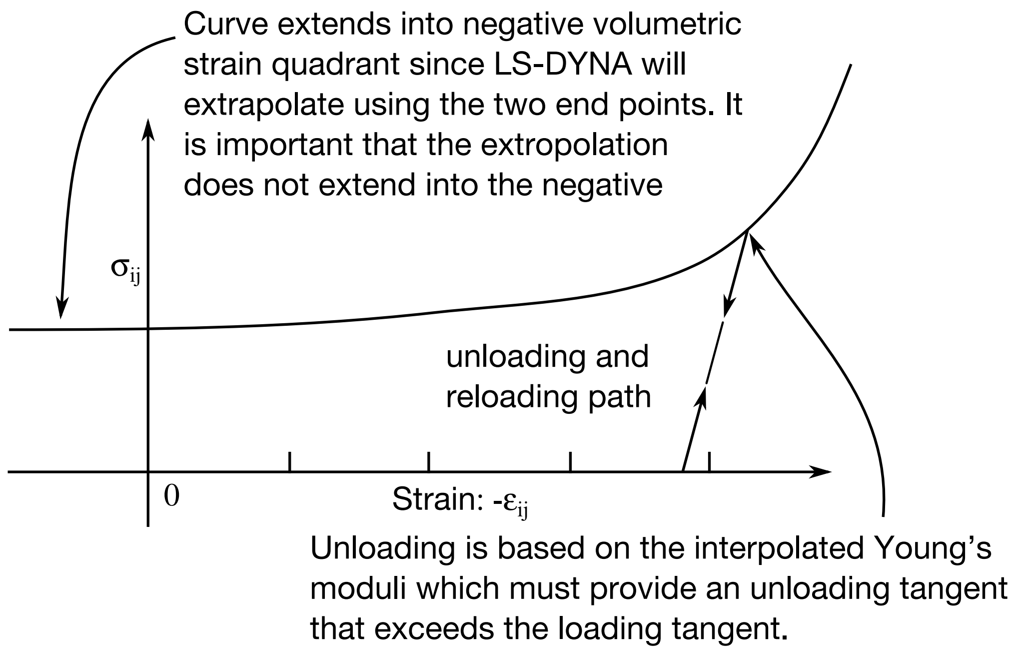


Figure M26-1. Stress quantity versus volumetric strain. Note that the "yield stress" at a volumetric strain of zero is non-zero. In the load curve definition, see *DEFINE_CURVE, the "time" value is the volumetric strain and the "function" value is the yield stress.

VARIABLE	DESCRIPTION
D1 D2 D3	Components of vector **d** for AOPT = 2.
V1 V2 V3	Define components of vector **v** for AOPT = 3 and 4.
TSEF	Tensile strain at element failure (element will erode).
SSEF	Shear strain at element failure (element will erode).

Remarks:

1. **Stress Load Curves.** For efficiency it is strongly recommended that the load curves with IDs LCA, LCB, LCC, LCS, LCAB, LCBC, and LCCA contain exactly the same number of points with corresponding strain values on the abscissa. If this recommendation is followed the cost of the table lookup is insignificant. Conversely, the cost increases significantly if the abscissa strain values are not consistent between load curves.

   The load curves define the magnitude of the average stress as the material changes density (relative volume); see Figure M26-1. There are two ways to define these curves, (1) as a function of relative volume, V, or (2) as a function of volumetric strain defined as:

$$\varepsilon_V = 1 - V \ .$$

In the former case, 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.*

2. **Elastic/Shear Moduli during Compaction.** The behavior before compaction is orthotropic where the components of the stress tensor are uncoupled, meaning 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:

$$E_{aa} = E_{aau} + \beta(E - E_{aau})$$
$$E_{bb} = E_{bbu} + \beta(E - E_{bbu})$$
$$E_{cc} = E_{ccu} + \beta(E - E_{ccu})$$
$$G_{ab} = E_{abu} + \beta(G - G_{abu})$$
$$G_{bc} = E_{bcu} + \beta(G - G_{bcu})$$
$$G_{ca} = E_{cau} + \beta(G - G_{cau})$$

where

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

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

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

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.

3. **Stress Updates.** 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 (see Remark 2) according to:

$$\begin{aligned}
\sigma_{aa}^{n+1^{\text{trial}}} &= \sigma_{aa}^{n} + E_{aa}\Delta\varepsilon_{aa} \\
\sigma_{bb}^{n+1^{\text{trial}}} &= \sigma_{bb}^{n} + E_{bb}\Delta\varepsilon_{bb} \\
\sigma_{cc}^{n+1^{\text{trial}}} &= \sigma_{cc}^{n} + E_{cc}\Delta\varepsilon_{cc} \\
\sigma_{ab}^{n+1^{\text{trial}}} &= \sigma_{ab}^{n} + 2G_{ab}\Delta\varepsilon_{ab} \\
\sigma_{bc}^{n+1^{\text{trial}}} &= \sigma_{bc}^{n} + 2G_{bc}\Delta\varepsilon_{bc} \\
\sigma_{ca}^{n+1^{\text{trial}}} &= \sigma_{ca}^{n} + 2G_{ca}\Delta\varepsilon_{ca}
\end{aligned}$$

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; for example, if

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

then

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

The stress components are found using the curves defined on Card 2. 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:

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

where the deviatoric strain increment is defined as

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

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

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

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

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

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

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

where

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

to obtain the final value for the Cauchy stress

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

After completing the stress the stresses are transformed back to the global configuration.

4. **Failure.** For *CONSTRAINED_TIED_NODES_WITH_FAILURE, the failure is based on the volume strain instead to the plastic strain.

*MAT_MOONEY-RIVLIN_RUBBER

This is Material Type 27. A two-parametric material model for rubber can be defined.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR	A	B	REF		
Type	A	F	F	F	F	F		

Least Squares Card. If the values on Card 2 are nonzero, then a least squares fit is computed from the uniaxial data provided by the curve LCID. In this case A and B are ignored. If the A and B fields on Card 1 are left blank, then the fields on Card 2 *must* be nonzero.

Card 2	1	2	3	4	5	6	7	8
Variable	SGL	SW	ST	LCID				
Type	F	F	F	I				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
PR	Poisson's ratio. A value between 0.49 and 0.5 is recommended; smaller values may not work. Setting to 0.5 for solid elements with implicit analysis activates a *U-P* formulation. See Remark 3 for details.
A	Constant; see literature and remarks below. This field is ignored if the fields on Card 2 are nonzero.
B	Constant; see literature and remarks below. This field is ignored if the fields on Card 2 are nonzero.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY.

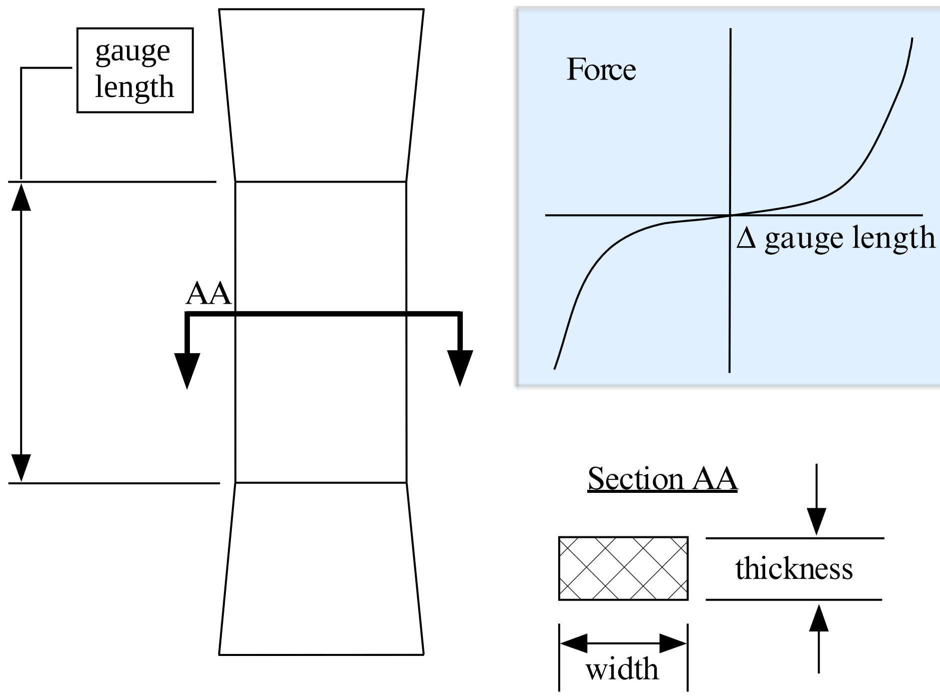


Figure M27-1. Uniaxial specimen for experimental data

VARIABLE	DESCRIPTION
	EQ.0.0: Off EQ.1.0: On
SGL	Specimen gauge length l_0; see Figure M27-1.
SW	Specimen width; see Figure M27-1.
ST	Specimen thickness; see Figure M27-1.
LCID	Curve ID, see *DEFINE_CURVE, giving the force versus actual change ΔL in the gauge length. See Remark 2. See also Figure M27-2 for an alternative definition. LS-DYNA computes a least squares fit from this data. A and B are ignored if this field is defined.

Remarks:

1. **Strain energy density function.** The strain energy density function is defined as:

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

where

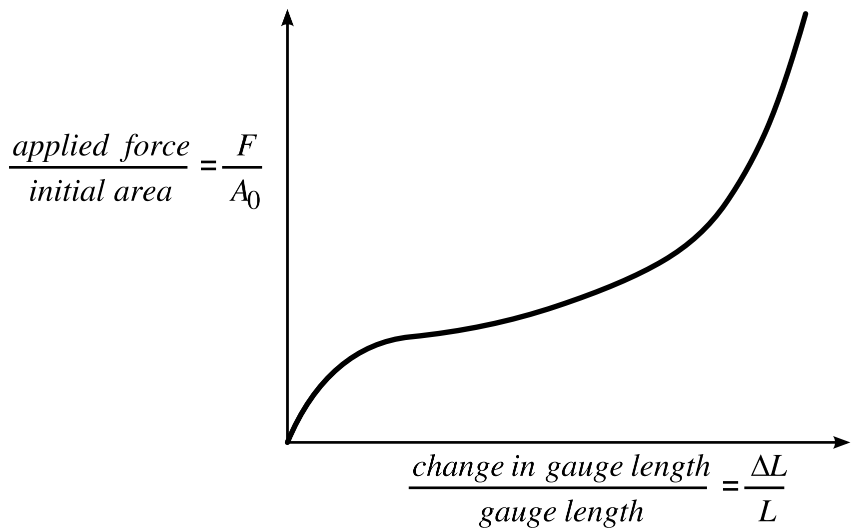


Figure M27-2 The stress as a function of strain curve can be used instead of the force as a function of the change in the gauge length by setting the gauge length, thickness, and width to unity (1.0) and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force. *MAT_077_O is a better alternative for fitting data resembling the curve above. *MAT_027 will provide a poor fit to a curve that exhibits a strong upturn in slope as strains become large.

$$C = 0.5A + B$$
$$D = \frac{A(5v - 2) + B(11v - 5)}{2(1 - 2v)}$$

Here, A and B are constants, v is the Poisson's ratio, $2(A + B)$ is the shear modulus of linear elasticity, and I, II, and III are the principal invariants of the right Cauchy-Green tensor, $\mathbf{C}$. Recall that $\mathbf{C} = \mathbf{F}^T\mathbf{F}$ where $\mathbf{F} = \nabla_{\mathbf{X}}\mathbf{x}$ is the deformation gradient, $\mathbf{x}$ is the current configuration, and $\mathbf{X}$ is the reference configuration.

2. **Experimental data for the material.** 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

$$\lambda_1 = \frac{L_0 + \Delta L}{L_0}$$

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

Alternatively, the stress as a function of 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 M27-1.

LS-DYNA performs a least square fit to the experimental data during the initialization phase. The d3hsp file provides a comparison between the fit and the actual. We recommend visually checking to make sure it is acceptable. d3hsp also contains the coefficients A and B. We also advise examining the material model with the material driver (see Appendix K).

3. **Incompressible material.** If the material is incompressible with a Poisson ratio of exactly 0.5, LS-DYNA uses a mixed finite element method of displacement-pressure (U-P) type to avoid volumetric locking. Note that this formulation is only available for solid elements with implicit analysis. With this formulation, we enforce the incompressibility constraint, $J = 1$, with $J = \det(F)$, strongly using a Lagrange multiplier technique. In the absence of inertial and external forces, this amounts to seeking a stationary point to the Lagrangian

$$L(\mathbf{u},\lambda) = \int W(\mathbf{C}) + \lambda(J-1)d\mathbf{x} \ ,$$

where $\mathbf{u} = \mathbf{x} - \mathbf{X}$ is the displacement, and λ is a pressure-like Lagrange multiplier for the constraint. The stiffness matrix resulting from the U-P formulation is a saddle-point type (i.e., indefinite), and may therefore require special consideration regarding the choice of linear solver and stiffness reformation limit.

4. **Output to effective plastic strain location in d3plot.** The history variable labeled as "effective plastic strain" in LS-PrePost is internal energy density in the case of *MAT_MOONEY-RIVLIN_RUBBER.

*MAT_RESULTANT_PLASTICITY

This is Material Type 28. It defines resultant formulation for beam and shell elements including elasto-plastic behavior. 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 LS-DYNA Theory 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 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN		
Type	A	F	F	F	F	F		
Default	none	none	none	none	none	0.0		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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. It is a force limited resultant formulation. 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 modeled. See also *MAT_139.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	DF	IAFLC	YTFLAG	ASOFT
Type	A	F	F	F	F	I	F	F
Default	none	none	none	none	0.0	0	0.0	0.0

Card 2	1	2	3	4	5	6	7	8
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	1	2	3	4	5	6	7	8
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	1	2	3	4	5	6	7	8
Variable	LPS1	SFS1	LPS2	SFS2	YMS1	YMS2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPS1	1.0	10^{20}	YMS1		

Card 5	1	2	3	4	5	6	7	8
Variable	LPT1	SFT1	LPT2	SFT2	YMT1	YMT2		
Type	F	F	F	F	F	F		
Default	0	1.0	LPT1	1.0	10^{20}	YMT1		

Card 6	1	2	3	4	5	6	7	8
Variable	LPR	SFR	YMR					
Type	F	F	F					
Default	0	1.0	10^{20}					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
DF	Damping factor; see Remark 2. *A proper control for the timestep must be maintained by the user.*

VARIABLE	DESCRIPTION
IAFLC	Axial force load curve option: EQ.0: axial load curves are force as a function of strain. EQ.1: axial load curves are force as a function of 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 as a function of (strain/change in length) curve. At least one must be defined. A maximum of 8 moments can be defined. The values should be in ascending order.
LC1, LC2, …, LC8	Load curve ID (see *DEFINE_CURVE) defining axial force (collapse load) as a function of 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 as a function of rotation about the *s*-axis at node 1. If zero, this load curve is ignored.
SFS1	Scale factor for plastic moment as a function of rotation curve about the *s*-axis at node 1. Default = 1.0.
LPS2	Load curve ID for plastic moment as a function of rotation about the *s*-axis at node 2. Default: LPS1.
SFS2	Scale factor for plastic moment as a function of rotation curve about the *s*-axis at node 2. Default: SFS1.
YMS1	Yield moment about the *s*-axis at node 1 for interaction calculations (default set to 10^{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 as a function of rotation about the *t*-axis at node 1. If zero, this load curve is ignored.

VARIABLE	DESCRIPTION
SFT1	Scale factor for plastic moment as a function of rotation curve about the *t*-axis at node 1. Default = 1.0.
LPT2	Load curve ID for plastic moment as a function of rotation about the *t*-axis at node 2. Default: LPT1.
SFT2	Scale factor for plastic moment as a function of rotation curve about the *t*-axis at node 2. Default: SFT1.
YMT1	Yield moment about the *t*-axis at node 1 for interaction calculations (default set to 10^{20} to prevent interactions)
YMT2	Yield moment about the *t*-axis at node 2 for interaction calculations (default set to YMT1)
LPR	Load curve ID for plastic torsional moment as a function of rotation. If zero, this load curve is ignored.
SFR	Scale factor for plastic torsional moment as a function of rotation (default = 1.0).
YMR	Torsional yield moment for interaction calculations (default set to 10^{20} to prevent interaction)

Remarks:

1. **Load Curves.** 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 as a function 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 (0.0, 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 as a function of 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 (0.0, 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.

2. **Damping.** Stiffness-proportional damping may be added using the damping factor λ. This is defined as follows:

$$\lambda = \frac{2 \times \xi}{\omega}$$

where ξ is the damping factor at the reference frequency ω (in radians per second). For example if 1% damping at 2Hz is required

$$\lambda = \frac{2 \times 0.01}{2\pi \times 2} = 0.001592 \ .$$

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 using 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).

3. **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:

$$\left(\frac{M_r}{M_{\mathrm{r_{yield}}}}\right)^2 + \left(\frac{M_s}{M_{\mathrm{s_{yield}}}}\right)^2 + \left(\frac{M_t}{M_{\mathrm{t_{yield}}}}\right)^2 \geq 1 \ ,$$

where

$$M_r,\ M_s,\ M_t = \text{current moment}$$

$$M_{r_{\mathrm{yield}}}, M_{s_{\mathrm{yield}}}, M_{t_{\mathrm{yield}}} = \text{yield moment}$$

Note that scale factors for hinge behavior defined in the input will also be applied to the yield moments: for example, $M_{s_{\mathrm{yield}}}$ 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

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

and similar conditions hold for $M_{s_{\mathrm{upper}}}$ and $M_{t_{\mathrm{upper}}}$.

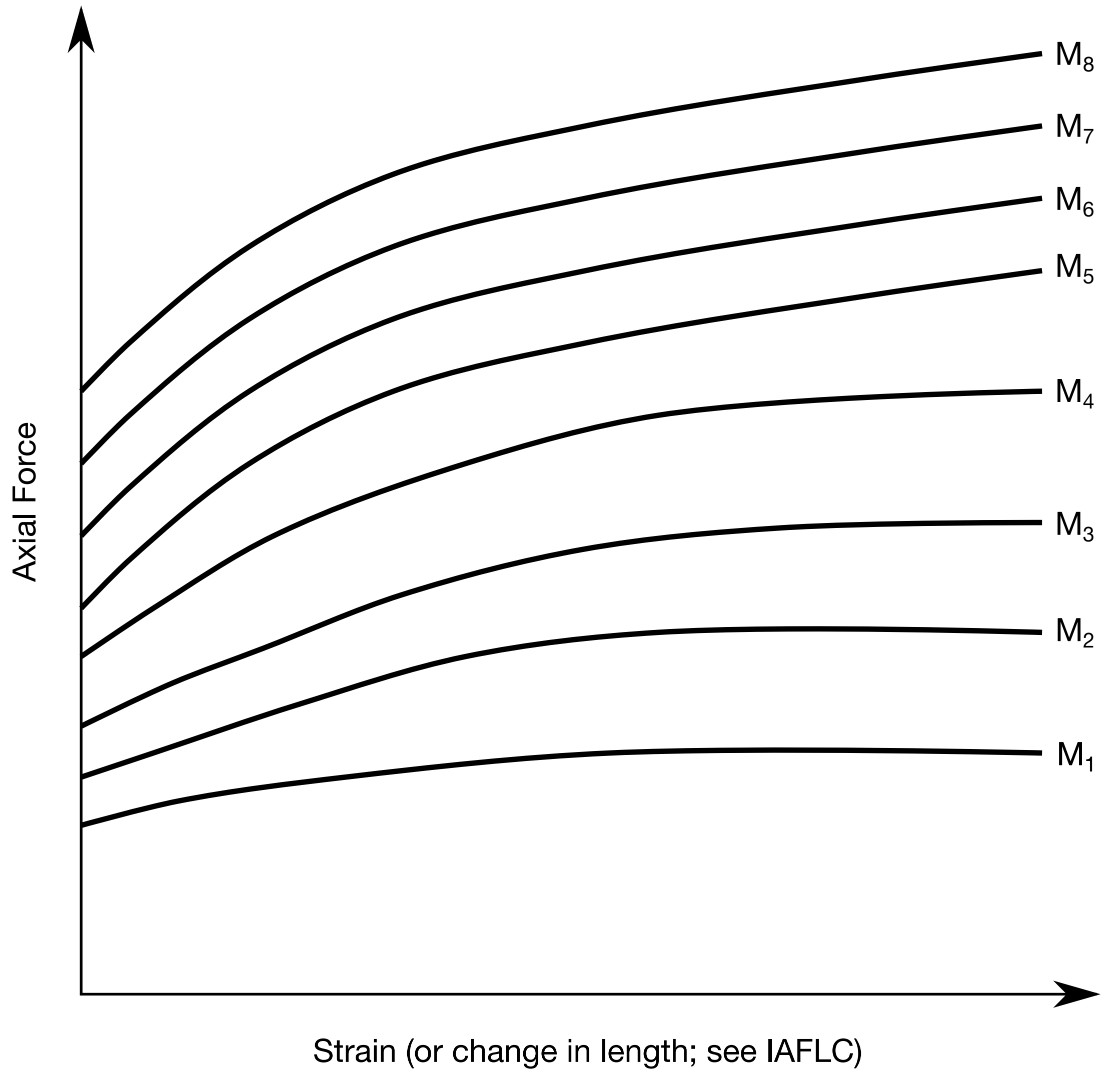


Figure M29-1. The force magnitude is limited by the applied end moment. For an intermediate value of the end moment LS-DYNA interpolates between the curves to determine the allowable force value.

Thereafter, the plastic moments will be given by

$$M_{r_p} = \min(M_{r_{\text{upper}}}, M_{r_{\text{curve}}})$$

where M_{r_p} is the current plastic moment and $M_{r_{\text{curve}}}$ is the moment from the load curve at the current rotation scaled by the scale factor. $M_{s_p}$and M_{t_p} satisfy similar conditions.

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 moment-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.

*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 M30-1). The material response is always characterized by a hysteresis loop. See the references by Auricchio, Taylor and Lubliner [1997] and Auricchio and Taylor [1997]. This model is available for shells, solids, and Hughes-Liu beam elements. The model supports von Mises isotropic plasticity with an arbitrary effective stress as a function of effective plastic strain curve.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	LCSS			
Type	A	F	F	F	I			
Default	none	none	none	none	0			

Card 2	1	2	3	4	5	6	7	8
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	

Optional Load Curve Card (starting with R7.1). Load curves for mechanically induced phase transitions.

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

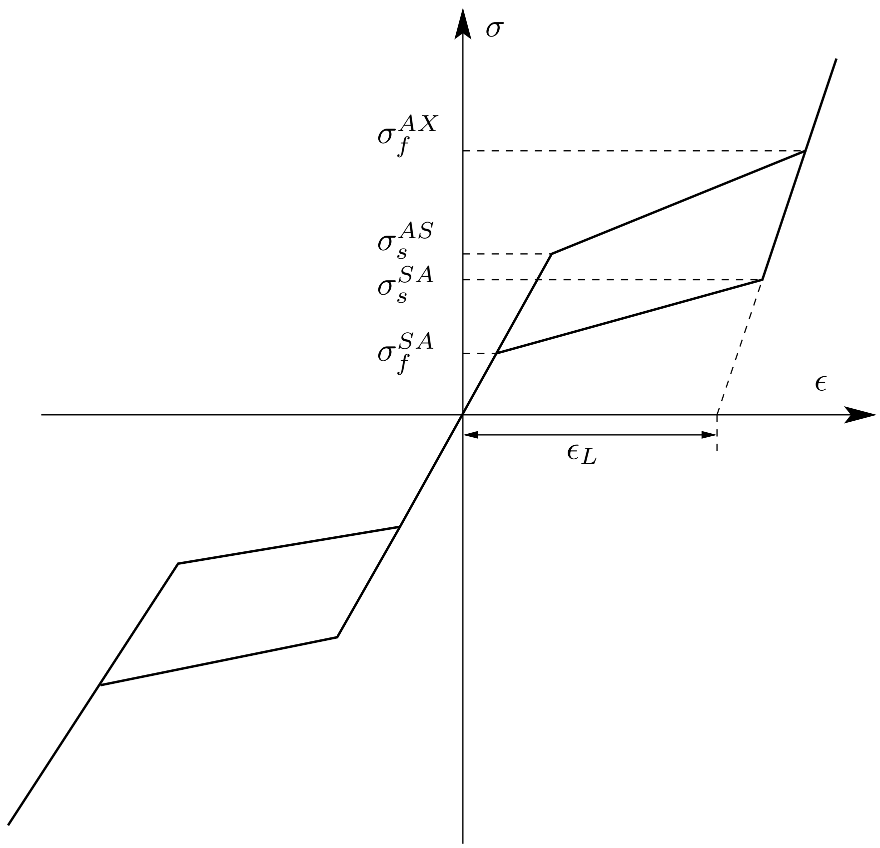


Figure M30-1. Superelastic Behavior for a Shape Memory Material

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
LCSS	The absolute value of LCSS is a load curve ID for effective stress as a function of effective plastic strain (this load curve is optional). The first data point, at zero plastic strain, indicates the initial yield stress. For a negative value of LCSS, negative values of SIG_ASS, SIG_ASF, SIG_SAS, SIG_SAF will indicate dependence on plastic strain; see below.
SIG_ASS	Starting value for the forward phase transformation (conversion of austenite into martensite) in the case of a uniaxial tensile state of

VARIABLE	DESCRIPTION
	stress. LT.0.0: -SIG_ASS is a load curve ID defining the starting value as a function of temperature. If LCSS is also negative, then -SIG_ASS is either a load curve specifying the starting value as a function of effective plastic strain or a table of such load curves for different temperatures.
SIG_ASF	Final value for the forward phase transformation (conversion of austenite into martensite) in the case of a uniaxial tensile state of stress. LT.0.0: -SIG_ASF is a load curve ID defining the final value as a function of temperature is specified. If LCSS is also negative, -SIG_ASF is either a load curve specifying the final value as a function of effective plastic strain or a table of such load curves for different temperatures.
SIG_SAS	Starting value for the reverse phase transformation (conversion of martensite into austenite) in the case of a uniaxial tensile state of stress. LT.0.0: -SIG_SAS is a load curve ID defining the starting value as a function of temperature. If LCSS is also negative, -SIG_SAS is either a load curve specifying the starting value as a function of effective plastic strain or a table of such load curves for different temperatures.
SIG_SAF	Final value for the reverse phase transformation (conversion of martensite into austenite) in the case of a uniaxial tensile state of stress. LT.0.0: -SIG_SAF is a load curve ID specifying the reverse value as a function of temperature. If LCSS is also negative, -SIG_SAF is either a load curve specifying the final value as a function of effective plastic strain or a table of such load curves for different temperatures.
EPSL	Recoverable strain or maximum residual strain. It is a measure of the maximum deformation obtainable for 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 remarks.

VARIABLE	DESCRIPTION
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.
LCID_AS	Load curve ID or table ID for the *forward* phase change (conversion of austenite into martensite). 1. *When LCID_AS is a load curve ID* the curve is taken to be effective stress as a function of martensite fraction (ranging from 0 to 1). 2. *When LCID_AS is a table ID* the table defines for each phase transition rate (derivative of martensite fraction) a load curve ID specifying the stress as a function of martensite fraction for that phase transition rate. The stress as a function of martensite fraction curve for the lowest value of the phase transition rate is used if the phase transition rate falls below the minimum value. Likewise, the stress as a function of martensite fraction curve for the highest value of phase transition rate is used if the phase transition rate exceeds the maximum value. 3. The values of SIG_ASS and SIG_ASF are overwritten when this option is used.
LCID_SA	Load curve ID or table ID for *reverse* phase change (conversion of martensite into austenite). 1. *When LCID_SA is a load curve ID*, the curve is taken to be effective stress as a function of martensite fraction (ranging from 0 to 1). 2. *When LCID_SA is a table ID*, the table defines for each phase transition rate (derivative of martensite fraction) a load curve ID specifying the stress as a function of martensite fraction for that phase transition rate. The stress as a function of martensite fraction curve for the lowest value of the phase transition rate is used if the phase transition rate falls below the minimum value. Likewise, the stress as a function of martensite fraction curve for the highest value of phase transition rate is used if phase transition rate exceeds the maximum value.

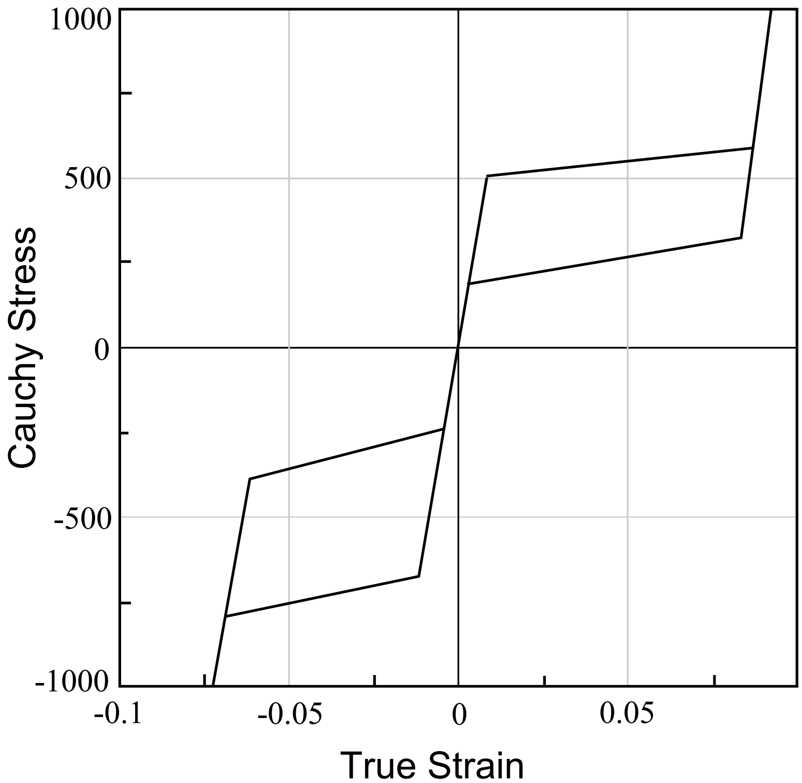


Figure M30-2. Complete loading-unloading test in tension and compression**.**

VARIABLE	DESCRIPTION
	3. The values of SIG_ASS and SIG_ASF are overwritten when this option is used.

Remarks:

The material parameter alpha, $-1 < \alpha < 1$, 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 from the expression

$$\alpha = \sqrt{\frac{2}{3}}\left(\frac{-\sigma_s^{AS,-} - \sigma_s^{AS,+}}{-\sigma_s^{AS,-} + \sigma_s^{AS,+}}\right),$$

where $\sigma_s^{AS,+} > 0$ and $\sigma_s^{AS,-} < 0$ are the values in tension and compression, respectively. From the input parameters α and $\sigma_s^{AS,+}$, the stress in compression is then

$$\sigma_s^{AS,-} = \frac{\alpha + 1}{\alpha - 1}\sigma_s^{AS,+}.$$

In Figure M30-2, we show the uniaxial Cauchy stress versus the logarithmic strain plot obtained from a simple test problem. The investigated problem is the complete loading-unloading test in tension and compression. We set the material properties to:

Property	Value	Property	Value
E	60000 MPa	SIG_SAF	200 MPa
PR	0.3	EPSL	0.07
SIG_ASS	520 MPa	ALPHA	0.12
SIG_ASF	600 MPa	YMRT	50000 MPa
SIG_SAS	300 MPa		

*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 Kenchington [1988]. See Remarks below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR	C100	C200	C300	C400	
Type	A	F	F	F	F	F	F	

Card 2	1	2	3	4	5	6	7	8
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 or label must be specified (see *PART).
RO	Mass density
PR	Poisson's ratio. Values between .49 and .50 are suggested.
C100	C_{100}, constant in strain energy functional. If a least squares fit is being used, set this constant to 1.0 if the term it belongs to in the strain energy functional is to be included. See Remarks.
C200	C_{200}, constant in strain energy functional. If a least squares fit is being used, set this constant to 1.0 if the term it belongs to in the strain energy functional is to be included. See Remarks.

VARIABLE	DESCRIPTION
C300	C_{300}, constant in strain energy functional. If a least squares fit is being used, set this constant to 1.0 if the term it belongs to in the strain energy functional is to be included. See Remarks.
C400	C_{400}, constant in strain energy functional. If a least squares fit is being used, set this constant to 1.0 if the term it belongs to in the strain energy functional is to be included. See Remarks.
C110	C_{110}, constant in strain energy functional. If a least squares fit is being used, set this constant to 1.0 if the term it belongs to in the strain energy functional is to be included. See Remarks.
C210	C_{210}, constant in strain energy functional. If a least squares fit is being used, set this constant to 1.0 if the term it belongs to in the strain energy functional is to be included. See Remarks.
C010	C_{010}, constant in strain energy functional. If a least squares fit is being used, set this constant to 1.0 if the term it belongs to in the strain energy functional is to be included. See Remarks.
C020	C_{020}, constant in strain energy functional. If a least squares fit is being used, set this constant to 1.0 if the term it belongs to in the strain energy functional is to be included. See Remarks.
EXIT	Exit option (only in explicit analysis): EQ.1.0: Stop if strain limits are exceeded (recommended) NE.1.0: 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 geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY (see there for more details). EQ.0.0: Off, EQ.1.0: On.
SGL	Specimen gauge length; see Figure M27-1.
SW	Specimen width; see Figure M27-1.

VARIABLE	DESCRIPTION
ST	Specimen thickness; see Figure M27-1.
LCID	Load curve ID (see DEFINE_CURVE) giving the force as a function of actual change in gauge length. See also Figure M27-2 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

$$U = C_{100}I_1 + C_{200}I_1^2 + C_{300}I_1^3 + C_{400}I_1^4 + C_{110}I_1I_2 + C_{210}I_1^2I_2 + C_{010}I_2 + C_{020}I_2^2 + f(J)$$

where the invariants I_1, I_2 and J can be expressed in terms of the deformation gradient matrix, $\mathbf{F}$, and the right stretch tensor, $\mathbf{C} = \mathbf{F}^T\mathbf{F}$:

$$\begin{aligned} J &= \det\mathbf{F} \\ I_1 &= \mathrm{tr}(\mathbf{C}) - 3 \\ I_2 &= \frac{1}{2}\left(\mathrm{tr}(\mathbf{C})^2 - \mathrm{tr}(\mathbf{C}^2)\right) - 3. \end{aligned}$$

The dependence on the third invariant is given as

$$f(J) = \frac{2C_{100}(\nu - 4) + 4C_{010}(11\nu - 5)}{1 - 2\nu}\left(\frac{J^2}{2} - \ln J\right) + \frac{1}{2}(C_{100} + 2C_{010})\frac{1}{J^4}$$

where ν is the Poisson's ratio. The first term on the right-hand side of this expression should be interpreted as the constitutive law for the pressure while the second is necessary for providing zero stress at the reference configuration.

The derivative of U with respect to $\mathbf{C}$ gives the 2nd Piola-Kirchhoff stress $\mathbf{S}$ as

$$\mathbf{S} = 2\frac{\partial U}{\partial \mathbf{C}}$$

and the Cauchy stress $\boldsymbol{\sigma}$ is then given by

$$\boldsymbol{\sigma} = \frac{1}{J}\mathbf{FSF}^T.$$

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 as a function of strain curve can also be input by setting the gauge length, thickness, and width to unity and defining the engineering strain in place of the change in gauge length and the nominal (engineering) stress in place of the force; see Figure M27-2 The least square fit to the experimental data is performed during the initialization phase, and 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} through 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 1	1	2	3	4	5	6	7	8
Variable	MID	RO	EG	PRG	SYG	ETG	EFG	EP
Type	A	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	PRP	SYP	ETP					
Type	F	F	F					

Integration Point Cards. Up to four of this card (specifying up to 32 values) may be input. This input is terminated by the next keyword (“*”) card.

Card 3	1	2	3	4	5	6	7	8
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 or label must be specified (see *PART).
RO	Mass density
EG	Young’s modulus for glass
PRG	Poisson’s ratio for glass
SYG	Yield stress for glass
ETG	Plastic hardening modulus for glass

VARIABLE	DESCRIPTION
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: EQ.0.0: glass (default) EQ.1.0: polymer A user-defined integration rule must be specified; see *INTEGRATION_SHELL. See Remarks below.

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*, must be given for the same number of integration points (NIPTS) as specified in the rule. A maximum of 32 layers is allowed.

If the recommended user defined rule is not defined, the default integration rules are used. The location of the integration points in the default rules are defined in the *SECTION_SHELL keyword description.

*MAT_BARLAT_ANISOTROPIC_PLASTICITY

This is Material Type 33. This model was developed by Barlat, Lege, and Brem [1991] for modeling 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 remarks below). For sheet forming problems, we recommend material 36 which is based on a 3-parameter model.

This material is available for shell, thick shell, and solid elements.

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

Card 2	1	2	3	4	5	6	7	8
Variable	A	B	C	F	G	H	LCID	
Type	F	F	F	F	F	F	F	

Card 3	1	2	3	4	5	6	7	8
Variable	AOPT	BETA	MACF					
Type	F	F	I					

Card 4	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus, E
PR	Poisson's ratio, ν
K	k, strength coefficient (see remarks below)
E0	ε_0, strain corresponding to the initial yield (see remarks below)
N	n, hardening exponent for yield strength
M	m, flow potential exponent in Barlat's Model
A	a, anisotropy coefficient in Barlat's Model
B	b, anisotropy coefficient in Barlat's Model
C	c, anisotropy coefficient in Barlat's Model
F	f, anisotropy coefficient in Barlat's Model
G	g, anisotropy coefficient in Barlat's Model
H	h, anisotropy coefficient in Barlat's Model
LCID	Option load curve ID defining effective stress as a function of effective plastic strain. If nonzero, this curve will be used to define the yield stress. The load curve is implemented for solid elements only.
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC, particularly the Material Directions section, for details): EQ.0.0: Locally orthotropic with material axes determined by

VARIABLE	DESCRIPTION
	element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. For shells only, the material axes are then rotated about the normal vector to the surface of the shell by the angle BETA.
	EQ.1.0: Locally orthotropic with material axes determined by a point, *P*, 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 a vector **v** and the normal vector to the plane of the element. 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then **a** and **b** are rotated about **c** by an angle BETA. BETA may be set in the keyword input for the element or in the input for this keyword. Note that for solids, the material axes may be switched depending on the choice of MACF. The switch may occur before or after applying BETA depending on the value of MACF.
	EQ.4.0: Locally orthotropic in a 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.
	LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_*OPTION*).
BETA	Material angle in degrees for AOPT = 1 (shells only) and AOPT = 3, may be overridden on the element card, see *ELEMENT_SHELL_-BETA, *ELEMENT_TSHELL_BETA, or *ELEMENT_SOLID_ORTHO.
MACF	Material axes change flag for solid elements:

VARIABLE	DESCRIPTION
	EQ.-4: Switch material axes b and c before BETA rotation
	EQ.-3: Switch material axes a and c before BETA rotation
	EQ.-2: Switch material axes a and b before BETA rotation
	EQ.1: No change, default
	EQ.2: Switch material axes a and b after BETA rotation
	EQ.3: Switch material axes a and c after BETA rotation
	EQ.4: Switch material axes b and c after BETA rotation
	Figure M2-2 indicates when LS-DYNA applies MACF during the process to obtain the final material axes. If BETA on *ELEMENT_-SOLID_*{OPTION}* is defined, then that BETA is used for the rotation for all AOPT options. Otherwise, if AOPT = 3, the BETA input on Card 3 rotates the axes. For all other values of AOPT, the material axes will be switched as specified by MACF, but no BETA rotation will be performed.
XP, YP, ZP	Coordinates of point p for AOPT = 1 and 4
A1, A2, A3	Components of vector **a** for AOPT = 2
V1, V2, V3	Components of vector **v** for AOPT = 3 and 4
D1, D2, D3	Components of vector **d** for AOPT = 2

Remarks:

The yield function Φ is defined as:

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

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

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

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

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

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

*MAT_BARLAT_YLD96

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

Card Summary:

Card 1. This card is required.

MID	RO	E	PR	K			

Card 2. This card is required.

E0	N	ESR0	M	HARD	A		

Card 3. This card is required.

C1	C2	C3	C4	AX	AY	AZ0	AZ1

Card 4. This card is required.

AOPT	BETA						

Card 5. This card is required.

			A1	A2	A3		

Card 6. This card is required.

V1	V2	V3	D1	D2	D3		

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	K			
Type	A	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus, E
PR	Poisson's ratio,ν
K	k, strength coefficient or a in Voce (see remarks below)

Card 2	1	2	3	4	5	6	7	8
Variable	E0	N	ESR0	M	HARD	A		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
E0	ε_0, strain corresponding to the initial yield or b in Voce (see remarks below)
N	n, hardening exponent for yield strength or c in Voce
ESR0	ε_{SR0}, in power law 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: Power law EQ.2.0: Voce
A	Flow potential exponent

Card 3	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	AX	AY	AZ0	AZ1
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
C1	c_1, see remarks below
C2	c_2, see remarks below
C3	c_3, see remarks below
C4	c_4, see remarks below
AX	a_x, see remarks below
AY	a_y, see remarks below
AZ0	a_{z_0}, see remarks below
AZ1	a_{z_1}, see remarks below

Card 4	1	2	3	4	5	6	7	8
Variable	AOPT	BETA						
Type	F	F						

VARIABLE	DESCRIPTION
AOPT	Material axes option: EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES, and then rotated about the shell element normal by an angle BETA 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

VARIABLE		DESCRIPTION
		offsetting the material axes by an angle, BETA, from a line determined by taking the cross product of the vector **v** with the normal to the plane of the element
	LT.0.0:	The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR).
BETA		Material angle in degrees for AOPT = 0 and 3. BETA may be overridden on the element card; see *ELEMENT_SHELL_BETA.

Card 5	1	2	3	4	5	6	7	8
Variable				A1	A2	A3		
Type				F	F	F		

VARIABLE	DESCRIPTION
A1, A2, A3	Components of vector **a** for AOPT = 2

Card 6	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
V1, V2, V3	Components of vector **v** for AOPT = 3
D1, D2, D3	Components of vector **d** for AOPT = 2

Remarks:

The yield stress σ_y is defined three ways. The first, the Swift equation, is given in terms of the input constants as:

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

The second, the Voce equation, is defined as:

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

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

The yield function Φ is defined as:

$$\Phi = \alpha_1|s_1 - s_2|^a + \alpha_2|s_2 - s_3|^a + \alpha_3|s_3 - s_1|^a = 2\sigma_y^a \ ,$$

Here s_i is a principle component of the deviatoric stress tensor. In vector notation:

$$\mathbf{s} = \mathbf{L}\sigma \ ,$$

where **L** is given as

$$\mathbf{L} = \begin{bmatrix} \frac{c_2 + c_3}{3} & \frac{-c_3}{3} & \frac{-c_2}{3} & 0 \\ \frac{-c_3}{3} & \frac{c_3 + c_1}{3} & \frac{-c_1}{3} & 0 \\ \frac{-c_2}{3} & \frac{-c_1}{3} & \frac{c_1 + c_2}{3} & 0 \\ 0 & 0 & 0 & c_4 \end{bmatrix}$$

A coordinate transformation relates the material frame to the principle directions of **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 **s** and the principal anisotropy axes.

*MAT_FABRIC

This is Material Type 34. This material is especially developed for airbag materials. The fabric model is a variation on the layered orthotropic composite model of material 22 and is valid for 3 and 4 node membrane elements only.

In addition to being a constitutive model, this model also invokes a special membrane element formulation which is more suited to the deformation experienced by fabrics under large deformation. For thin fabrics, buckling can result in an inability to support compressive stresses; thus a flag is included for this option. A linearly elastic liner is also included which can be used to reduce the tendency for these elements to be crushed when the no-compression option is invoked. An isotropic elastic option is also available.

Card Summary:

Card 1. This card is required.

MID	RO	EA	EB		PRBA	PRAB	

Card 2. This card is required.

GAB			CSE	EL	PRL	LRATIO	DAMP

Card 3a. Include this card if 0 < X0 < 1 (see Card 5).

AOPT	X2	X3	ELA	LNRC	FORM	FVOPT	TSRFAC

Card 3b. Include this card if X0 = 0 or X0 = -1 (see Card 5) and FVOPT < 7.

AOPT	FLC	FAC	ELA	LNRC	FORM	FVOPT	TSRFAC

Card 3c. Include this card if X0 = 0 or X0 = -1 (see Card 5) and FVOPT ≥ 7.

AOPT	FLC	FAC	ELA	LNRC	FORM	FVOPT	TSRFAC

Card 3d. Include this card if X0 = 1 (see Card 5) and FVOPT < 7.

AOPT	FLC	FAC	ELA	LNRC	FORM	FVOPT	TSRFAC

Card 3e. Include this card if X0 = 1 (see Card 5) and FVOPT ≥ 7.

AOPT	FLC	FAC	ELA	LNRC	FORM	FVOPT	TSRFAC

Card 4. Include this card if FVOPT < 0.

L	R	C1	C2	C3			

Card 5. This card is required.

	RGBRTH	A0REF	A1	A2	A3	X0	X1

Card 6. This card is required.

V1	V2	V3				BETA	ISREFG

Card 7. Include this card if FORM = 4, 14, or -14.

LCA	LCB	LCAB	LCUA	LCUB	LCUAB	RL	

Card 8. Include this card if FORM = -14.

LCAA	LCBB	H	DT		ECOAT	SCOAT	TCOAT

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB		PRBA	PRAB	
Type	A	F	F	F		F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
EA	Young's modulus - longitudinal direction. For an isotropic elastic fabric material, *only* EA and PRBA are defined; they 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.
PRBA	ν_{ba}, Minor Poisson's ratio *ba* direction
PRAB	ν_{ab}, Major Poisson's ratio *ab* direction (see Remark 15)

Card 2	1	2	3	4	5	6	7	8
Variable	GAB			CSE	EL	PRL	LRATIO	DAMP
Type	F			F	F	F	F	F
Remarks				1	4	4	4	

VARIABLE	DESCRIPTION
GAB	G_{ab}, shear modulus in the *ab* direction. Set to zero for an isotropic elastic material.
CSE	Compressive stress elimination option (see Remark 1): EQ.0.0: Do not eliminate compressive stresses (*default*). EQ.1.0: Eliminate compressive stresses. This option does not apply to the liner.
EL	Young's modulus for elastic liner (required if LRATIO > 0)
PRL	Poisson's ratio for elastic liner (required if LRATIO > 0)
LRATIO	A non-zero value activates the elastic liner and defines the ratio of liner thickness to total fabric thickness (optional).
DAMP	Rayleigh damping coefficient. A 0.05 coefficient is recommended corresponding to 5% of critical damping. Sometimes larger values are necessary.

This card is included if and only if 0 < X0 < 1 (see Card 5).

Card 3a	1	2	3	4	5	6	7	8
Variable	AOPT	X2	X3	ELA	LNRC	FORM	FVOPT	TSRFAC
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC for a more complete description). Also, please refer to Remark 5 for

VARIABLE	DESCRIPTION
	additional information specific to fiber directions for fabrics: EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES, and then rotated about the element normal by an angle BETA 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 LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR).
X2	Coefficient of the porosity from the equation in Anagonye and Wang [1999]
X3	Coefficient of the porosity equation of Anagonye and Wang [1999]
ELA	Effective leakage area for blocked fabric, ELA (see Remark 3): LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of time. The default value of zero assumes that no leakage occurs. A value of .10 would assume that 10% of the blocked fabric is leaking gas.
LNRC	Flag to turn off compression in liner until the reference geometry is reached, that is, the fabric element becomes tensile (see Remark 4): EQ.0.0: Off EQ.1.0: On EQ.2.0: Liner's resistance force follows the strain restoration factor, TSRFAC.
FORM	Flag to modify membrane formulation for fabric material: EQ.0.0: Default. Least costly and very reliable. EQ.1.0: Invariant local membrane coordinate system

VARIABLE	DESCRIPTION
	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.
	EQ.12.0: Enhanced version of formulation 2. See Remark 11.
	EQ.13.0: Enhanced version of formulation 3. See Remark 11.
	EQ.14.0: Enhanced version of formulation 4. See Remark 11.
	EQ.-14.0: Same as formulation 14 but invokes reading of Card 8. See Remark 14.
	EQ.24.0: Enhanced version of formulation 14. See Remark 11.
FVOPT	Fabric venting option (see Remark 9).
	EQ.1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered.
	EQ.2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered.
	EQ.3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered.
	EQ.4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered.
	EQ.5: Leakage formulas based on flow through a porous media are used. Blockage is not considered.
	EQ.6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.
	EQ.7: Leakage is based on gas volume outflow versus pressure load curve [Lian, 2000]. Blockage is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC in the *MAT_FABRIC card.
	EQ.8: Leakage is based on gas volume outflow versus pressure load curve [Lian 2000]. Blockage of venting or porous area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as

VARIABLE	DESCRIPTION
	FAC in the *MAT_FABRIC card. LT.0: |FVOPT| defines the same fabric venting options as above, but a new formula for the leakage area is used to replace the element area. See Remark 16. Note: See Remark 17 for FVOPT option for CPM (*AIRBAG_PARTICLE) bags.
TSRFAC	Strain restoration factor (see Remark 10): LT.0: |TSRFAC| is the ID of a curve defining TSRFAC as a function of time. GT.0 and LT.1: TSRFAC applied from time 0. GT.1: TSRFAC is the ID of a curve that defines TSRFAC as a function of time using an alternate method (not available for FORM = 0 or 1).

This card is included if X0 = 0 or X = -1 and FVOPT < 7.

Card 3b	1	2	3	4	5	6	7	8
Variable	AOPT	FLC	FAC	ELA	LNRC	FORM	FVOPT	TSRFAC
Type	F	F	F	F	F	F	F	F
Remarks		2	2	3	4	11	9	10

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC for a more complete description). Also, please refer to Remark 5 for additional information specific to fiber directions for fabrics: EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES, and then rotated about the element normal by an angle BETA 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,

VARIABLE	DESCRIPTION
	BETA, from a line in the plane of the element defined by the cross product of the vector **v** with the element normal LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR).
FLC	Optional porous leakage flow coefficient. (See theory manual.) GE.0: Porous leakage flow coefficient. LT.0: |FLC| is interpreted as a load curve ID defining FLC as a function of time.
FAC	Optional characteristic fabric parameter. (See theory manual.) GE.0: Characteristic fabric parameter LT.0: |FAC| is interpreted as a load curve ID defining FAC as a function of absolute pressure.
ELA	Effective leakage area for blocked fabric, ELA (see Remark 3): LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of time. The default value of zero assumes that no leakage occurs. A value of .10 would assume that 10% of the blocked fabric is leaking gas.
LNRC	Flag to turn off compression in liner until the reference geometry is reached, that is, the fabric element becomes tensile: EQ.0.0: Off EQ.1.0: On
FORM	Flag to modify membrane formulation for fabric material: EQ.0.0: Default. Least costly and very reliable. EQ.1.0: Invariant local membrane coordinate system EQ.2.0: Green-Lagrange strain formulation EQ.3.0: Large strain with nonorthogonal material angles. See Remark 5. EQ.4.0: Large strain with nonorthogonal material angles and nonlinear stress strain behavior. Define optional load curve IDs on optional card.

VARIABLE	DESCRIPTION
	EQ.12.0: Enhanced version of formulation 2. See Remark 11.
	EQ.13.0: Enhanced version of formulation 3. See Remark 11.
	EQ.14.0: Enhanced version of formulation 4. See Remark 11.
	EQ.-14.0: Same as formulation 14 but invokes reading of Card 8. See Remark 14.
	EQ.24.0: Enhanced version of formulation 14. See Remark 11.
FVOPT	Fabric venting option. EQ.1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ.3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ.5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ.6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered. LT.0: |FVOPT| defines the same fabric venting options as above, but a new formula for the leakage area is used to replace the element area. See Remark 16. Note: See Remark 17 for FVOPT option for CPM (*AIRBAG_PARTICLE) bags.
TSRFAC	Strain restoration factor: LT.0: |TSRFAC| is the ID of a curve defining TSRFAC as a function of time. GT.0 and LT.1: TSRFAC applied from time 0. GE.1: TSRFAC is the ID of a curve that defines TSRFAC as a function of time using an alternate method (not available for FORM = 0 or 1).

This card is included if and only if X0 = 0 or X0 = -1 and FVOPT ≥ 7.

Card 3c	1	2	3	4	5	6	7	8
Variable	AOPT	FLC	FAC	ELA	LNRC	FORM	FVOPT	TSRFAC
Type	F	F	F	F	F	F	F	F
Remarks		2	2	3	4	11	9	10

VARIABLE	DESCRIPTION

AOPT

Material axes option (see MAT_*OPTION*TROPIC_ELASTIC for a more complete description). Also, please refer to Remark 5 for additional information specific to fiber directions for fabrics:

EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES, and then rotated about the element normal by an angle BETA

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

LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR).

FLC

Optional porous leakage flow coefficient. (See theory manual.)

GE.0: Porous leakage flow coefficient

LT.0: |FLC| is interpreted as a load curve ID defining FLC as a function of time.

FAC

Optional characteristic fabric parameter. (See theory manual.)

GE.0: Characteristic fabric parameter

VARIABLE	DESCRIPTION
	LT.0: |FAC| is interpreted as a load curve ID giving *leakage volume flux rate* as a function of absolute pressure. The volume flux (per area) rate (per time) has the dimensions of $$d(\mathrm{vol}_{\mathrm{flux}})/\mathrm{dt} \approx [\mathrm{length}]^3/([\mathrm{length}]^2[\mathrm{time}]) \approx [\mathrm{length}]/[\mathrm{time}],$$ equivalent to relative porous gas speed.
ELA	Effective leakage area for blocked fabric, ELA (see Remark 3): LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of time. The default value of zero assumes that no leakage occurs. A value of .10 would assume that 10% of the blocked fabric is leaking gas.
LNRC	Flag to turn off compression in liner until the reference geometry is reached, that is, the fabric element becomes tensile: EQ.0.0: Off EQ.1.0: On
FORM	Flag to modify membrane formulation for fabric material: EQ.0.0: Default. Least costly and very reliable. EQ.1.0: Invariant local membrane coordinate system EQ.2.0: Green-Lagrange strain formulation EQ.3.0: Large strain with nonorthogonal material angles. See Remark 5. EQ.4.0: Large strain with nonorthogonal material angles and nonlinear stress strain behavior. Define optional load curve IDs on optional card. EQ.12.0: Enhanced version of formulation 2. See Remark 11. EQ.13.0: Enhanced version of formulation 3. See Remark 11. EQ.14.0: Enhanced version of formulation 4. See Remark 11. EQ.-14.0: Same as formulation 14 but invokes reading of Card 8. See Remark 14. EQ.24.0: Enhanced version of formulation 14. See Remark 11.
FVOPT	Fabric venting option: EQ.7: Leakage is based on gas volume outflow versus pressure

VARIABLE	DESCRIPTION
	load curve [Lian, 2000]. Blockage is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC in the *MAT_FABRIC card. EQ.8: Leakage is based on gas volume outflow versus pressure load curve [Lian 2000]. Blockage of venting or porous area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC in the *MAT_FABRIC card. Note: See Remark 17 for FVOPT option for CPM (*AIRBAG_PARTICLE) bags.
TSRFAC	Strain restoration factor: LT.0: |TSRFAC| is the ID of a curve defining TSRFAC as a function of time. GT.0 and LT.1: TSRFAC applied from time 0. GE.1: TSRFAC is the ID of a curve that defines TSRFAC as a function of time using an alternate method (not available for FORM = 0 or 1).
ELA	Effective leakage area for blocked fabric, ELA (see Remark 3): LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of 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.

This card is included if X0 = 1 and FVOPT < 7.

Card 3d	1	2	3	4	5	6	7	8
Variable	AOPT	FLC	FAC	ELA	LNRC	FORM	FVOPT	TSRFAC
Type	F	F	F	F	F	F	F	F
Remarks		2	2	3	4	11	9	10

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC for a more complete description). Also, please refer to Remark 5 for

VARIABLE	DESCRIPTION
	additional information specific to fiber directions for fabrics: EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES, and then rotated about the element normal by an angle BETA 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 LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR).
FLC	Optional porous leakage flow coefficient. (See theory manual.) GE.0: Porous leakage flow coefficient. LT.0: |FLC| is interpreted as a load curve ID defining FLC as a function of the stretching ratio defined as $r_s = A/A_0$. See notes below.
FAC	Optional characteristic fabric parameter. (See theory manual.) GE.0: Characteristic fabric parameter LT.0: |FAC| is interpreted as a load curve defining FAC as a function of the pressure ratio $r_p = P_{\mathrm{air}}/P_{\mathrm{bag}}$. See Remark 2 below.
ELA	Effective leakage area for blocked fabric, ELA (see Remark 3): LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of time. The default value of zero assumes that no leakage occurs. A value of .10 would assume that 10% of the blocked fabric is leaking gas.
LNRC	Flag to turn off compression in liner until the reference geometry is reached, that is, the fabric element becomes tensile: EQ.0.0: Off

VARIABLE	DESCRIPTION
	EQ.1.0: On
FORM	Flag to modify membrane formulation for fabric material: EQ.0.0: Default. Least costly and very reliable. EQ.1.0: Invariant local membrane coordinate system EQ.2.0: Green-Lagrange strain formulation EQ.3.0: Large strain with nonorthogonal material angles. See Remark 5. EQ.4.0: Large strain with nonorthogonal material angles and nonlinear stress strain behavior. Define optional load curve IDs on optional card. EQ.12.0: Enhanced version of formulation 2. See Remark 11. EQ.13.0: Enhanced version of formulation 3. See Remark 11. EQ.14.0: Enhanced version of formulation 4. See Remark 11. EQ.-14.0: Same as formulation 14 but invokes reading of Card 8. See Remark 14. EQ.24.0: Enhanced version of formulation 14. See Remark 11.
FVOPT	Fabric venting option. EQ.1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ.3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ.5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ.6: Leakage formulas based on flow through a porous media are LT.0: |FVOPT| defines the same fabric venting options as above, but a new formula for the leakage area is used to replace the element area. See Remark 16.

VARIABLE	DESCRIPTION
	Note: See Remark 17 for FVOPT option for CPM (*AIRBAG_PARTICLE) bags.
TSRFAC	Strain restoration factor: LT.0: |TSRFAC| is the ID of a curve defining TSRFAC as a function of time. GT.0 and LT.1: TSRFAC applied from time 0. GE.1: TSRFAC is the ID of a curve that defines TSRFAC as a function of time using an alternate method (not available for FORM = 0 or 1).

This card is included if X0 = 1 and FVOPT ≥ 7.

Card 3e	1	2	3	4	5	6	7	8
Variable	AOPT	FLC	FAC	ELA	LNRC	FORM	FVOPT	TSRFAC
Type	F	F	F	F	F	F	F	F
Remarks		2	2	3	4	11	9	10

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC for a more complete description). Also, please refer to Remark 5 for additional information specific to fiber directions for fabrics: EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES, and then rotated about the element normal by an angle BETA 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 LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES,

VARIABLE	DESCRIPTION
	*DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR).
FLC	Optional porous leakage flow coefficient. (See theory manual.) GE.0: Porous leakage flow coefficient. LT.0: |FLC| is interpreted as a load curve ID defining FLC as a function of the stretching ratio defined as $r_s = A/A_0$. See notes below.
FAC	Optional characteristic fabric parameter. (See theory manual.) GE.0: Characteristic fabric parameter LT.0: $\vert\text{FAC}\vert$ is interpreted as a load curve defining leakage volume flux rate versus the pressure ratio defined as $r_p = P_{\text{air}}/P_{\text{bag}}$. See Remark 2 below. The volume flux (per area) rate (per time) has the unit of $$d(\text{vol}_{\text{flux}})/\text{dt} \approx [\text{length}]^3/([\text{length}]^2[\text{time}]) \approx [\text{length}]/[\text{time}]\,,$$ equivalent to relative porous gas speed.
ELA	Effective leakage area for blocked fabric, ELA (see Remark 3): LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of time. The default value of zero assumes that no leakage occurs. A value of .10 would assume that 10% of the blocked fabric is leaking gas.
LNRC	Flag to turn off compression in liner until the reference geometry is reached, that is, the fabric element becomes tensile: EQ.0.0: Off EQ.1.0: On
FORM	Flag to modify membrane formulation for fabric material: EQ.0.0: Default. Least costly and very reliable. EQ.1.0: Invariant local membrane coordinate system EQ.2.0: Green-Lagrange strain formulation EQ.3.0: Large strain with nonorthogonal material angles. See Remark 5.

VARIABLE	DESCRIPTION
	EQ.4.0: Large strain with nonorthogonal material angles and nonlinear stress strain behavior. Define optional load curve IDs on optional card.
	EQ.12.0: Enhanced version of formulation 2. See Remark 11.
	EQ.13.0: Enhanced version of formulation 3. See Remark 11.
	EQ.14.0: Enhanced version of formulation 4. See Remark 11.
	EQ.-14.0: Same as formulation 14 but invokes reading of Card 8. See Remark 14.
	EQ.24.0: Enhanced version of formulation 14. See Remark 11.
FVOPT	Fabric venting option.
	EQ.7: Leakage is based on gas volume outflow versus pressure load curve [Lian, 2000]. Blockage is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC in the *MAT_FABRIC card.
	EQ.8: Leakage is based on gas volume outflow versus pressure load curve [Lian 2000]. Blockage of venting or porous area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC in the *MAT_FABRIC card.
	Note: See Remark 17 for FVOPT option for CPM (*AIRBAG_PARTICLE) bags.
TSRFAC	Strain restoration factor:
	LT.0: |TSRFAC| is the ID of a curve defining TSRFAC as a function of time.
	GT.0 and LT.1: TSRFAC applied from time 0.
	GE.1: TSRFAC is the ID of a curve that defines TSRFAC as a function of time using an alternate method (not available for FORM = 0 or 1).

Additional card for FVOPT < 0.

Card 4	1	2	3	4	5	6	7	8
Variable	L	R	C1	C2	C3			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
L	Dimension of unit cell (length)
R	Radius of yarn (length)
C1	Pressure coefficient (dependent on unit system)
C2	Pressure exponent
C3	Strain coefficient

Card 5	1	2	3	4	5	6	7	8
Variable		RGBRTH	A0REF	A1	A2	A3	X0	X1
Type		F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
RGBRTH	Material dependent birth time of airbag reference geometry. Nonzero RGBRTH overwrites the birth time defined in the *AIRBAG_REFERENCE_GEOMETRY_BIRTH keyword. RGBRTH also applies to reference geometry defined by *AIRBAG_SHELL_REFERENCE_GEOMETRY.
A0REF	Calculation option of initial area, A_0, used for airbag porosity leakage calculation. EQ.0.: Default. Use the initial geometry defined in *NODE. EQ.1.: Use the reference geometry defined in *AIRBAG_REFERENCE_GEOMETRY or *AIRBAG_SHELL_REFERENCE_GEOMETRY.
A1, A2, A3	Components of vector **a** for AOPT = 2

VARIABLE	DESCRIPTION
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $A_{\text{leak}} = A_0\left(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p\right)$ X0.EQ.-1: Compressing seal vent option. The leakage area is evaluated as $A_{\text{leak}} = \max(A_{\text{current}} - A_0, 0)$.

Card 6	1	2	3	4	5	6	7	8
Variable	V1	V2	V3				BETA	ISREFG
Type	F	F	F				F	I

VARIABLE	DESCRIPTION
V1, V2, V3	Components of vector **v** for AOPT = 3
BETA	Material angle in degrees for AOPT = 0 and 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.
ISREFG	Initialize stress by *AIRBAG_REFERENCE_GEOMETRY. This option applies only to FORM = 12. Note that *MAT_FABRIC cannot be initialized using a dynain file because *INITIAL_STRESS_SHELL is not applicable to *MAT_FABRIC. EQ.0.0: Default. Not active. EQ.1.0: Active

Additional card for FORM = 4, 14, -14, or 24.

Card 7	1	2	3	4	5	6	7	8
Variable	LCA	LCB	LCAB	LCUA	LCUB	LCUAB	RL	
Type	I	I	I	I	I	I	F	

VARIABLE	DESCRIPTION
LCA	Load curve or table ID. Load curve ID defines the stress as a function of uniaxial strain along the *a*-axis fiber. Table ID defines for each strain rate a load curve representing stress as a function of uniaxial strain along the *a*-axis fiber. The load curve is available for

VARIABLE	DESCRIPTION
	FORM = 4, 14, –14, and 24 while the table is allowed only for FORM = -14. If zero, EA is used. For FORM = 14, -14, and 24, this curve can be defined in both tension and compression; see Remark 6 below.
LCB	Load curve or table ID. Load curve ID defines the stress as a function of uniaxial strain along the *b*-axis fiber. Table ID defines for each strain rate a load curve representing stress as a function of uniaxial strain along the *b*-axis fiber. The load curve is available for FORM = 4, 14, -14, and 24 while the table is allowed only for FORM = -14. If zero, EB is used. For FORM = 14, -14, and 24, this curve can be defined in both tension and compression; see Remark 6 below.
LCAB	Load curve ID for shear stress as a function of shear strain in the *ab*-plane. If zero, GAB is used.
LCUA	Unload/reload curve ID for stress as a function of strain along the *a*-axis fiber. If zero, LCA is used.
LCUB	Unload/reload curve ID for stress as a function of strain along the *b*-axis fiber. If zero, LCB is used.
LCUAB	Unload/reload curve ID for shear stress as a function of shear strain in the *ab*-plane. If zero, LCAB is used.
RL	Optional reloading parameter for FORM = 14 and 24. Values between 0.0 (reloading on unloading curve-default) and 1.0 (reloading on a minimum linear slope between unloading curve and loading curve) are possible.

Additional card for FORM = -14.

Card 8	1	2	3	4	5	6	7	8
Variable	LCAA	LCBB	H	DT		ECOAT	SCOAT	TCOAT
Type	I	I	F	F		F	F	F

VARIABLE	DESCRIPTION
LCAA	Load curve or table ID. Load curve ID defines the stress along the *a*-axis fiber as a function of biaxial strain. Table ID defines for each

VARIABLE	DESCRIPTION
	directional strain rate a load curve representing stress along the *a*-axis fiber as a function of biaxial strain. If zero, LCA is used.
LCBB	Load curve or table ID. Load curve ID defines the stress along the *b*-axis fiber as a function of biaxial strain. Table ID defines for each directional strain rate a load curve representing stress along the *b*-axis fiber as a function of biaxial strain. If zero, LCB is used.
H	Normalized hysteresis parameter between 0 and 1
DT	Strain rate averaging option: EQ.0.0: Strain rate is evaluated using a running average LT.0.0: Strain rate is evaluated using average of last 11 time steps GT.0.0: Strain rate is averaged over the last DT time units
ECOAT	Young's modulus of coat material; see Remark 14.
SCOAT	Yield stress of coat material; see Remark 14.
TCOAT	Thickness of coat material, may be positive or negative; see Remark 14.

Remarks:

1. **The compressive stress elimination option for airbag wrinkling.** Setting CSE = 1 switches off compressive stress in the fabric, thereby eliminating wrinkles. Without this "no compression" option, the geometry of the bag's wrinkles controls the amount of mesh refinement. In eliminating the wrinkles, this feature reduces the number of elements needed to attain an accurate solution.

   The no compression option can allow elements to collapse to a line which can lead to elements becoming tangled. The elastic liner option is one way to add some stiffness in compression to prevent this, see Remark 4. Alternatively, when using fabric formulations 14, -14, or 24 (see FORM) tangling can be reduced by defining stress/strain curves that include negative strain and stress values. See Remark 6.

2. **Porosity.** The parameters FLC and FAC are optional for the Wang-Nefske and Hybrid inflation models. It is possible for the airbag to be constructed of multiple fabrics having different values for porosity and permeability. Typically, FLC and FAC must be determined experimentally and their variations in time or with pressure are optional to allow for maximum flexibility.

3. **Effects of airbag-structure interaction on porosity.** To calculate the leakage of gas through the fabric it is necessary to accurately determine the leakage area. The dynamics of the airbag may cause the leakage area to change during the course of the simulation. In particular, the deformation may change the leakage area, but the leakage area may also decrease when the contact between the air-bag and the structure blocks the flow. LS-DYNA can check the interaction of the bag with the structure and split the areas into regions that are blocked and un-blocked depending on whether the regions are in or not in contact, respectively. Blockage effects may be controlled with the ELA field.

4. **Elastic liner.** An optional elastic liner can be defined using EL, PRL and LRA-TIO. The liner is an isotropic layer that acts in both tension and compression. However, setting, LNRC to 1.0 eliminates compressive stress in the liner until both principle stresses are tensile. The compressive stress elimination option, CSE = 1, has no influence on the liner behavior.

5. **Fiber axes.** For formulations 0, 1, and 2, (see FORM) 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, 4, 13, or 14, the fiber directions *are not assumed to be 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.

   When reference geometry is defined, the material axes are computed using co-ordinates from the reference geometry. The material axes are determined by computing the angle between the element system and the material direction.

6. **Stress as a function of strain curves.** For formulations (see FORM) 4, 14, -14, and 24, 2nd Piola-Kirchhoff stress as a function of Green's strain curves may be defined for *a*-axis, *b*-axis, and shear stresses for loading and also for unloading and reloading. Alternatively, the *a*-axis and *b*-axis curves can be input using engineering stress as a function of strain by setting DATTYP = -2 on *DEFINE_-CURVE.

   Additionally, for formulations 14, -14, and 24, the uniaxial loading curves LCA and LCB may be defined for negative values of strain and stress, that is, a straightforward extension of the curves into the compressive region. This is available for modeling the compressive stresses resulting from tight folding of airbags.

   The *a*-axis and *b*-axis stress follow the curves for the entire defined strain region and if compressive behavior is desired the user should preferably make sure the curve covers all strains of interest. For strains below the first point on the curve, the curve is extrapolated using the stiffness from the constant values, EA or EB.

Shear stress/strain behavior is assumed symmetric and curves should be defined for positive strain only. However, formulations 14, -14, and 24 allow the extending of the curves in the negative strain region to model asymmetric behavior. The asymmetric option cannot be used with a shear stress unload curve. If a load curve is omitted, the stress is calculated from the appropriate constant modulus, EA, EB, or GAB.

7. **Yield behavior.** When formulations 4, 14, -14, and 24 (see FORM) are used with loading and unloading curves the initial yield strain is set equal to the strain of the first point in the load curve having a stress greater than zero. When the current strain exceeds the yield strain, the stress follows 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 and then the stress is calculated from the shifted curve. When using unloading curves, compressive stress elimination should be active to prevent the fibers from developing compressive stress during unloading when the strain remains tensile. *To use this option, the unload curve should have a nonnegative second derivate so that the curve will shift right as the yield stress increases. In fact, if a shift to the left would be needed, the unload curves is not used and unloading will follow the load curve instead.*

   If LCUA, LCUB, or LCUAB are input with negative curve ID values, then unloading is handled differently. Instead of shifting the unload curve along the x-axis, the curve is stretched in both the x-direction and y-direction such that the first point remains anchored at (0,0) and the initial intersection point of the curves is moved to the current yield point. This option guarantees the stress remains tensile while the strain is tensile, so compressive stress elimination is not necessary. *To use this option the unload curve should have an initial slope less steep than the load curve and should steepen such that it intersects the load curve at some positive strain value.*

8. **Shear unload-reload, fabric formulation, and LS-DYNA version.** With release 6.0.0 of version 971, LS-DYNA changed the way that unload/reload curves for shear stress-strain relations are interpreted. Let f be the shear stress unload-reload curve LCUAB. Then,

$$\sigma_{ab} = c_2 f(c_1 \varepsilon_{ab})$$

where the scale factors c_1 and c_2 depend on the fabric form (see FORM).

Fabric form	c_1	c_2
4	2	1
14 and -14	1	2
24	1	1

When switching fabric forms or versions, the curve scale factors SFA and SFO on *DEFINE_CURVE can be used to offset this behavior.

9. **Per material venting option.** The FVOPT flag allows an airbag fabric venting equation to be assigned to a material. The anticipated use for this option is to allow a vent to be defined using FVOPT = 1 or 2 for one material and fabric leakage to be defined for using FVOPT = 3, 4, 5, or 6 for other materials. In order to use FVOPT, a venting option must first be defined for the airbag using the OPT parameter on *AIRBAG_WANG_NEFSKE or *AIRBAG_HYBRID. If OPT = 0, then FVOPT is ignored. If OPT is defined and FVOPT is omitted, then FVOPT is set equal to OPT.

10. **TSRFAC option to restore element strains.** Airbags that use a reference geometry will typically have nonzero strains at the start of the calculation. To prevent such initial strains from prematurely opening an airbag, initial strains are stored and subtracted from the measured strain throughout the calculation.

$$\sigma = f(\varepsilon - \varepsilon_{\text{initial}})$$

- Fabric formulations 2, 3, and 4 (see FORM) subtract off only the initial tensile strains so these forms are typically used with CSE = 1 and LNRC = 1.
- Fabric formulations 12, 13, 14, -14, and 24 subtract off the total initial strains, so these forms may be used with CSE = 0 or 1 and LNRC = 0 or 1. A side effect of this strain modification is that airbags may not achieve the correct volume when they open. Therefore, the TSRFAC option is implemented to reduce the stored initial strain values over time thereby restoring the total stain which drives the airbag towards the correct volume.

During each cycle, the stored initial strains are scaled by $(1.0 - \text{TSRFAC})$. A small value on the order of 0.0001 is typically sufficient to restore the strains in a few milliseconds of simulation time.

$$\sigma = f\left(\varepsilon - \varepsilon_{\text{adjustment}}\right)$$

The adjustment to restore initial strain is then,

$$\varepsilon_{\text{adjustment}} = \varepsilon_{\text{initial}} \prod_i [1 - \text{TSFRAC}].$$

a) *Time Dependent TSRFAC.* When TSRFAC < 0, |TSRFAC| becomes the ID of a curve that defines TSRFAC as a function of time. To delay the effect of TSRFAC, the curve ordinate value should be initially zero and should ramp up to a small number to restore the strain at an appropriate time during the simulation. The adjustment to restore initial strain is then,

$$\varepsilon_{\text{adjustment}}(t_i) = \varepsilon_{\text{initial}} \prod_i [1 - \text{TSFRAC}(t_i)].$$

To prevent airbags from opening prematurely, it is recommended to use the load curve option of TSRFAC to delay the strain restoration until the airbag is partially opened due to pressure loading.

b) *Alternate Time Dependent TSRFAC.* For fabric formulations 2 and higher, a second curve option is invoked by setting TSRFAC ≥ 1 where TSRFAC is again the ID of a curve that defines TSRFAC as a function of time. Like the first curve option, the stored initial strain values are scaled by (1.0 − TSRFAC), but the modified initial strains are not saved, so the effect of TSRFAC does not accumulate. In this case the adjustment to eliminate initial strain

$$\varepsilon_{\text{adjustment}}(t_i) = [1 - \text{TSFRAC}(t_i)]\varepsilon_{\text{initial}}.$$

Therefore, the curve should ramp up from zero to one to fully restore the strain. This option gives the user better control of the rate of restoring the strain as it is a function of time rather than solution time step.

11. **Enhancements to the material formulations.** Material formulations (see FORM) 12, 13, and 14 are enhanced versions of formulations 2, 3, and 4, respectively. The most notable difference in their behavior is apparent when a reference geometry is used for the fabric. As discussed in Remark 10, the strain is modified to prevent initial strains from prematurely opening an airbag at the start of a calculation.

    Formulations 2, 3, and 4 subtract the initial tensile strains, while the enhanced formulations subtract the total initial strains. Therefore, the enhanced formulations can be used without setting CSE = 1 and LNRC = 1 since compressive stress cutoff is not needed to prevent initial airbag movement. Formulations 2, 3, and 4 need compressive stress cutoff when used with a reference geometry or they can generate compressive stress at the start of a calculation. Available for formulation 12 only, the ISREFG parameter activates an option to calculate the initial stress by using a reference geometry.

    Material formulation 24 is an enhanced version of formulation 14 implementing a correction for Poisson's effects when stress as a function of strain curves are input for the *a*-fiber or *b*-fiber. Also, for formulation 24, the outputted stress and strain in the elout or d3plot database files is engineering stress and strain rather than the 2nd Piola Kirchoff and Green's strain used by formulations other than 0 and 1.

12. **Noise reduction for the strain rate measure.** If tables are used, then the strain rate measure is the time derivative of the Green-Lagrange strain in the direction of interest. To suppress noise, the strain rate is averaged according to the value of DT. If DT > 0, it is recommended to use a large enough value to suppress the noise, while being small enough to not lose important information in the signal.

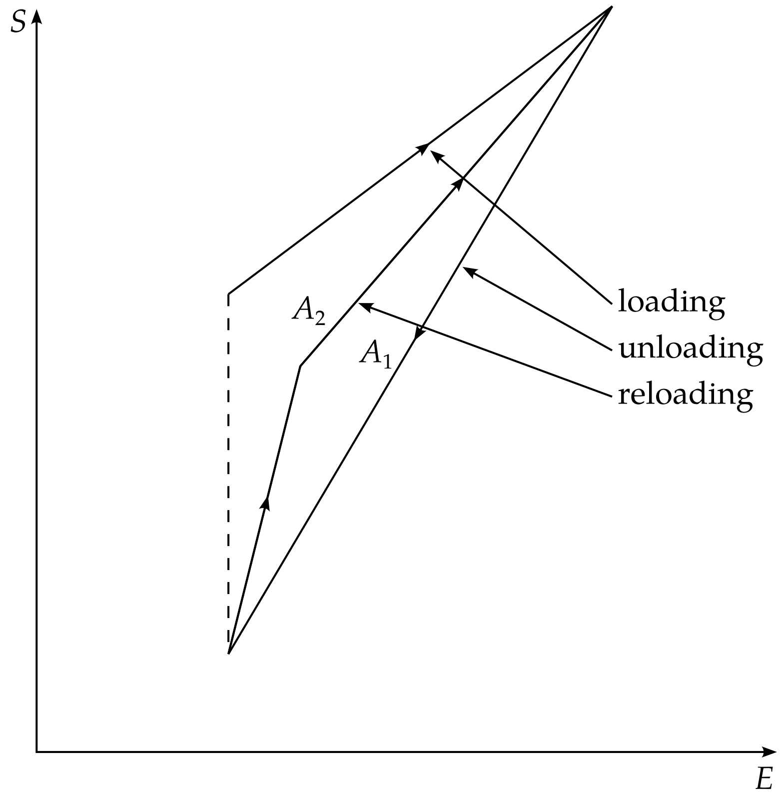


Figure M34-1. Hysteresis curve

13. **Hysteresis.** The hysteresis parameter H defines the fraction of dissipated energy during a load cycle in terms of the maximum possible dissipated energy. Referring to the Figure M34-1,

$$H \approx \frac{A_1}{A_1 + A_2} \ .$$

14. **Coating feature for additional rotational resistance.** It is possible to model coating of the fabric using a sheet of elastic-ideal-plastic material where the Young's modulus, yield stress, and thickness is specified for the coat material. This coating feature adds rotational resistance to the fabric for more realistic behavior of coated fabrics. To read these properties set FORM = -14, which adds an extra card containing the three fields ECOAT, SCOAT and TCOAT, corresponding to the three coat material properties mentioned above. The coating model includes transverse shear stiffness to avoid nonphysical zig-zagging. To adjust this stiffness, set SHRF on *SECTION_SHELL.

    The thickness, TCOAT, applies to both sides of the fabric. The coat material for a certain fabric element deforms along with this and all elements connected to this element, which is how the rotations are "captured." Note that unless TCOAT is set to a negative value, the coating will add to the membrane stiffness. For negative values of TCOAT the thickness is set to |TCOAT| and the membrane contribution from the coating is suppressed. For this feature to work, the fabric parts must not include any T-intersections, and all of the surface normal vectors of connected fabric elements must point in the same direction. This option increases the computational complexity of this material.

15. **Poisson's ratios.** Fabric forms 12, 13, 14, -14, and 24 allow input of both the minor Poisson's ratio, ν_{ba}, and the major Poisson's ratio, ν_{ab}. This allows asymmetric Poisson's behavior to be modelled. If the major Poisson's ratio is left blank or input as zero, then it will be calculated using $\nu_{ab} = \nu_{ba} \frac{E_a}{E_b}$.

16. **St. Venant-Wantzel leakage.** If a negative value for the fabric venting option FVOPT is used (only -1 and -2 are supported), the mass flux through a fabric membrane is calculated according St.Venant-Wantzel by

$$\dot{m} = A_{\mathrm{eff}} \Psi \sqrt{2 p_i \rho_i}$$

where p_i describes the internal pressure, ρ_i is the density of the outlet gas, and the effluence function ψ depends on the character of the flow, the adiabatic exponent κ and the pressure difference between the inside (p_i) and the outside (p_a) of the membrane. For subsonic flow it is formulated as:

$$\Psi = \sqrt{\frac{\kappa}{\kappa - 1}\left[\left(\frac{p_a}{p_i}\right)^{\frac{2}{\kappa}} - \left(\frac{p_a}{p_i}\right)^{\frac{\kappa+1}{\kappa}}\right]}$$

and for sonic or critical flow as:

$$\Psi = \sqrt{\frac{\kappa}{2}\left(\frac{2}{\kappa + 1}\right)^{\frac{\kappa+1}{\kappa-1}}}$$

The effective venting area of the membrane is determined according to M. Schlenger:

$$A_{\mathrm{eff}} = \frac{A_0}{L^2}\left[\left(C_1 \Delta p^{C_2} - C_3\right)(L - 2r)^2 + C_3\left(L\lambda_1 - \frac{2r}{\sqrt{\lambda_2}}\right)\left(L\lambda_2 - \frac{2r}{\sqrt{\lambda_1}}\right)\right]\sin(\alpha_{12})$$

where λ_i is the stretch in fiber direction i and α_{12} is the angle between the fibers. The initial membrane area is equal to A_0. r and L represent the radius of the fabric fiber and the edge length of the fabric set, respectively. The coefficients, L, r, C_1, C_2, and C_3, must be defined on additional Card 4. This option is supported for *AIRBAG_WANG_NESFKE, *AIRBAG_HYBRID, and *AIRBAG_-PARTICLE. No additional input in *AIRBAG cards is needed. All FORM options are supported, whereas α_{12} can only be different from 90 degree for FORM = 3, 4, 13, or 14.

17. **CPM (*AIRBAG_PARTICLE) bags.** Only FVOPT = -1, -2, 7, and 8 are supported for CPM bags. If FVOPT = 0 is used, it defaults to FVOPT = 8. For FVOPT = -1 and -2, FLC is active and can be either a scalar or a curve defining the porous leakage flow coefficient as a function of time. The FAC coefficient, however, is *inactive* as the porous leakage velocity is computed using the formula specified

in Remark 16 from the coefficient defined in Card 4. Note that for uniform pressure airbags (*AIRBAG_HYBRID_...) *both* the FLC and FAC coefficients are active.

*MAT_FABRIC_MAP

This is Material Type 34 in which the stress response is given exclusively by tables, or maps, and where some obsolete features in *MAT_FABRIC have been deliberately excluded to allow for a clean input and better overview of the model. The response can be made temperature dependent.

Card Summary:

Card 1. This card is required.

MID	RO	PXX	PYY	SXY	DAMP	TH	T0

Card 1.1. Include this card if T0 > 0.

T1	T2	T3	T4	T5	T6	T7	T8

Card 1.2. Include this card if T0 > 0.

PXX1	PXX2	PXX3	PXX4	PXX5	PXX6	PXX7	PXX8

Card 1.3. Include this card if T0 > 0.

PYY1	PYY2	PYY3	PYY4	PYY5	PYY6	PYY7	PYY8

Card 1.4. Include this card if T0 > 0.

SXY1	SXY2	SXY3	SXY4	SXY5	SXY6	SXY7	SXY8

Card 2a. Include this card if 0 < X0 < 1.

FVOPT	X0	X1	X2	X3	ELA		

Card 2b. Include this card if X0 = 0 or X0 = -1 and FVOPT < 7.

FVOPT	X0	X1	FLC	FAC	ELA		

Card 2c. Include this card if X0 = 0 or X0 = -1 and FVOPT ≥ 7.

FVOPT	X0	X1	FLC	FAC	ELA		

Card 2d. Include this card if X0 = 1 and FVOPT < 7.

FVOPT	X0	X1	FLC	FAC	ELA		

Card 2e. Include this card if X0 = 1 and FVOPT ≥ 7.

FVOPT	X0	X1	FLC	FAC	ELA		

Card 3. This card is required.

ISREFG	CSE	SRFAC	BULKC	JACC	FXX	FYY	DT

Card 4. This card is required.

AOPT	ECOAT	SCOAT	TCOAT				

Card 5. This card is required.

XP	YP	ZP	A1	A2	A3		

Card 6. This card is required.

V1	V2	V3	D1	D2	D3	BETA	

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PXX	PYY	SXY	DAMP	TH	T0
Type	A	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
PXX	Table giving engineering local XX-stress as function of engineering local XX-strain and YY-strain
PYY	Table giving engineering local YY-stress as function of engineering local YY-strain and XX-strain
SXY	Curve giving local 2nd Piola-Kirchhoff XY-stress as function of local Green XY-strain.
DAMP	Damping coefficient for numerical stability
TH	Table giving hysteresis factor $0 \leq H < 1$ as function of engineering local XX-strain and YY-strain:

VARIABLE	DESCRIPTION
	GT.0.0: TH is table ID.
	LE.0.0: -TH is used as constant value for hysteresis factor.
T0	Flag to indicate temperature dependence and temperature corresponding to tables PXX, PYY, and SXY: EQ.0.0: Do not consider temperature dependence for this model (default). GT.0.0: Consider temperature dependence considered. T0 gives the temperature corresponding to tables PXX, PYY, and SXY. LS-DYNA expects Cards 1.1 through 1.4 to provide additional positive temperatures that correspond to similar tables. T0 represents a typical work temperature. It may be anywhere inside or outside the range between T1 - T8 defined in Card 1.1, but it cannot be equal to any of those individual values. Note that we are assuming that no matter the units the relevant temperatures are positive.

Include this card if T0 > 0.

Card 1.1	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

VARIABLE	DESCRIPTION
T*i*	Temperature values for which the tables and curves specified in Cards 1.2 - 1.4 apply. Temperature values must be increasing and positive, meaning T1 > 0, T2 > T1, T3 > T2, etc. If needing fewer than 8 temperature points, then set the first unused temperature value to zero. T0 may not take any of the positive T*i* values but will be properly inserted into the range so that all positive temperatures defined are in increasing order.

Include this card if T0 > 0.

Card 1.2	1	2	3	4	5	6	7	8
Variable	PXX1	PXX2	PXX3	PXX4	PXX5	PXX6	PXX7	PXX8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
PXX*i*	Table giving engineering local *XX*-stress as a function of engineering local *XX*-strain and *YY*-strain for temperature T*i*.

Include this card if T0 > 0.

Card 1.3	1	2	3	4	5	6	7	8
Variable	PYY1	PYY2	PYY3	PYY4	PYY5	PYY6	PYY7	PYY8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
PYY*i*	Table giving engineering local *YY*-stress as a function of engineering local *YY*-strain and *XX*-strain for temperature T*i*.

Include this card if T0 > 0.

Card 1.4	1	2	3	4	5	6	7	8
Variable	SXY1	SXY2	SXY3	SXY4	SXY5	SXY6	SXY7	SXY8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
SXY*i*	Curve giving local 2nd Piola-Kirchhoff *XY*-stress as function of local Green *XY*-strain for temperature T*i*.

This card is included if 0 < X0 < 1.

Card 2a	1	2	3	4	5	6	7	8
Variable	FVOPT	X0	X1	X2	X3	ELA		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
FVOPT	Fabric venting option (see *MAT_FABRIC): EQ.1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ.3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ.5: Leakage formulas based on flow through a porous media are used. Blockage is not considered. EQ.6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered. EQ.7: Leakage is based on gas volume outflow versus pressure load curve [Lian, 2000]. Blockage is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC. EQ.8: Leakage is based on gas volume outflow versus pressure load curve [Lian 2000]. Blockage of venting or porous area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC. LT.0: |FVOPT| defines the same fabric venting options as above, but a new formula for the leakage area is used to replace the element area. See Remark 16 of *MAT_FABRIC. Note: See Remark 17 of *MAT_FABRIC for FVOPT option for CPM (*AIRBAG_PARTICLE) bags.

VARIABLE	DESCRIPTION
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $A_{\text{leak}} = A_0(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p)$; see *MAT_FABRIC.
X2	Coefficient of the porosity from the equation in Anagonye and Wang [1999]
X3	Coefficient of the porosity equation of Anagonye and Wang [1999]
ELA	Effective leakage area for blocked fabric, ELA (see Remark 3 of *MAT_FABRIC): LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of 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.

This card is included if and only if X0 = 0 or X0 = -1 and FVOPT < 7.

Card 2b	1	2	3	4	5	6	7	8
Variable	FVOPT	X0	X1	FLC	FAC	ELA		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
FVOPT	Fabric venting option (see *MAT_FABRIC): EQ.1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered. EQ.2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered. EQ.3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered. EQ.4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered. EQ.5: Leakage formulas based on flow through a porous media are used. Blockage is not considered.

VARIABLE	DESCRIPTION
	EQ.6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.
	LT.0: |FVOPT| defines the same fabric venting options as above, but a new formula for the leakage area is used to replace the element area. See Remark 16 of *MAT_FABRIC.
	Note: See Remark 17 of *MAT_FABRIC for FVOPT option for CPM (*AIRBAG_PARTICLE) bags.
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $A_{\mathrm{leak}} = A_0(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p)$; see *MAT_-FABRIC.
	X0.EQ.-1: Compressing seal vent option. The leakage area is evaluated as $A_{\mathrm{leak}} = \max(A_{\mathrm{current}} - A_0, 0)$.
FLC	Optional porous leakage flow coefficient. (See theory manual.)
	GE.0.0: Porous leakage flow coefficient.
	LT.0.0: |FLC| is a load curve ID defining FLC as a function of time.
FAC	Optional characteristic fabric parameter. (See theory manual.)
	GE.0.0: Characteristic fabric parameter
	LT.0.0: |FAC| is a load curve ID defining FAC as a function of absolute pressure.
ELA	Effective leakage area for blocked fabric, ELA (see Remark 3 of *MAT_FABRIC):
	LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of 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.

This card is included if and only if X0 = 0 or X0 = -1 and FVOPT ≥ 7.

Card 2c	1	2	3	4	5	6	7	8
Variable	FVOPT	X0	X1	FLC	FAC	ELA		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
FVOPT	Fabric venting option (see *MAT_FABRIC): EQ.7: Leakage is based on gas volume outflow versus pressure load curve [Lian, 2000]. Blockage is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC. EQ.8: Leakage is based on gas volume outflow versus pressure load curve [Lian 2000]. Blockage of venting or porous area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC. Note: See Remark 17 of *MAT_FABRIC for FVOPT option for CPM (*AIRBAG_PARTICLE) bags.
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $A_{\text{leak}} = A_0\left(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p\right)$; see *MAT_-FABRIC. X0.EQ.-1: Compressing seal vent option. The leakage area is evaluated as $A_{\text{leak}} = \max(A_{\text{current}} - A_0, 0)$.
FLC	Optional porous leakage flow coefficient. (See theory manual.) GE.0.0: Porous leakage flow coefficient LT.0.0: |FLC| is a load curve ID defining FLC as a function of time.
FAC	Optional characteristic fabric parameter. (See theory manual.) GE.0.0: Characteristic fabric parameter LT.0.0: |FAC| is a load curve ID giving leakage volume flux rate as a function of absolute pressure. The volume flux (per area) rate (per time) has the dimensions of

VARIABLE	DESCRIPTION

$$d(\mathrm{vol}_{\mathrm{flux}})/\mathrm{dt} \approx [\mathrm{length}]^3/([\mathrm{length}]^2[\mathrm{time}])$$

$$\approx [\mathrm{length}]/[\mathrm{time}]$$

equivalent to relative porous gas speed.

ELA — Effective leakage area for blocked fabric, ELA (see Remark 3 of *MAT_FABRIC):

LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of 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.

This card is included if and only if X0 = 1 and FVOPT < 7.

Card 2d	1	2	3	4	5	6	7	8
Variable	FVOPT	X0	X1	FLC	FAC	ELA		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION

FVOPT — Fabric venting option (see *MAT_FABRIC):

EQ.1: Wang-Nefske formulas for venting through an orifice are used. Blockage is not considered.

EQ.2: Wang-Nefske formulas for venting through an orifice are used. Blockage of venting area due to contact is considered.

EQ.3: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage is not considered.

EQ.4: Leakage formulas of Graefe, Krummheuer, and Siejak [1990] are used. Blockage of venting area due to contact is considered.

EQ.5: Leakage formulas based on flow through a porous media are used. Blockage is not considered.

EQ.6: Leakage formulas based on flow through a porous media are used. Blockage of venting area due to contact is considered.

VARIABLE	DESCRIPTION
	LT.0: |FVOPT| defines the same fabric venting options as above, but a new formula for the leakage area is used to replace the element area. See Remark 16 of *MAT_FABRIC. Note: See Remark 17 of *MAT_FABRIC for FVOPT option for CPM (*AIRBAG_PARTICLE) bags.
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $A_{\mathrm{leak}} = A_0(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p)$; see *MAT_-FABRIC.
FLC	Optional porous leakage flow coefficient. (See theory manual and *MAT_FABRIC.) GE.0.0: Porous leakage flow coefficient. LT.0.0: |FLC| is interpreted as a load curve ID defining FLC as a function of the stretching ratio defined as $r_s = A/A_0$.
FAC	Optional characteristic fabric parameter. (See theory manual and *MAT_FABRIC.) GE.0.0: Characteristic fabric parameter LT.0.0: |FAC| is interpreted as a load curve defining FAC as a function of the pressure ratio $r_p = P_{\mathrm{air}}/P_{\mathrm{bag}}$.
ELA	Effective leakage area for blocked fabric, ELA (see Remark 3 of *MAT_FABRIC): LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of 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.

This card is included if and only if X0 = 1 and FVOPT ≥ 7.

Card 2e	1	2	3	4	5	6	7	8
Variable	FVOPT	X0	X1	FLC	FAC	ELA		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
FVOPT	Fabric venting option (see *MAT_FABRIC): EQ.7: Leakage is based on gas volume outflow versus pressure load curve [Lian, 2000]. Blockage is not considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC. EQ.8: Leakage is based on gas volume outflow versus pressure load curve [Lian 2000]. Blockage of venting or porous area due to contact is considered. Absolute pressure is used in the porous-velocity-versus-pressure load curve, given as FAC. Note: See Remark 17 of *MAT_FABRIC for FVOPT option for CPM (*AIRBAG_PARTICLE) bags.
X0, X1	Coefficients of Anagonye and Wang [1999] porosity equation for the leakage area: $A_{\mathrm{leak}} = A_0(X_0 + X_1 r_s + X_2 r_p + X_3 r_s r_p)$; see *MAT_-FABRIC.
FLC	Optional porous leakage flow coefficient. (See theory manual.) GE.0: Porous leakage flow coefficient. LT.0: |FLC| is interpreted as a load curve ID defining FLC as a function of the stretching ratio defined as $r_s = A/A_0$.
FAC	Optional characteristic fabric parameter. (See theory manual.) GE.0: Characteristic fabric parameter LT.0: $\vert\mathrm{FAC}\vert$ is interpreted as a load curve defining leakage volume flux rate versus the pressure ratio defined as $r_p = P_{\mathrm{air}}/P_{\mathrm{bag}}$. The volume flux (per area) rate (per time) has the unit of $d(\mathrm{vol}_{\mathrm{flux}})/\mathrm{dt} \approx [\mathrm{length}]^3/([\mathrm{length}]^2[\mathrm{time}]) \approx [\mathrm{length}]/[\mathrm{time}]$, equivalent to relative porous gas speed.
ELA	Effective leakage area for blocked fabric, ELA (see Remark 3 of *MAT_FABRIC): LT.0.0: |ELA| is the load curve ID of the curve defining ELA as a function of time. The default value of zero assumes that

VARIABLE	DESCRIPTION
	no leakage occurs. A value of .10 would assume that 10% of the blocked fabric is leaking gas.

Card 3	1	2	3	4	5	6	7	8
Variable	ISREFG	CSE	SRFAC	BULKC	JACC	FXX	FYY	DT
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
ISREFG	Initial stress by reference geometry: EQ.0.0: Not active EQ.1.0: Active
CSE	Compressive stress elimination option: EQ.0.0: Do not eliminate compressive stresses. EQ.1.0: Eliminate compressive stresses.
SRFAC	Load curve ID for smooth stress initialization when using a reference geometry
BULKC	Bulk modulus for fabric compaction
JACC	Jacobian for the onset of fabric compaction
FXX	Load curve giving scale factor of uniaxial stress in first material direction as function of engineering strain rate
FYY	Load curve giving scale factor of uniaxial stress in second material direction as function of engineering strain rate
DT	Time window for smoothing strain rates used for FXX and FYY

Card 4	1	2	3	4	5	6	7	8
Variable	AOPT	ECOAT	SCOAT	TCOAT				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC for a more complete description). Also, please refer to Remark 5 of *MAT_FABRIC for additional information specific to fiber directions for fabrics: EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES, and then rotated about the element normal by an angle BETA 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 LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
ECOAT	Young’s modulus of coat material to include bending properties. This together with the following two parameters (SCOAT and TCOAT) encompass the same coating/bending feature as in *MAT_FABRIC. Please refer to these manual pages and associated remarks.
SCOAT	Yield stress of coat material, see *MAT_FABRIC.
TCOAT	Thickness of coat material, may be positive or negative, see *MAT_FABRIC.

Card 5	1	2	3	4	5	6	7	8
Variable				A1	A2	A3		
Type				F	F	F		

VARIABLE	DESCRIPTION
A1, A2, A3	Components of vector **a** for AOPT = 2

Card 6	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
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 = 0 and 3. It may be overridden on the element card; see *ELEMENT_SHELL_BETA.

Remarks:

1. **Material Model.** This material model invokes a special membrane element formulation regardless of the element choice. It is an anisotropic hyperelastic model, where the 2nd Piola-Kirchhoff stress **S** is a function of the Green-Lagrange strain **E** and possibly its history and temperature. Due to anisotropy, this strain is transformed to obtain the strains in each of the fiber directions, E_{XX} and E_{YY}, together with the shear strain, E_{XY}. The associated stress components in the local system are given as functions of the strain components and temperature

$$S_{XX} = \gamma S_{XX}(E_{XX}, E_{YY}, T)\vartheta$$
$$S_{YY} = \gamma S_{YY}(E_{YY}, E_{XX}, T)\vartheta$$
$$S_{XY} = \gamma S_{XY}(E_{XY}, T)\vartheta$$

The factor γ is used for dissipative effects and is described in more detail in Remark 5. For TH = 0, $\gamma = 1$. The function ϑ represents a strain rate scale factor (see Remark 7); for FXX = FYY = 0, this factor is 1. While the input curve SXY directly gives the shear relation, the tabular input of the fiber stress components PXX and PYY is for the sake of convenience. PXX and PYY give the engineering stress as a function of engineering strain and optionally temperature, that is,

$$P_{XX} = P_{XX}(e_{XX}, e_{YY}, T)$$
$$P_{YY} = P_{YY}(e_{YY}, e_{XX}, T)$$

Because of this, the following conversion formulae are used between stresses and strains

$$e = \sqrt{1+2E} - 1$$
$$S = \frac{P}{1+e}$$

which are applied in each of the two fiber directions.

2. **Temperature Dependence.** We apply temperature dependence through input tables and curves for up to 9 different temperature values (see T0 and Cards 1.1 through 1.4). Whenever the temperature in an element is between two defined temperature values, interpolation of the values for the two temperature points gives the resulting value. If the temperature in an element falls below the smallest temperature defined or above the largest temperature defined, the resulting value is not extrapolated, but the first and last defined table/curve is used, respectively. Note that LS-DYNA inserts T0 and its associated data at the appropriate location so that all temperature values are internally in increasing order. For determing dissipation in the material, we use the properties at temperature T0.

3. **Compressive Stress Elimination.** Compressive stress elimination is optional through the CSE parameter, and when activated the principal components of the 2nd Piola-Kirchhoff stress is restricted to positive values.

4. **Reference Geometry and Smooth Stress Initialization.** If a reference geometry is used, then SRFAC is the curve ID for a function $\alpha(t)$ that should increase from zero to unity during a short time span. During this time span, the Green-Lagrange strain used in the formulae in Remark 1 above is substituted with

$$\tilde{\mathbf{E}} = \mathbf{E} - [1 - \alpha(t)]\mathbf{E}_0,$$

where $\mathbf{E}_0$ is the strain at time zero. This is done in order to smoothly initialize the stress resulting from using a reference geometry different from the geometry at time zero.

5. **Dissipative Effects.** The factor γ is a function of the strain history and is initially set to unity. It specifically depends on the internal work, ϵ, given by the stress power

$$\dot{\epsilon} = \mathbf{S} : \dot{\mathbf{E}}.$$

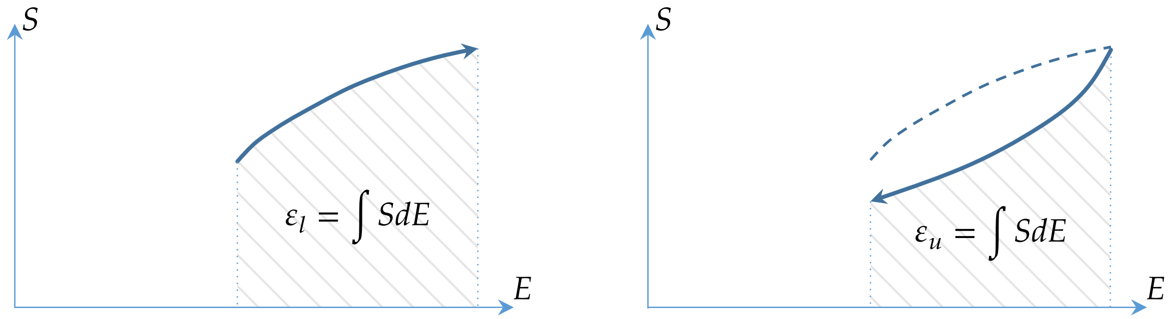


Figure M34M-1. Cyclic loading model for hysteresis model H

The evolution of γ is related to the stress power since it increases on loading and decreases on unloading. As a result, it introduces dissipation. The exact mathematical formula is too complicated to reveal, but basically the function looks like

$$\gamma = \begin{cases} 1 - H(\bar{e}_{XX}, \bar{e}_{YY}) + H(\bar{e}_{XX}, \bar{e}_{YY})\exp[\beta(\epsilon - \overline{\epsilon})] & \dot{\epsilon} < 0 \\ 1 - H(\bar{e}_{XX}, \bar{e}_{YY})\exp[-\beta(\epsilon - \underline{\epsilon})] & \dot{\epsilon} \geq 0 \end{cases}$$

Here $\overline{\epsilon}$ is the maximum attained internal work up to this point in time, $\bar{e}_{XX}$ and $\bar{e}_{YY}$ are the engineering strain values associated with value. $H(\bar{e}_{XX}, \bar{e}_{YY})$ is the hysteresis factor specified through input parameter TH; it may or may not depend on the strains. β is a decay constant that depends on $\bar{e}_{XX}$ and $\bar{e}_{YY}$, and $\underline{\epsilon}$ is the minimum attained internal work at any point in time after $\overline{\epsilon}$ was attained. In other words, on unloading, γ will exponentially decay to $1 - H$, and on loading it will exponentially grow to 1 and always be restricted by the lower and upper bounds, $1 - H < \gamma \leq 1$. This formulation requires inputting a proper hysteresis factor H. With reference to a general loading/unloading cycle illustrated in Figure M34M-1, the relation $1 - H = \epsilon_u/\epsilon_l$ should hold with proper input. LS-DYNA uses the properties at the work temperature T0 for this dissipative treatment.

6. **Packing of Yarn in Compression.** To account for the packing of yarns in compression, a compaction effect is modeled by adding a term to the strain energy function of the form

$$W_c = K_c J \left\{ \ln\left(\frac{J}{J_c}\right) - 1 \right\}, \text{ for } J \leq J_c \ .$$

Here K_c (BULKC) is a physical bulk modulus, $J = \det(\mathbf{F})$ is the jacobian of the deformation and J_c (JACC) is the critical jacobian for when the effect commences. $\mathbf{F}$ is the deformation gradient. This contributes to the pressure with a term

$$p = K_c \ln\left(\frac{J_c}{J}\right), \text{ for } J \leq J_c$$

and thus prevents membrane elements from collapsing or inverting when subjected to compressive loads. The bulk modulus K_c should be selected with the slopes in the stress map tables in mind, presumably some order of magnitude(s) smaller.

7. **Strain Rate Scale Factor.** As an option, the local membrane stress can be scaled based on the engineering strain rates via the function $\vartheta = \vartheta(\dot{e}, \mathbf{S})$. We set

$$\dot{e} = \max\left(\frac{\dot{\epsilon}}{\|\mathbf{FS}\|}, 0\right)$$

to be the equivalent engineering strain rate in the direction of loading and define

$$\vartheta(\dot{e}, S) = \frac{F_{XX}(\dot{e})|S_{XX}| + F_{YY}(\dot{e})|S_{YY}| + 2|S_{XY}|}{|S_{XX}| + |S_{YY}| + 2|S_{XY}|},$$

meaning that the strain rate scale factor defaults to the user input data FXX and FYY for uniaxial loading in the two material directions, respectively. Note that we only consider strain rate scaling in loading and not in unloading, and furthermore that the strain rates used in evaluating the curves are pre-filtered using the time window DT to avoid excessive numerical noise. It is, therefore, recommended to set DT to a time corresponding to at least hundred time steps or so.

*MAT_PLASTIC_GREEN-NAGHDI_RATE

This is Material Type 35. It 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. It is available for solid, thick shell (formulations 3, 5,and 7), and SPH elements.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR				
Type	A	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. A unique number or label must be specified (see *PART).
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_{OPTION}

This is Material Type 36. This model was developed by Barlat and Lian [1989] for modeling sheets with anisotropic materials under plane stress conditions. Lankford parameters may be used to define the anisotropy. This particular development is due to Barlat and Lian [1989]. *MAT_FLD_3-PARAMETER_BARLAT is a version of this material model that includes a flow limit diagram failure option.

Available options include:

<BLANK>

NLP

The NLP option estimates failure using the Formability Index (F.I.), which accounts for the non-linear strain paths seen in metal forming applications (see the Remarks). The NLP field in Card 4b *must* be defined when using this option. The NLP option is also available for *MAT_037, *MAT_125, and *MAT_226.

Card Summary:

Card 1. This card is required.

MID	RO	E	PR	HR	P1	P2	ITER

Card 2a. This card is included if PB = 0 (see Card 4a/4b).

M	R00	R45	R90	LCID	E0	SPI	P3

Card 2b. This card is included if PB > 0. (see Card 4a/4b).

M	AB	CB	HB	LCID	E0	SPI	P3

Card 3. This card is included if M < 0.

CRC1	CRA1	CRC2	CRA2	CRC3	CRA3	CRC4	CRA4

Card 4a. This card is included if the keyword option is unset (<BLANK>).

AOPT	C	P	VLCID		PB	HTA	HTB

Card 4b. This card is included if the keyword option is NLP.

AOPT	C	P	VLCID		PB	NLP	

Card 5. This card is required.

			A1	A2	A3	HTC	HTD

Card 6. This card is required.

V1	V2	V3	D1	D2	D3	BETA	HTFLAG

Card 7. This card is optional.

USRFAIL	LCBI	LCSH					

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	HR	P1	P2	ITER
Type	A	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus, E GT.0.0: constant value LT.0.0: load curve ID = |E|, which defines Young's Modulus as a function of plastic strain. See Remarks.
PR	Poisson's ratio, ν
HR	Hardening rule: EQ.1.0: linear (default) EQ.2.0: exponential (Swift) EQ.3.0: load curve or table with strain rate effects EQ.4.0: exponential (Voce) EQ.5.0: exponential (Gosh)

VARIABLE	DESCRIPTION
	EQ.6.0: exponential (Hocket-Sherby)
	EQ.7.0: load curves in three directions
	EQ.8.0: table with temperature dependence
	EQ.9.0: three-dimensional table with temperature and strain rate dependence
	EQ.10.0: table with pre-strain dependence. See Remarks.
P1	Material parameter:
	HR.EQ.1.0: tangent modulus
	HR.EQ.2.0: k, strength coefficient for Swift exponential hardening
	HR.EQ.4.0: a, coefficient for Voce exponential hardening
	HR.EQ.5.0: k, strength coefficient for Gosh exponential hardening
	HR.EQ.6.0: a, coefficient for Hocket-Sherby exponential hardening
	HR.EQ.7.0: load curve ID for hardening in the 45°-direction. See Remarks.
P2	Material parameter:
	HR.EQ.1.0: yield stress
	HR.EQ.2.0: n, exponent for Swift exponential hardening
	HR.EQ.4.0: c, coefficient for Voce exponential hardening
	HR.EQ.5.0: n, exponent for Gosh exponential hardening
	HR.EQ.6.0: c, coefficient for Hocket-Sherby exponential hardening
	HR.EQ.7.0: load curve ID for hardening in the 90°-direction. See Remarks.
ITER	Iteration flag for speed:
	EQ.0.0: fully iterative
	EQ.1.0: fixed at three iterations
	Generally, ITER = 0 is recommended. ITER = 1, however, is somewhat faster and may give acceptable results in most problems.

Lankford Parameters Card. This card is included if PB = 0 (see Card 4a/4b).

Card 2a	1	2	3	4	5	6	7	8
Variable	M	R00	R45	R90	LCID	E0	SPI	P3
Type	F	F	F	F	I	F	F	F

VARIABLE	DESCRIPTION
M	m, exponent in Barlat's yield surface. If negative, the absolute value is used.
R00	R_{00}, Lankford parameter in 0°-direction: GT.0.0: constant value LT.0.0: load curve or table ID = |R00| which defines R_{00} as a function of plastic strain (curve) or as a function of temperature and plastic strain (table). See Remarks.
R45	R_{45}, Lankford parameter in 45°-direction: GT.0.0: constant value LT.0.0: load curve or table ID = |R45| which defines R_{45} as a function of plastic strain (curve) or as a function of temperature and plastic strain (table). See Remarks.
R90	R_{90}, Lankford parameter in 90°-direction: GT.0.0: constant value LT.0.0: load curve or table ID = |R90| which defines R_{90} as a function of plastic strain (curve) or as a function of temperature and plastic strain (table). See Remarks.
LCID	Load curve/table ID for hardening in the 0°-direction. See Remarks.
E0	Material parameter: HR.EQ.2.0: ε_0 for determining initial yield stress for Swift exponential hardening (default = 0.0) HR.EQ.4.0: b, coefficient for Voce exponential hardening HR.EQ.5.0: ε_0 for determining initial yield stress for Gosh exponential hardening (default = 0.0)

VARIABLE	DESCRIPTION
	HR.EQ.6.0: b, coefficient for Hocket-Sherby exponential hardening
SPI	Case I: If HR = 2.0 and E0 is zero, then ε_0 is determined by: EQ.0.0: $\varepsilon_0 = \left(\frac{E}{k}\right)^{\left[1/(n-1)\right]}$, default LE.0.02: $\varepsilon_0 = \mathrm{SPI}$ GT.0.02: $\varepsilon_0 = \left(\frac{\mathrm{SPI}}{k}\right)^{\left[1/n\right]}$ Case II: If HR = 5.0, then the strain at plastic yield is determined by an iterative procedure based on the same principles as for HR = 2.0.
P3	Material parameter: HR.EQ.5.0: p, parameter for Gosh exponential hardening HR.EQ.6.0: n, exponent for Hocket-Sherby exponential hardening

BARLAT89 Parameters Card. This card is included if PB > 0 (see Card 4a/4b).

Card 2b	1	2	3	4	5	6	7	8
Variable	M	AB	CB	HB	LCID	E0	SPI	P3
Type	F	F	F	F	I	F	F	F

VARIABLE	DESCRIPTION
M	m, exponent in Barlat's yield surface. If negative, the absolute value is used.
AB	a, Barlat89 parameter
CB	c, Barlat89 parameter
HB	h, Barlat89 parameter
LCID	Load curve/table ID for hardening in the 0°-direction. See Remarks.

VARIABLE	DESCRIPTION
E0	Material parameter: HR.EQ.2.0: ε_0 for determining initial yield stress for Swift exponential hardening (default = 0.0) HR.EQ.4.0: b, coefficient for Voce exponential hardening HR.EQ.5.0: ε_0 for determining initial yield stress for Gosh exponential hardening (default = 0.0) HR.EQ.6.0: b, coefficient for Hocket-Sherby exponential hardening
SPI	Case I: If HR = 2.0 and E0 is zero, then ε_0 is determined by: EQ.0.0: $\varepsilon_0 = \left(\frac{E}{k}\right)^{\left[1/(n-1)\right]}$, default LE.0.02: $\varepsilon_0 = \mathrm{SPI}$ GT.0.02: $\varepsilon_0 = \left(\frac{\mathrm{SPI}}{k}\right)^{\left[1/n\right]}$ Case II: If HR = 5.0, then the strain at plastic yield is determined by an iterative procedure based on the same principles as for HR = 2.0.
P3	Material parameter: HR.EQ.5.0: p, parameter for Gosh exponential hardening HR.EQ.6.0: n, exponent for Hocket-Sherby exponential hardening

Define the following card if and only if M < 0

Card 3	1	2	3	4	5	6	7	8
Variable	CRC1	CRA1	CRC2	CRA2	CRC3	CRA3	CRC4	CRA4
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
CRCn	Chaboche-Rousselier hardening parameters; see Remarks.
CRAn	Chaboche-Rousselier hardening parameters; see Remarks.

This card is included if the keyword option is not used (<BLANK>)

Card 4a	1	2	3	4	5	6	7	8
Variable	AOPT	C	P	VLCID		PB	HTA	HTB
Type	F	F	F	I		F	F	F

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 and then rotated about the shell element normal by an angle BETA. 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. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available with the R3 release of Version 971 and later.
C	C in Cowper-Symonds strain rate model
P	p in Cowper-Symonds strain rate model. Set P to zero for no strain rate effects.
VLCID	Volume correction curve ID defining the relative volume change (change in volume relative to the initial volume) as a function of the effective plastic strain. This is only used when nonzero. See Remarks.
PB	Barlat89 parameter, p. If PB > 0, parameters AB, CB, and HB are read instead of R00, R45, and R90. See Remarks below.

VARIABLE	DESCRIPTION
HTA	Load curve/Table ID for postforming parameter *a* in heat treatment
HTB	Load curve/Table ID for postforming parameter *b* in heat treatment

This card is included if the keyword option is NLP.

Card 4b	1	2	3	4	5	6	7	8
Variable	AOPT	C	P	VLCID		PB	NLP	
Type	F	F	F	I		F	I	

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 and then rotated about the shell element normal by an angle BETA. 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. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available with the R3 release of Version 971 and later.
C	*C* in Cowper-Symonds strain rate model
P	*p* in Cowper-Symonds strain rate model. Set P to zero for no strain rate effects.

VARIABLE	DESCRIPTION
VLCID	Volume correction curve ID defining the relative volume change (change in volume relative to the initial volume) as a function of the effective plastic strain. This is only used when nonzero. See Remarks.
PB	Barlat89 parameter, p. If PB > 0, parameters AB, CB, and HB are read instead of R00, R45, and R90. See Remarks below.
NLP	ID of a load curve of the Forming Limit Diagram (FLD) under linear strain paths. In the load curve, abscissas represent minor strains while ordinates represent major strains. Define only when option NLP is used. See Remarks.

Card 5	1	2	3	4	5	6	7	8
Variable				A1	A2	A3	HTC	HTD
Type				F	F	F	F	F

VARIABLE	DESCRIPTION
A1, A2, A3	Components of vector **a** for AOPT = 2
HTC	Load curve/table ID for postforming parameter c in heat treatment
HTD	Load curve/table ID for postforming parameter d in heat treatment

Card 6	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	HTFLAG
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
V1, V2, V3	Components of vector **v** for AOPT = 3
D1, D2, D3	Components of vector **d** for AOPT = 2

VARIABLE	DESCRIPTION
BETA	Material angle in degrees for AOPT = 0 and 3. It may be overridden on the element card; see *ELEMENT_SHELL_BETA.
HTFLAG	Heat treatment flag (see Remarks): EQ.0: preforming stage EQ.1: heat treatment stage EQ.2: postforming stage

Optional card.

Card 7	1	2	3	4	5	6	7	8
Variable	USRFAIL	LCBI	LCSH					
Type	F	F	F					

VARIABLE	DESCRIPTION
USRFAIL	User defined failure flag: EQ.0: no user subroutine is called. EQ.1: user subroutine matusr_24 in dyn21.f is called.
LCBI	HR.EQ.7: load curve defining biaxial stress as a function of biaxial strain for hardening rule; see discussion in the formulation section below for a definition. HR.NE.7: ignored
LCSH	HR.EQ.7: load curve defining shear stress as a function of shear strain for hardening; see discussion in the formulation section below for a definition. HR.NE.7: ignored

Formulation:

The effective plastic strain used in this model is defined to be plastic work equivalent. A consequence of this is that for parameters defined as functions of effective plastic strain, the rolling (00) direction should be used as reference direction. For instance, the hardening curve for HR = 3 is the stress as function of strain for uniaxial tension in the rolling direction, VLCID curve should give the relative volume change as function of strain for uniaxial tension in the rolling direction and load curve given by $-E$ should give the Young's modulus as function of strain for uniaxial tension in the rolling direction. Optionally, the curve can be substituted for a table defining hardening as function of plastic strain rate (HR = 3), temperature (HR = 8), or pre-strain (HR = 10).

Exceptions from the rule above are curves defined as functions of plastic strain in the 45° and 90° directions, i.e., $P1$ and $P2$ for HR = 7 and negative R45 or R90, see Fleischer et.al. [2007]. The hardening curves are here defined as measured stress as function of measured plastic strain for uniaxial tension in the direction of interest, i.e., as determined from experimental testing using a standard procedure. The optional biaxial and shear hardening curves require some further elaboration, as we assume that a biaxial or shear test reveals that the true stress tensor in the material system expressed as

$$\boldsymbol{\sigma} = \begin{pmatrix} \sigma & 0 \\ 0 & \pm\sigma \end{pmatrix}, \qquad \sigma \geq 0,$$

is a function of the (plastic) strain tensor

$$\boldsymbol{\varepsilon} = \begin{pmatrix} \varepsilon_1 & 0 \\ 0 & \pm\varepsilon_2 \end{pmatrix}, \qquad \varepsilon_1 \geq 0, \qquad \varepsilon_2 \geq 0,$$

The input hardening curves are σ as function of $\varepsilon_1+\varepsilon_2$. The $\pm$ sign above distinguishes between the biaxial (+) and the shear (−) cases. Moreover, the curves defining the R-values are as function of the measured plastic strain for uniaxial tension in the direction of interest. These curves are transformed internally to be used with the effective stress and strain properties in the actual model. The effective plastic strain does not coincide with the plastic strain components in other directions than the rolling direction and may be somewhat confusing to the user. Therefore the von Mises work equivalent plastic strain is output as history variable #2 if HR = 7 or if any of the R-values is defined as function of the plastic strain.

The R-values in curves are defined as the ratio of instantaneous width change to instantaneous thickness change. That is, assume that the width W and thickness T are measured as function of strain. Then the corresponding R-values is given by:

$$R = \frac{\frac{dW}{d\varepsilon}/W}{\frac{dT}{d\varepsilon}/T}$$

The anisotropic yield criterion Φ for plane stress is defined as:

$$\Phi = a|K_1 + K_2|^m + a|K_1 - K_2|^m + c|2K_2|^m = 2\sigma_Y^m$$

where σ_Y is the yield stress and $K_{i=1,2}$ are given by:

$$K_1 = \frac{\sigma_x + h\sigma_y}{2}$$

$$K_2 = \sqrt{\left(\frac{\sigma_x - h\sigma_y}{2}\right)^2 + p^2\tau_{xy}^2}$$

If PB = 0, the anisotropic material constants $a, c, h \; and \; p$ are obtained through $R_{00}, R_{45} \; and \; R_{90}$:

$$\begin{aligned} a &= 2 - 2\sqrt{\left(\frac{R_{00}}{1 + R_{00}}\right)\left(\frac{R_{90}}{1 + R_{90}}\right)} \\ c &= 2 - a \\ h &= \sqrt{\left(\frac{R_{00}}{1 + R_{00}}\right)\left(\frac{1 + R_{90}}{R_{90}}\right)} \end{aligned}$$

The anisotropy parameter p is calculated implicitly. According to Barlat and Lian the R-value, width to thickness strain ratio, for any angle ϕ can be calculated from:

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

where σ_ϕ is the uniaxial tension in the ϕ direction. This expression can be used to iteratively calculate the value of p. Let $\phi = 45$ and define a function g as:

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

An iterative search is used to find the value of p. If PB > 0, material parameters a(AB), c(CB), h(HB), and p(PB) are used directly.

The effective stress, given as

$$\sigma_{\mathrm{eff}} = \left\{\frac{1}{2}\left(a|K_1 + K_2|^m + a|K_1 - K_2|^m + c|2K_2|^m\right)\right\}^{1/m}$$

can be output to the D3plot database through *DEFINE_MATERIAL_HISTORIES.

**DEFINE_MATERIAL_HISTORIES Properties*

Label	Attributes				Description
Effective Stress	-	-	-	-	Effective stress σ_{eff}, see above

For face centered cubic (FCC) materials $m = 8$ is recommended and for body centered cubic (BCC) materials $m = 6$ may be used. The yield strength of the material can be expressed in terms of k and n:

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

where ε_{yp} is the elastic strain to yield and $\bar{\varepsilon}^p$is the effective plastic strain (logarithmic). If SIGY is set to zero, the strain to yield 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:

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

If SIGY yield is nonzero and greater than 0.02 then:

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

The other available hardening models include the Voce equation given by:

$$\sigma_{\mathrm{Y}}(\varepsilon_p) = a - be^{-c\varepsilon_p},$$

the Gosh equation given by:

$$\sigma_{\mathrm{Y}}(\varepsilon_p) = k(\varepsilon_0 + \varepsilon_p)^n - p,$$

and finally the Hocket-Sherby equation given by:

$$\sigma_{\mathrm{Y}}(\varepsilon_p) = a - be^{-c\varepsilon_p^n}.$$

For the Gosh hardening law, the interpretation of the variable SPI is the same, i.e., if set to zero the strain at yield is determined implicitly from the intersection of the strain hardening equation with the linear elastic equation.

To include strain rate effects in the model we multiply the yield stress by a factor depending on the effective plastic strain rate. We use the Cowper-Symonds' model; hence the yield stress can be written as:

$$\sigma_{\mathrm{Y}}(\varepsilon_p, \dot{\varepsilon}_p) = \sigma_{\mathrm{Y}}^s(\varepsilon_p)\left\{1 + \left(\frac{\dot{\varepsilon}_p}{C}\right)^{1/p}\right\}$$

where σ_{Y}^s denotes the static yield stress, C and p are material parameters, and $\dot{\varepsilon}_p$ is the effective plastic strain rate. With HR.EQ.3 strain rate effects can be defined using a table, in which each load curve in the table defines the yield stress as function of plastic strain for a given strain rate. In contrast to material 24, when the strain rate is larger than that of any curve in the table, the table is extrapolated in the strain rate direction to find the appropriate yield stress.

A kinematic hardening model is implemented following the works of Chaboche and Roussilier. A back stress $\boldsymbol{\alpha}$ is introduced such that the effective stress is computed as:

$$\sigma_{\mathrm{eff}} = \sigma_{\mathrm{eff}}(\sigma_{11} - 2\alpha_{11} - \alpha_{22}, \sigma_{22} - 2\alpha_{22} - \alpha_{11}, \sigma_{12} - \alpha_{12})$$

The back stress is the sum of up to four terms according to:

$$\alpha_{ij} = \sum_{k=1}^{4} \alpha_{ij}^{k}$$

and the evolution of each back stress component is as follows:

$$\delta\alpha_{ij}^{k} = C_k \left(a_k \frac{s_{ij} - \alpha_{ij}}{\sigma_{\text{eff}}} - \alpha_{ij}^{k} \right) \delta\varepsilon_p$$

where C_k and a_k are material parameters, s_{ij} is the deviatoric stress tensor, σ_{eff} is the effective stress and ε_p is the effective plastic strain. The yield condition is for this case modified according to

$$f(\sigma,\alpha,\varepsilon_p) = \sigma_{\text{eff}}(\sigma_{11} - 2\alpha_{11} - \alpha_{22}, \sigma_{22} - 2\alpha_{22} - \alpha_{11}, \sigma_{12} - \alpha_{12}) \\ - \left\{ \sigma_Y^t(\varepsilon_p, \dot{\varepsilon}_p, 0) - \sum_{k=1}^{4} a_k \left[1 - \exp(-C_k \varepsilon_p \ \right] \right\} \leq 0$$

in order to get the expected stress strain response for uniaxial stress. The calculated effective stress is stored in history variable #7.

With hardening rule HR = 10, the flow curves in a table definition can be based on different pre-strain values. Hence flow curves can have varying shapes as defined in the corresponding table. For example, the plastic strain distribution as obtained in a first step of a two-stage procedure is initialized in the next stage with *INITIAL_STRESS_SHELL and corresponding values for EPS. With HR = 10 this pre-strain is initially transferred to history variable #9 and all stresses and other history variables are set to zero assuming that the part was subjected to an annealing phase. With EPS now stored on history variable #9 the table lookup for the actual yield value may now be used to interpolate on differently shaped flow curves.

A failure criterion for nonlinear strain paths (NLP) in sheet metal forming:

When the option NLP is used, a necking failure criterion is activated to account for the non-linear strain path effect in sheet metal forming. Based on the traditional Forming Limit Diagram (FLD) for the linear strain path, the Formability Index (F.I.) is calculated for every element in the model throughout the simulation duration. The entire F.I. time history for every element is stored in history variable #1 in d3plot files, accessible from *Post/History* menu in *LS-PrePost* v4.0. In addition to the F.I. output, other useful information stored in other history variables can be found as follows,

1. Formability Index: #1

2. Strain ratio (in-plane minor strain increment/major strain increment): #2

3. Effective strain from the planar isotropic assumption: #3

To enable the output of these history variables to the d3plot files, NEIPS on the *DATABASE_EXTENT_BINARY card must be set to at least 3. The history variables can also be plotted on the formed sheet blank as a color contour map, accessible from *Post/FriComp/Misc* menu. The index value starts from 0.0, with the onset of necking failure when it reaches 1.0. The F.I. is calculated based on critical effect strain method, as explained in manual pages in *MAT_037. The theoretical background based on two papers can also be found in manual pages in *MAT_037.

When d3plot files are used to plot the history variable #1 (the F.I.) in color contour, the value in the *Max* pull-down menu in *Post/FriComp* needs to be set to *Min*, meaning that the necking failure occurs only when all integration points through the thickness have reached the critical value of 1.0 (refer to *Tharrett and Stoughton's paper in 2003 SAE 2003-01-1157*). It is also suggested to set the variable "MAXINT" in *DATABASE_EXTENT_BINARY to the same value as the variable "NIP" in *SECTION_SHELL. In addition, the value in the *Avg* pull-down menu in *Post/FriRang* needs to be set to *None*. The strain path history (major vs. minor strain) of each element can be plotted with the radial dial button *Strain Path* in *Post/FLD*.

An example of a partial input for the material is provided below, where the FLD for the linear strain path is defined by the variable NLP with load curve ID 211, where abscissas represent minor strains and ordinates represent major strains.

```
*MAT_3-PARAMETER_BARLAT_NLP
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$      MID        RO         E        PR        HR        P1        P2      ITER
         1 2.890E-09  6.900E04     0.330     3.000
$        M       R00       R45       R90      LCID        E0       SPI        P3
     8.000     0.800     0.600     0.550        99
$     AOPT         C         P     VLCID                           NLP
     2.000                                                         211
$                                     A1        A2        A3
                                   0.000     1.000     0.000
$       V1        V2        V3        D1        D2        D3      BETA

$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$ Hardening Curve
*DEFINE_CURVE
        99
              0.000                130.000
              0.002                134.400
              0.006                143.000
              0.010                151.300
              0.014                159.300
                ⋮                     ⋮
              0.900                365.000
              1.000                365.000
$ FLD Definition
*DEFINE_CURVE
211
               -0.2                  0.325
            -0.1054                 0.2955
            -0.0513                 0.2585
             0.0000                 0.2054
             0.0488                 0.2240
             0.0953                 0.2396
             0.1398                 0.2523
```

```
            0.1823              0.2622
               ⋮                   ⋮
```

Shown in Figures M36-1, M36-2 and M36-3, predictions and validations of forming limit curves (FLC) of various nonlinear strain paths on a single shell element was done using this new option, for an Aluminum alloy with $R_{00} = 0.8, R_{45} = 0.6\ and\ R_{90} = 0.55$ and the yield at 130.0 MPa. In each case, the element is further strained in three different paths (uniaxial stress – U.A., plane strain – P.S., and equi-biaxial strain – E.B.) separately, following a pre-straining in uniaxial, plane strain and equi-biaxial strain state, respectively. The forming limits are determined at the end of the secondary straining for each path, when the F.I. has reached the value of 1.0. It is seen that the predicted FLCs (dashed curves) in case of the nonlinear strain paths are totally different from the FLCs under the linear strain paths. It is noted that the current predicted FLCs under nonlinear strain path are obtained by connecting the ends of the three distinctive strain paths. More detailed FLCs can be obtained by straining the elements in more paths between U.A. and P.S. and between P.S. and E.B. In Figure M36-4, time-history plots of F.I., strain ratio and effective strain are shown for uniaxial pre-strain followed by equi-biaxial strain path on the same single element.

Typically, to assess sheet formability, F.I. contour of the entire part should be plotted. Based on the contour plot, non-linear strain path and the F.I. time history of a few elements in the area of concern can be plotted for further study. These plots are similar to those shown in manual pages of *MAT_037.

Smoothing of the strain ratio β:

*CONTROL_FORMING_STRAIN_RATIO_SMOOTH applies a smoothing algorithm to reduce output noise level of the strain ratio β (in-plane minor strain increment/major strain increment) which is used to calculate the Formability Index.

Support of non-integer flow potential exponent m:

Starting in Dev139482, non-integer value of the exponent m is supported for the option NLP.

Heat treatment with variable HTFLAG:

Heat treatment for increasing the formability of prestrained aluminum sheets can be simulated through the use of HTFLAG, where the intention is to run a forming simulation in steps involving preforming, springback, heat treatment and postforming. In each step the history is transferred to the next via the use of dynain (see *INTERFACE_SPRINGBACK). The first two steps are performed with HTFLAG = 0 according to standard procedures, resulting in a plastic strain field ε_p^0corresponding to the prestrain. The heat treatment step is performed using HTFLAG = 1 in a coupled thermomechanical simulation, where the blank is heated. The coupling between thermal and mechanical is only that the maximum temperature T^0 is stored as a history variable in the material model, this corresponding to the heat treatment temperature. Here it is important to export all history variables to the dynein file for the postforming step. In the final postforming step, HTFLAG = 2, the yield stress is then augmented by the Hocket-Sherby like term:

$$\Delta\sigma = b - (b - a)\exp\left[-c\left(\varepsilon_p - \varepsilon_p^0\right)^d\right]$$

where $a, b, c\ and\ d$ are given as tables as functions of the heat treatment temperature T^0 and prestrain ε_p^0. That is, in the table definitions each load curve corresponds to a given prestrain and the load curve value is with respect to the heat treatment temperature,

$$a = a(T^0, \varepsilon_p^0) \quad b = b(T^0, \varepsilon_p^0) \quad c = c(T^0, \varepsilon_p^0) \quad d = d(T^0, \varepsilon_p^0)$$

The effect of heat treatment is that the material strength decreases but hardening increases, thus typically:

$$a \le 0 \quad b \ge a \quad c > 0 \quad d > 0$$

Revision information:

The option NLP is available starting in Dev 95576 in explicit dynamic analysis, and in SMP and MPP.

1. Smoothing of β is available starting in Revision 109781.
2. Dev139482: non-integer value of the exponent m is supported for the option NLP.

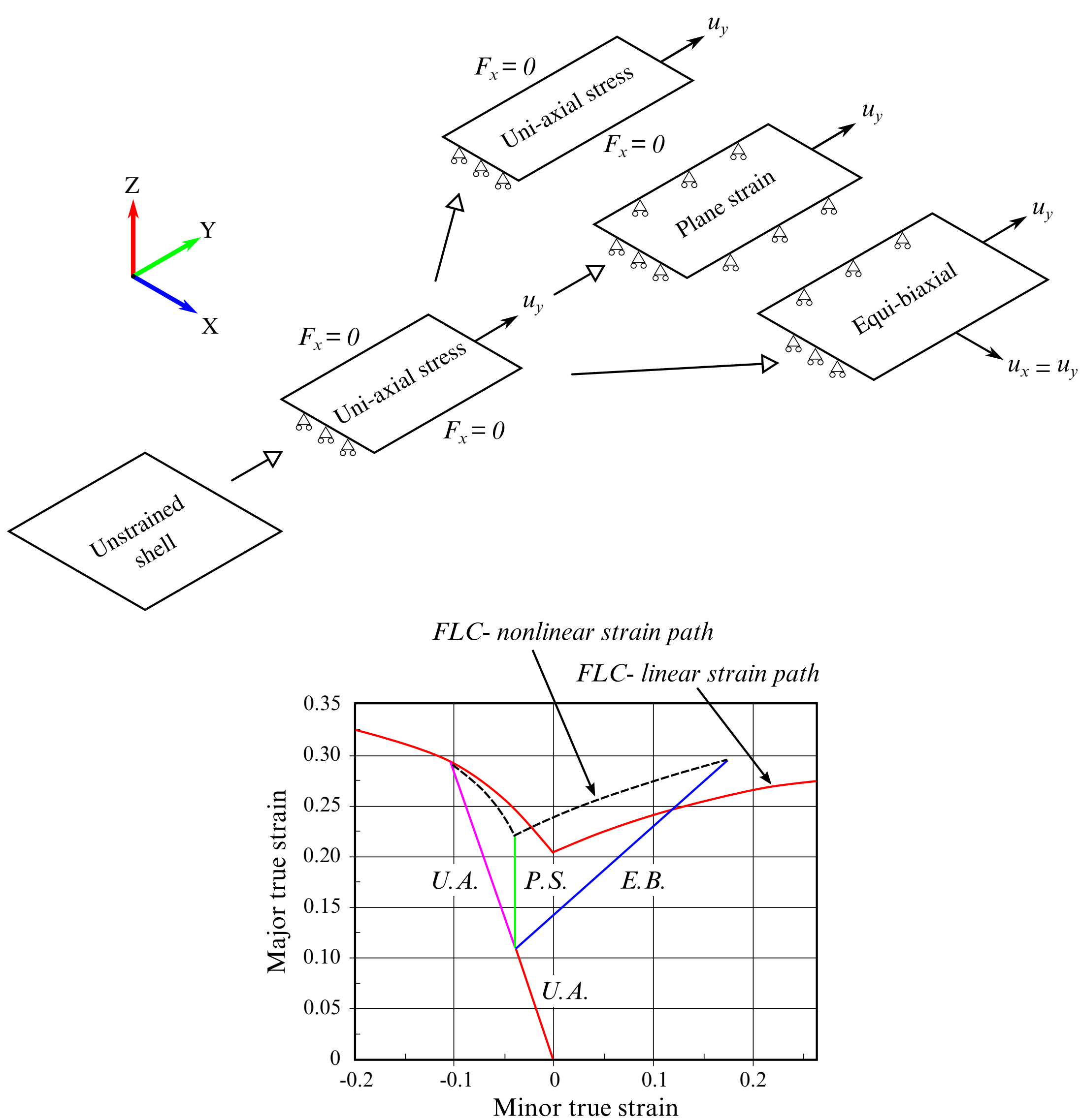


Figure M36-1. Nonlinear FLD prediction with uniaxial pre-straining.

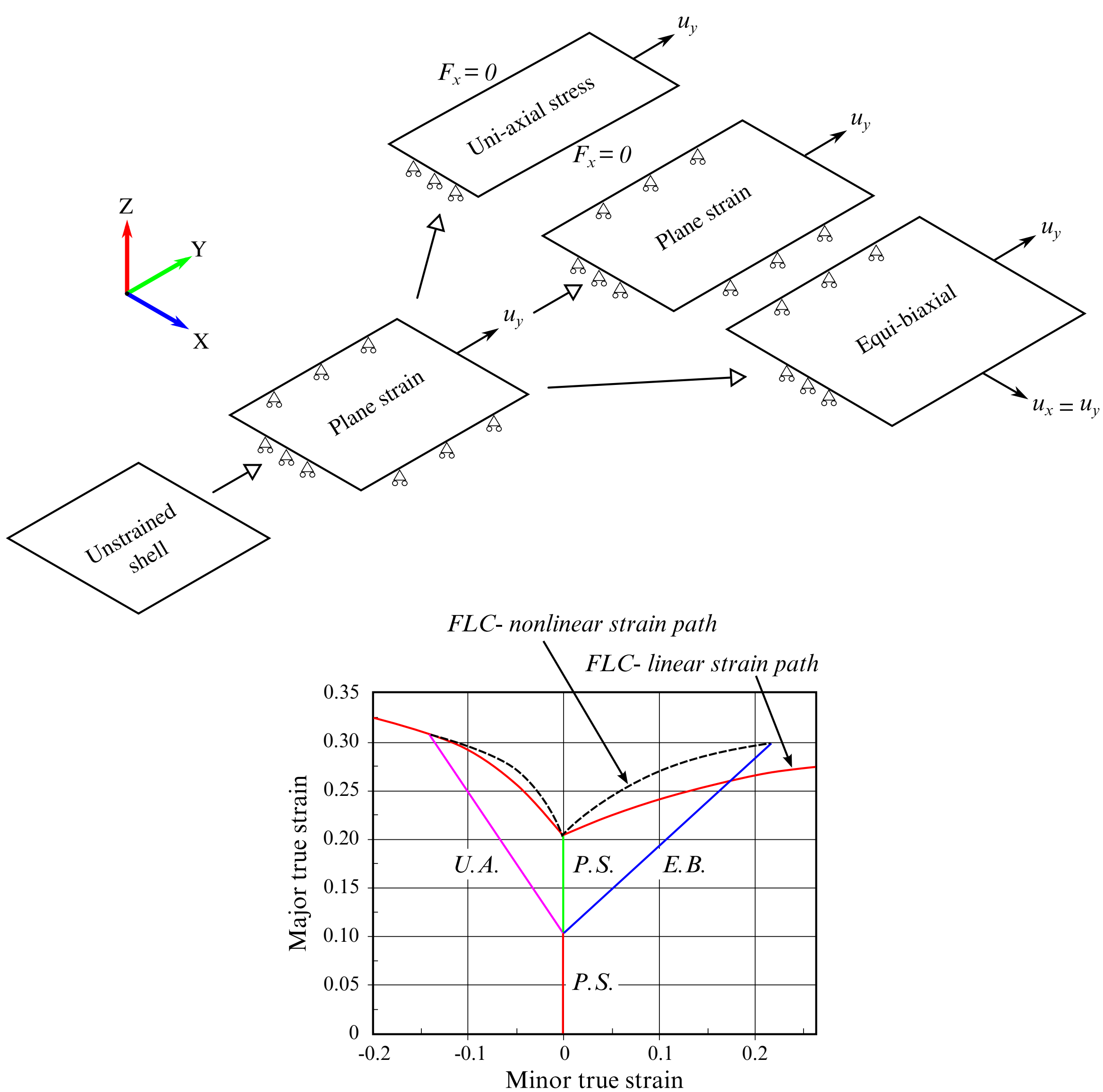


Figure M36-2. Nonlinear FLD prediction with plane strain pre-straining.

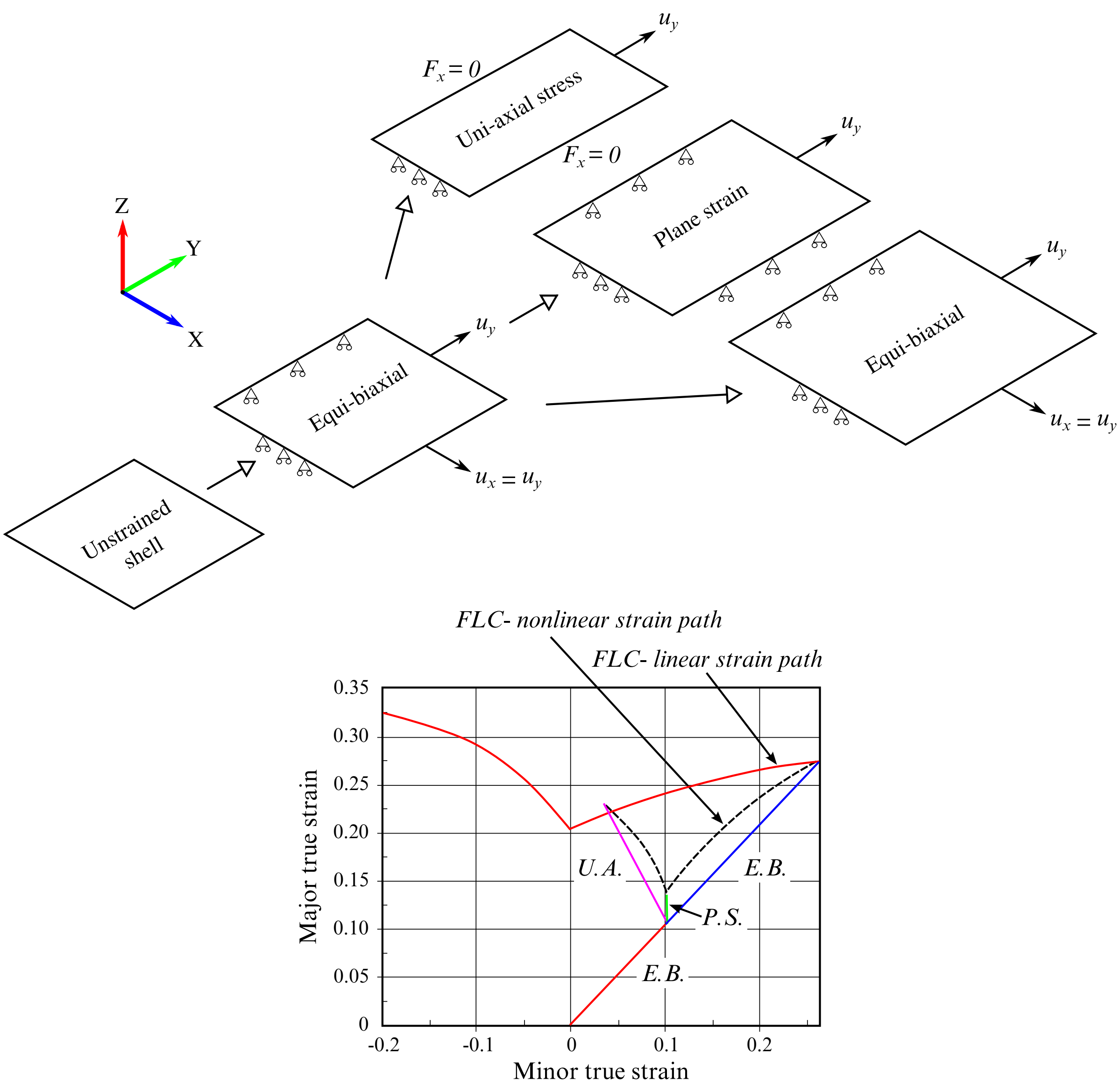


Figure M36-3. Nonlinear FLD prediction with equi-biaxial pre-straining.

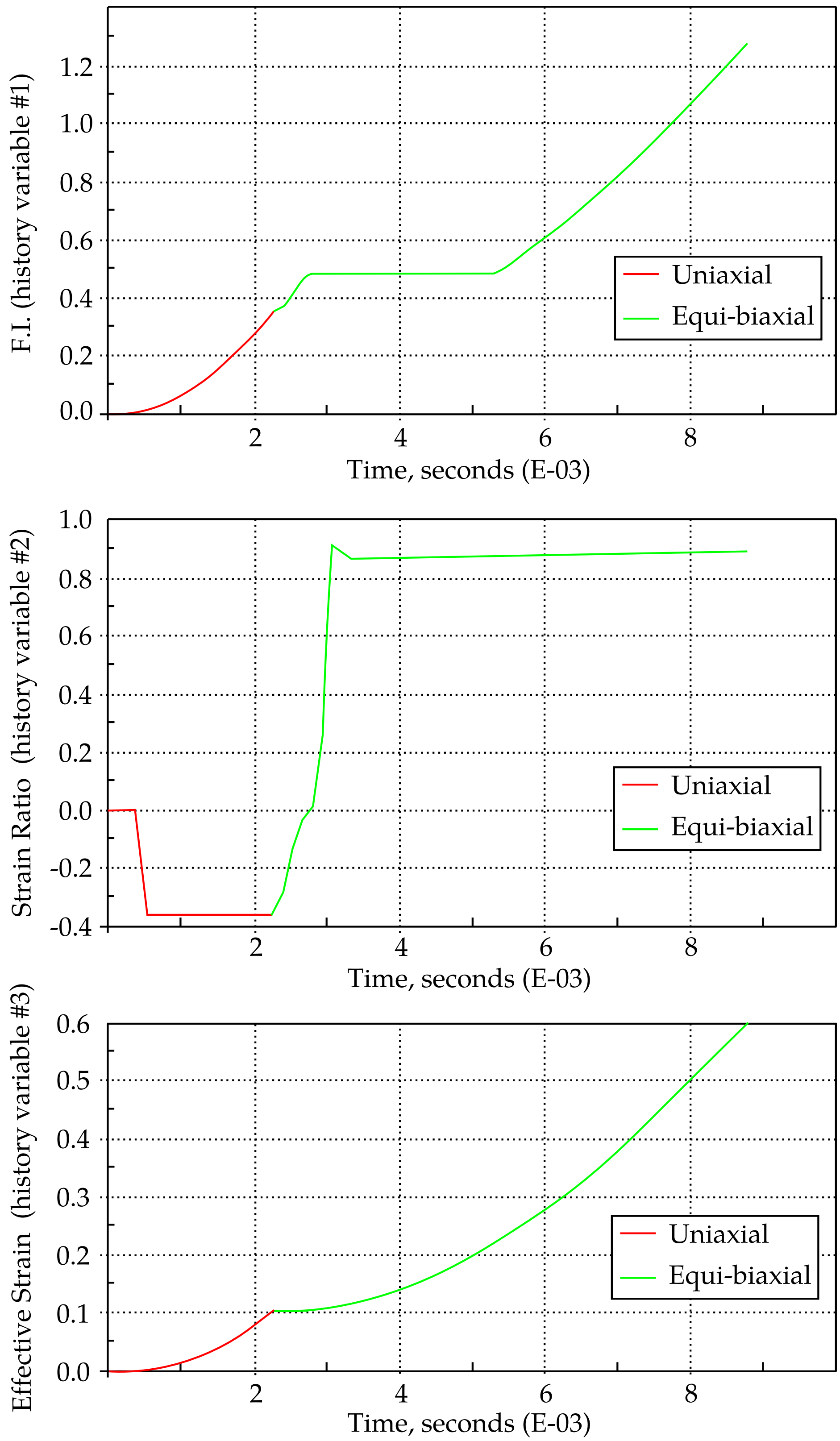


Figure M36-4. Time-history plots of the three history variables.

*MAT_EXTENDED_3-PARAMETER_BARLAT

This is Material Type 36E. This model is an extension to the standard 3-parameter Barlat model and allows for different hardening curves and R-values in different directions, see Fleischer et.al. [2007]. The directions in this context are the three uniaxial directions (0, 45 and 90 degrees) and optionally biaxial and shear.

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

Card 2	1	2	3	4	5	6	7	8
Variable	LCH00	LCH45	LCH90	LCHBI	LCHSH	HOSF		
Type	F	F	F	F	F	F		

Card 3	1	2	3	4	5	6	7	8
Variable	LCR00	LCR45	LCR90	LCRBI	LCRSH	M		
Type	F	F	F	F	F	F		

Card 4	1	2	3	4	5	6	7	8
Variable	AOPT							
Type	F							

Card 5	1	2	3	4	5	6	7	8
Variable				A1	A2	A3		
Type				F	F	F		

Card 6	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 or label must be specified (see *PART).
RO	Mass density.
E	Young's modulus, E.
PR	Poisson's ratio, ν.
LCH*XX*	Load curve/table defining uniaxial stress vs. uniaxial strain and strain rate in the given direction (*XX* is either 00, 45, 90). The exact definition is discussed in the Remarks below. LCH00 must be defined, the other defaults to LCH00 if not defined.
LCHBI	Load curve/table defining biaxial stress vs. biaxial strain and strain rate, see discussion in the Remarks below for a definition. If not defined this is determined from LCH00 and the initial R-values to yield a response close to the standard 3-parameter Barlat model.
LCHSH	Load curve/table defining shear stress vs. shear strain and strain rate, see discussion in the Remarks below for a definition. If not defined this is ignored to yield a response close to the standard 3-parameter Barlat model.
HOSF	Hosford option for enhancing convexity of yield surface, set to 1 to activate.
LCR*XX*	Load curve defining standard R-value vs. uniaxial strain in the given direction (*XX* is either 00, 45, 90). The exact definition is discussed in the Remarks below. Default is a constant R-value of 1.0, a negative input will result in a constant R-value of –LCRXX.

VARIABLE	DESCRIPTION
LCRBI	Load curve defining biaxial R-value vs. biaxial strain, see discussion in the Remarks below for a definition. Default is a constant R-value of 1.0, a negative input will result in a constant R-value of –LCRBI.
LCRSH	Load curve defining shear R-value vs. shear strain, see discussion in the Remarks below for a definition. Default is a constant R-value of 1.0, a negative input will result in a constant R-value of –LCRSH.
M	Barlat flow exponent, m, must be an integer value.
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, and then rotated about the shell element normal by an angle BETA. 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. LT.0.0: the absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_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.
BETA	Material angle in degrees for AOPT = 0 and 3, may be overridden on the element card, see *ELEMENT_SHELL_BETA.

Formulation:

The standard 3-parameter Barlat model incorporates plastic anisotropy in a fairly moderate sense, allowing for the specification of R-values in three different directions, together with a stress level in the reference direction (termed *rolling* or *0 degree* direction), but not more than that. To allow for a more accurate representation of a more severe anisotropic material, like in rolled aluminium sheet components, one could migrate to the Barlat YLD2000 model (material 133 in LS-DYNA) which also allows for specifying stress levels in the two remaining directions as well as stress and strain data at an arbitrary point on the yield surface. The properties of extruded aluminium however, are such that neither of these two material models are sufficient to describe its extreme anisotropy. One particular observation from experiments is that anisotropy evolves with deformation, a feature that is not captured in any of the material models discussed so far. The present extended version of material 36 was therefore developed in an attempt to fill this void in the LS-DYNA material library. In short, this material allows for R-values and stress levels in the three directions, together with similar data in biaxial and shear directions. *And*, these properties are functions of the effective plastic strain so as to allow for deformation induced anisotropy. The following is an explanation of its parameters.

The hardening curves or tables LCH00, LCH45 and LCH90 are here defined as measured stress as function of measured plastic strain (and potentially rate) for uniaxial tension in the direction of interest, i.e., as determined from experimental testing using a standard procedure. The optional biaxial and shear hardening curves LCHBI and LCHSH require some further elaboration, as we assume that a biaxial or shear test reveals that the true stress tensor in the material system expressed as

$$\boldsymbol{\sigma} = \begin{pmatrix} \sigma & 0 \\ 0 & \pm\sigma \end{pmatrix}, \qquad \sigma \geq 0,$$

is a function of the (plastic) strain tensor

$$\boldsymbol{\varepsilon} = \begin{pmatrix} \varepsilon_1 & 0 \\ 0 & \pm\varepsilon_2 \end{pmatrix}, \qquad \varepsilon_1 \geq 0, \qquad \varepsilon_2 \geq 0,$$

The input hardening curves are σ as function of $\varepsilon_1+\varepsilon_2$. The $\pm$ sign above distinguishes between the biaxial (+) and the shear (−) cases.

Moreover, the curves LCR00, LCR45 and LCR90 defining the R values are as function of the measured plastic strain for uniaxial tension in the direction of interest. The R-values in themselves are defined as the ratio of instantaneous width change to instantaneous thickness change. That is, assume that the width W and thickness T are measured as function of strain. Then the corresponding R-value is given by:

$$R_\varphi = \frac{\frac{dW}{d\varepsilon}/W}{\frac{dT}{d\varepsilon}/T}.$$

These curves are transformed internally to be used with the effective stress and strain properties in the actual model. The effective plastic strain does not coincide with the plastic strain components in other directions than the rolling direction and may be somewhat confusing to the user. Therefore the von Mises work equivalent plastic strain is output as history variable #2. As for hardening, the optional biaxial and shear R-value curves LCRBI and LCRSH are defined in a special way for which we return to the local plastic strain tensor $\boldsymbol{\varepsilon}$ as defined above. The biaxial and shear R-values are defined as

$$R_{b/s} = \frac{\dot{\varepsilon}_1}{\dot{\varepsilon}_2}$$

and again the curves are $R_{b/s}$ as function of $\varepsilon_1 + \varepsilon_2$. Note here that the suffix b assumes loading biaxially and s assumes loading in shear, so the R-values to be defined are always positive.

The option HOSF = 0 is equivalent to the standard Barlat model with HR = 7 whose yield function can be expressed by the potential Φ as given in the remarks for *MAT_3-PARAMETER_BARLAT. The HOSF = 1 allows for a "Hosford-based" effective stress in the yield function instead of using the Barlat-based effective stress. If the material and principal axes are coincident, the plastic potential Φ for HOSF = 1 can be written as

$$\Phi(\sigma) = \frac{1}{2}\left(|\sigma_1|^m + |\sigma_2|^m + |\sigma_1 - \sigma_2|^m\right) - \sigma_y^m$$

The main difference is that the Barlat-based effective stress contains the orthotropic parameters a, c, h and p in the yield function meanwhile the Hosford-based effective stress does not contain any information about the anisotropy. For HOSF = 1, the information about direction dependent yielding is directly obtained from the hardening curves LCH00, LCH45 and LCH90. For materials exhibiting very dissimilar R −values in the different material directions (e.g. typical aluminum extrusion), HOSF = 0 might (but does not necessarily) lead to concave yield surfaces which, in turn, might lead to numerical instabilities under certain circumstances. HOSF = 1 tends to reduce this effect.

More information on the theoretical and numerical foundations of HOSF = 1 can be found on the paper by Andrade, Borrvall, DuBois and Feucht, *A Hosford-based orthotropic plasticity model in LS-DYNA* (2019).

MAT_TRANSVERSELY_ANISOTROPIC_ELASTIC_PLASTIC_*{OPTION}

This is Material Type 37. This model is for simulating sheet forming processes with an anisotropic material. This model only considers transverse anisotropy. Optionally, a load curve can specify an arbitrary dependency of stress and effective plastic strain. This plasticity model is fully iterative and is available only for shell elements.

Available options include:

<BLANK>

ECHANGE

NLP_FAILURE

NLP2

The ECHANGE option allows the Young's Modulus to change during the simulation. See Remark 4.

The NLP_FAILURE option estimates failure using the Formability Index (F.I.) which accounts for the nonlinear strain paths common in metal forming applications (see Remarks 5 and 7). The option NLP is also available for *MAT_036, *MAT_125, and *MAT_226. A related keyword is *CONTROL_FORMING_STRAIN_RATIO_SMOOTH, which applies a smoothing algorithm to reduce the noise level of the strain ratio β (in-plane minor strain increment/major strain increment) when calculating the F.I.

The NLP_FAILURE option uses effective plastic strain to calculate the onset of necking, whichassumes the necking happens in an instant. However, necking may occur over a longer duration. We developed the keyword option NLP2 to address this issue. NLP2 calculates the damage during forming and accumulates it to predict the sheet metal failure. History variable #1 when output to d3plot gives this accumulated damage.

Card Summary:

Card 1. This card is required.

MID	RO	E	PR	SIGY	ETAN	R	HLCID

Card 2a. Include this card for the ECHANGE keyword option.

IDSCALE	EA	COE					

Card 2b. Include this card for the NLP_FAILURE keyword option.

			ICFLD		STRAINLT		

Card 2c. Include this card for the NLP2 keyword option.

			ICFLD				

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Plastic hardening modulus. When this value is negative, normal stresses (either from contact or applied pressure) are considered and *LOAD_SURFACE_STRESS must be used to capture the stresses. This feature is applicable to both shell element types 2 and 16. It is found in some cases this inclusion can improve accuracy. The negative local z-stresses caused by the contact pressure can be viewed from d3plot files.
R	Anisotropic parameter $\bar{r}$, also commonly called r-bar, in sheet metal forming literature. Its interpretation is given in Remark 1. GT.0: Standard formulation LT.0: The anisotropic parameter is set to $\lvert R \rvert$. When R is set to a negative value the algorithm is modified for better stability in sheet thickness or thinning for sheet metal forming involving high strength steels or in cases when the simulation time is long. This feature is available to both element formulations 2 and 16. See Remark 2 and Figure M37-1.

VARIABLE	DESCRIPTION
HLCID	Load curve ID expressing effective yield stress as a function of effective plastic strain in uniaxial tension.

ECHANGE Card. Additional card included if the using the ECHANGE keyword option.

Card 2a	1	2	3	4	5	6	7	8
Variable	IDSCALE	EA	COE					
Type	I	F	F					

VARIABLE	DESCRIPTION
IDSCALE	Load curve ID expressing the scale factor for the Young's modulus as a function of effective plastic strain. If the EA and COE fields are specified, this curve is unnecessary. See Remark 4.
EA, COE	Coefficients defining the Young's modulus with respect to the effective plastic strain, EA is E^A and COE is ζ. If IDSCALE is defined, these two parameters are not necessary. See Remark 4.

NLP_FAILURE Card. Additional card included if using the NLP_FAILURE keyword option.

Card 2b	1	2	3	4	5	6	7	8
Variable				ICFLD		STRAINLT		
Type				F		F		

VARIABLE	DESCRIPTION
ICFLD	ID of a load curve of the Forming Limit Diagram (FLD) under linear strain paths (see Remark 6). In the load curve, abscissas represent minor strains while ordinates represent major strains.
STRAINLT	Critical strain value at which strain averaging is activated. See Remark 8.

NLP2 Card. Additional card included if using NLP2 keyword option.

Card 2	1	2	3	4	5	6	7	8
Variable				ICFLD				
Type				F				

VARIABLE	DESCRIPTION
ICFLD	ID of a load curve of the Forming Limit Diagram (FLD) under linear strain paths (see Remark 6). In the load curve, abscissas represent minor strains while ordinates represent major strains.

Remarks:

1. **Formulation.** 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 as:

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

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:

$$2F=\frac{1}{\sigma_{y2}^2}+\frac{1}{\sigma_{y3}^2}-\frac{1}{\sigma_{y1}^2}$$

$$2G=\frac{1}{\sigma_{y3}^2}+\frac{1}{\sigma_{y1}^2}-\frac{1}{\sigma_{y2}^2}$$

$$2H=\frac{1}{\sigma_{y1}^2}+\frac{1}{\sigma_{y2}^2}-\frac{1}{\sigma_{y3}^2}$$

$$2L=\frac{1}{\sigma_{y23}^2}$$

$$2M=\frac{1}{\sigma_{y31}^2}$$

$$2N=\frac{1}{\sigma_{y12}^2}$$

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:

$$\begin{aligned} 2F = 2G &= \frac{1}{\sigma_{y3}^2} \\ 2H &= \frac{2}{\sigma_y^2} - \frac{1}{\sigma_{y3}^2} \\ N &= \frac{2}{\sigma_y^2} - \frac{1}{2\sigma_{y3}^2} \end{aligned}$$

where it has been assumed that $\sigma_{y1} = \sigma_{y2} = \sigma_y$.

Letting $K = \sigma_y / \sigma_{y3}$, the yield criteria can be written as:

$$F(\sigma) = \sigma_e = \sigma_y \ ,$$

where

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

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

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

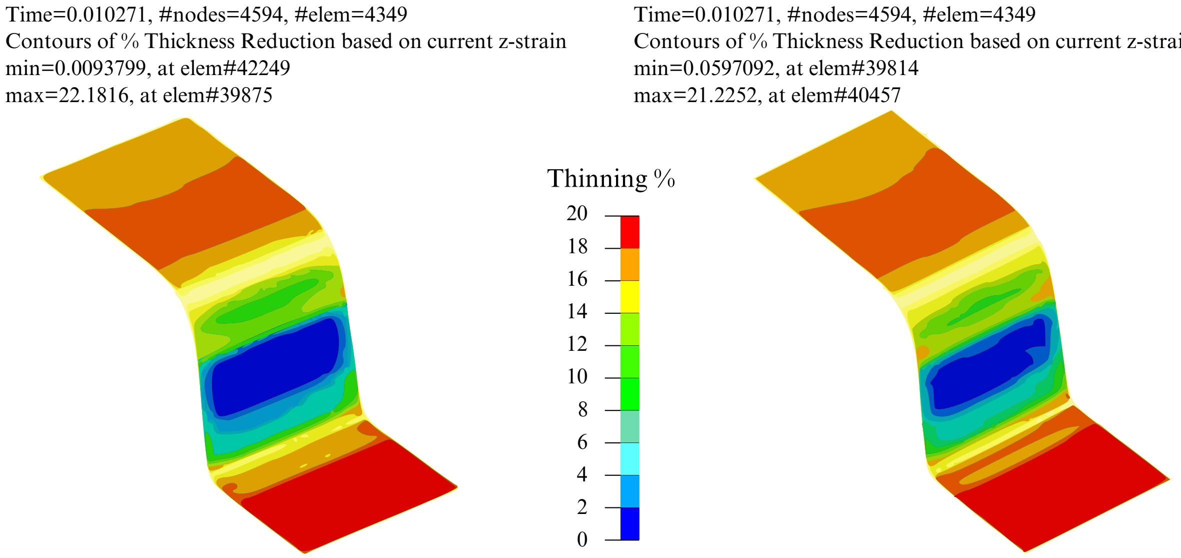
Now consider the case of plane stress, where $\sigma_{33} = 0$. Also, define the anisotropy input parameter, R, as the ratio of the in-plane plastic strain rate to the out-of-plane plastic strain rate,

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

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 as:



With negative R-value *With positive R-value*

Figure M37-1. Thinning contour comparison.

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

2. **Anisotropic Parameter R.** When the R value is set to a negative value, it stabilizes the sheet thickness or thinning in sheet metal forming for some high strength types of steel or in cases where the simulation time is long. In Figure M37-1, a comparison of thinning contours is shown on a U-channel forming (one-half model) using negative and positive R values. Maximum thinning on the draw wall is slight higher in the negative R case than that in the positive R case.

3. **Comparison to other Material Models.** This model and other plasticity models for shell elements, such as *MAT_PIECEWISE_LINEAR_PLASTICITY, differ in several ways. First, the yield function for plane stress does not include the transverse shear stress components which are updated elastically. Secondly, this model is always fully iterative. Consequently, when 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.

4. **ECHANGE.** In the original implementation, we assume that the Young's modulus is constant. However, some researchers have found that the Young's modulus decreases with respect to the increase of effective plastic strain. To accommodate this observation, we added the keyword option ECHANGE.

   We implemented two methods for defining the change of Young's modulus. For the first method, you specify a load curve to define the scale factor of the Young's modulus with respect to the effective plastic strain. The value of this scale factor

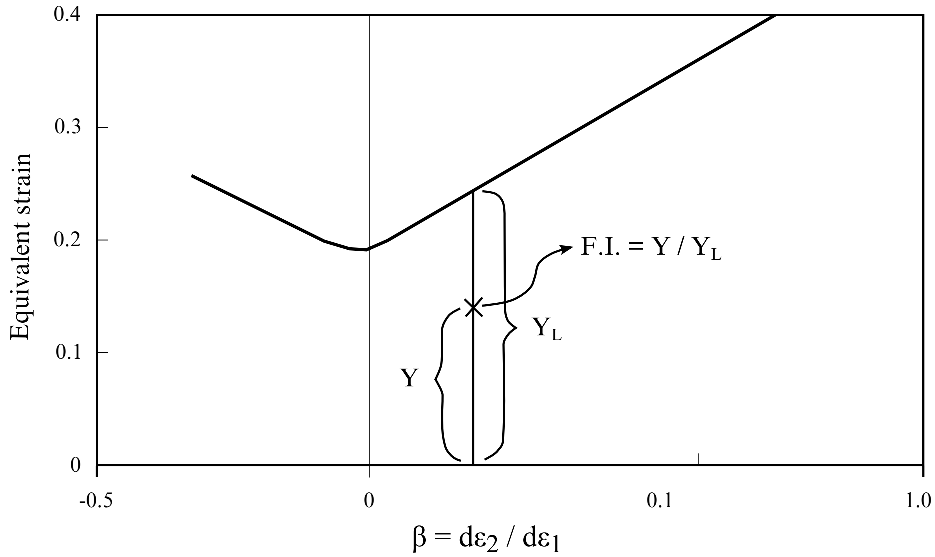


Figure M37-2. Calculation of F.I. based on critical equivalent strain method.

should decrease from 1.0 to 0.0 with the increase of effective plastic strain. The second method uses a function as proposed by Yoshida [2003]:

$$E = E^0 - \left(E^0 - E^A\right)[1 - \exp(-\zeta\bar{\varepsilon})].$$

5. **Nonlinear Strain Paths.** When the keyword option NLP_FAILURE is used, a necking failure criterion independent of strain path changes is activated. In sheet metal forming, as strain path history (plotted on in-plane major and minor strain space) of an element becomes non-linear, the position and shape of a traditional strain-based Forming Limit Diagram (FLD) changes. This option provides a simple formability index (F.I.) which remains invariant regardless of the presence of the non-linear strain paths in the model and can be used to identify if the element has reached its necking limit.

   Formability index (F.I) is calculated, as illustrated in Figure M37-2, for every element in the sheet blank throughout the simulation duration. The value of F.I. is 0.0 for virgin material and reaches maximum of 1.0 when the material fails. The theoretical background can be found in two papers: *1) T.B. Stoughton, X. Zhu, "Review of Theoretical Models of the Strain-Based FLD and their Relevance to the Stress-Based FLD, International Journal of Plasticity", V. 20, Issues 8-9, P. 1463-1486, 2003; and 2) Danielle Zeng, Xinhai Zhu, Laurent B. Chappuis, Z. Cedric Xia, "A Path Independent Forming Limited Criterion for Sheet Metal Forming Simulations", 2008 SAE Proceedings, Detroit MI, April, 2008.*

6. **ICFLD.** The load curve input for ICFLD follows keyword format in *DEFINE_-CURVE, with abscissas as minor strains and ordinates as major strains.

   ICFLD can also be specified using the *DEFINE_CURVE_FLC keyword where the sheet metal thickness and strain hardening value are used. Detailed usage information can be found in the manual entry for *DEFINE_CURVE_FLC.

7. **Formability Index Output.** The formability index is output as a history variable #1 in d3plot files. In addition to the F.I. values, starting in Revision 95599, the strain ratio β and effective plastic strain $\bar{\varepsilon}$ are written to the d3plot database as history variables #2 and #3, respectively provided NEIPS on the second field of the first card of *DATABASE_EXTENT_BINARY is set to at least 3. The contour map of history variables can be plotted in LS-PrePost, accessible in *Post*/*FriComp*, under *Misc*, and by *Element*, under *Post/History*. It is suggested that variable MAXINT in *DATABASE_EXTENT_BINARY be set to the same value of as the NIP field for the *SECTION_SHELL keyword.

8. **STRAINLT.** By setting the STRAINLT field, strains (and strain ratios) can be averaged to reduce noise, which, in turn, affect the calculation of the formability index. The strain STRAINLT causes the formability index calculation to use only time averaged strains. Reasonable STRAINLT values range from 5×10^{-3} to 10^{-2}.

*MAT_BLATZ-KO_FOAM

This is Material Type 38. This model is for the definition of rubber like foams of polyurethane. It is a simple one-parameter model with a fixed Poisson's ratio of .25.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	REF				
Type	A	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
G	Shear modulus
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY. EQ.0.0: off EQ.1.0: on

Remarks:

The strain energy functional for the compressible foam model is given by

$$W = \frac{G}{2}\left(\frac{\mathrm{II}}{\mathrm{III}} + 2\sqrt{\mathrm{III}} - 5\right) \ .$$

Blatz and Ko [1962] suggested this form for a 47 percent volume polyurethane foam rubber with a Poisson's ratio of 0.25. In terms of the strain invariants, I, II, and III, the second Piola-Kirchhoff stresses are given as

$$S^{ij} = G\left[\left(I\delta_{ij} - C_{ij}\right)\frac{1}{\mathrm{III}} + \left(\sqrt{\mathrm{III}} - \frac{\mathrm{II}}{\mathrm{III}}\right)C_{ij}^{-1}\right] \ ,$$

where C_{ij} is the right Cauchy-Green strain tensor. This stress measure is transformed to the Cauchy stress, σ_{ij}, according to the relationship

$$\sigma^{ij} = \mathrm{III}^{-1/2} F_{ik} F_{jl} S_{lk} \ ,$$

where F_{ij} is the deformation gradient tensor.

*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 using 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-PrePost. This plasticity model is fully iterative and is available only for shell elements. Also see the Remarks below.

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

Card 2	1	2	3	4	5	6	7	8
Variable	LCFLD							
Type	F							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Plastic hardening modulus; see Remarks for MAT 37.
R	Anisotropic hardening parameter; see Remarks for MAT 37.
HLCID	Load curve ID defining effective stress as a function of effective plastic strain. The yield stress and hardening modulus are ignored with this option.

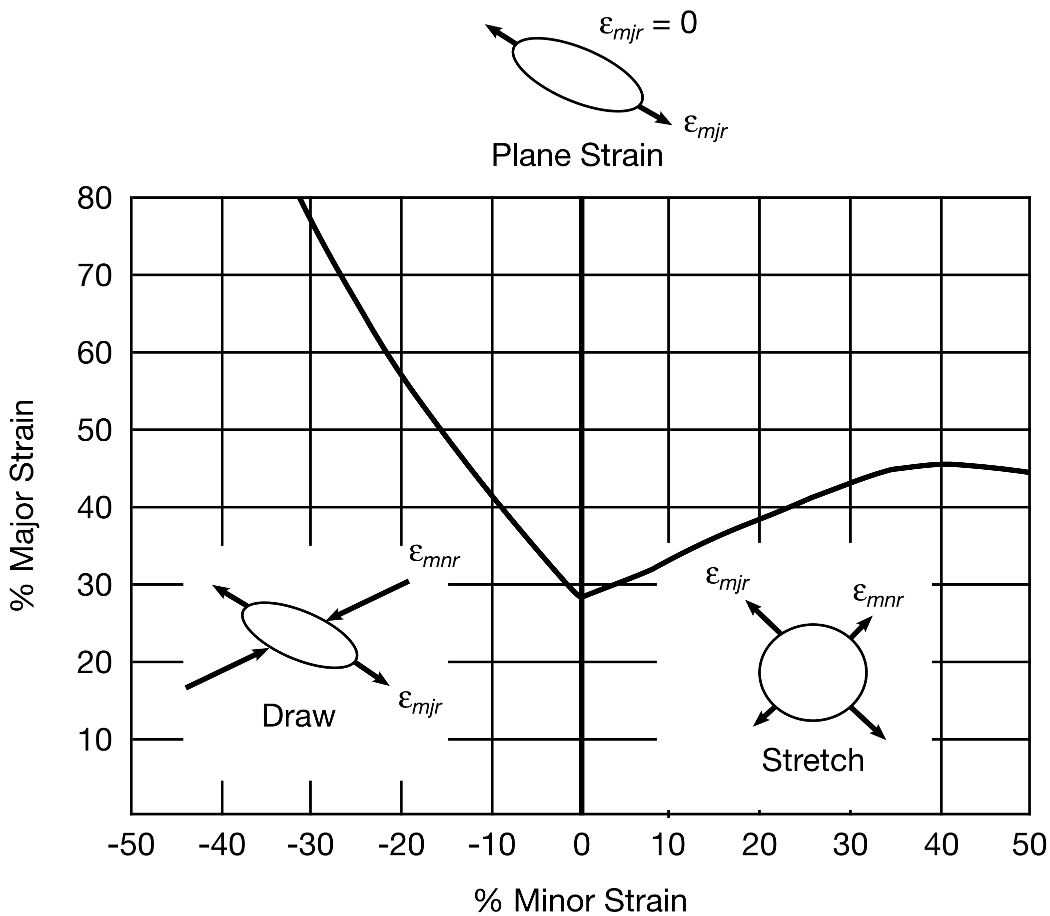


Figure M39-1. Forming limit diagram.

VARIABLE	DESCRIPTION
LCFLD	Load curve ID defining the Forming Limit Diagram. Minor strains in percent are defined as abscissa values and major strains in percent are defined as ordinate values. The forming limit diagram is shown in Figure M39-1. 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:

$$\frac{\varepsilon_{\mathrm{major}_{\mathrm{workpiece}}}}{\varepsilon_{\mathrm{major}_{\mathrm{fld}}}} ,$$

corresponding to $\varepsilon_{\mathrm{minor}_{\mathrm{workpiece}}}$.

*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, solid elements, and thick shell formulations 3, 5, and 7.

WARNING: 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 1	1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	PRBA	PRCA	PRCB
Type	A	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	0.0	0.0		

Card 3	1	2	3	4	5	6	7	8
Variable	LCIDA	LCIDB	EFAIL	DTFAIL	CDAMP	AOPT	MACF	ATRACK
Type	F	F	F	F	F	F	I	I
Default	0.0	0.0	0.0	0.0	0.0	0.0	0	0

Card 4	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	
Type	F	F	F	F	F	F	F	

Optional Card 6 (Applies to solid elements only)

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 or label must be specified (see *PART).
RO	Mass density.

VARIABLE	DESCRIPTION
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_{ba}, 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.
DT	Temperature increment for isotropic stress initialization. This option can be used during dynamic relaxation.
TRAMP	Time to ramp up to the final temperature.
ALPHA	Thermal expansion coefficient.
LCIDA	Optional load curve ID defining the nominal stress versus strain along a-axis. Strain is defined as $\lambda_a - 1$ where λ_a is the stretch ratio along the a-axis.
LCIDB	Optional load curve ID defining the nominal stress versus strain along b-axis. Strain is defined as $\lambda_b - 1$ where λ_b is the stretch ratio along the b-axis.
EFAIL	Failure strain, $\lambda - 1$.
DTFAIL	Time step for automatic element erosion
CDAMP	Damping coefficient.
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC, particularly the Material Directions section, for details): EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. For shells only, the material axes are then rotated about the normal vector to the surface of

VARIABLE	DESCRIPTION
	the shell by the angle BETA.
	EQ.1.0: Locally orthotropic with material axes determined by a point, *P*, 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 a vector **v** and the normal vector to the plane of the element. 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then **a** and **b** are rotated about **c** by an angle BETA. BETA may be set in the keyword input for the element or in the input for this keyword. Note that for solids, the material axes may be switched depending on the choice of MACF. The switch may occur before or after applying BETA depending on the value of MACF.
	EQ.4.0: Locally orthotropic in a 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.
	LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_*OPTION*).
MACF	Material axes change flag for solid elements:
	EQ.-4: Switch material axes *b* and *c* before BETA rotation
	EQ.-3: Switch material axes *a* and *c* before BETA rotation
	EQ.-2: Switch material axes *a* and *b* before BETA rotation
	EQ.1: No change, default
	EQ.2: Switch material axes *a* and *b* after BETA rotation
	EQ.3: Switch material axes *a* and *c* after BETA rotation

VARIABLE	DESCRIPTION
	EQ.4: Switch material axes b and c after BETA rotation Figure M2-2 indicates when LS-DYNA applies MACF during the process to obtain the final material axes. If BETA on *ELEMENT_-SOLID_*{OPTION}* is defined, then that BETA is used for the rotation for all AOPT options. Otherwise, if AOPT = 3, the BETA input on Card 5 rotates the axes. For all other values of AOPT, the material axes will be switched as specified by MACF, but no BETA rotation will be performed.
ATRACK	Material a-axis tracking flag (shell elements only) EQ.0: a-axis rotates with element (default) EQ.1: a-axis also tracks deformation
XP, YP, ZP	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 = 0 (shells and thick shells only) and AOPT = 3. BETA may be overridden on the element card, see *ELEMENT_SHELL_BETA. *ELEMENT_TSHELL_BETA, and *ELEMENT_SOLID_ORTHO.
LCIDC	Load curve ID defining the nominal stress versus strain along c-axis. Strain is defined as $\lambda_c - 1$ where λ_c is the stretch ratio along the c-axis.
LCIDAB	Load curve ID defining the nominal *ab* shear stress versus *ab*-strain in the *ab*-plane. Strain is defined as the $\sin(\gamma_{ab})$ where γ_{ab} is the shear angle.
LCIDBC	Load curve ID defining the nominal *bc* shear stress versus *bc*-strain in the *bc*-plane. Strain is defined as the $\sin(\gamma_{bc})$where γ_{bc} is the shear angle.
LCIDCA	Load curve ID defining the nominal *ca* shear stress versus *ca*-strain in the *ca*-plane. Strain is defined as the $\sin(\gamma_{ca})$ where γ_{ca} is the shear angle.

Remarks:

1. **The ATRACK field.** The initial material directions are set using AOPT and the related data. By default, the material directions in shell elements are updated each cycle based on the rotation of the 1-2 edge, or else the rotation of all edges if the invariant node numbering option is set on *CONTROL_ACCURACY. When ATRACK=1, an optional scheme is used in which the a-direction of the material tracks element deformation as well as rotation. For more information, see Remark 2 of *MAT_COMPOSITE_DAMAGE.

2. **Computing stresses.** The stress versus stretch curves LCIDA, LCIDB, LCIDC, LCIDAB, LCIDBC, and LCIDCA are only used to obtain the slope (stiffness) to fill up the $|C|$ matrix and are not used directly to compute the stresses. The stresses are computed using the $|C|$ matrix and the Green-St Venant strain tensor.

*MAT_USER_DEFINED_MATERIAL_MODELS

These are Material Types 41 - 50. The user must provide a material subroutine. See also Appendix A. This keyword input is used to define material properties for the subroutine. Isotropic, anisotropic, thermal, and hyperelastic material models with failure can be handled.

Card Summary:

Card 1. This card is required.

MID	RO	MT	LMC	NHV	IORTHO	IBULK	IG

Card 2. This card is required.

IVECT	IFAIL	ITHERM	IHYPER	IEOS	LMCA	EXT	EPSHV

Card 3. Include this card if IORTHO = 1 or 3.

AOPT	MACF	XP	YP	ZP	A1	A2	A3

Card 4. Include this card if IORTHO = 1 or 3.

V1	V2	V3	D1	D2	D3	BETA	IEVTS

Card 5. Include as many instantiations of this card as required to define LMC fields.

P1	P2	P3	P4	P5	P6	P7	P8

Card 6. Include as many instantiations of this card as required to define LMCA fields.

P1	P2	P3	P4	P5	P6	P7	P8

Data Card Definitions:

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
MT	User material type (41 - 50 inclusive). A number between 41 and 50 must be chosen. If MT < 0, subroutine `rwumat` in dyn21.f is called, where the material parameter reading can be modified. **WARNING**: If two or more materials in an input deck share the same MT value, those materials must have the same values of other variables on Cards 1 and 2 except for MID and RO.
LMC	Length of material constant array which is equal to the number of material constants to be input. See Remark 2.
NHV	Number of history variables to be stored; see Appendix A. When the model is to be used with an equation of state, NHV must be increased by 4 to allocate the storage required by the equation of state.
IORTHO/ ISPOT	Orthotropic/spot weld thinning flag: EQ.0: If the material is not orthotropic and is not used with spot weld thinning EQ.1: If the material is orthotropic EQ.2: If material is used with spot weld thinning EQ.3: If material is orthotropic and used with spot weld thinning
IBULK	Address of bulk modulus in material constants array; see Appendix A.
IG	Address of shear modulus in material constants array; see Appendix A.

Card 2	1	2	3	4	5	6	7	8
Variable	IVECT	IFAIL	ITHERM	IHYPER	IEOS	LMCA	EXT	EPSHV
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
IVECT	Vectorization flag: EQ.0: Off EQ.1: On. A vectorized user subroutine must be supplied.
IFAIL	Failure flag. EQ.0: No failure EQ.1: Allows failure of shell and solid elements LT.0: |IFAIL| is the address of NUMINT in the material constants array. NUMINT is defined as the number of failed integration points that will trigger element deletion. This option applies only to shell and solid elements (release 5 of version 971).
ITHERM	Temperature flag: EQ.0: Off EQ.1: On. Compute element temperature.
IHYPER	Deformation gradient flag (see Appendix A): EQ.0: Do not compute deformation gradient. EQ.-1: Same as 1, except if IORTHO = 1 or 3, the deformation gradient is in the global coordinate system. EQ.-10: Same as -1, except that this will enforce full integration for elements -1, -2 and 2. EQ.1: Compute deformation gradient for bricks and shells. If IORTHO = 1 or 3, the deformation gradient is in the local coordinate system instead of the global coordinate system. EQ.10: Same as 1, except that this will enforce full integration for elements -1, -2 and 2.

VARIABLE	DESCRIPTION
	EQ.3: Compute deformation gradient for shells from the nodal coordinates in the global coordinate system.
IEOS	Equation of state flag: EQ.0: Off EQ.1: On
LMCA	Length of additional material constant array
EXT	Flag to call external user material routines from other codes. See the file dyn21extumat.F for documentation.
EPSHV	Indicates which history variable is used to store effective plastic strain (if used). EPSHV is used in conjunction with EXT $\neq 0$ to facilitate post-processing.

Orthotropic Card 1. Additional card for IORTHO = 1 or 3.

Card 3	1	2	3	4	5	6	7	8
Variable	AOPT	MACF	XP	YP	ZP	A1	A2	A3
Type	F	I	F	F	F	F	F	F

VARIABLE	DESCRIPTION
AOPT	Material axes option (see *MAT_002 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, and then, for shells only, rotated about the shell element normal by an angle BETA 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.

VARIABLE	DESCRIPTION
	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.
	LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR). Available in R3 version of 971 and later.
MACF	Material axes change flag for brick elements for quick changes:
	EQ.1: No change, default
	EQ.2: Switch material axes *a* and *b*
	EQ.3: Switch material axes *a* and *c*
	EQ.4: Switch material axes *b* and *c*
XP, YP, ZP	Coordinates of point *p* for AOPT = 1 and 4
A1, A2, A3	Components of vector **a** for AOPT = 2

Orthotropic Card 2. Additional card for IORTHO = 1 or 3.

Card 4	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	IEVTS
Type	F	F	F	F	F	F	F	I

VARIABLE	DESCRIPTION
V1, V2, V3	Components of vector **v** for AOPT = 3 and 4
D1, D2, D3	Components of vector **d** for AOPT = 2

VARIABLE	DESCRIPTION
BETA	Material angle in degrees for AOPT = 0 (shells only) and AOPT = 3. BETA may be overridden on the element card; see *ELEMENT_SHELL_BETA and *ELEMENT_SOLID_ORTHO.
IEVTS	Address of E_a for orthotropic material with thick shell formulation 5 (see Remark 4)

Define LMC material parameters using 8 parameters per card. See Remark 2.

Card 5	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

Define LMCA material parameters using 8 parameters per card.

Card 6	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
P1	First material parameter.
P2	Second material parameter.
P3	Third material parameter.
P4	Fourth material parameter.
⋮	⋮
PLMC	LMCth material parameter.

Remarks:

1. **Cohesive Elements.** Material models for the cohesive element (solid element type 19) uses the first two *material param*eters to set flags in the *element formulation*.

a) *P1.* The P1 field controls how the density is used to calculate the mass when determining the tractions at mid-surface (tractions are calculated on a surface midway between the surfaces defined by nodes 1-2-3-4 and 5-6-7-8). If P1 is set to 1.0, then the density is per unit area of the mid-surface instead of per unit volume. Note that the cohesive element formulation permits the element to have zero or negative volume.

b) *P2.* The second parameter, P2, specifies the number of integration points (one to four) that are required to fail for the element to fail. If it is zero, the element will not fail regardless of IFAIL. The recommended value for P2 is 1.

c) *Other Parameters.* The cohesive element only uses MID, RO, MT, LMC, NHV, IFAIL and IVECT in addition to the material parameters.

d) *Appendix R.* See Appendix R for the specifics of the `umat` subroutine requirements for the cohesive element.

2. **Material Constants.** If IORTHO = 0, LMC must be ≤ 48. If IORTHO = 1, LMC must be ≤ 40. If more material constants are needed, LMCA may be used to create an additional material constant array. There is no limit on the size of LMCA.

3. **Spot weld thinning.** If the user-defined material is used for beam or brick element spot welds that are tied to shell elements, and SPOTHIN > 0 on *CONTROL_CONTACT, then spot weld thinning will be done for those shells if ISPOT = 2. Otherwise, it will not be done.

4. **Thick Shell Formulation 5.** IEVTS is optional and is used only by thick shell formulation 5. It points to the position of E_a in the material constants array. Following E_a, the next 5 material constants must be E_b, E_c, ν_{ba}, ν_{ca}, and ν_{cb}. This data enables thick shell formulation 5 to calculate an accurate thickness strain, otherwise the thickness strain will be based on the elastic constants pointed to by IBULK and IG.

*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 [Bammann 1989].

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

Card 2	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 3	1	2	3	4	5	6	7	8
Variable	C9	C10	C11	C12	C13	C14	C15	C16
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	C17	C18	A1	A2	A4	A5	A6	KAPPA
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus (psi)

VARIABLE	DESCRIPTION
PR	Poisson’s ratio
T	Initial temperature (°R, degrees Rankine)
HC	Heat generation coefficient (°R/psi)
C1	psi
C2	°R
C3	psi
C4	°R
C5	s^{-1}
C6	°R
C7	1/psi
C8	°R
C9	psi
C10	°R
C11	1/psi-s
C12	°R
C13	1/psi
C14	°R
C15	psi
C16	°R
C17	1/psi-s
C18	°R
A1	α_1, initial value of internal state variable 1
A2	α_2, initial value of internal state variable 2. Note: $\alpha_3 = -(\alpha_1 + \alpha_2)$
A4	α_4, initial value of internal state variable 3

VARIABLE	DESCRIPTION
A5	α_5, initial value of internal state variable 4
A6	α_6, initial value of internal state variable 5
KAPPA	κ, initial value of internal state variable 6

Unit Conversion Table

sec × psi × ^oR	sec × MPa × ^oR	sec × MPA × ^oK
C_1	$\times\ ^1\!/_{145}$	$\times\ ^1\!/_{145}$
C_2	—	$\times\ ^5\!/_9$
C_3	$\times\ ^1\!/_{145}$	$\times\ ^1\!/_{145}$
C_4	—	$\times\ ^5\!/_9$
C_5	—	—
C_6	—	$\times\ ^5/_9$
C_7	× 145	× 145
C_8	—	$\times\ ^5\!/_9$
C_9	$\times\ ^1\!/_{145}$	$\times\ ^1\!/_{145}$
C_{10}	—	$\times\ ^5\!/_9$
C_{11}	× 145	× 145
C_{12}	—	$\times\ ^5\!/_9$
C_{13}	× 145	× 145
C_{14}	—	$\times\ ^5\!/_9$
C_{15}	$\times\ ^1\!/_{145}$	$\times\ ^1\!/_{145}$
C_{16}	—	$\times\ ^5\!/_9$
C_{17}	× 145	× 145
C_{18}	—	$\times\ ^5\!/_9$
C0 = HC	× 145	$\times\ (145)(^5\!/_9)$
E	$\times\ ^1\!/_{145}$	$\times\ ^1\!/_{145}$
υ	—	—
T	—	$\times\ ^5\!/_9$

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:

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

where the Cauchy stress $\boldsymbol{\sigma}$ is convected with the elastic spin $\mathbf{W}^e$ as,

$$\overset{o}{\boldsymbol{\sigma}} = \dot{\boldsymbol{\sigma}} - \mathbf{W}^e\boldsymbol{\sigma} + \boldsymbol{\sigma}\mathbf{W}^e \ .$$

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 $\mathbf{D}^e$ and the elastic spin $\mathbf{W}^e$,

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

Within this structure it is now necessary to prescribe an equation for the plastic spin $\mathbf{W}^p$ in addition to the normally prescribed flow rule for $\mathbf{D}^p$ and the stretching due to the thermal expansion D^{th}. As proposed, we assume a flow rule of the form,

$$\mathbf{D}^p = f(T)\sinh\left[\frac{|\boldsymbol{\xi}| - \kappa - Y(T)}{V(T)}\right]\frac{\boldsymbol{\xi}'}{|\boldsymbol{\xi}'|} \ .$$

where T is the temperature, κ is the scalar hardening variable, and $\boldsymbol{\xi}'$ is the difference between the deviatoric Cauchy stress $\boldsymbol{\sigma}'$ and the tensor variable $\boldsymbol{\alpha}'$,

$$\boldsymbol{\xi}' = \boldsymbol{\sigma}' - \boldsymbol{\alpha}' \ ,$$

and $f(T)$, $Y(T)$, and $V(T)$ 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,

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

The evolution of the internal variables $\boldsymbol{\alpha}$ and $\boldsymbol{\kappa}$ are prescribed in a hardening minus recovery format as,

$$\overset{o}{\boldsymbol{\alpha}} = h(T)\mathbf{D}^p - [r_d(T)|\mathbf{D}^p| + r_s(T)]|\boldsymbol{\alpha}|\boldsymbol{\alpha}$$
$$\dot{\boldsymbol{\kappa}} = H(T)\mathbf{D}^p - [R_d(T)|\mathbf{D}^p| + R_s(T)]\boldsymbol{\kappa}^2$$

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 $\mathbf{W}^p = 0$, 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,

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

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,

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

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

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

$$\begin{aligned}
V(T) &= \mathrm{C1}\exp(-\mathrm{C2}/T) & r_s(T) &= \mathrm{C11}\exp(-\mathrm{C12}/T)\\
Y(T) &= \mathrm{C3}\exp(\mathrm{C4}/T) & R_d(T) &= \mathrm{C13}\exp(-\mathrm{C14}/T)\\
f(T) &= \mathrm{C5}\exp(-\mathrm{C6}/T) & H(T) &= \mathrm{C15}\exp(\mathrm{C16}/T)\\
r_d(T) &= \mathrm{C7}\exp(-\mathrm{C8}/T) & R_s(T) &= \mathrm{C17}\exp(-\mathrm{C18}/T)\\
h(T) &= \mathrm{C9}\exp(\mathrm{C10}/T) & &
\end{aligned}$$

and the heat generation coefficient is

$$\mathrm{HC} = \frac{0.9}{\rho C_v}.$$

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

Card 2	1	2	3	4	5	6	7	8
Variable	C1	C2	C3	C4	C5	C6	C7	C8
Type	F	F	F	F	F	F	F	F

Card 3	1	2	3	4	5	6	7	8
Variable	C9	C10	C11	C12	C13	C14	C15	C16
Type	F	F	F	F	F	F	F	F

Card 4	1	2	3	4	5	6	7	8
Variable	C17	C18	A1	A2	A3	A4	A5	A6
Type	F	F	F	F	F	F	F	F

Card 5	1	2	3	4	5	6	7	8
Variable	N	D0	FS					
Type	F	F	F					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus (psi)
PR	Poisson's ratio
T	Initial temperature (°R, degrees Rankine)
HC	Heat generation coefficient $\left({}^{\circ}R/_{psi}\right)$
C1	Psi
C2	°R
C3	Psi
C4	°R
C5	$^1/s$
C6	°R
C7	$^1/psi$
C8	°R
C9	Psi
C10	ºR
C11	$^1/psi\text{-}s$
C12	°R
C13	$^1/psi$
C14	°R
C15	psi
C16	°R

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

Remarks:

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

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

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 material models low density, closed cell polyurethane foam. It is for simulating impact limiters in automotive applications. The effect of the confined air pressure is included with the air being treated as an ideal gas. The general behavior is isotropic with uncoupled components of the stress tensor.

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

Card 2	1	2	3	4	5	6	7	8
Variable	GAMMA0	LCID						
Type	F	I						

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus
A	a, factor for yield stress definition; see Remarks below.
B	b, factor for yield stress definition; see Remarks below.
C	c, factor for yield stress definition; see Remarks below.
P0	Initial foam pressure, p_0
PHI	Ratio of foam to polymer density, ϕ
GAMMA0	Initial volumetric strain,γ_0. The default is zero.
LCID	Optional load curve defining the von Mises yield stress as a function of $-\gamma$. 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

VARIABLE	DESCRIPTION
	load curve is defined in the positive quadrant, that is, positive values of γ are defined as negative values on the abscissa.

Remarks:

A rigid, low density, closed cell, polyurethane foam model developed at Sandia Laboratories [Neilsen, Morgan and Krieg 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.

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

where $\sigma_{ij}^{\mathrm{sk}}$ is the skeletal stress and σ^{air} is the air pressure. σ^{air} is computed from the equation:

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

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

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

Here, 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:

$$\sigma_{ij}^{\mathrm{sk}} = \sigma_{ij} + \sigma_{ij}\sigma^{\mathrm{air}}$$

and compute the trial stress, $\sigma_{ij}^{\mathrm{skt}}$

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

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

$$\sigma_i^{\mathrm{sk}} = \min\left(\sigma_y, \left|\sigma_i^{\mathrm{skt}}\right|\right)\frac{\sigma_i^{\mathrm{skt}}}{\left|\sigma_i^{\mathrm{skt}}\right|} \;.$$

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

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

*MAT_ENHANCED_COMPOSITE_DAMAGE

These are Material Types 54 - 55 which are enhanced versions of the composite model material type 22. Arbitrary orthotropic materials, such as 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 1987b] or [Tsai and Wu 1971]. In addition, special measures are taken for failure under compression. See [Matzenmiller and Schweizerhof 1991].

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. A damage model for transverse shear strain to model interlaminar shear failure is available. The definition of minimum stress limits is available for thin/thick shells and solids.

NOTE: *MAT_054 is supported for shell, solid, and thick shell elements. *MAT_055 is only supported for shell elements and thick shell formulations 1, 2, and 6. If *MAT_055 is used for solids, LS-DYNA automatically switches to *MAT_054.

NOTE: We recommend using *MAT_054 over *MAT_055.

Card Summary:

Card 1. This card is required.

MID	RO	EA	EB	EC	PRBA	PRCA	PRCB

Card 2. This card is required.

GAB	GBC	GCA	(KF)	AOPT	2WAY	TI	

Card 3. This card is required.

XP	YP	ZP	A1	A2	A3	MANGLE	

Card 4a. Include this card if the material is *MAT_054 and DFAILT ≠ 0.0.

V1	V2	V3	D1	D2	D3	DFAILM	DFAILS

Card 4b. Include this card if Card 4a is not included, meaning the material is *MAT_055 or the material is *MAT_054 with DFAILT = 0.0.

V1	V2	V3	D1	D2	D3		

Card 5a. Include this card if the material is *MAT_054.

TFAIL	ALPH	SOFT	FBRT	YCFAC	DFAILT	DFAILC	EFS

Card 5b. Include this card if the material is *MAT_055.

TFAIL	ALPH	SOFT	FBRT				

Card 6a. Include this card if the 2WAY flag is 0.

XC	XT	YC	YT	SC	CRIT	BETA	

Card 6b. Include this card if the 2WAY flag is 1.

XC	XT	YC	YT	SC	CRIT	BETA	

Card 7. Only include this card for *MAT_054 (CRIT = 54).

PFL	EPSF	EPSR	TSMD	SOFT2			

Card 8a. Only include this card for *MAT_054 (CRIT = 54) and 2WAY = 0.

SLIMT1	SLIMC1	SLIMT2	SLIMC2	SLIMS	NCYRED	SOFTG	

Card 8b. Only include this card for *MAT_054 (CRIT = 54) and 2WAY = 1.

SLIMT1	SLIMC1	SLIMT2	SLIMC2	SLIMS	NCYRED	SOFTG	

Card 9. Only include this card for *MAT_054 (CRIT = 54).

LCXC	LCXT	LCYC	LCYT	LCSC	DT		

Data Card Definitions:

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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
PRBA	ν_{ba}, Poisson's ratio *ba*
PRCA	ν_{ca}, Poisson's ratio *ca*
PRCB	ν_{cb}, Poisson's ratio *cb*

Card 2	1	2	3	4	5	6	7	8
Variable	GAB	GBC	GCA	(KF)	AOPT	2WAY	TI	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
GAB	G_{ab}, shear modulus *ab*
GBC	G_{bc}, shear modulus *bc*
GCA	G_{ca}, shear modulus *ca*

VARIABLE	DESCRIPTION
(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, and then, for shells only, rotated about the shell element normal by an angle MANGLE. 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 (MANGLE) 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. LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_NODES, *DEFINE_COORDINATE_SYSTEM or *DEFINE_COORDINATE_VECTOR).
2WAY	Flag to turn on 2-way fiber action: EQ.0.0: Standard unidirectional behavior, meaning fibers run only in the *a*-direction EQ.1.0: 2-way fiber behavior, meaning fibers run in both the *a*- and *b*-directions. The meaning of the fields DFAILT, DFAILC, YC, YT, SLIMT2 and SLIMC2 are altered if this flag is set. This option is only available for *MAT_054 using thin shells.
TI	Flag to turn on transversal isotropic behavior for *MAT_054 solid elements.

VARIABLE	DESCRIPTION
	EQ.0.0: Standard unidirectional behavior EQ.1.0: Transversal isotropic behavior (see Remark 5)

Card 3	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3	MANGLE	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
XP, YP, ZP	Coordinates of point *p* for AOPT = 1 and 4
A1, A2, A3	Components of vector **a** for AOPT = 2
MANGLE	Material angle in degrees for AOPT = 0 (shells only) and AOPT = 3. MANGLE may be overridden on the element card; see *ELEMENT_SHELL_BETA and *ELEMENT_SOLID_ORTHO.

This card is included if the material is *MAT_054 and DFAILT (see Card 5a) is nonzero.

Card 4a	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	DFAILM	DFAILS
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
V1, V2, V3	Define components of vector **v** for AOPT = 3.
D1, D2, D3	Define components of vector **d** for AOPT = 2.
DFAILM	Maximum strain for matrix straining in tension or compression (active only for *MAT_054 and only if DFAILT > 0). 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 tensorial shear strain (active only for *MAT_054 and only if DFAILT > 0). The layer in the element is completely removed after the maximum shear strain is reached. The input value

VARIABLE	DESCRIPTION
	is always positive.

This card is included if Card 4a is not included.

Card 4b	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
V1, V2, V3	Define components of vector **v** for AOPT = 3.
D1, D2, D3	Define components of vector **d** for AOPT = 2.

This card is included if the material is *MAT_054.

Card 5a	1	2	3	4	5	6	7	8
Variable	TFAIL	ALPH	SOFT	FBRT	YCFAC	DFAILT	DFAILC	EFS
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
TFAIL	Time step size criteria for element deletion: LE.0.0: No element deletion by time step size. The crashfront algorithm only works if TFAIL is set to a value greater than zero. GT.0.0.and.LE.0.1: Element is deleted when its time step is smaller than the given value. GT.0.1: 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 *MAT_022.
SOFT	Softening reduction factor for material strength in crashfront elements (default = 1.0). TFAIL must be greater than zero to activate

VARIABLE	DESCRIPTION
	this option. Crashfront elements are elements that are direct neighbors of failed (deleted) elements. See Remark 1.
FBRT	Softening for fiber tensile strength: EQ.0.0: Tensile strength = XT GT.0.0: Tensile strength = XT, reduced to XT × FBRT after failure has occurred in compressive matrix mode
YCFAC	Reduction factor for compressive fiber strength after matrix compressive failure. The compressive strength in the fiber direction after compressive matrix failure is reduced to: $X_c = \text{YCFAC} \times Y_c$, (default: YCFAC = 2.0)
DFAILT	Maximum strain for fiber tension (*MAT_054 only). A value of 1 is 100% tensile strain. The layer in the element is completely removed after the maximum tensile strain in the fiber direction is reached. If a nonzero value is given for DFAILT (recommended), a nonzero, negative value must also be provided for DFAILC. If the 2-way fiber flag is set, then DFAILT is the fiber tensile failure strain in the *a* and *b* directions.
DFAILC	Maximum strain for fiber compression. A value of -1 is 100% compression strain. The layer in the element is completely removed after the maximum compressive strain in the fiber direction is reached. The input value should be negative and is required if DFAILT > 0. If the 2-way fiber flag is set, then DFAILC is the fiber compressive failure strain in the *a* and *b* directions.
EFS	Effective failure strain

This card is included if the material is *MAT_055.

Card 5b	1	2	3	4	5	6	7	8
Variable	TFAIL	ALPH	SOFT	FBRT				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
TFAIL	Time step size criteria for element deletion: LE.0.0: No element deletion by time step size. The crashfront algorithm only works if TFAIL is set to a value greater than zero. GT.0.0.and.LE.0.1: Element is deleted when its time step is smaller than the given value. GT.0.1: 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 *MAT_022.
SOFT	Softening reduction factor for material strength in crashfront elements (default = 1.0). TFAIL must be greater than zero to activate this option. Crashfront elements are elements that are direct neighbors of failed (deleted) elements.
FBRT	Softening for fiber tensile strength: EQ.0.0: Tensile strength = XT GT.0.0: Tensile strength = XT, reduced to XT × FBRT after failure has occurred in compressive matrix mode

This card is included if 2WAY = 0.

Card 6a	1	2	3	4	5	6	7	8
Variable	XC	XT	YC	YT	SC	CRIT	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
XC	Longitudinal compressive strength (absolute value is used): GE.0.0: Poisson effect (PRBA) after failure is active. LT.0.0: Poisson effect after failure is not active, meaning PRBA = 0.
XT	Longitudinal tensile strength; see Material Formulation below.

VARIABLE	DESCRIPTION
YC	Transverse compressive strength, b-axis (positive value). See Material Formulation below.
YT	Transverse tensile strength, b-axis. See Material Formulation below.
SC	Shear strength, ab-plane. See the Material Formulation below.
CRIT	Failure criterion (material number): EQ.54.0: Chang-Chang criterion for matrix failure (as *MAT_022) (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.

This card is included if 2WAY = 1 (CRIT must be 54 in this case).

Card 6b	1	2	3	4	5	6	7	8
Variable	XC	XT	YC	YT	SC	CRIT	BETA	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
XC	Longitudinal compressive strength (absolute value is used): GE.0.0: Poisson effect (PRBA) after failure is active. LT.0.0: Poisson effect after failure is not active, meaning PRBA = 0.
XT	Longitudinal tensile strength; see Material Formulation below.
YC	Fiber compressive failure stress in the b-direction. See Material Formulation below.
YT	Fiber tensile failure stress in the b-direction. See Material Formulation below.
SC	Shear strength, ab-plane. See the Material Formulation below.

VARIABLE	DESCRIPTION
CRIT	Failure criterion (material number): EQ.54.0: Chang-Change criterion for matrix failure (as *MAT_-022) (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.

Optional Card 7 (only for CRIT = 54). This card is included for *MAT_054 only.

Card 7	1	2	3	4	5	6	7	8
Variable	PFL	EPSF	EPSR	TSMD	SOFT2			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
PFL	Percentage of layers which must fail before crashfront is initiated (thin and thick shells only). For example, if |PFL| = 80.0, then 80% of the layers must fail before strengths are reduced in neighboring elements. Default: all layers must fail. The sign of PFL determines how many in-plane integration points must fail for a single layer to fail: GT.0.0: A single layer fails if 1 in-plane IP fail. LT.0.0: A single layer fails if 4 in-plane IPs fail.
EPSF	Damage initiation transverse shear strain
EPSR	Final rupture transverse shear strain LT.0.0: |EPSR| is final rupture transverse shear strain. In addition, the element erodes if transverse shear damage reaches TSMD.
TSMD	Transverse shear maximum damage (default = 0.90)
SOFT2	Optional "orthogonal" softening reduction factor for material strength in crashfront elements (default = 1.0). See Remark 1 (thin and thick shells only).

Optional Card 8 (only for CRIT = 54). This card is included for *MAT_054 only and 2WAY = 0.

Card 8a	1	2	3	4	5	6	7	8
Variable	SLIMT1	SLIMC1	SLIMT2	SLIMC2	SLIMS	NCYRED	SOFTG	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
SLIMT1	Factor to determine the minimum stress limit after stress maximum (fiber tension). Similar to *MAT_058.
SLIMC1	Factor to determine the minimum stress limit after stress maximum (fiber compression). Similar to *MAT_058.
SLIMT2	Factor to determine the minimum stress limit after stress maximum (matrix tension). Similar to *MAT_058.
SLIMC2	Factor to determine the minimum stress limit after stress maximum (matrix compression). Similar to *MAT_058.
SLIMS	Factor to determine the minimum stress limit after stress maximum (shear). Similar to *MAT_058.
NCYRED	Number of cycles for stress reduction from maximum to minimum for DFAILT > 0.
SOFTG	Softening reduction factor for transverse shear moduli GBC and GCA in crashfront elements (thin and thick shells). Default = 1.0.

Optional Card 8 (only for CRIT = 54). This card is included for *MAT_054 only and 2WAY = 1.

Card 8b	1	2	3	4	5	6	7	8
Variable	SLIMT1	SLIMC1	SLIMT2	SLIMC2	SLIMS	NCYRED	SOFTG	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
SLIMT1	Factor to determine the minimum stress limit after stress maximum (fiber tension) in the *a*-direction. Similar to *MAT_058.
SLIMC1	Factor to determine the minimum stress limit after stress maximum (fiber compression) in the *a*-direction. Similar to *MAT_058.
SLIMT2	Factor to determine the minimum stress limit after tensile failure stress is reached in the *b* fiber direction
SLIMC2	Factor to determine the minimum stress limit after compressive failure stress is reached in the *b* fiber direction
SLIMS	Factor to determine the minimum stress limit after stress maximum (shear). Similar to *MAT_058.
NCYRED	Number of cycles for stress reduction from maximum to minimum for DFAILT > 0.
SOFTG	Softening reduction factor for transverse shear moduli GBC and GCA in crashfront elements (thin and thick shells). Default = 1.0.

Optional Card 9 (only for CRIT = 54). This card is included for *MAT_054 only.

Card 9	1	2	3	4	5	6	7	8
Variable	LCXC	LCXT	LCYC	LCYT	LCSC	DT		
Type	I	I	I	I	I	F		

VARIABLE	DESCRIPTION
LCXC	Load curve ID for XC as a function of strain rate (XC is ignored with this option)
LCXT	Load curve ID for XT as a function strain rate (XT is ignored with this option)
LCYC	Load curve ID for YC as a function of strain rate (YC is ignored with this option)
LCYT	Load curve ID for YT as a function of strain rate (YT is ignored with this option)

VARIABLE	DESCRIPTION
LCSC	Load curve ID for SC as a function of strain rate (SC is ignored with this option)
DT	Strain rate averaging option: EQ.0.0: Strain rate is evaluated using a running average. LT.0.0: Strain rate is evaluated using an average of the last 11 time steps. GT.0.0: Strain rate is averaged over the last DT time units.

Material Formulation:

***MAT_054 Failure Criteria**
The Chang-Chang (*MAT_054) criteria is given as follows:

1. For the tensile fiber mode,

$$\sigma_{aa} > 0 \Rightarrow e_f^2 = \left(\frac{\sigma_{aa}}{X_t}\right)^2 + \beta\left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1, \qquad \begin{array}{l} e_f^2 \geq 0 \Rightarrow \text{failed} \\ e_f^2 < 0 \Rightarrow \text{elastic} \end{array}$$

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

2. For the compressive fiber mode,

$$\sigma_{aa} < 0 \Rightarrow e_c^2 = \left(\frac{\sigma_{aa}}{X_c}\right)^2 - 1, \qquad \begin{array}{l} e_c^2 \geq 0 \Rightarrow \text{failed} \\ e_c^2 < 0 \Rightarrow \text{elastic} \end{array}$$

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

3. For the tensile matrix mode,

$$\sigma_{bb} > 0 \Rightarrow e_m^2 = \left(\frac{\sigma_{bb}}{Y_t}\right)^2 + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1, \qquad \begin{array}{l} e_m^2 \geq 0 \Rightarrow \text{failed} \\ e_m^2 < 0 \Rightarrow \text{elastic} \end{array}$$

$$E_b = \nu_{ba} = 0 \Rightarrow G_{ab} = 0$$

4. For the compressive matrix mode,

$$\sigma_{bb} < 0 \Rightarrow e_d^2 = \left(\frac{\sigma_{bb}}{2S_c}\right)^2 + \left[\left(\frac{Y_c}{2S_c}\right)^2 - 1\right]\frac{\sigma_{bb}}{Y_c} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1, \qquad \begin{array}{l} e_d^2 \geq 0 \Rightarrow \text{failed} \\ e_d^2 < 0 \Rightarrow \text{elastic} \end{array}$$

$$E_b = \nu_{ba} = \nu_{ab} = 0 \Rightarrow G_{ab} = 0$$

$$X_c = 2Y_c, \text{ for 50\% fiber volume}$$

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

***MAT_054 with 2-Way Fiber Flag Failure Criteria**

If the 2-way fiber flag is set, then the failure criteria for tensile and compressive fiber failure in the local x-direction are unchanged. For the local y-direction, the same failure criteria as for the x-direction fibers are used.

1. For the tensile fiber mode in the local y-direction,

$$\sigma_{bb} > 0 \Rightarrow e_f^2 = \left(\frac{\sigma_{bb}}{Y_t}\right)^2 + \beta\left(\frac{\sigma_{ab}}{S_c}\right) - 1, \qquad \begin{aligned} e_f^2 &\geq 0 \Rightarrow \text{failed} \\ e_f^2 &< 0 \Rightarrow \text{elastic} \end{aligned}$$

2. For the compressive fiber mode in the local y-direction,

$$\sigma_{bb} < 0 \Rightarrow e_c^2 = \left(\frac{\sigma_{bb}}{Y_c}\right)^2 - 1, \qquad \begin{aligned} e_c^2 &\geq 0 \Rightarrow \text{failed} \\ e_c^2 &< 0 \Rightarrow \text{elastic} \end{aligned}$$

3. For 2WAY the matrix only fails in shear,

$$e_m^2 = \left(\frac{\sigma_{ab}}{S_c}\right)^2 - 1, \qquad \begin{aligned} e_m^2 &\geq 0 \Rightarrow \text{failed} \\ e_m^2 &< 0 \Rightarrow \text{elastic} \end{aligned}$$

***MAT_055 Failure Criteria**

For the Tsai-Wu (*MAT_055) criteria, the tensile and compressive fiber modes are treated the same as in the Chang-Chang criteria. The failure criterion for the tensile and compressive matrix mode is given as:

$$e_{m/d}^2 = \frac{\sigma_{bb}^2}{Y_c Y_t} + \left(\frac{\sigma_{ab}}{S_c}\right)^2 + \frac{(Y_c - Y_t)\,\sigma_{bb}}{Y_c Y_t} - 1, \qquad \begin{aligned} e_{m/d}^2 &\geq 0 \Rightarrow \text{failed} \\ e_{m/d}^2 &< 0 \Rightarrow \text{elastic} \end{aligned}$$

Remarks:

1. **Integration point failure.** In *MAT_054, failure can occur in any of four different ways:
   - If DFAILT is zero, failure occurs if the Chang-Chang failure criterion is satisfied in the tensile fiber mode.
   - If DFAILT is greater than zero, failure occurs if:
     - the fiber strain is greater than DFAILT or less than DFAILC
     - if absolute value of matrix strain is greater than DFAILM
     - if absolute value of tensorial shear strain is greater than DFAILS
   - If EFS is greater than zero, failure occurs if the effective strain is greater than EFS.
   - If TFAIL is greater than zero, failure occurs according to the element timestep as described in the definition of TFAIL above.

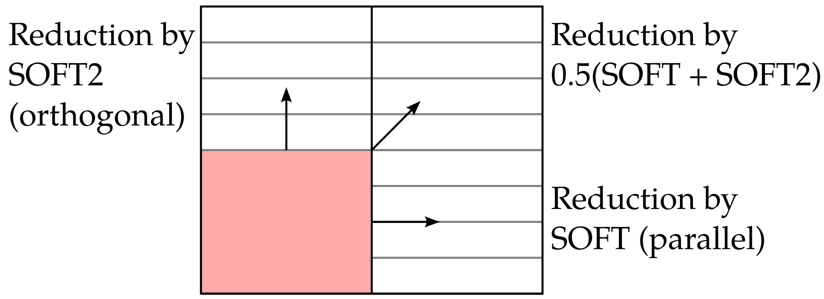


Figure M54-1. Direction dependent softening

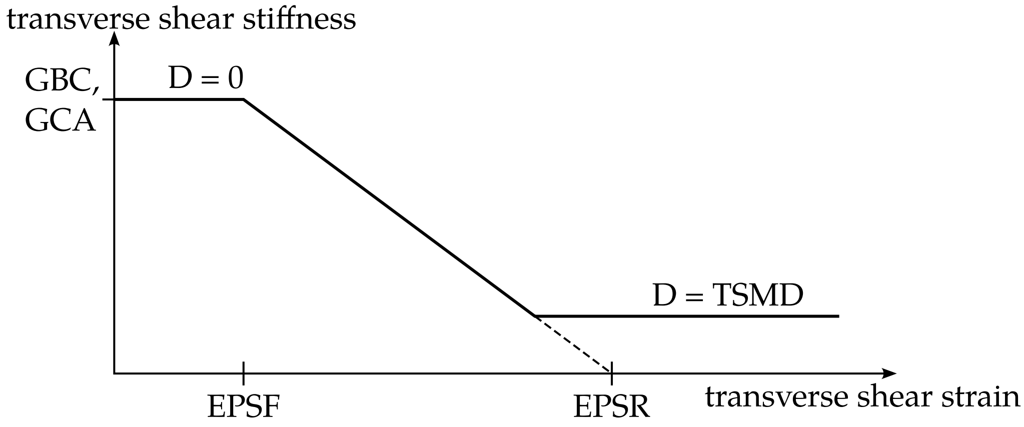


Figure M54-2. Linear Damage for transverse shear behavior

In *MAT_055, an integration point is deleted (all stresses go to zero) only if the tensile stress at that point reaches XT. Other strengths, XC, YT, YC, SC serve to cap stresses but do not delete the integration point.

When failure has occurred in all the composite layers (through-thickness integration points), the element is deleted. For bricks, the element is deleted after one integration point has met the failure criteria.

2. **Crashfront elements and strength reduction.** Elements that share nodes with a deleted element become "crashfront" elements and can have their strengths reduced by using the SOFT parameter with TFAIL greater than zero. An earlier initiation of crashfront elements is possible by using parameter PFL.

   An optional direction dependent strength reduction can be invoked by setting 0 < SOFT2 < 1. Then, SOFT equals a strength reduction factor for fiber parallel failure and SOFT2 equals a strength reduction factor for fiber orthogonal failure. Linear interpolation is used for angles in between. See Figure M54-1.

3. **Transverse shear strain damage model.** In an optional damage model for transverse shear strain, out-of-plane stiffness (GBC and GCA) can linearly

decrease to model interlaminar shear failure. Damage starts when effective transverse shear strain

$$\varepsilon_{56}^{\mathrm{eff}} = \sqrt{\varepsilon_{yz}^2 + \varepsilon_{zx}^2}$$

reaches EPSF. Final rupture occurs when effective transverse shear strain reaches EPSR. A maximum damage of TSMD (0.0 < TSMD < 0.99) cannot be exceeded. See Figure M54-2.

4. **Failure/damage status.** The status in each layer (integration point) and element can be plotted using additional integration point history variables. NEIPH and NEIPS on *DATABASE_EXTENT_BINARY sets the number of additional integration point history variables output for solids and shells, respectively. The number of additional integration point history variables for shells and solids written to the LS-DYNA database is input by the *DATABASE_EXTENT_BINARY definition as variable . For Models 54 and 55 these additional history variables are tabulated below (i = integration point):

Table M54-1. Additional history variables for *MAT_054

History Variable #	Description for shells and thick shell types 1, 2, and 6	Description for solids and thick shell types 3, 5, and 7	Value
1	Tensile fiber failure mode, ef(i)	Tensile fiber failure mode, ef(i)	1: elastic 0: failed
2	Compressive fiber failure mode, ec(i)	Compressive fiber failure mode, ec(i)	1: elastic 0: failed
3	Tensile (shear for 2WAY) matrix mode, $em(i)$	Tensile (shear for 2WAY) matrix mode, $em(i)$	1: elastic 0: failed
4	Compressive matrix mode, $ed(i)$	Compressive matrix mode, $ed(i)$	1: elastic 0: failed
5	Total failure	Total failure	1: elastic 0: failed
6	Damage parameter (SOFT)	Damage parameter (SOFT)	-1: element intact 10^{-8}: element in crashfront 1: element failed
8	$\cos(\alpha)$, where α is the in-plane angle between the material coordinate system and		

History Variable #	Description for shells and thick shell types 1, 2, and 6	Description for solids and thick shell types 3, 5, and 7	Value
	the element coordinate system		
9	sin α		
10	Local strain in the a-direction		
11	Local strain in the b-direction		
12	Local shear strain (ab-plane)		
15		Local strain in the a-direction	
16	Transverse shear damage	Local strain in the b-direction	
17		Local shear strain (ab-plane)	

Table M54-2. Additional history variables for *MAT_055

History Variable #	Description for shells and thick shell types 1, 2, and 6	Value
1	Tensile fiber failure mode, $ef(i)$	1: elastic 0: failed
2	Compressive fiber failure mode, $ec(i)$	1: elastic 0: failed
3	Tensile matrix mode, $em(i)$	1: elastic 0: failed
4	Compressive matrix mode, $ed(i)$	1: elastic 0: failed
5	Total failure	1: elastic 0: failed
6	Damage parameter (SOFT)	-1: element intact 10^{-8}: element in crashfront 1: element failed
8	$\cos(\alpha)$, where α is the in-plane angle between the material coordinate system and the element coordinate system	
9	$\sin(\alpha)$	

The three element history variables in the table below represent the fraction of elastic (non-failed) integration points in tensile fiber, compressive fiber, and tensile matrix failure modes. They are labeled as "effective plastic strain" by LS-PrePost for integration points 1, 2, and 3. In the table i indexes the integration points in the element and nip is the number of integration points in the element.

Description	Integration Point
$\frac{1}{\mathrm{nip}}\sum_{i=1}^{\mathrm{nip}} \mathrm{ef}(i)$	1
$\frac{1}{\mathrm{nip}}\sum_{i=1}^{\mathrm{nip}} \mathrm{ec}(i)$	2
$\frac{1}{\mathrm{nip}}\sum_{i=1}^{\mathrm{nip}} \mathrm{em}(i)$	3

5. **TI flag.** This applies only to transversal isotropic behavior for *MAT_054 solid elements. The behavior in the bc-plane is assumed to be isotropic, thus the elastic constants EC, PRCA and GCA are ignored and set according to the given values EA, EB, PRAB, and GAB. Damage in transverse shear (EPSF, EPSR, TSMD, SOFTG) is ignored. The failure criterion is evaluated by replacing σ_{bb} and σ_{ab} with the corresponding stresses σ_{11} and σ_{a1} in a principal stress frame rotated around the local a-axis. The principal axes 1 and 2 in the bc-plane are chosen such that $|\sigma_{11}| \geq |\sigma_{22}|$ is fulfilled.

6. **Material parameters.** PRBA is the minor Poisson's ratio if EA>EB, and the major Poisson's ratio will be equal to PRBA*(EA/EB). If EB > EA, then PRBA is the major Poisson's ratio. PRCA and PRCB are similarly defined. They are the minor Poisson's ratio if EA > EC or EB > EC, and the major Poisson's ratio if the EC > EA or EC > EB.

   Care should be taken when using material parameters from third party products regarding the directional indices a, b and c, as they may differ from the definition used in LS-DYNA. For the direction indices used in LS-DYNA see Material Directions in *MAT_002/*MAT_*OPTION*TROPIC_ELASTIC.

*MAT_LOW_DENSITY_FOAM

This is Material Type 57 for modeling highly compressible low density foams. Its main applications are for seat cushions and padding on the Side Impact Dummies (SID). Optionally, a tension cut-off failure can be defined. A table can be defined if thermal effects are considered in the nominal stress as a function of strain behavior. Also, see the remarks below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	LCID	TC	HU	BETA	DAMP
Type	A	F	F	F	F	F	F	F
Default	none	none	none	none	10^{20}	1.	none	0.05
Remarks						3	1	

Card 2	1	2	3	4	5	6	7	8
Variable	SHAPE	FAIL	BVFLAG	ED	BETA1	KCON	REF	
Type	F	F	F	F	F	F	F	
Default	1.0	0.0	0.0	0.0	0.0	0.0	0.0	
Remarks	3		2	5	5	6		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus used in tension. For implicit problems E is set to the initial slope of load curve LCID.
LCID	Load curve ID (see *DEFINE_CURVE) or table ID defining the nominal stress as a function of nominal strain. If a table is used, a

VARIABLE	DESCRIPTION
	family of curves is defined each corresponding to a discrete temperature; see *DEFINE_TABLE.
TC	Cut-off for the nominal tensile stress, τ_i.
HU	Hysteretic unloading factor between 0.0 and 1.0 (default = 1.0, that is, no energy dissipation); see also Figure M57-1.
BETA	Decay constant to model creep in unloading, β.
DAMP	Viscous coefficient (.05 < recommended value < .50) to model damping effects. LT.0.0: |DAMP| is the load curve ID, which defines the damping constant as a function of the maximum strain in compression defined as: $$\varepsilon_{\max} = \max(1-\lambda_1, 1-\lambda_2, 1.-\lambda_3)\ .$$ In tension, the damping constant is set to the value corresponding to the strain at 0.0. The abscissa should be defined from 0.0 to 1.0.
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 M57-1.
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: EQ.0.0: no bulk viscosity (recommended) EQ.1.0: bulk viscosity active
ED	Optional Young's relaxation modulus, E_d, for rate effects.
BETA1	Optional decay constant, β_1.
KCON	Stiffness coefficient for contact interface stiffness. If undefined the maximum slope in the stress as a function of 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

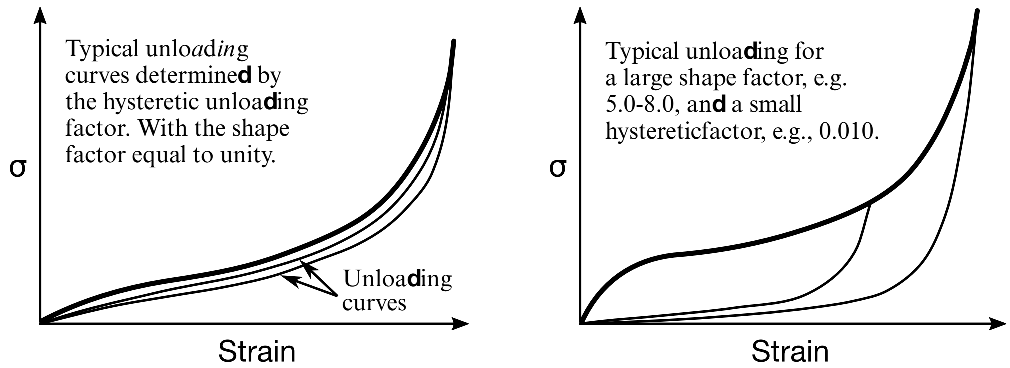


Figure M57-1. Behavior of the low density urethane foam model

VARIABLE	DESCRIPTION
	be significantly smaller, so defining a reasonable stiffness is recommended.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY. EQ.0.0: Off EQ.1.0: On

Material Formulation:

The compressive behavior is illustrated in Figure M57-1 where hysteresis upon 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 negative of the principal elongations (negative of principal engineering strains) are stored in an arbitrary order as extra history variables 16, 17, and 18 if ED = 0 and as extra history variables 28, 29, and 30 if ED > 0. (See NEIPH in *DATABASE_EXTENT_BINARY for output of extra history variables.) The stretch ratios are found by solving for the eigenvalues of the left stretch tensor, V_{ij}, which is obtained using 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.

Remarks:

1. **Decay constant and hysteretic unloading.** 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. **Bulk viscosity.** The bulk viscosity, which generates a rate dependent pressure, may cause an unexpected volumetric response and, consequently, it is optional with this model.

3. **Hysteretic unloading factor.** The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown in Figure M57-1 This unloading provides energy dissipation which is reasonable in certain kinds of foam.

4. **Output.** Note that since this material has no effective plastic strain, the internal energy per initial volume is written into the output databases.

5. **Rate effects.** 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:

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

given by,

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

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

6. **Time step size.** The time step size is based on the current density and the maximum of the instantaneous loading slope, E, and KCON. If KCON is undefined, the maximum slope in the loading curve is used instead.

MAT_LAMINATED_COMPOSITE_FABRIC_*{OPTION}

Available options include:

SOLID

Without the keyword option, this model supports shell elements (and thick shell element types 1, 2, and 6). The *SOLID* option allows the model to work for solid elements (and thick shell element types ELFORM = 3, 5, and 7).

This is Material Type 58. Depending on the type of failure surface, this material can model composite materials with unidirectional layers, complete laminates, and woven fabrics. We implemented this model for shell, thick shell, and solid elements. Shell elements (and thick shell types 1, 2, and 6) require no keyword option, while solid elements (and thick shell element types 3, 5, and 7) require the SOLID keyword option.

Card Summary:

Card 1. This card is required.

MID	RO	EA	EB	EC	PRBA	TAU1	GAMMA1

Card 2. This card is required.

GAB	GBC	GCA	SLIMT1	SLIMC1	SLIMT2	SLIMC2	SLIMS

Card 3. This card is required.

AOPT	TSIZE	ERODS	SOFT	FS	EPSF	EPSR	TSMD

Card 4. This card is required.

XP	YP	ZP	A1	A2	A3	PRCA	PRCB

Card 5. This card is required.

V1	V2	V3	D1	D2	D3	BETA	LCDFAIL

Card 6. This card is required.

E11C	E11T	E22C	E22T	GMS			

Card 7. This card is required.

XC	XT	YC	YT	SC			

Card 8.1. This card is required for the SOLID keyword option.

E33C	E33T	GMS23	GMS31				

Card 8.2. This card is required for the SOLID keyword option.

ZC	ZT	SC23	SC31				

Card 8.3. This card is required for the SOLID keyword option.

SLIMT3	SLIMC3	SLIMS23	SLIMS31	TAU2	GAMMA2	TAU3	GAMMA3

Card 9. This card is optional. (shells and solids)

LCXC	LCXT	LCYC	LCYT	LCSC	LCTAU	LCGAM	DT

Card 10. This card is optional. (shells and solids)

LCE11C	LCE11T	LCE22C	LCE22T	LCGMS	LCEFS		

Card 11. This card is optional. (solids only!)

LCZC	LCZT	LCSC23	LCSC31	LCTAU2	LCGAM2	LCTAU3	LCGAM3

Card 12. This card is optional. (solids only!)

LCE33C	LCE33T	LCGMS23	LCGMS31				

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	EA	EB	EC	PRBA	TAU1	GAMMA1
Type	A	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
EA	GT.0.0: E_a, Young's modulus - longitudinal direction

VARIABLE	DESCRIPTION
	LT.0.0: Load Curve ID or Table ID = (-EA). See Remark 8. **Load Curve.** When -EA is equal to a load curve ID, it is taken as defining the uniaxial elastic stress as a function of strain behavior in the longitudinal direction. Negative data points correspond to compression, and positive values to tension. **Tabular Data.** When -EA is equal to a table ID, it defines a load curve ID for each strain rate value. The load curves give the uniaxial elastic stress as a function of strain behavior in the longitudinal direction. **Logarithmically Defined Tables.** Suppose the first uniaxial elastic stress as a function of strain curve in the table corresponds to a negative strain rate. In that case, LS-DYNA assumes that the natural logarithm of the strain rate value is used for *all* stress-strain curves.
EB	GT.0.0: E_b, Young's modulus - transverse direction LT.0.0: Load Curve ID or Table ID = (-EB). See Remark 8. **Load Curve.** When -EB is equal to a load curve ID, it is taken as defining the uniaxial elastic stress as a function of strain behavior in the transverse direction. Negative data points correspond to compression, and positive values to tension. **Tabular Data.** When -EB corresponds to a table ID, it specifies a load curve ID for each strain rate value. The load curves give the uniaxial elastic stress as a function of strain behavior in the transverse direction. **Logarithmically Defined Tables.** Suppose the first uniaxial elastic stress as a function of strain curve in the table corresponds to a negative strain rate. In that case, LS-DYNA assumes that the natural logarithm of the strain rate value is used for *all* stress-strain curves.
EC	E_c, Young's modulus - normal direction (used only by thick shells and solids). See Remark 6. GT.0.0: E_c, Young's modulus - normal direction LT.0.0: Load Curve ID or Table ID = (-EC) (solids only). See Remark 8. **Load Curve.** When -EC is equal to a load curve ID, it is

VARIABLE	DESCRIPTION
	taken as defining the uniaxial elastic stress as a function of strain behavior in the transverse direction. Negative data points correspond to compression, and positive values to tension. **Tabular Data.** When -EC corresponds to a table ID, it specifies a load curve ID for each strain rate value. The load curves give the uniaxial elastic stress as a function of strain behavior in the transverse direction. **Logarithmically Defined Tables.** Suppose the first uniaxial elastic stress as a function of strain curve in the table corresponds to a negative strain rate. In that case, LS-DYNA assumes that the natural logarithm of the strain rate value is used for *all* stress-strain curves.
PRBA	ν_{ba}, Poisson's ratio *ba*
TAU1	τ_1, stress limit of the first slightly nonlinear part of the shear stress as a function of shear strain curve. The values τ_1 and γ_1 help define a shear stress as a function of shear strain curve. Input these values if you set FS to -1 (see Card 3).
GAMMA1	γ_1, strain limit of the first slightly nonlinear part of the shear stress as a function of engineering shear strain curve

Card 2	1	2	3	4	5	6	7	8
Variable	GAB	GBC	GCA	SLIMT1	SLIMC1	SLIMT2	SLIMC2	SLIMS
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
GAB	GT.0.0: G_{ab}, shear modulus in the *ab*-direction LT.0.0: Load Curve ID or Table ID = (-GAB) **Load Curve.** When -GAB is equal to a load curve ID, it is taken as defining the elastic shear stress as a function of shear strain behavior in the *ab*-direction. **Tabular Data.** When -GAB corresponds to a table ID, it defines a load curve ID for each strain rate value. The

VARIABLE	DESCRIPTION
	load curves give the elastic shear stress as a function of shear strain behavior in the *ab*-direction. **Logarithmically Defined Tables.** If the *first* elastic shear stress as a function of shear strain curve in the table corresponds to a negative strain rate, LS-DYNA assumes that the natural logarithm of the strain rate value is used for *all* shear stress-shear strain curves.
GBC	GT.0.0: G_{bc}, shear modulus in the *cb*-direction LT.0.0: Load Curve ID or Table ID = (-GBC) (solids only) **Load Curve.** When -GBC is equal to a load curve ID, it is taken as defining the elastic shear stress as a function of shear strain behavior in the *bc*-direction. **Tabular Data.** When -GBC corresponds to a table ID, it defines a load curve ID for each strain rate value. The load curves give the elastic shear stress as a function of shear strain behavior in the *bc*-direction. **Logarithmically Defined Tables.** If the *first* elastic shear stress as a function of shear strain curve in the table corresponds to a negative strain rate, LS-DYNA assumes that the natural logarithm of the strain rate value is used for *all* shear stress-shear strain curves.
GCA	GT.0.0: G_{ca}, shear modulus in the *ca*-direction LT.0.0: Load Curve ID or Table ID = (-GCA) (solids only) **Load Curve.** When -GCA is equal to a load curve ID, it is taken as defining the elastic shear stress as a function of shear strain behavior in the *ca*-direction. **Tabular Data.** When -GCA refers to a table ID, it defines a load curve ID for each strain rate value. The load curves give the elastic shear stress as a function of shear strain behavior in the *ca*-direction. **Logarithmically Defined Tables.** If the *first* elastic shear stress as a function of shear strain curve in the table corresponds to a negative strain rate, LS-DYNA assumes that the natural logarithm of the strain rate value is used for *all* shear stress-shear strain curves.

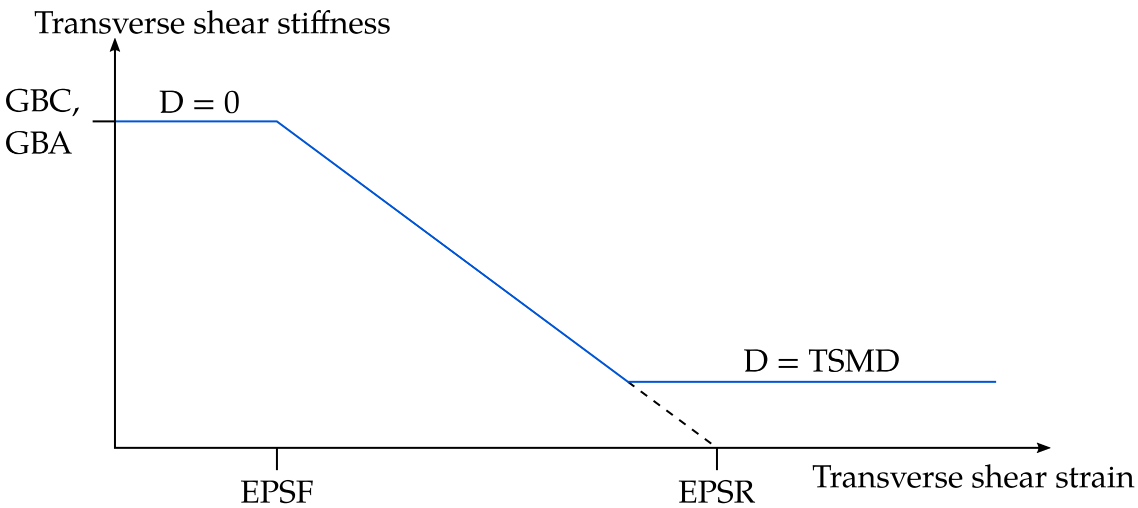


Figure M58-1. Linear Damage for Transverse Shear Behavior

VARIABLE	DESCRIPTION
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 determine the minimum stress limit after stress maximum (shear)

Card 3	1	2	3	4	5	6	7	8
Variable	AOPT	TSIZE	ERODS	SOFT	FS	EPSF	EPSR	TSMD
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
AOPT	Material axes option (see *MAT_002 for a complete description): EQ.0.0: Locally orthotropic with material axes determined by element nodes, as shown in Figure M2-1. For shells only, the material axes are then rotated about the normal

VARIABLE	DESCRIPTION
	vector to the surface of the shell by the angle BETA.
	EQ.1.0: Locally orthotropic with material axes determined by a point, *P*, 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 **a** and **d** input below, as with *DEFINE_COORDINATE_VECTOR
	EQ.3.0: Locally orthotropic material axes determined by a vector **v** and the normal vector to the plane of the element (see Figure M2-1). 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then an angle BETA, which you set in the element's keyword input or the input for this keyword, rotates **a** and **b** about **c**.
	EQ.4.0: Locally orthotropic in a 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.
	LT.0.0: |AOPT| is a coordinate system ID (see *DEFINE_COORDINATE_*OPTION*).
TSIZE	Time step for automatic element deletion
ERODS	Maximum effective strain for element layer failure. A value of unity would equal 100% strain (see Remark 1).
	GT.0.0: Fails when effective strain calculated assuming the material is volume preserving exceeds ERODS (old way)
	LT.0.0: Fails when effective strain calculated from the full strain tensor exceeds |ERODS|
SOFT	Softening reduction factor for strength in the crashfront (see Remark 3)

VARIABLE	DESCRIPTION
FS	Failure surface type (see Remarks 4 and 5): EQ.1.0: Smooth failure surface with a quadratic criterion for both the fiber (a) and transverse (b) directions. This option can be used with complete laminates and fabrics. EQ.0.0: Smooth failure surface in the transverse (*b*) direction with a limiting value in the fiber (*a*) direction. This model is appropriate for unidirectional (UD) layered composites only. EQ.-1.0: Faceted failure surface. When the strength values are reached, damage evolves in tension and compression for the fiber and transverse direction. Shear behavior is also considered. This option can be used with complete laminates and fabrics.
EPSF	Damage initiation transverse shear strain
EPSR	Final rupture transverse shear strain
TSMD	Transverse shear maximum damage; default = 0.90.

Card 4	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3	PRCA	PRCB
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
XP, YP, ZP	Coordinates of point p for AOPT = 1 and 4
A1, A2, A3	Components of vector **a** for AOPT = 2
PRCA	ν_{ca}, Poisson's ratio *ca* (default = PRBA)
PRCB	ν_{cb}, Poisson's ratio *cb* (default = PRBA)

Card 5	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	D1	D2	D3	BETA	LCDFAIL
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
V1, V2 V3	Components of vector **v** for AOPT = 3 and 4
D1, D2, D3	Components of vector **d** for AOPT = 2
BETA	Angle in degrees of a material rotation about the c-axis, available for AOPT = 0 (shells and tshells only) and AOPT = 3 (all element types). This angle may be overridden on the element card; see *ELEMENT_SHELL_BETA, *ELEMENT_TSHELL_BETA, and *ELEMENT_SOLID_ORTHO.
LCDFAIL	Load curve ID, which defines orientation-dependent failure strains. The ordinate values in the load curve define the various failure strains in the following order: 1. EF_11T: tensile failure strain in longitudinal *a*-direction 2. EF_11C: compressive failure strain in longitudinal *a*-direction 3. EF_22T: tensile failure strain in transverse *b*-direction 4. EF_22C: compressive failure strain in transverse *b*-direction 5. EF_12: in-plane shear failure strain in *ab*-plane 6. EF_33T: tensile failure strain in transverse *c*-direction 7. EF_33C: compressive failure strain in transverse *c*-direction 8. EF_23: out-of-plane shear failure strain in *bc*-plane 9. EF_31: out-of-plane shear failure strain in *ca*-plane Thus, the load curve to define these values must have either five (shells) or nine (solids) entries in its definition. You may input a

VARIABLE	DESCRIPTION
	load curve with nine entries for shells, but LS-DYNA ignores the last four entries. The ignored abscissa values need to be ascending, such as 1.0, 2.0, …, 9.0.

Card 6	1	2	3	4	5	6	7	8
Variable	E11C	E11T	E22C	E22T	GMS			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
E11C	Strain at longitudinal compressive strength, *a*-axis (positive)
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	Engineering shear strain at shear strength, *ab*-plane

Card 7	1	2	3	4	5	6	7	8
Variable	XC	XT	YC	YT	SC			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
XC	Longitudinal compressive strength (positive value); see Remark 2.
XT	Longitudinal tensile strength; see Remark 2.
YC	Transverse compressive strength, *b*-axis (positive value); see Remark 2.
YT	Transverse tensile strength, *b*-axis; see Remark 2.
SC	Shear strength, *ab*-plane; see below Remark 2.

Card 8.1 for SOLID Keyword Option.

Card 8.1	1	2	3	4	5	6	7	8
Variable	E33C	E33T	GMS23	GMS31				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
E33C	Strain at transverse compressive strength, *c*-axis.
E33T	Strain at transverse tensile strength, *c*-axis.
GMS23	Engineering shear strain at shear strength, *bc*-plane.
GMS31	Engineering shear strain at shear strength, *ca*-plane.

Card 8.2 for SOLID Keyword Option.

Card 8.2	1	2	3	4	5	6	7	8
Variable	ZC	ZT	SC23	SC31				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
ZC	Transverse compressive strength, *c*-axis (positive value).
ZT	Transverse tensile strength, *c*-axis.
SC23	Shear strength, *bc*-plane.
SC31	Shear strength, *ca*-plane.

Card 8.3 for SOLID Keyword Option.

Card 8.3	1	2	3	4	5	6	7	8
Variable	SLIMT3	SLIMC3	SLIMS23	SLIMS31	TAU2	GAMMA2	TAU3	GAMMA3
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
SLIMT3	Factor to determine the minimum stress limit after stress maximum (matrix tension, *c*-axis).
SLIMC3	Factor to determine the minimum stress limit after stress maximum (matrix compression, *c*-axis).
SLIMS23	Factor to determine the minimum stress limit after stress maximum (shear, *bc*-plane).
SLIMS31	Factor to determine the minimum stress limit after stress maximum (shear, *ca*-plane).
TAU2	τ_2, stress limit of the first slightly nonlinear part of the shear stress as a function of shear strain curve. The values τ_2 and γ_2 are used to define a shear stress as a function of shear strain curve. Input these values if FS = -1 (see Card 3). These values are for the *bc*-plane.
GAMMA2	γ_2, strain limit of the first slightly nonlinear part of the shear stress as a function of engineering shear strain curve (*bc*-plane).
TAU3	τ_3, stress limit of the first slightly nonlinear part of the shear stress as a function of shear strain curve. The values τ_3 and γ_3 help define a shear stress as a function of shear strain curve. Input these values if FS = -1 on Card 3 (*ca*-plane).
GAMMA3	γ_3, strain limit of the first slightly nonlinear part of the shear stress as a function of engineering shear strain curve (*bc*-plane).

First Optional Strain Rate Dependence Card. (shells and solids)

Card 9	1	2	3	4	5	6	7	8
Variable	LCXC	LCXT	LCYC	LCYT	LCSC	LCTAU	LCGAM	DT
Type	I	I	I	I	I	I	I	F

VARIABLE	DESCRIPTION
LCXC	Load curve ID defining longitudinal compressive strength XC as a function of strain rate (XC is ignored with that option). If the first strain rate value in the curve is negative, LS-DYNA assumes that

VARIABLE	DESCRIPTION
	you input all the strain rate values as the natural logarithm of the strain rate.
LCXT	Load curve ID defining longitudinal tensile strength XT as a function of strain rate (XT is ignored with this option). If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
LCYC	Load curve ID defining transverse compressive strength YC as a function of strain rate (YC is ignored with this option). If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
LCYT	Load curve ID defining transverse tensile strength YT as a function of strain rate (YT is ignored with this option). If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
LCSC	Load curve ID defining shear strength SC as a function of strain rate (SC is ignored with this option). If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
LCTAU	Load curve ID defining TAU1 as a function of strain rate (TAU1 is ignored with this option). This value is only used for FS = -1. If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
LCGAM	Load curve ID defining GAMMA1 as a function of strain rate (GAMMA1 is ignored with this option). This value is only used for FS = -1. If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
DT	Strain rate averaging option: EQ.0.0: Strain rate is evaluated using a running average. LT.0.0: Strain rate is evaluated using the average over the last 11 time steps. GT.0.0: Strain rate is averaged over the last DT time units.

Second Optional Strain Rate Dependence Card. (shells and solids)

Card 10	1	2	3	4	5	6	7	8
Variable	LCE11C	LCE11T	LCE22C	LCE22T	LCGMS	LCEFS		
Type	I	I	I	I	I	I		

VARIABLE	DESCRIPTION
LCE11C	Load curve ID defining E11C as a function of strain rate (E11C is ignored with this option). If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
LCE11T	Load curve ID defining E11T as a function of strain rate (E11T is ignored with this option). If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
LCE22C	Load curve ID defining E22C as a function of strain rate (E22C is ignored with this option). If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
LCE22T	Load curve ID defining E22T as a function of strain rate (E22T is ignored with this option). If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
LCGMS	Load curve ID defining GMS as a function of strain rate (GMS is ignored with this option). If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.
LCEFS	Load curve ID defining ERODS as a function of strain rate (ERODS is ignored with this option). LS-DYNA uses the full strain tensor to compute the equivalent strain. If the first strain rate value in the curve is negative, LS-DYNA assumes that you input all the strain rate values as the natural logarithm of the strain rate.

Third Optional Strain Rate Dependence Card. (solid only!)

Card 11	1	2	3	4	5	6	7	8
Variable	LCZC	LCZT	LCSC23	LCSC31	LCTAU2	LCGAM2	LCTAU3	LCGAM3
Type	I	I	I	I	I	I	I	I

VARIABLE	DESCRIPTION
LCZC	Load curve ID defining transverse compressive strength ZC as a function of strain rate (ZC is ignored with that option). If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.
LCZT	Load curve ID defining transverse tensile strength ZT as a function of strain rate (ZT is ignored with this option). If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.
LCSC23	Load curve ID defining shear strength SC23 as a function of strain rate (SC23 is ignored with this option). If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.
LCSC31	Load curve ID defining shear strength SC31 as a function of strain rate (SC31 is ignored with this option). If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.
LCTAU2	Load curve ID defining TAU2 as a function of strain rate (TAU2 is ignored with this option). This value is only used for FS = -1. If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.
LCGAM2	Load curve ID defining GAMMA2 as a function of strain rate (GAMMA2 is ignored with this option). This value is only used for FS = -1. If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.
LCTAU3	Load curve ID defining TAU3 as a function of strain rate (TAU3 is ignored with this option). This value is only used for FS = -1. If the first strain rate value in the curve is negative, all the strain rate

VARIABLE	DESCRIPTION
	values are assumed to be given as the natural logarithm of the strain rate.
LCGAM3	Load curve ID defining GAMMA3 as a function of strain rate (GAMMA3 is ignored with this option). This value is only used for FS = -1. If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.

Fourth Optional Strain Rate Dependence Card. (solids only!)

Card 12	1	2	3	4	5	6	7	8
Variable	LCE33C	LCE33T	LCGMS23	LCGMS31				
Type	I	I	I	I				

VARIABLE	DESCRIPTION
LCE33C	Load curve ID defining E33C as a function of strain rate (E33C is ignored with this option). If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.
LCE33T	Load curve ID defining E33T as a function of strain rate (E33T is ignored with this option). If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.
LCGMS23	Load curve ID defining GMS23 as a function of strain rate (GMS23 is ignored with this option). If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.
LCGMS31	Load curve ID defining GMS31 as a function of strain rate (GMS31 is ignored with this option). If the first strain rate value in the curve is negative, all the strain rate values are assumed to be given as the natural logarithm of the strain rate.

Remarks:

1. **Failure of an element layer.** ERODS, the maximum effective strain, controls the failure of an element layer. The maximum value of ERODS, 1, is 100% straining. The layer in the element is completely removed after the maximum effective strain (compression/tension including shear) is reached.

2. **Stress limits.** The stress limits are factors used to limit the stress in the softening part to a given value,

$$\sigma_{\min} = \text{SLIM}xx \times \text{strength}.$$

Thus, the damage value is slightly modified to achieve elastoplastic-like behavior with the threshold stress. The SLIM*xx* fields may range between 0.0 and 1.0. With a factor of 1.0, the stress remains at a maximum value identical to the strength, similar to ideal elastoplastic behavior. A small value for SLIMT*x* is often reasonable for tensile failure; however, SLIMCx = 1.0 is preferred for compression. This is also valid for the corresponding shear value.

If SLIM*xx* is smaller than 1.0, then localization can be observed depending on the total behavior of the layer. If intentionally using SLIM*xx* < 1.0, we generally recommend avoiding a drop to zero and setting the value to something between 0.05 and 0.10. Then elastoplastic behavior is achieved in the limit, which often leads to fewer numerical problems. The defaults for SLIM*xx* are 10^{-8}.

3. **Crashfront.** To start the crashfront algorithm, input a value for TSIZE. Note that the time step size, with element elimination after the actual time step, becomes smaller than TSIZE.

4. **Damage.** The damage parameters can be written to the post-processing 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 better for complete laminates and fabrics, as all directions are treated similarly.

For FS = 1, the model assumes an interaction between the normal and shear stresses for damage evolution in the *a* and *b*-directions. The shear damage is always the maximum damage value from the criterion in the *a* or *b*-direction.

For FS = -1, we assume that the damage evolution is independent of any of the other stresses. The elastic material parameters and the complete structure provide the only coupling. In the tensile and compression directions, as well as in the *b*-direction, the material can have different failure surfaces. The damage values monotonically increase. Thus, a load reversal from tension to compression, or compression to tension, does not reduce damage.

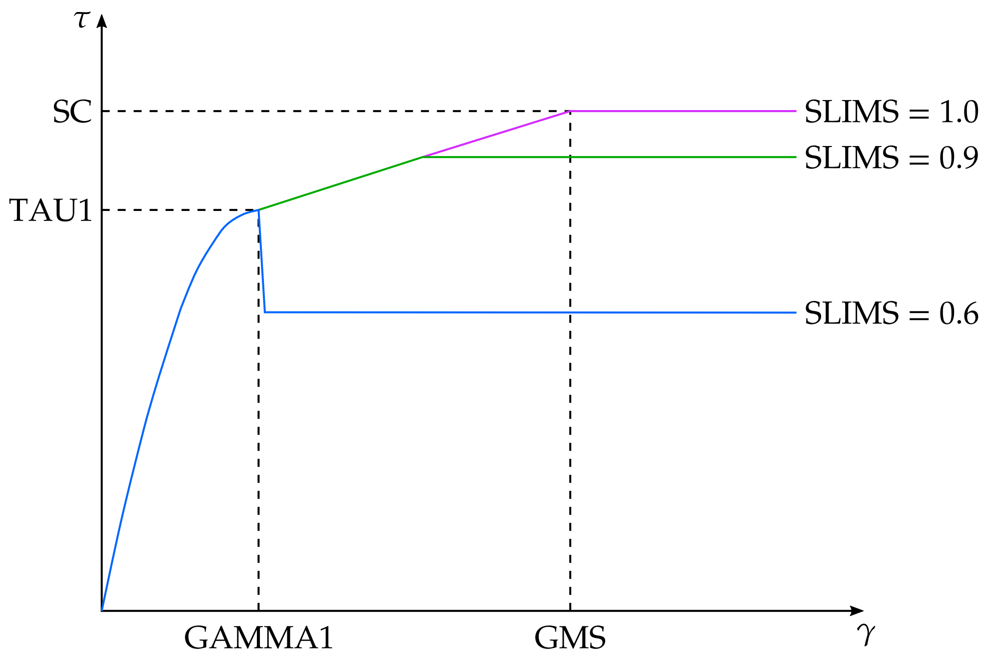


Figure M58-2. Stress-strain diagram for shear

5. **Shear failure of fabrics.** For fabric materials, we can assume a nonlinear stress-strain curve for the shear part for failure surface FS = -1, as given below. This is not possible for other values of FS.

   Three points define the curve as shown in Figure M58-2:

   a) the origin (0,0) is assumed,

   b) the limit of the first slightly nonlinear part (must be input), stress (TAU1) and strain (GAMMA1), and

   c) the shear strength at failure and shear strain at failure.

   In addition, a stress limiter can be used to keep the stress constant using the SLIMS field. This value must be positive and less than or equal to 1.0. It leads to elastoplastic behavior for the shear part. The default is 10^{-8}, assuming almost brittle failure once the strength limit SC is reached.

6. **EC.** The EC field is ignored when thin shells use this material model. When used with thick shell elements of form 1, 2, or 6, a positive EC value will be used to evaluate thickness stress. If EC is set to zero or a negative number, then the minimum of EA and EB is used for the thickness stress calculation.

7. **Strain rate.** LS-DYNA uses the smoothed, direction-appropriate strain rate for any property specified to be strain-rate-dependent. For example, LS-DYNA uses strain rate in the a-direction when assessing properties in the a-direction. LS-

DYNA, however, uses the effective strain rate when determining the rate-dependence of ERODS for load curve LCEFS.

8. **EA / EB / EC < 0.0.** If a load curve specifies the uniaxial elastic stress as a function of strain behavior, the range of the strain space (abscissa values) must span from at least 5% negative (compressive) to 5% positive (tensile) strain.

*MAT_COMPOSITE_FAILURE_*OPTION*_MODEL

This is Material Type 59.

Available options include:

SHELL

SOLID

SPH

depending on the element type the material is to be used with; see *PART. An equation of state (*EOS) is optional for SPH elements and is invoked by setting EOSID to a nonzero value in *PART. If an equation of state is used, only the deviatoric stresses are calculated by the material model and the pressure is calculated by the equation of state.

Card Summary:

Card 1. This card is required.

MID	RO	EA	EB	EC	PRBA	PRCA	PRCB

Card 2. This card is required.

GAB	GBC	GCA	KF	AOPT	MACF		

Card 3. This card is required.

XP	YP	ZP	A1	A2	A3		

Card 4. This card is required.

V1	V2	V3	D1	D2	D3	BETA	

Card 5a.1. This card is included if the SHELL keyword option is used.

TSIZE	ALP	SOFT	FBRT	SR	SF		

Card 5a.2. This card is included if the SHELL keyword option is used.

XC	XT	YC	YT	SC			

Card 5b.1. This card is included if either the SOLID or SPH keyword option is used.

SBA	SCA	SCB	XXC	YYC	ZZC		

Card 5b.2. This card is included if either the SOLID or SPH keyword option is used.

XXT	YYT	ZZT					

Data Card Definition:

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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*

Card 2	1	2	3	4	5	6	7	8
Variable	GAB	GBC	GCA	KF	AOPT	MACF		
Type	F	F	F	F	F	I		

VARIABLE	DESCRIPTION
GAB	G_{ab}, shear modulus
GBC	G_{bc}, shear modulus

VARIABLE	DESCRIPTION
GCA	G_{ca}, shear modulus
KF	Bulk modulus of failed material
AOPT	Material axes option (see MAT_*OPTION*TROPIC_ELASTIC, particularly the Material Directions section, for details): EQ.0.0: Locally orthotropic with material axes determined by element nodes 1, 2, and 4, as with *DEFINE_COORDINATE_NODES. For shells only, the material axes are then rotated about the normal vector to the surface of the shell by the angle BETA. EQ.1.0: Locally orthotropic with material axes determined by a point, P, 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 a vector **v** and the normal vector to the plane of the element. 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. Thus, for solid elements, AOPT = 3 is only available for hexahedrons. **a** is determined by taking the cross product of **v** with the normal vector, **b** is determined by taking the cross product of the normal vector with **a**, and **c** is the normal vector. Then **a** and **b** are rotated about **c** by an angle BETA. BETA may be set in the keyword input for the element or in the input for this keyword. Note that for solids, the material axes may be switched depending on the choice of MACF. The switch may occur before or after applying BETA depending on the value of MACF. EQ.4.0: Locally orthotropic in a 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. LT.0.0: The absolute value of AOPT is a coordinate system ID number (CID on *DEFINE_COORDINATE_*OPTION*).

VARIABLE	DESCRIPTION
MACF	Material axes change flag for solid elements: EQ.-4: Switch material axes *b* and *c* before BETA rotation EQ.-3: Switch material axes *a* and *c* before BETA rotation EQ.-2: Switch material axes *a* and *b* before BETA rotation EQ.1: No change, default EQ.2: Switch material axes *a* and *b* after BETA rotation EQ.3: Switch material axes *a* and *c* after BETA rotation EQ.4: Switch material axes *b* and *c* after BETA rotation Figure M2-2 indicates when LS-DYNA applies MACF during the process to obtain the final material axes. If BETA on *ELEMENT_SOLID_*{OPTION}* is defined, then that BETA is used for the rotation for all AOPT options. Otherwise, if AOPT = 3, the BETA input on Card 4 rotates the axes. For all other values of AOPT, the material axes will be switched as specified by MACF, but no BETA rotation will be performed.

Card 3	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	A1	A2	A3		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
XP YP ZP	Coordinates of point *p* for AOPT = 1 and 4
A1 A2 A3	Components of vector **a** for AOPT = 2

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
V1 V2 V3	Components of vector **v** for AOPT = 3 and 4
D1 D2 D3	Components of vector **d** for AOPT = 2
BETA	Material angle in degrees for AOPT = 0 (shells only) and AOPT = 3. BETA may be overridden on the element card; see *ELEMENT_-SHELL_BETA or *ELEMENT_SOLID_ORTHO.

Card 5 for SHELL Keyword Option.

Card 5a.1	1	2	3	4	5	6	7	8
Variable	TSIZE	ALP	SOFT	FBRT	SR	SF		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
TSIZE	Time step for automatic element deletion
ALP	Nonlinear shear stress parameter
SOFT	Softening reduction factor for material strength in crashfront elements
FBRT	Softening of fiber tensile strength
SR	s_r, reduction factor (default = 0.447)
SF	s_f, softening factor (default = 0.0)

Card 6 for SHELL Keyword Option.

Card 5a.2	1	2	3	4	5	6	7	8
Variable	XC	XT	YC	YT	SC			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
XC	Longitudinal compressive strength, a-axis (positive value)

VARIABLE	DESCRIPTION
XT	Longitudinal tensile strength, *a*-axis
YC	Transverse compressive strength, *b*-axis (positive value)
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

Card 5 for SPH and SOLID Keyword Options.

Card 5b.1	1	2	3	4	5	6	7	8
Variable	SBA	SCA	SCB	XXC	YYC	ZZC		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
SBA	In plane shear strength
SCA	Transverse shear strength
SCB	Transverse shear strength
XXC	Longitudinal compressive strength *a*-axis (positive value)
YYC	Transverse compressive strength *b*-axis (positive value)
ZZC	Normal compressive strength *c*-axis (positive value)

Card 6 for SPH and SOLID Keyword Options.

Card 5b.2	1	2	3	4	5	6	7	8
Variable	XXT	YYT	ZZT					
Type	F	F	F					

VARIABLE	DESCRIPTION
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 (such as 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 1	1	2	3	4	5	6	7	8
Variable	MID	RO	V0	A	B	C	LCID	
Type	A	F	F	F	F	F	F	

Card 2	1	2	3	4	5	6	7	8
Variable	PR1	PR2	PR3	PR4	PR5	PR6	PR7	PR8
Type	F	F	F	F	F	F	F	F

Card 3	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 4	1	2	3	4	5	6	7	8
Variable	V1	V2	V3	V4	V5	V6	V7	V8
Type	F	F	F	F	F	F	F	F

Card 5	1	2	3	4	5	6	7	8
Variable	E1	E2	E3	E4	E5	E6	E7	E8
Type	F	F	F	F	F	F	F	F

Card 6	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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
V0	Temperature independent dynamic viscosity coefficient, V_0. If defined, the temperature dependent viscosity defined below is skipped; see type i and ii definitions for viscosity below.
A	Dynamic viscosity coefficient; see type i and ii definitions below.
B	Dynamic viscosity coefficient; see type i and ii definitions below.
C	Dynamic viscosity coefficient; see type i and ii definitions below.
LCID	Load curve (see *DEFINE_CURVE) defining viscosity as a function of temperature; see type iii. (Optional.)
T1, T2, …, TN	Temperatures, T_i, define up to 8 values
PR1, PR2, …, PRN	Poisson's ratios for the temperatures T_i
V1, V2, …, VN	Corresponding dynamic 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)

VARIABLE	DESCRIPTION
ALPHA1, …, ALPHAN.	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:

$$\dot{\boldsymbol{\varepsilon}}'_{\text{total}} = \dot{\boldsymbol{\varepsilon}}'_{\text{elastic}} + \dot{\boldsymbol{\varepsilon}}'_{\text{viscous}} = \frac{\dot{\boldsymbol{\sigma}}'}{2G} + \frac{\boldsymbol{\sigma}'}{2v} ,$$

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

$$d\sigma' = 2G\dot{\varepsilon}_{\text{total}}dt - \frac{G}{v}dt\,\boldsymbol{\sigma}' .$$

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

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

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

$$\begin{aligned}\dot{\varepsilon}_{33} &= -a(\dot{\varepsilon}_{11} + \dot{\varepsilon}_{22}) + bp \\ a &= \frac{\left(K - \frac{2}{3}G\right)}{\left(K + \frac{4}{3}G\right)} \\ b &= \frac{Gdt}{v\left(K + \frac{4}{3}G\right)}\end{aligned}$$

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) $V = V_0 \times 10^{\left(\frac{A}{T-B}+C\right)}$

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

NOTE: Viscosity is inactive during dynamic relaxation.

*MAT_ELASTIC_WITH_VISCOSITY_CURVE

This is Material Type 60 which was developed to simulate the forming of glass products (such as 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. Load curves are used to represent the temperature dependence of Poisson's ratio, Young's modulus, the coefficient of expansion, and the viscosity.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	V0	A	B	C	LCID	
Type	A	F	F	F	F	F	F	

Card 2	1	2	3	4	5	6	7	8
Variable	PR_LC	YM_LC	A_LC	V_LC	V_LOG			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
V0	Temperature independent dynamic viscosity coefficient, V_0. If defined, the temperature dependent viscosity curve, V_LC, is skipped; see type i and ii definitions for viscosity below.
A	Dynamic viscosity coefficient; see type i and ii definitions below.
B	Dynamic viscosity coefficient; see type i and ii definitions below.
C	Dynamic viscosity coefficient; see type i and ii definitions below.
LCID	Load curve (see *DEFINE_CURVE) defining factor on dynamic viscosity as a function of temperature; see type iii. (Optional).

VARIABLE	DESCRIPTION
PR_LC	Load curve (see *DEFINE_CURVE) defining Poisson's ratio as a function of temperature.
YM_LC	Load curve (see *DEFINE_CURVE) defining Young's modulus as a function of temperature.
A_LC	Load curve (see *DEFINE_CURVE) defining the coefficient of thermal expansion as a function of temperature.
V_LC	Load curve (see *DEFINE_CURVE) or table for defining the dynamic viscosity GT.0: Load curve ID for defining dynamic viscosity as a function of temperature LT.0: |V_LC| is table ID giving dynamic viscosity as a function of shear strain rate and temperature. The dynamic viscosity as a function of temperature curves are indexed by the shear strain rate.
V_LOG	Flag for the form of V_LC: EQ.1.0: The value specified in V_LC is the natural logarithm of the viscosity, $\ln V$. The value interpolated from the curve is then exponentiated to obtain the viscosity. The logarithmic form is useful if the value of the viscosity changes by orders of magnitude over the temperature range of the data. EQ.0.0: The value specified in V_LC is the viscosity.

Remarks:

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

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

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

$$d\boldsymbol{\sigma}' = 2G\dot{\boldsymbol{\varepsilon}}'_{\text{total}}\,dt - \frac{G}{v}\,dt\,\boldsymbol{\sigma}'$$

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

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

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

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

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

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

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 curve, V_LC.

ii) $V = V_0 \times 10^{\left(\frac{A}{T-B}+C\right)}$

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

NOTE: Viscosity is inactive during dynamic relaxation.

*MAT_KELVIN-MAXWELL_VISCOELASTIC

This is Material Type 61. This material is a classical Kelvin-Maxwell model for modeling viscoelastic bodies, such as foams. This model is valid for solid elements only. See Remarks below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	G0	GI	DC	FO	SO
Type	A	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 or label must be specified (see *PART).
RO	Mass density
BULK	Bulk modulus (elastic)
G0	Short-time shear modulus, G_0
GI	Long-time (infinite) shear modulus, G_∞
DC	Constant depending on formulation: FO.EQ.0.0: Maxwell decay constant FO.EQ.1.0: Kelvin relaxation constant
FO	Formulation option: EQ.0.0: Maxwell EQ.1.0: Kelvin
SO	Strain (logarithmic) output option to control what is written as component 7 to the d3plot database. (LS-PrePost always blindly labels this component as effective plastic strain.) The maximum values are updated for each element each time step: EQ.0.0: Maximum principal strain that occurs during the calculation

VARIABLE	DESCRIPTION
	EQ.1.0: Maximum magnitude of the principal strain values that occurs during the calculation
	EQ.2.0: Maximum effective strain that occurs during the calculation

Remarks:

The shear relaxation behavior is described for the Maxwell model by:

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

A Jaumann rate formulation is used

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

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

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

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

*MAT_VISCOUS_FOAM

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

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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 Remarks below.
N2	Exponent in power law for viscosity, n_2
PR	Poisson's ratio, ν

Remarks:

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

$$E_1^t = E_1(V^{-n_1})$$
$$V_2^t = V_2|1 - V|^{n_2}$$

Here, V is the relative volume defined by the ratio of the current to initial volume. Viscosity generates a shear stress given by

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

$\dot{\gamma}$ is the engineering shear strain rate.

Table showing typical values (units of N, mm, s):

Variable	Value
E1	0.0036
N1	4.0
V2	0.0015
E2	100.0
N2	0.2
PR	0.05

*MAT_CRUSHABLE_FOAM

This is Material Type 63. This material type models crushable foam with optional damping and tension cutoff. Unloading is fully elastic. The model treats tension as elastic-perfectly-plastic at the tension cut-off value. A modified version of this model, *MAT_-MODIFIED_CRUSHABLE_FOAM, includes strain rate effects.

Setting MODEL = 1 or 2 on Card 1 invokes alternative formulations for modeling crushable foam. They both incorporate an elliptical yield surface in *p*-*q* space and include independent definitions of elastic and plastic Poisson's ratio. They also both support rate dependence. See Remarks 2 and 3 for further details.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	LCID	TSC	DAMP	MODEL
Type	A	F	F	F	F	F	F	I
Default	none	none	none	none	none	0.0	0.10	0

Optional card.

Card 2	1	2	3	4	5	6	7	8
Variable	PRP	K	RFILTF	BVFLAG	SRCRT	ESCAL	KT	
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 or label must be specified (see *PART).
RO	Mass density
E	Young's modulus. For MODEL = 0, E may affect contact stiffness but otherwise is not used. The final slope of the curve LCID determines the elastic stiffness for loading and unloading. The time step calculation also uses this slope. For MODEL = 1 or 2, the material law uses E as the Young's modulus.

VARIABLE	DESCRIPTION
PR	(Elastic) Poisson's ratio
LCID	MODEL.EQ.0: Load curve ID defining yield stress as a function of volumetric strain, γ (see Figure M63-1). MODEL.GE.1: Load curve, table ID, or 3D table ID. If specifying a load curve ID, the load curve defines uniaxial yield stress under compression, σ_c, as a function of equivalent plastic strain. If specifying a table ID, each strain rate references a load curve ID that gives uniaxial yield stress as a function of equivalent plastic strain. If specifying a 3D table ID, uniaxial yield stress is given as a function of history variable #8 (3D table), strain rate (table), and equivalent plastic strain (curve).
TSC	Tensile stress cutoff (only for MODEL = 0). A nonzero, positive value is strongly recommended for realistic behavior.
DAMP	Rate sensitivity via damping coefficient (.05 < recommended value < .50). Only available for MODEL = 0.
MODEL	Choice of material model formulation: EQ.0: Original approach (default), EQ.1: Elliptical yield surface in *p*-*q* space with *symmetric* tension-compression behavior (isotropic hardening), EQ.2: Elliptical yield surface in *p*-*q* space with *asymmetric* tension-compression behavior (volumetric hardening).
PRP	Plastic Poisson's ratio (only for MODEL = 1 or 2). PRP determines the yield potential, that is, the plastic flow direction. It ranges from -1 to 0.5.
K	Ratio of σ_c^0, initial uniaxial yield stress, to p_c^0, initial hydrostatic yield stress under compression (only for MODEL = 1 or 2). K determines the shape of the yield ellipse.
RFILTF	Rate filtering parameter for MODEL = 1 or 2 (0.0 ≤ RFILTF < 1.0): EQ.0.0: Plastic strain rates are used if LCID is a table (default). GT.0.0: Filtered total strain rates are used if LCID is a table. See Remark 2.

VARIABLE	DESCRIPTION
BVFLAG	Bulk viscosity deactivation flag (for MODEL = 1 or 2): EQ.0.0: Bulk viscosity active (default). EQ.1.0: No bulk viscosity
SRCRT	Critical stretch ratio for high compression regime (MODEL = 1 or 2). For instance, a value of 0.1 for SRCRT means adding an elastic stiffness at 10% residual length to avoid excessive compression.
ESCAL	Scale factor for high compression stiffness as a multiple of the Young's modulus (MODEL = 1 or 2).
KT	Ratio of p_t, absolute yield stress in hydrostatic tension, to p_c^0, initial yield stress in hydrostatic compression (only for MODEL = 2). KT defines the shift of the yield ellipse in the direction of the p-axis. With KT = 1, the initial yield ellipse is symmetric (with respect to the q-axis), but it always becomes unsymmetric through hardening.

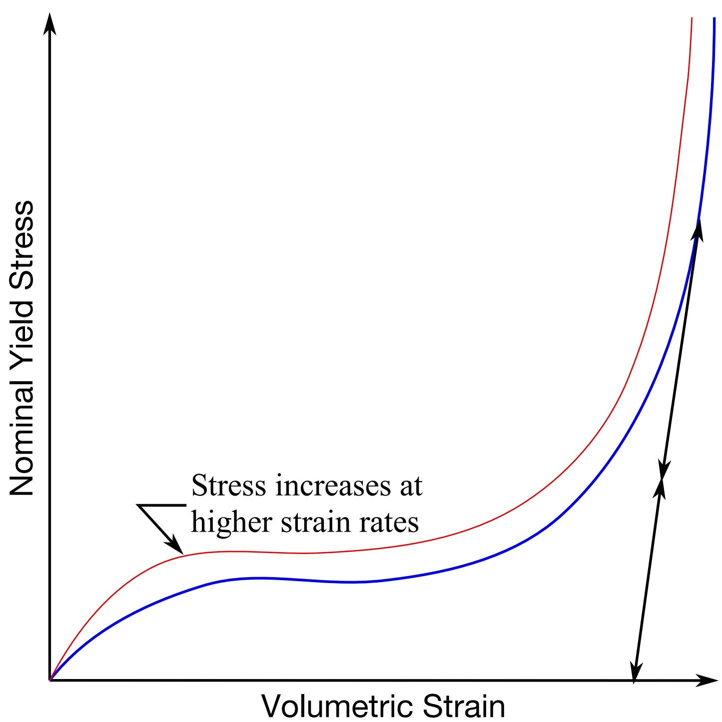


Figure M63-1. Behavior of strain rate sensitive crushable foam. Unloading is elastic to the tension cutoff. Subsequent reloading follows the unloading curve.

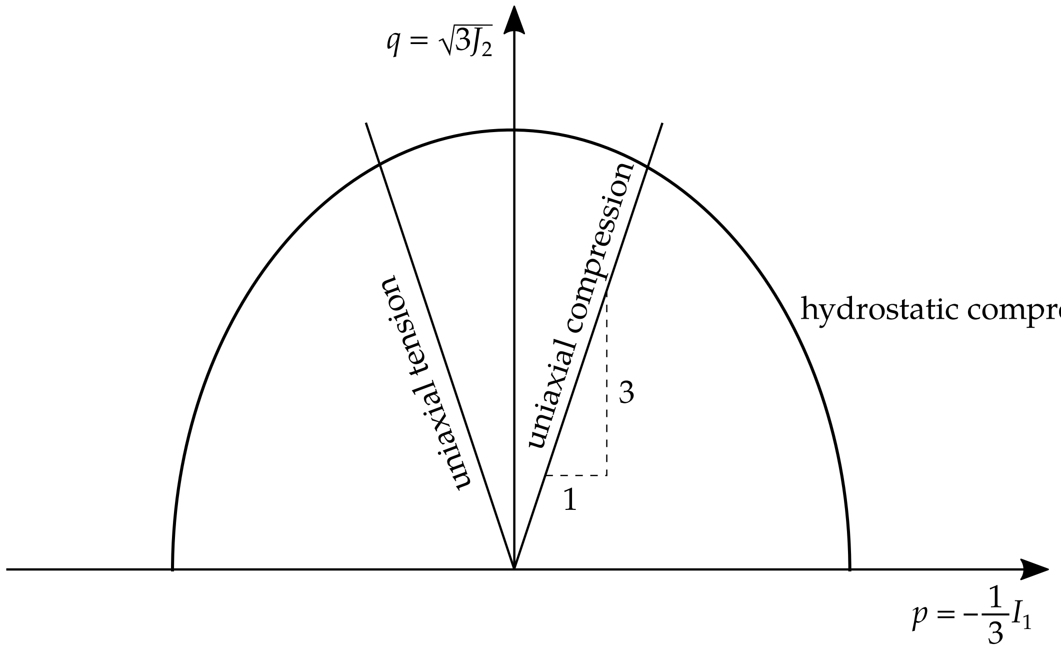


Figure M63-2. Yield surface for MODEL = 1 in *p*-*q* space

Remarks:

1. **Volumetric strain.** The volumetric strain is defined in terms of the relative volume, V, as:

$$\gamma = 1 - V$$

The relative volume is the ratio of the current volume to the initial volume. In place of the effective plastic strain in the d3plot database, the integrated volumetric strain (natural logarithm of the relative volume) is output.

2. **Symmetric elliptical yield surface formulation.** Setting MODEL = 1 invokes an alternative formulation for crushable with the following yield condition:

$$F = \sqrt{q^2 + \alpha^2 p^2} - Y_s = 0 \ ,$$

This yield condition corresponds to an elliptical yield surface in the pressure (p) – deviator Mises stress (q) space; see Figure M63-2. In the above yield condition,

$$p = -\frac{1}{3}I_1$$

$$q = \sqrt{3J_2}$$

$$\alpha = \frac{3k}{\sqrt{9 - k^2}}$$

$$Y_s = \sigma_c \sqrt{1 + \left(\frac{\alpha}{3}\right)^2}$$

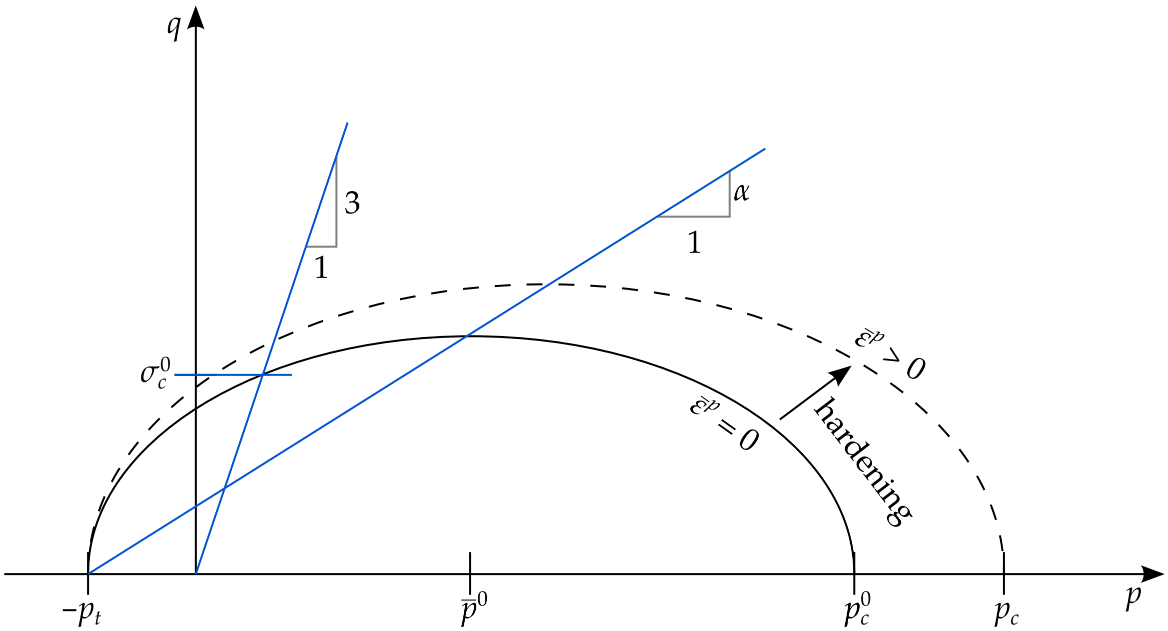


Figure M63-3. Yield surface for MODEL = 2 in p-q space

Y_s, the yield stress, gives the size of the elliptical yield surface. k, the stress ratio, is given by

$$k = \frac{\sigma_{\mathrm{c}}^0}{p_{\mathrm{c}}^0} \ .$$

k describes the shape of the yield surface and is input as field K. It ranges from 0 (von Mises) to less than 3.

For lateral straining, define individual Poisson's ratios for the elastic (PR) and the plastic (PRP) regimes. The associated flow potential is given by

$$G = \sqrt{q^2 + \beta^2 p^2} \ ,$$

where

$$\beta = \frac{3}{\sqrt{2}} \sqrt{\frac{1 - 2\nu^{\mathrm{pl}}}{1 + \nu^{\mathrm{pl}}}}$$

with plastic Poisson's ratio ν^{pl} (PRP). A yield curve or table specified with LCID defines the hardening. If LCID is a yield curve, it relates uniaxial yield stress, σ_c, as a function of equivalent plastic strain. To consider rate dependence, make LCID a table. RFILTF determines if the algorithm uses plastic strain rates (RFILT = 0.0) or filtered total strain rates (0.0 < RFILTF < 1.0). In the latter case, we use exponential smoothing:

$$\dot{\varepsilon}^{\mathrm{avg}} = \mathrm{RFILTF} \times \dot{\varepsilon}_{n-1}^{\mathrm{avg}} + (1 - \mathrm{RFILTF}) \times \dot{\varepsilon}_n^{\mathrm{cur}} \ ,$$

Thus, as RFILTF increases, more filtering occurs.

3. **Asymmetric elliptical yield surface formulation.** Setting MODEL = 2 invokes another formulation for crushable foam with different plastic deformation behavior under tension and compression. It has the following yield condition:

$$F = \sqrt{q^2 + \alpha^2 (p - \bar{p})^2} - Y_s = 0 \ ,$$

corresponding to an elliptical yield surface in the pressure (p) – deviator Mises stress (q) space with its center at $\bar{p} = (p_c - p_t)/2$; see Figure M63-3. In the above yield condition,

$$p = -\frac{1}{3} I_1$$
$$q = \sqrt{3 J_2}$$
$$\alpha = \frac{3k}{\sqrt{(3k_t + k)(3 - k)}}$$
$$Y_s = \alpha \frac{p_c + p_t}{2}$$

The yield stress, Y_s, gives the size of the elliptical yield surface. The yield stress in hydrostatic compression, p_c, is a function of the uniaxial yield stress, σ_c, through this equation:

$$p_c = \frac{\sigma_c\left(\sigma_c\left(\frac{1}{\alpha^2} + \frac{1}{9}\right) + \frac{1}{3} p_t\right)}{p_t + \frac{1}{3}\sigma_c} \ .$$

The following equations give the stress ratios k and k_t:

$$k = \frac{\sigma_c^0}{p_c^0} \ , \qquad k_t = \frac{p_t}{p_c^0} \ .$$

k describes the shape of the yield surface. Input field K sets k. It ranges from 0 (von Mises) to less than 3. k_t describes the shift of the yield ellipse on the p-axis. Input field KT sets k_t.

The flow potential G is the same as for MODEL = 1. The other statements from Remark 2 on plastic Poisson's ratio PRP, yield curve input with LCID, and strain rate filtering with RFILTF also hold for MODEL = 2.

*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 1	1	2	3	4	5	6	7	8
Variable	MID	R0	E	PR	K	M	N	E0
Type	A	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0001	none	0.0002

Card 2	1	2	3	4	5	6	7	8
Variable	VP	EPS0	RFILTF					
Type	F	F	F					
Default	0.0	1.0	0.0					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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	Quasi-static threshold strain rate. See description under *MAT_-015.
RFILTF	Smoothing factor on the effective strain rate for solid elements when VP = 0: $\dot{\varepsilon}_n^{\text{avg}} = \text{RFILTF} \times \dot{\varepsilon}_{n-1}^{\text{avg}} + (1 - \text{RFILTF}) \times \dot{\varepsilon}_n$

Remarks:

1. **Constitutive Relationship.** This material model follows a constitutive relationship of the form:

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

where σ is the yield stress, ε is the effective plastic strain, and $\dot{\varepsilon}$ is the effective total strain rate (VP = 0), or the effective plastic strain rate (VP = 1). 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, such as 0.0001.

The initial yield stress is obtained through:

$$\sigma_0 = k\varepsilon_0^m \dot{\varepsilon}^n \quad ,$$

with an initial effective strain of

$$\varepsilon_0 = \max\left(0.001, \left(\frac{E}{k\dot{\varepsilon}^n}\right)^{1/(m-1)}\right) \quad .$$

2. **Superplastic Forming.** This model can be combined with the superplastic forming input (see *LOAD_SUPERPLASTIC_FORMING) to control the magnitude of the pressure in the pressure boundary conditions. Controlling the pressure limits the effective plastic strain rate so that it does not exceed a maximum value at any integration point within the model.

3. **Viscoplastic Formulation.** A fully viscoplastic formulation is optional. An additional cost is incurred, but the improvement in results can be dramatic.

*MAT_MODIFIED_ZERILLI_ARMSTRONG

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

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

Optional FCC Metal Card. This card is optional.

Card 4	1	2	3	4	5	6	7	8
Variable	M							
Type	F							

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
G	Shear modulus

VARIABLE	DESCRIPTION
E0	$\dot{\varepsilon}_0$, factor to normalize strain rate
N	n, exponent for bcc metal
TROOM	T_r, room temperature
PC	p_0, Pressure cutoff
SPALL	Spall Type: EQ.1.0: minimum pressure limit EQ.2.0: maximum principal stress EQ.3.0: minimum pressure cutoff
C*i*	C_i, coefficients for flow stress; see Remark 1 below.
EFAIL	Failure strain for erosion
VP	Formulation for rate effects: EQ.0.0: scale yield stress (default) EQ.1.0: viscoplastic formulation
B*i*	B_i, coefficients for polynomial to represent temperature dependency of flow stress yield
G*i*	G_i, coefficients for defining heat capacity and temperature dependency of heat capacity
BULK	Bulk modulus defined for shell elements only. Do not input for solid elements.
M	m, exponent for FCC metal (default = 0.5). This field is only used when N = 0.0 on Card 1.

Remarks:

1. **Flow Stress.** The Zerilli-Armstrong Material Model expresses the flow stress as follows.

   a) For FCC metals ($n = 0$),

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

where ε^p is the effective plastic strain and $\dot{\varepsilon}^*$ is the effective plastic strain rate defined as

$$\dot{\varepsilon}^* = \frac{\dot{\varepsilon}}{\dot{\varepsilon}_0} \ .$$

$\dot{\varepsilon}_0 = 1$, 10^{-3}, 10^{-6} for time units of seconds, milliseconds, and microseconds, respectively.

b) For BCC metals ($n > 0$),

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

where

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

2. **Heat Capacity.** The relationship between heat capacity (specific heat) and temperature may be characterized by a cubic polynomial equation as follows:

$$C_p = G_1 + G_2 T + G_3 T^2 + G_4 T^3$$

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

*MAT_LINEAR_ELASTIC_DISCRETE_BEAM

This is Material Type 66. This material model is defined for simulating the effects of a linear elastic beam by using six springs each acting about one of the six local degrees-of-freedom. The two nodes defining a beam may be coincident to give a zero length beam or offset to give a finite length beam. For finite length discrete beams the absolute value of the variable SCOOR in the *SECTION_BEAM input should be set to a value of 2.0, which causes the local r-axis to be aligned along the two nodes of the beam, to give physically correct behavior. The distance between the nodes of a beam should not affect the behavior of this model. A triad is used to orient the beam for the directional springs. Translational/rotational stiffness and viscous damping effects are considered for a local cartesian system; see Remark 1. Applications for this element include the modeling of joint stiffnesses.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	TKR	TKS	TKT	RKR	RKS	RKT
Type	A	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	TDR	TDS	TDT	RDR	RDS	RDT		
Type	F	F	F	F	F	F		

Card 3	1	2	3	4	5	6	7	8
Variable	FOR	FOS	FOT	MOR	MOS	MOT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density; see also "volume" in the *SECTION_BEAM definition.

VARIABLE	DESCRIPTION
TKR	Translational stiffness along local r-axis; see Remarks 1 and 2 below.
TKS	Translational stiffness along local s-axis
TKT	Translational stiffness along 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
TDR	Translational viscous damper along local r-axis (optional)
TDS	Translational viscous damper along local s-axis (optional)
TDT	Translational viscous damper along local t-axis (optional)
RDR	Rotational viscous damper about the local r-axis (optional)
RDS	Rotational viscous damper about the local s-axis (optional)
RDT	Rotational viscous damper about the local t-axis (optional)
FOR	Preload force in r-direction (optional)
FOS	Preload force in s-direction (optional)
FOT	Preload force in t-direction (optional)
MOR	Preload moment about r-axis (optional)
MOS	Preload moment about s-axis (optional)
MOT	Preload moment about t-axis (optional)

Remarks:

1. **Coordinate System and Orientation.** 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).

2. **Null Stiffness.** For null stiffness coefficients, no forces corresponding to these null values will develop. The viscous damping coefficients are optional.

3. **Rotational Displacement.** Rotational displacement is measured in radians.

*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 aligns the local *r*-axis 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 remarks below.

Card Summary:

Card 1. This card is required.

MID	RO	LCIDTR	LCIDTS	LCIDTT	LCIDRR	LCIDRS	LCIDRT

Card 2. This card is required.

LCIDTDR	LCIDTDS	LCIDTDT	LCIDRDR	LCIDRDS	LCIDRDT		

Card 3. This card is required.

FOR	FOS	FOT	MOR	MOS	MOT		

Card 4. To consider failure, this card must be defined. Otherwise it is optional.

FFAILR	FFAILS	FFAILT	MFAILR	MFAILS	MFAILT		

Card 5. To consider failure, this card must be defined. Otherwise it is optional.

UFAILR	UFAILS	UFAILT	TFAILR	TFAILS	TFAILT		

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	LCIDTR	LCIDTS	LCIDTT	LCIDRR	LCIDRS	LCIDRT
Type	A	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density; see also volume in *SECTION_BEAM.
LCIDTR	Load curve ID defining translational force resultant along local *r*-axis as a function of relative translational displacement; see Remarks 1 and 3 and Figure M67-1.
LCIDTS	Load curve ID defining translational force resultant along local *s*-axis as a function of relative translational displacement.
LCIDTT	Load curve ID defining translational force resultant along local *t*-axis as a function of relative translational displacement.
LCIDRR	Load curve ID defining rotational moment resultant about local *r*-axis as a function of relative rotational displacement.
LCIDRS	Load curve ID defining rotational moment resultant about local *s*-axis as a function of relative rotational displacement.
LCIDRT	Load curve ID defining rotational moment resultant about local *t*-axis as a function of relative rotational displacement.

Card 2	1	2	3	4	5	6	7	8
Variable	LCIDTDR	LCIDTDS	LCIDTDT	LCIDRDR	LCIDRDS	LCIDRDT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
LCIDTDR	Load curve ID defining translational damping force resultant along local *r*-axis as a function of relative translational velocity.
LCIDTDS	Load curve ID defining translational damping force resultant along local *s*-axis as a function of relative translational velocity.
LCIDTDT	Load curve ID defining translational damping force resultant along local *t*-axis as a function of relative translational velocity.
LCIDRDR	Load curve ID defining rotational damping moment resultant about local *r*-axis as a function of relative rotational velocity.

VARIABLE	DESCRIPTION
LCIDRDS	Load curve ID defining rotational damping moment resultant about local s-axis as a function of relative rotational velocity.
LCIDRDT	Load curve ID defining rotational damping moment resultant about local t-axis as a function of relative rotational velocity.

Card 3	1	2	3	4	5	6	7	8
Variable	FOR	FOS	FOT	MOR	MOS	MOT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
FOR	Preload force in r-direction (optional).
FOS	Preload force in s-direction (optional).
FOT	Preload force in t-direction (optional).
MOR	Preload moment about r-axis (optional).
MOS	Preload moment about s-axis (optional).
MOT	Preload moment about t-axis (optional).

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		

VARIABLE	DESCRIPTION
FFAILR	Optional failure parameter. If zero, the corresponding force, F_r, is not considered in the failure calculation. See Remark 4.

VARIABLE	DESCRIPTION
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.
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.

Card 5	1	2	3	4	5	6	7	8
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
UFAILR	Optional failure parameter. If zero, the corresponding displacement, u_r, is not considered in the failure calculation. See Remark 4.
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.

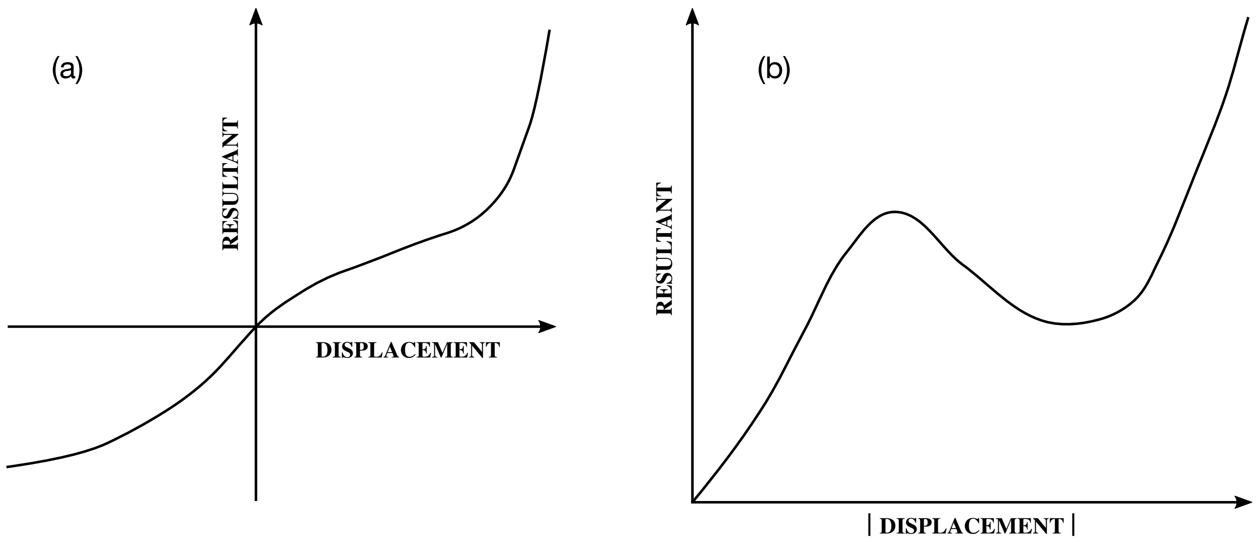


Figure M67-1. The resultant forces and moments are determined by a table lookup. If the origin of the load curve is at [0,0] as in (b) and tension and compression responses are symmetric.

VARIABLE	DESCRIPTION
TFAILT	Optional failure parameter. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation.

Remarks:

1. **Null Load Curve IDs.** For null load curve IDs, no forces are computed.

2. **Discrete Beam Formulation.** 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).

3. **Tension and Compression.** 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 M67-1 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, that is, start the load curve at the origin [0,0].

4. **Failure.** Catastrophic failure based on force resultants occurs if the following inequality is satisfied.

$$\left(\frac{F_r}{F_r^{\text{fail}}}\right)^2+\left(\frac{F_s}{F_s^{\text{fail}}}\right)^2+\left(\frac{F_t}{F_t^{\text{fail}}}\right)^2+\left(\frac{M_r}{M_r^{\text{fail}}}\right)^2+\left(\frac{M_s}{M_s^{\text{fail}}}\right)^2+\left(\frac{M_t}{M_t^{\text{fail}}}\right)^2-1.\geq 0\ .$$

After failure the discrete element is deleted. Likewise, catastrophic failure based on displacement resultants occurs if the following inequality is satisfied:

$$\left(\frac{u_r}{u_r^{\text{fail}}}\right)^2+\left(\frac{u_s}{u_s^{\text{fail}}}\right)^2+\left(\frac{u_t}{u_t^{\text{fail}}}\right)^2+\left(\frac{\theta_r}{\theta_r^{\text{fail}}}\right)^2+\left(\frac{\theta_s}{\theta_s^{\text{fail}}}\right)^2+\left(\frac{\theta_t}{\theta_t^{\text{fail}}}\right)^2-1.\geq 0\ .$$

After failure, the discrete element is deleted. If failure is included, either or both of the criteria may be used.

5. **Rotational Displacement.** Rotational displacement is measured in radians.

*MAT_NONLINEAR_PLASTIC_DISCRETE_BEAM

This is Material Type 68. This material model is 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 aligns the local r-axis 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 orients the beam for the directional springs. Translational/rotational stiffness and damping effects can be considered. The plastic behavior is modeled using force/moment curves as a function of displacements/rotation. Optionally, failure can be specified based on a force/moment criterion and a displacement rotation criterion. See also the remarks below.

Card Summary:

Card 1. This card is required

MID	RO	TKR	TKS	TKT	RKR	RKS	RKT

Card 2. This card is required.

TDR	TDS	TDT	RDR	RDS	RDT	RYLD	

Card 3. This card is required.

LCPDR	LCPDS	LCPDT	LCPMR	LCPMS	LCPMT		

Card 4. This card is required.

FFAILR	FFAILS	FFAILT	MFAILR	MFAILS	MFAILT		

Card 5. This card is required.

UFAILR	UFAILS	UFAILT	TFAILR	TFAILS	TFAILT		

Card 6. This card is required.

FOR	FOS	FOT	MOR	MOS	MOT		

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	TKR	TKS	TKT	RKR	RKS	RKT
Type	A	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density; see also volume on *SECTION_BEAM definition.
TKR	Translational stiffness along local r-axis LT.0.0: |TKR| is the load curve ID defining the elastic translational force along the local r-axis as a function of relative translational displacement. Useful for nonlinear elastic behavior.
TKS	Translational stiffness along local s-axis LT.0.0: |TKS| is the load curve ID for defining the elastic translational force along the local s-axis as a function of relative translational displacement. Useful for nonlinear elastic behavior.
TKT	Translational stiffness along local t-axis LT.0.0: |TKT| is the load curve ID defining the elastic translational force along the local t-axis as a function of relative translational displacement. Useful for nonlinear elastic behavior.
RKR	Rotational stiffness about the local r-axis LT.0.0: |RKR| is the load curve ID defining the elastic rotational moment along the local r-axis as a function of relative rotational displacement. Useful for nonlinear elastic behavior.

VARIABLE	DESCRIPTION
RKS	Rotational stiffness about the local *s*-axis LT.0.0: |RKS| is the load curve ID defining the elastic rotational moment along the local *s*-axis as a function of relative rotational displacement. Useful for nonlinear elastic behavior.
RKT	Rotational stiffness about the local *t*-axis LT.0.0: |RKT| is the load curve ID defining the elastic rotational moment along the local *t*-axis as a function of relative rotational displacement. Useful for nonlinear elastic behavior.

Card 2	1	2	3	4	5	6	7	8
Variable	TDR	TDS	TDT	RDR	RDS	RDT	RYLD	
Type	F	F	F	F	F	F	I	
Default	none	none	none	none	none	none	0	

VARIABLE	DESCRIPTION
TDR	Translational viscous damper along local *r*-axis
TDS	Translational viscous damper along local *s*-axis
TDT	Translational viscous damper along 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
RYLD	Flag for method of computing plastic yielding: EQ.0: Original method of determining plastic yielding (default) EQ.1: Compute yield displacement/rotation by taking the first point of the relevant curve as the yield force/moment and dividing it by the relevant stiffness

Card 3	1	2	3	4	5	6	7	8
Variable	LCPDR	LCPDS	LCPDT	LCPMR	LCPMS	LCPMT		
Type	I	I	I	I	I	I		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
LCPDR	Load curve (or table) ID for yield force as a function of plastic displacement along the local *r*-axis (and translational velocity in the *r*-direction, if table). If the curve/table ID is zero, and TKR is nonzero, then elastic behavior is obtained for this component.
LCPDS	Load curve (or table) ID-yield force as a function of plastic displacement along the *s*-axis (and translational velocity in the *s*-direction, if table). If the curve/table ID is zero, and TKS is nonzero, then elastic behavior is obtained for this component.
LCPDT	Load curve (or table) ID-yield force as a function of plastic displacement along the *t*-axis (and translational velocity in the *t*-direction, if table). If the curve/table ID is zero, and TKT is nonzero, then elastic behavior is obtained for this component.
LCPMR	Load curve (or table) ID-yield moment as a function of plastic rotation about the *r*-axis (and rotational velocity about the *r*-axis, if table). If the curve/table ID is zero, and RKR is nonzero, then elastic behavior is obtained for this component.
LCPMS	Load curve (or table) ID-yield moment as a function of plastic rotation about the *s*-axis (and rotational velocity about the *s*-axis, if table). If the curve/table ID is zero, and RKS is nonzero, then elastic behavior is obtained for this component.
LCPMT	Load curve (or table) ID-yield moment as a function of plastic rotation about the *t*-axis (and rotational velocity about the *t*-axis, if table). If the curve/table ID is zero, and RKT is nonzero, then elastic behavior is obtained for this component.

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		

VARIABLE	DESCRIPTION
FFAILR	Optional failure parameter. If zero, the corresponding force, F_r, is not considered in the failure calculation. LT.0.0: |FFAILR| is the load curve ID defining F_r as a function of translational velocity along the local r-axis.
FFAILS	Optional failure parameter. If zero, the corresponding force, F_s, is not considered in the failure calculation. LT.0.0: |FFAILS| is the load curve ID defining F_s as a function of translational velocity along the local s-axis.
FFAILT	Optional failure parameter. If zero, the corresponding force, F_t, is not considered in the failure calculation. LT.0.0: |FFAILT| is the load curve ID defining F_t as a function of translational velocity along the local t-axis.
MFAILR	Optional failure parameter. If zero, the corresponding moment, M_r, is not considered in the failure calculation. LT.0.0: |MFAILR| is the load curve ID defining M_r as a function of rotational velocity about the local r-axis.
MFAILS	Optional failure parameter. If zero, the corresponding moment, M_s, is not considered in the failure calculation. LT.0.0: |MFAILS| is the load curve ID defining M_s as a function of rotational velocity about the local s-axis.
MFAILT	Optional failure parameter. If zero, the corresponding moment, M_t, is not considered in the failure calculation. LT.0.0: |MFAILT| is the load curve ID defining M_t as a function of rotational velocity about the local t-axis.

Card 5	1	2	3	4	5	6	7	8
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
UFAILR	Optional failure parameter. If zero, the corresponding displacement, u_r, is not considered in the failure calculation. LT.0.0: |UFAILR| is the load curve ID defining u_r as a function of translational velocity along the local r-axis.
UFAILS	Optional failure parameter. If zero, the corresponding displacement, u_s, is not considered in the failure calculation. LT.0.0: |UFAILS| is the load curve ID defining u_s as a function of translational velocity along the local s-axis.
UFAILT	Optional failure parameter. If zero, the corresponding displacement, u_t, is not considered in the failure calculation. LT.0.0: |UFAILT| is the load curve ID defining u_t as a function of translational velocity along the local t-axis.
TFAILR	Optional failure parameter. If zero, the corresponding rotation, θ_r, is not considered in the failure calculation. LT.0.0: |TFAILR| is the load curve ID defining θ_r as a function of rotational velocity about the local r-axis.
TFAILS	Optional failure parameter. If zero, the corresponding rotation, θ_s, is not considered in the failure calculation. LT.0.0: |TFAILS| is the load curve ID defining θ_s as a function of rotational velocity about the local s-axis.
TFAILT	Optional failure parameter. If zero, the corresponding rotation, θ_t, is not considered in the failure calculation. LT.0.0: |TFAILT| is the load curve ID defining θ_t as a function of rotational velocity about the local t-axis.

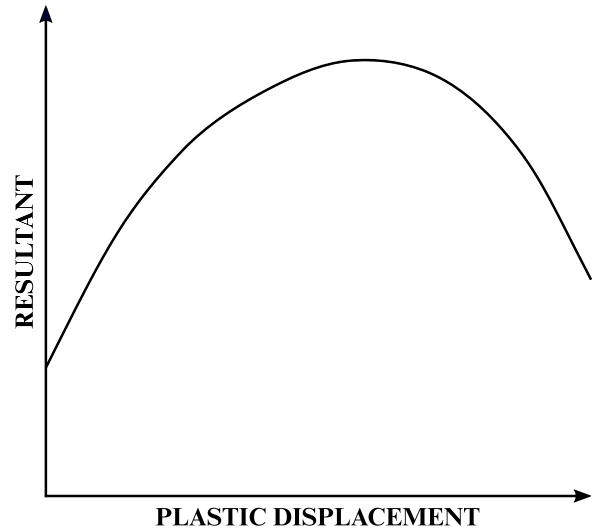


Figure M68-1. The resultant forces and moments are limited by the yield definition. The initial yield point corresponds to a plastic displacement of zero.

Card 6	1	2	3	4	5	6	7	8
Variable	FOR	FOS	FOT	MOR	MOS	MOT		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
FOR	Preload force in *r*-direction (optional)
FOS	Preload force in *s*-direction (optional)
FOT	Preload force in *t*-direction (optional)
MOR	Preload moment about *r*-axis (optional)
MOS	Preload moment about *s*-axis (optional)
MOT	Preload moment about *t*-axis (optional)

Remarks:

1. **Elastic behavior.** For the translational and rotational degrees of freedom where elastic behavior is desired, set the load curve ID to zero.

2. **Plastic displacement.** The plastic displacement for the load curves is defined as:

$$\text{plastic displacement} = \text{total displacement} - \text{yield force}/\text{elastic stiffness}\ .$$

3. **Discrete beam formulation.** 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).

4. **Failure.** Catastrophic failure based on force resultants occurs if the following inequality is satisfied.

$$\left(\frac{F_r}{F_r^{\text{fail}}}\right)^2 + \left(\frac{F_s}{F_s^{\text{fail}}}\right)^2 + \left(\frac{F_t}{F_t^{\text{fail}}}\right)^2 + \left(\frac{M_r}{M_r^{\text{fail}}}\right)^2 + \left(\frac{M_s}{M_s^{\text{fail}}}\right)^2 + \left(\frac{M_t}{M_t^{\text{fail}}}\right)^2 - 1. \geq 0.$$

After failure, the discrete element is deleted. Likewise, catastrophic failure based on displacement resultants occurs if the following inequality is satisfied:

$$\left(\frac{u_r}{u_r^{\text{fail}}}\right)^2 + \left(\frac{u_s}{u_s^{\text{fail}}}\right)^2 + \left(\frac{u_t}{u_t^{\text{fail}}}\right)^2 + \left(\frac{\theta_r}{\theta_r^{\text{fail}}}\right)^2 + \left(\frac{\theta_s}{\theta_s^{\text{fail}}}\right)^2 + \left(\frac{\theta_t}{\theta_t^{\text{fail}}}\right)^2 - 1. \geq 0.$$

After failure the discrete element is deleted. If failure is included, either or both of the criteria may be used.

5. **Rotational displacement.** Rotational displacement is measured in radians.

6. **History variables.** The following additional history variables are available for this material by setting NEIPB on *DATABASE_EXTENT_BINARY:

History Variable #	Description
12	Flag for failure from resultant forces: EQ.0: Intact EQ.1: Failed
13	Flag for failure from displacement resultants: EQ.0: Intact EQ.1: Failed

*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 as a function of relative velocity curves since the force characteristics are dependent on the displacement of the piston. See Remarks below.

Card Summary:

Card 1. This card is required.

MID	RO	ST	D	R	H	K	C

Card 2. This card is required.

C3	STF	RHOF	C1	C2	LCIDF	LCIDD	S0

Card 3. Include one card per orifice. Read in up to 15 orifice locations. The next keyword ("*") card terminates this input.

ORFLOC	ORFRAD	SF	DC				

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	ST	D	R	H	K	C
Type	A	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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 M69-1 below.
D	d, piston diameter
R	R, default orifice radius
H	h, orifice controller position

VARIABLE	DESCRIPTION
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

Card 2	1	2	3	4	5	6	7	8
Variable	C3	STF	RHOF	C1	C2	LCIDF	LCIDD	S0
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
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 as a function of piston displacement, s, that is, 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 as a function of piston displacement, s, that is, $g(s+s_0)$. 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.

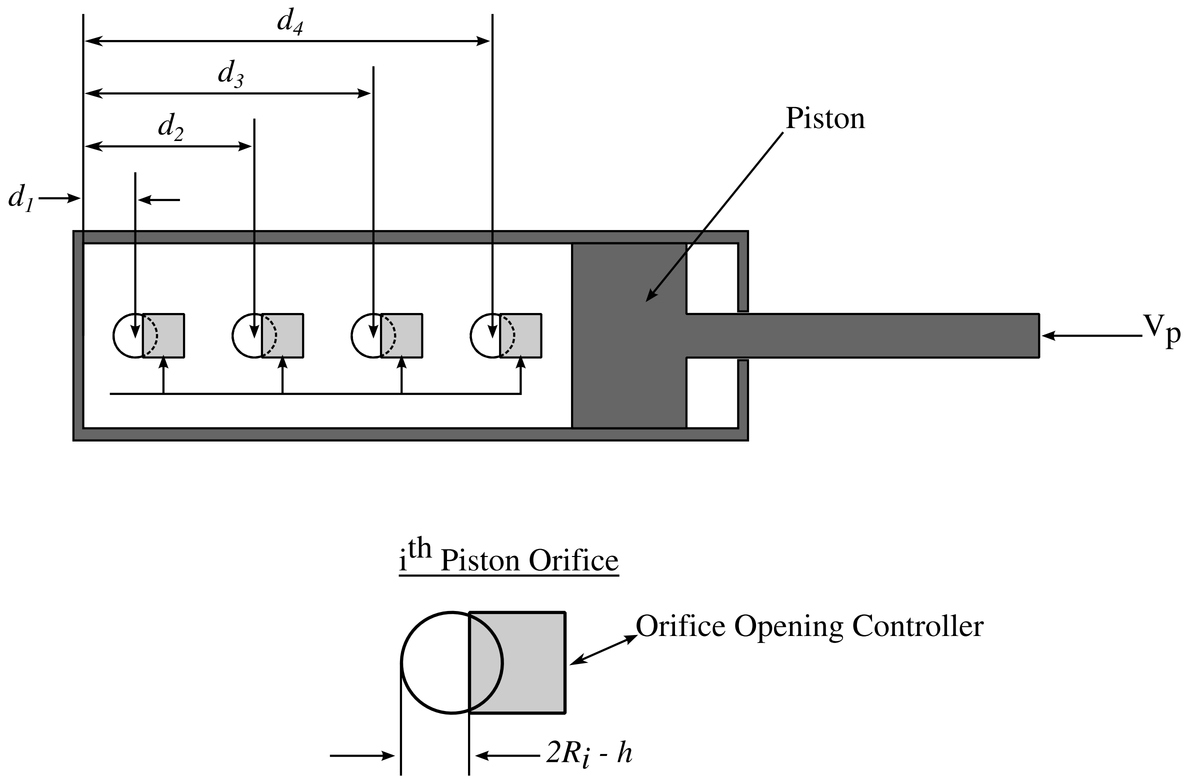


Figure M69-1. Mathematical model for the Side Impact Dummy damper.

Orifice Cards. Include one card per orifice. Read in up to 15 orifice locations. The next keyword ("*") card terminates this input. On the first card below the optional input parameters SF and DF may be specified.

Cards 3	1	2	3	4	5	6	7	8
Variable	ORFLOC	ORFRAD	SF	DC				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
ORFLOC	d_i, orifice location of i^{th} orifice relative to the fixed end
ORFRAD	r_i, orifice radius of i^{th} orifice EQ.0.0: R on Card 1 is used.
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

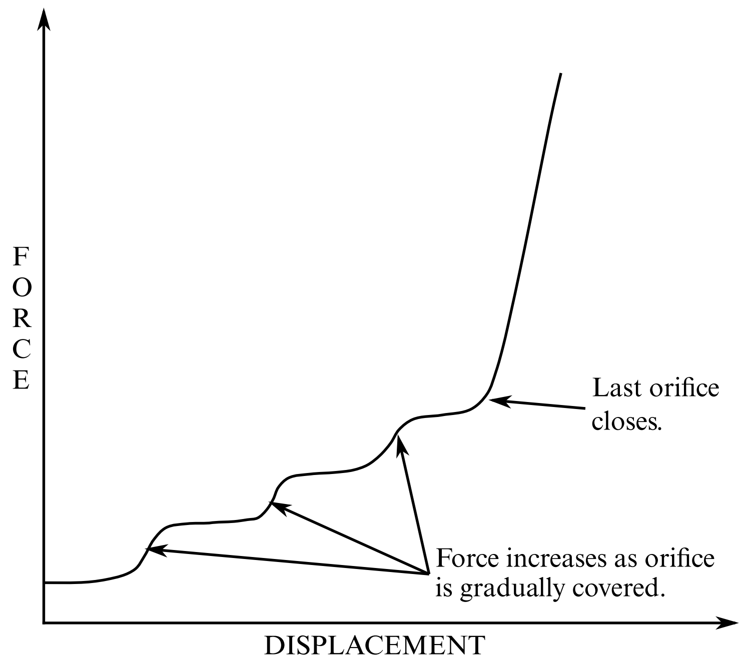


Figure M69-2. Force as a function of displacement as orifices are covered at a constant relative velocity. Only the linear velocity term is active.

Remarks:

As the damper moves, the fluid flows through the open orifices to provide the necessary damping resistance. While moving as shown in Figure M69-1, 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,

$$F = \mathrm{SF} \times \left\{ KA_p V_p \left\{ \frac{C_1}{A_0^t} + C_2 |V_p| \rho_{\text{fluid}} \left[\left(\frac{A_p}{CA_0^t} \right)^2 - 1 \right] \right\} - f(s + s_0) + V_p g(s + s_0) \right\} ,$$

where K is a user defined constant or a tabulated function of the absolute value of the relative velocity, V_p is the piston velocity, C is the discharge coefficient, A_p is the piston area, A_0^t is the total open areas of orifices at time t, ρ_{fluid} is the fluid density, C_1 is the coefficient for the linear term, and C_2 is the coefficient for the quadratic term.

In the implementation, the orifices are assumed to be circular with partial covering by the orifice controller. As the piston closes, the closure of the orifice is gradual. This gradual closure is properly taken into account to insure a smooth response. If the piston stroke is exceeded, the stiffness value, k, limits further movement, meaning 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 as a function of displacement curve at constant relative velocity is shown in Figure M69-2.

The factor, SF, which scales the force defaults to 1.0 and is analogous to the adjusting ring on the damper.

*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 M70-1. 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 the remarks below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	CO	N	P0	PA	AP	KH
Type	A	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	LCID	FR	SCLF	CLEAR				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label not must be specified (see *PART).
RO	Mass density, see also volume in *SECTION_BEAM definition.
CO	Length of gas column, C_0
N	Adiabatic constant, n
P0	Initial gas pressure, P_0
PA	Atmospheric pressure, P_a
AP	Piston cross sectional area, A_p
KH	Hydraulic constant, K
LCID	Load curve ID (see *DEFINE_CURVE) defining the orifice area, a_0, as a function of element deflection S

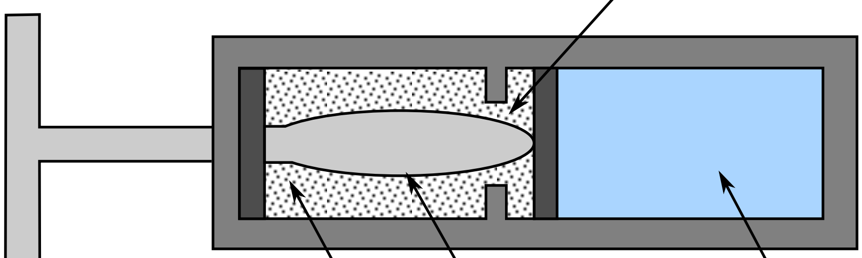


Figure M70-1. Schematic of Hydraulic/Gas damper.

VARIABLE	DESCRIPTION
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.

Remarks:

As the damper is compressed two actions contribute to the force which develops. First, the gas is adiabatically compressed into a smaller volume. Secondly, oil is forced through an orifice. A profiled pin may occupy some of the cross-sectional area of the orifice; thus, the orifice area available for the oil varies with the stroke. The force is assumed proportional to the square of the velocity and inversely proportional to the available area.

The equation for this element is:

$$F = \mathrm{SCLF} \times \left\{ K_h \left(\frac{V}{a_0} \right)^2 + \left[P_0 \left(\frac{C_0}{C_0 + S} \right)^n - P_a \right] A_p \right\}$$

where S is the element deflection (positive in tension) 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 modeled; thus, no force will develop in compression.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	LCID	F0	TMAXF0	TRAMP	IREAD
Type	A	F	F	F	F	F	F	I
Default	none	none	none	none	0	0	0	0

Additional card for IREAD > 1.

Card 2	1	2	3	4	5	6	7	8
Variable	OUTPUT	TSTART	FRACL0	MXEPS	MXFRC			
Type	I	F	F	F	F			
Default	0	0	0	1.0E+20	1.0E+20			

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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.
TMAXF0	Time for which pre-tension force will be held

VARIABLE	DESCRIPTION
TRAMP	Ramp-up time for pre-tension force
IREAD	Set to 1 to read second line of input
OUTPUT	Flag = 1 to output axial strain (see note below concerning OUTPUT)
TSTART	Time at which the ramp-up of pre-tension begins
FRACL0	Fraction of initial length that should be reached over time period of TRAMP. Corresponding tensile force builds up as necessary to reach cable length = FRACL0 × L0 at time t = TRAMP.
MXEPS	Maximum strain at failure
MXFRC	Maximum force at failure

Remarks:

The force, F, generated by the cable is nonzero if and only if the cable is tension. The force is given by:

$$F = \max(F_0 + K\Delta L, 0.)$$

where ΔL is the change in length

$$\Delta L = \text{current length} - (\text{initial length} - \text{offset})$$

and the stiffness (E > 0.0 only) is defined as:

$$K = \frac{E \times \text{area}}{(\text{initial length} - \text{offset})}$$

Note that a constant force element can be obtained by setting:

$$F_0 > 0 \text{ and } K = 0$$

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.

By default, cable pretension is applied only at the start of the analysis. If the cable is attached to flexible structure, deformation of the structure will result in relaxation of the cables, which will therefore lose some or all of the intended preload.

This can be overcome by using TMAXF0. In this case, it is expected that the structure will deform under the loading from the cables and that this deformation will take time to occur during the analysis. The unstressed length of the cable will be continuously adjusted until time TMAXF0 such that the force is maintained at the user-defined pre-tension force – this is analogous to operation of the pre-tensioning screws in real cables. After time TMAXF0, the unstressed length is fixed and the force in the cable is determined in the normal way using the stiffness and change of length.

Sudden application of the cable forces at time zero may result in an excessively dynamic response during pre-tensioning. A ramp-up time TRAMP may optionally be defined. The cable force ramps up from zero at time TSTART to the full pre-tension F0 at time TSTART + TRAMP. TMAXF0, if set less than TSTART + TRAMP by the user, will be internally reset to TSTART + TRAMP.

If the model does not use dynamic relaxation, it is recommended that damping be applied during pre-tensioning so that the structure reaches a steady state by time TMAXF0.

If the model uses dynamic relaxation, TSTART, TRAMP, and TMAXF0 apply only during dynamic relaxation. The cable preload at the end of dynamic relaxation carries over to the start of the subsequent transient analysis.

The cable mass will be calculated from length × area × density if VOL is set to zero on *SECTION_BEAM. Otherwise, VOL × density will be used.

If OUTPUT is set to 1, one additional history variable representing axial strain is output to d3plot for the cable elements. This axial strain can be plotted by LS-PrePost by selecting the beam component labeled as “axial stress”. Though the label says “axial stress”, it is actually axial strain.

If the stress-strain load curve option, LCID, is combined with preload, two types of behavior are available:

1. If the preload is applied using the TMAXF0/TRAMP method, the initial strain is calculated from the stress-strain curve to achieve the desired preload.

2. If TMAXF0/TRAMP are not used, the preload force is taken as additional to the force calculated from the stress/strain curve. Thus, the total stress in the cable will be higher than indicated by the stress/strain curve.

*MAT_CONCRETE_DAMAGE

This is Material Type 72. This model has been used to analyze buried steel reinforced concrete structures subjected to impulsive loadings. A newer version of this model is available as *MAT_CONCRETE_DAMAGE_REL3.

Card Summary:

Card 1. This card is required.

MID	RO	PR					

Card 2. This card is required.

SIGF	A0	A1	A2				

Card 3. This card is required.

A0Y	A1Y	A2Y	A1F	A2F	B1	B2	B3

Card 4. This card is required.

PER	ER	PRR	SIGY	ETAN	LCP	LCR	

Card 5. This card is required.

$\lambda 1$	$\lambda 2$	$\lambda 3$	$\lambda 4$	$\lambda 5$	$\lambda 6$	$\lambda 7$	$\lambda 8$

Card 6. This card is required.

$\lambda 9$	$\lambda 10$	$\lambda 11$	$\lambda 12$	$\lambda 13$			

Card 7. This card is required.

$\eta 1$	$\eta 2$	$\eta 3$	$\eta 4$	$\eta 5$	$\eta 6$	$\eta 7$	$\eta 8$

Card 8. This card is required.

$\eta 9$	$\eta 10$	$\eta 11$	$\eta 12$	$\eta 13$			

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR					
Type	A	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
PR	Poisson’s ratio

Card 2	1	2	3	4	5	6	7	8
Variable	SIGF	A0	A1	A2				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
SIGF	Maximum principal stress for failure
A0	Cohesion
A1	Pressure hardening coefficient
A2	Pressure hardening coefficient

Card 3	1	2	3	4	5	6	7	8
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

VARIABLE	DESCRIPTION
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

Card 4	1	2	3	4	5	6	7	8
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	

VARIABLE	DESCRIPTION
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.

Card 5	1	2	3	4	5	6	7	8
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	1	2	3	4	5	6	7	8
Variable	λ9	λ10	λ11	λ12	λ13			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
λ1 - λ13	Tabulated damage function

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	1	2	3	4	5	6	7	8
Variable	η9	η10	η11	η12	η13			
Type	F	F	F	F	F			
Default	none	none	none	none	none			

VARIABLE	DESCRIPTION
η1 - η13	Tabulated scale factor.

Remarks:

1. **Cohesion.** Cohesion for failed material $a_{0f} = 0$.

2. **B3.** B3 must be positive or zero.

3. **Damage Function.** $\lambda_n \leq \lambda_{n+1}$. The first point must be zero.

*MAT_CONCRETE_DAMAGE_REL3

This is Material Type 72R3. The Karagozian & Case (K&C) Concrete Model - Release III is a three-invariant model, uses three shear failure surfaces, includes damage and strain-rate effects, and has origins based on the Pseudo-TENSOR Model (Material Type 16). The most significant user improvement provided by Release III is a model parameter generation capability, based solely on the unconfined compression strength of the concrete. The implementation of Release III significantly changed the user input, thus previous input files using Material Type 72 prior to LS-DYNA Version 971, are not compatible with the present input format.

An open source reference, that precedes the parameter generation capability, is provided in Malvar et al. [1997]. A workshop proceedings reference, Malvar et al. [1996], is useful, but may be difficult to obtain. More recent, but *limited distribution* reference materials, such as Malvar et al. [2000], may be obtained by contacting Karagozian & Case.

Seven card images are required to define the *complete* set of model parameters for the K&C Concrete Model. An Equation-of-State is also required for the pressure-volume strain response. Brief descriptions of all the input parameters are provided below, however it is expected that this model will be used primarily with the option to automatically generate the model parameters based on the unconfined compression strength of the concrete. These generated material parameters, along with the generated parameters for *EOS_TABULATED_COMPACTION, are written to the d3hsp file.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	R0	PR					
Type	A	F	F					
Default	none	none	none					

Card 2	1	2	3	4	5	6	7	8
Variable	FT	A0	A1	A2	B1	OMEGA	A1F	
Type	F	F	F	F	F	F	F	
Default	none	0.0	0.0	0.0	0.0	none	0.0	

Card 3	1	2	3	4	5	6	7	8
Variable	Sλ	NOUT	EDROP	RSIZE	UCF	LCRATE	LOCWID	NPTS
Type	F	F	F	F	F	I	F	F
Default	none	none	none	none	none	none	none	none

Card 4	1	2	3	4	5	6	7	8
Variable	λ01	λ02	λ03	λ04	λ05	λ06	λ07	λ08
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 5	1	2	3	4	5	6	7	8
Variable	λ09	λ10	λ11	λ12	λ13	B3	A0Y	A1Y
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	0.0	0.0

Card 6	1	2	3	4	5	6	7	8
Variable	η01	η02	η03	η04	η05	η06	η07	η08
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 7	1	2	3	4	5	6	7	8
Variable	η09	η10	η11	η12	η13	B2	A2F	A2Y
Type	F	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 or label must be specified (see *PART).
RO	Mass density
PR	Poisson's ratio, ν
FT	Uniaxial tensile strength, f_t
A0	Maximum shear failure surface parameter, a_0, **or** $-f_c'$ for **parameter generation** (recommended)
A1	Maximum shear failure surface parameter, a_1
A2	Maximum shear failure surface parameter, a_2
B1	Compressive damage scaling parameter, b_1
OMEGA	Fractional dilatancy, ω
A1F	Residual failure surface coefficient, a_{1f}
Sλ	λ stretch factor, s
NOUT	Output selector for effective plastic strain (see table)
EDROP	Post peak dilatancy decay, N^{α}
RSIZE	Unit conversion factor for length (inches/user-unit). For example, set to 39.37 if user length unit in meters.
UCF	Unit conversion factor for stress (psi/user-unit). For instance set to 145 if f_c' in MPa.

VARIABLE	DESCRIPTION
LCRATE	Define (load) curve number for strain-rate effects; effective strain rate on abscissa (negative = tension) and strength enhancement on ordinate. If LCRATE is set to -1, strain rate effects are automatically included, based on equations provided in Wu, Crawford, Lan, and Magallanes [2014]. LCRATE = -1 is applicable to models which use time units of seconds, for other time units, the strain-rate effects should be input by means of a curve.
LOCWID	Three times the maximum aggregate diameter (input in user length units).
NPTS	Number of points in λ versus η damage relation; must be 13 points.
λ01	1st value of damage function, (a.k.a., 1st value of "modified" effective plastic strain; see references for details).
λ02	2nd value of damage function,
λ03	3rd value of damage function,
λ04	4th value of damage function,
λ05	5th value of damage function,
λ06	6th value of damage function,
λ07	7th value of damage function,
λ08	8th value of damage function,
λ09	9th value of damage function,
λ10	10th value of damage function,
λ11	11th value of damage function,
λ12	12th value of damage function,
λ13	13th value of damage function.
B3	Damage scaling coefficient for triaxial tension, b_3.
A0Y	Initial yield surface cohesion, a_{0y}.
A1Y	Initial yield surface coefficient, a_{1y}.
η01	1st value of scale factor,

VARIABLE	DESCRIPTION
η02	2nd value of scale factor,
η03	3rd value of scale factor,
η04	4th value of scale factor,
η05	5th value of scale factor,
η06	6th value of scale factor,
η07	7th value of scale factor,
η08	8th value of scale factor,
η09	9th value of scale factor,
η10	10th value of scale factor,
η11	11th value of scale factor,
η12	12th value of scale factor,
η13	13th value of scale factor.
B2	Tensile damage scaling exponent, b_2.
A2F	Residual failure surface coefficient, a_{2f}.
A2Y	Initial yield surface coefficient, a_{2y}.

λ, sometimes referred to as "modified" effective plastic strain, is computed internally as a function of effective plastic strain, strain rate enhancement factor, and pressure. η is a function of λ as specified by the η as a function of λ curve. The η value, which is always between 0 and 1, is used to interpolate between the yield failure surface and the maximum failure surface or between the maximum failure surface and the residual failure surface, depending on whether λ is to the left or right of the first peak in the η as a function of λ curve. The "scaled damage measure" ranges from 0 to 1 as the material transitions from the yield failure surface to the maximum failure surface, and thereafter ranges from 1 to 2 as the material ranges from the maximum failure surface to the residual failure surface. See the references for details.

Output of Selected Variables:

The quantity labeled as "plastic strain" by LS-PrePost is actually the quantity described in Table M72-1, in accordance with the input value of NOUT (see Card 3 above).

NOUT	Function	Description
1		Current shear failure surface radius
2	$\delta = 2\lambda/(\lambda + \lambda_m)$	Scaled damage measure
3	$\dot{\sigma}_{ij}\dot{\varepsilon}_{ij}$	Strain energy (rate)
4	$\dot{\sigma}_{ij}\dot{\varepsilon}_{ij}^p$	Plastic strain energy (rate)

Table M72-1. Description of quantity labeled "plastic strain" by LS-PrePost.

An additional six extra history variables as shown in Table M72-2 may be written by setting NEIPH = 6 on the keyword *DATABASE_EXTENT_BINARY. The extra history variables are labeled as "history var#1" through "history var#6" in LS-PrePost.

Label	Description
history var#1	Internal energy
history var#2	Pressure from bulk viscosity
history var#3	Volume in previous time step
history var#4	Plastic volumetric strain
history var#5	Slope of damage evolution (η vs. λ) curve
history var#6	"Modified" effective plastic strain (λ)

Table M72-2. Extra History Variables for *MAT_072R3

Sample Input for Concrete:

As an example of the K&C Concrete Model material parameter generation, the following sample input for a 45.4 MPa (6,580 psi) unconfined compression strength concrete is provided. The basic units for the provided parameters are length in millimeters (mm), time in milliseconds (msec), and mass in grams (g). This base unit set yields units of force in Newtons (N) and pressure in Mega-Pascals (MPa).

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	PR					
Value	72	2.3E-3						

Card 2	1	2	3	4	5	6	7	8
Variable	FT	A0	A1	A2	B1	OMEGA	A1F	
Value		-45.4						

Card 3	1	2	3	4	5	6	7	8
Variable	Sλ	NOUT	EDROP	RSIZE	UCF	LCRATE	LOCWID	NPTS
Value				3.94E-2	145.0	723		

Card 4	1	2	3	4	5	6	7	8
Variable	λ01	λ02	λ03	λ04	λ05	λ06	λ07	λ08
Value								

Card 5	1	2	3	4	5	6	7	8
Variable	λ09	λ10	λ11	λ12	λ13	B3	A0Y	A1Y
Value								

Card 6	1	2	3	4	5	6	7	8
Variable	η01	η02	η03	η04	η05	η06	η07	η08
Value								

Card 7	1	2	3	4	5	6	7	8
Variable	η09	η10	η11	η12	η13	B2	A2F	A2Y
Value								

Shear strength enhancement factor as a function of effective strain rate is given by a curve (*DEFINE_CURVE) with LCID 723. The sample input values, see Malvar & Ross [1998], are given in Table M72-3.

Strain-Rate (1/ms)	Enhancement
-3.0E+01	9.70
-3.0E-01	9.70
-1.0E-01	6.72
-3.0E-02	4.50
-1.0E-02	3.12
-3.0E-03	2.09
-1.0E-03	1.45
-1.0E-04	1.36
-1.0E-05	1.28
-1.0E-06	1.20
-1.0E-07	1.13
-1.0E-08	1.06
0.0E+00	1.00
3.0E-08	1.00
1.0E-07	1.03
1.0E-06	1.08
1.0E-05	1.14
1.0E-04	1.20
1.0E-03	1.26
3.0E-03	1.29
1.0E-02	1.33

Strain-Rate (1/ms)	Enhancement
3.0E-02	1.36
1.0E-01	2.04
3.0E-01	2.94
3.0E+01	2.94

Table M72-3. Enhancement as a function of effective strain rate for 45.4 MPa concrete (sample). When defining curve LCRATE, input negative (tensile) values of effective strain rate first. The enhancement should be positive and should be 1.0 at a strain rate of zero.

*MAT_LOW_DENSITY_VISCOUS_FOAM

This is Material Type 73. This material model is for Modeling Low Density Urethane Foam with high compressibility and 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. See the remarks below and the description of material 57.

Card Summary:

Card 1. This card is required.

MID	RO	E	LCID	TC	HU	BETA	DAMP

Card 2. This card is required.

SHAPE	FAIL	BVFLAG	KCON	LCID2	BSTART	TRAMP	NV

Card 3a. This card is included if and only if LCID2 = 0. Include up to 6 of this card. The next keyword ("*") card terminates this input.

G*i*	BETA*i*	REF					

Card 3b. This card is included if and only if LCID2 = -1.

LCID3	LCID4	SCALEW	SCALEA				

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	LCID	TC	HU	BETA	DAMP
Type	A	F	F	I	F	F	F	F
Default	none	none	none	none	10^{20}	1.	none	0.05

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density

VARIABLE	DESCRIPTION
E	Young's modulus used in tension. For implicit problems E is set to the initial slope of load curve LCID.
LCID	Load curve ID (see *DEFINE_CURVE) for nominal stress as a function of strain
TC	Tension cut-off stress
HU	Hysteretic unloading factor between 0 and 1 (default = 1, that is, no energy dissipation); see Figure M57-1 and Remark 3.
BETA	β, decay constant to model creep in unloading (see Remark 3) EQ.0.0: no relaxation
DAMP	Viscous coefficient (.05 < recommended value <.50) to model damping effects. LT.0.0: |DAMP| is the load curve ID which defines the damping constant as a function of the maximum strain in compression defined as: $\varepsilon_{max} = \max(1-\lambda_1, 1-\lambda_2, 1.-\lambda_3)$. In tension, the damping constant is set to the value corresponding to the strain at 0. The abscissa should be defined from 0 to 1.

Card 2	1	2	3	4	5	6	7	8
Variable	SHAPE	FAIL	BVFLAG	KCON	LCID2	BSTART	TRAMP	NV
Type	F	F	F	F	I	F	F	I
Default	1.0	0.0	0.0	0.0	0	0.0	0.0	6

VARIABLE	DESCRIPTION
SHAPE	Shape factor for unloading which is active for nonzero values of HU. SHAPE less than one reduces the energy dissipation and greater than one increases dissipation; see Figure M57-1.
FAIL	Failure option after cutoff stress is reached:

VARIABLE	DESCRIPTION
	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 remarks below): EQ.0.0: no bulk viscosity (recommended) EQ.1.0: bulk viscosity active
KCON	Stiffness coefficient for contact interface stiffness. If undefined, the maximum slope in the stress as a function of 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, so defining a reasonable stiffness is recommended.
LCID2	Load curve ID of relaxation curve. If LCID2 > 0, constants G_i and β_i are determined using a least squares fit. An example is shown in Figure M76-1. This model ignores the constant stress.
BSTART	Fit parameter. In the fit, β_1 is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2, β_4 is 10 times greater than β_3 , and so on. If zero, BSTART = .01.
TRAMP	Optional ramp time for loading
NV	Number of terms in fit. Currently, the maximum number is 6. Since each term used adds significantly to the cost, 2 or 3 terms is recommended. Caution should be exercised when taking the results from the fit. Preferably, all generated coefficients should be positive because negative values may lead to unstable results. Once a satisfactory fit has been achieved, we recommend using the output coefficients in future runs.

Relaxation Constant Cards. If LCID2 = 0, then include this card. Up to 6 cards may be input. The next keyword ("*") card terminates this input.

Card 3a	1	2	3	4	5	6	7	8
Variable	G*i*	BETA*i*	REF					
Type	F	F	F					

VARIABLE	DESCRIPTION
G*i*	Optional shear relaxation modulus for the i^{th} term
BETA*i*	Optional decay constant if i^{th} term
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY. EQ.0.0: off EQ.1.0: on

Frequency Dependence Card. If LCID2 = -1 then include this card.

Card 3b	1	2	3	4	5	6	7	8
Variable	LCID3	LCID4	SCALEW	SCALEA				
Type	I	I	I	I				

VARIABLE	DESCRIPTION
LCID3	Load curve ID giving the magnitude of the shear modulus as a function of the frequency. LCID3 must use the same frequencies as LCID4.
LCID4	Load curve ID giving the phase angle of the shear modulus as a function of the frequency. LCID4 must use the same frequencies as LCID3.
SCALEW	Flag for the form of the frequency data: EQ.0: frequency is in cycles per unit time. EQ.1: circular frequency
SCALEA	Flag for the units of the phase angle: EQ.0: degrees EQ.1: radians

Remarks:

1. **Material Formulation.** This viscoelastic foam material formulation models highly compressible viscous foams. The hyperelastic formulation of this model follows that of Material 57.

2. **Rate Effects.** 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:

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

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.

Frequency data can be fit to the Prony series. Using Fourier transforms the relationship between the relaxation function and the frequency dependent data is

$$G_s(\omega) = \alpha_0 + \sum_{m=1}^{N} \frac{\alpha_m(\omega/\beta_m)^2}{1+(\omega/\beta_m)^2}$$

$$G_\ell(\omega) = \sum_{m=1}^{N} \frac{\alpha_m\omega/\beta_m}{1+\omega/\beta_m}$$

where the storage modulus and loss modulus are defined in terms of the frequency dependent magnitude G and phase angle ϕ given by load curves LCID3 and LCID4 respectively,

$$G_s(\omega) = G(\omega)\cos[\phi(\omega)]$$

$$G_l(\omega) = G(\omega)\sin[\phi(\omega)]$$

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

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

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

The hysteretic unloading factor results in the unloading curve to lie beneath the loading curve as shown in Figure M57-1. This unloading provides energy dissipation which is reasonable in certain kinds of foam.

*MAT_ELASTIC_SPRING_DISCRETE_BEAM

This is Material Type 74. This model permits elastic springs with damping to be combined and represented with a discrete beam element type 6. Linear stiffness and damping coefficients can be defined, and, for nonlinear behavior, a force as a function of deflection and force as a function of rate curves can be used. Displacement based failure and an initial force are optional.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	K	F0	D	CDF	TDF	
Type	A	F	F	F	F	F	F	

Card 2	1	2	3	4	5	6	7	8
Variable	FLCID	HLCID	C1	C2	DLE	GLCID		
Type	F	F	F	F	F	I		

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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.

VARIABLE	DESCRIPTION
FLCID	Load curve ID (see *DEFINE_CURVE) defining force as a function of deflection for nonlinear behavior.
HLCID	Load curve ID (see *DEFINE_CURVE) defining force as a function of 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, that is, only the sign changes.
C1	Damping coefficient for nonlinear behavior (optional).
C2	Damping coefficient for nonlinear behavior (optional).
DLE	Factor to scale time units. The default is unity.
GLCID	Optional load curve ID (see *DEFINE_CURVE) defining a scale factor as a function of deflection for load curve ID, HLCID. If zero, a scale factor of unity is assumed.

Remarks:

If the linear spring stiffness is used, the force, F, is given by:

$$F = F_0 + \mathrm{K}\Delta L + \mathrm{D}\Delta\dot{L} \quad .$$

But if the load curve ID is specified, the force is then given by:

$$F = F_0 + \mathrm{K}f(\Delta L)\left\{1 + \mathrm{C1}\times\Delta\dot{L} + \mathrm{C2}\times\mathrm{sgn}(\Delta\dot{L})\ln\left[\max\left(1.,\frac{\Delta\dot{L}}{\mathrm{DLE}}\right)\right]\right\} + \mathrm{D}\Delta\dot{L} + g(\Delta L)h(\Delta\dot{L}) \quad .$$

In these equations, ΔL is the change in length, that is,

$$\Delta L = \text{current length} - \text{initial length} \quad .$$

The cross-sectional area is defined on the section card for the discrete beam elements; see *SECTION_BEAM. The square root of this area is used as the contact thickness offset if these elements are included in the contact treatment.

*MAT_BILKHU/DUBOIS_FOAM

This is Material Type 75. This model is for simulating isotropic crushable foams. Uniaxial and triaxial test data are used to describe the behavior.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	LCPY	LCUYS	VC	PC	VPC
Type	A	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	TSC	VTSC	LCRATE	PR	KCON	ISFLG	NCYCLE	
Type	I	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus, E
LCPY	Load curve ID giving pressure for plastic yielding as a function of volumetric strain; see Figure M75-1.
LCUYS	Load curve ID giving uniaxial yield stress as a function of volumetric strain (see Figure M75-1). All abscissa values should be positive if only the results of a compression test are included. Optionally the results of a tensile test can be added (corresponding to negative values of the volumetric strain); in this case PC, VPC, TC and VTC will be ignored.
VC	Viscous damping coefficient (0.05 < recommended value < 0.50; default is 0.05)
PC	Pressure cutoff for hydrostatic tension. If zero, the default is set to one-tenth of p_0, the yield pressure corresponding to a volumetric strain of zero. PC will be ignored if TC is nonzero.

VARIABLE	DESCRIPTION
VPC	Variable pressure cutoff for hydrostatic tension as a fraction of pressure yield value. If nonzero this will override the pressure cutoff value PC.
TC	Tension cutoff for uniaxial tensile stress. Default is zero. A nonzero value is recommended for better stability.
VTC	Variable tension cutoff for uniaxial tensile stress as a fraction of the uniaxial compressive yield strength. If nonzero; this will override the tension cutoff value TC.
LCRATE	Load curve ID giving a scale factor for the previous yield curves, dependent upon the volumetric strain rate
PR	Poisson's ratio, which applies to both elastic and plastic deformations. It must be smaller then 0.5.
KCON	Stiffness coefficient for contact interface stiffness. If undefined one-third of Young's modulus, E, is used. KCON is also considered in the element time step calculation; therefore, large values may reduce the element time step size.
ISFLG	Flag for tensile response (active only if negative abscissa are present in load curve LCUYS): EQ.0: load curve abscissa in tensile region correspond to volumetric strain EQ.1: load curve abscissa in tensile region correspond to effective strain (for large PR, when volumetric strain vanishes)
NCYCLE	Number of cycles to determine the average volumetric strain rate. NCYCLE is 1 by default (no smoothing) and cannot exceed 100.

Remarks:

1. **Volumetric Strain.** The logarithmic volumetric strain is defined in terms of the relative volume, V, as:

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

If option ISFLG = 1 is used, the effective strain is defined in the usual way:

$$\varepsilon_{\text{eff}} = \sqrt{\frac{2}{3}\text{tr}(\boldsymbol{\varepsilon}^{\text{t}}\boldsymbol{\varepsilon})}$$

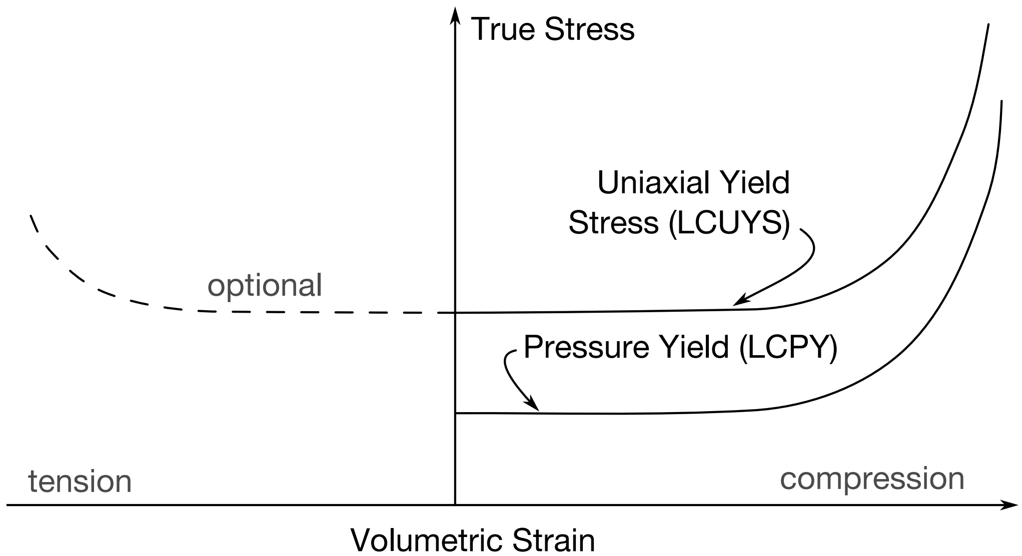


Figure M75-1. Behavior of crushable foam. Unloading is elastic.

The stress and strain pairs in load curve LCPY should be positive values starting with a volumetric strain value of zero.

2. **LCUYS.** The load curve LCUYS can optionally contain the results of the tensile test (corresponding to negative values of the volumetric strain); if it does, then the load curve information will override PC, VPC, TC and VTC. This is the recommended approach, because the necessary continuity between tensile and compressive regime becomes obvious (see Figure M75-1).

3. **Yield Surface.** The yield surface is defined as an ellipse in the equivalent pressure and von Mises stress plane. This ellipse is characterized by three points:

   a) the hydrostatic compression limit (LCPY),

   b) the uniaxial compression limit (LCUYS), and

   c) either the pressure cutoff for hydrostatic stress (PC,VPC), the tension cutoff for uniaxial tension (TC,VTC), or the optional tensile part of LCUYS.

4. **High Frequency Oscillations.** To prevent high frequency oscillations in the strain rate from causing similar high frequency oscillations in the yield stress, a modified volumetric strain rate is used to obtain the scaled yield stress. The modified strain rate is obtained as follows. If NYCLE is > 1, then the modified strain rate is obtained by a time average of the actual strain rate over NCYCLE solution cycles. The averaged strain rate is stored in history variable #3.

*MAT_GENERAL_VISCOELASTIC_{OPTION}

The available options include:

<BLANK>

MOISTURE

This is Material Type 76. This material model provides a general viscoelastic Maxwell model having up to 18 terms in the Prony series expansion and is useful for modeling dense continuum rubbers and solid explosives. Either the coefficients of the Prony series expansion or a relaxation curve may be specified to define the viscoelastic deviatoric and bulk behavior.

The material model can also be used with laminated shells. Either an elastic or viscoelastic layer can be defined with the laminated formulation. To activate laminated shells, you must set the laminated formulation flag on *CONTROL_SHELL. With the laminated option you must also define an integration rule. The addition of an elastic or viscoelastic layer was implemented by Professor Ala Tabiei, and the laminated shells feature was developed and implemented by Professor Ala Tabiei.

Card Summary:

Card 1. This card is required.

MID	RO	BULK	PCF	EF	TREF	A	B

Card 2. Leave blank if the Prony Series Cards (Card 4) are used below. Also, leave blank if an elastic layer is defined in a laminated shell.

LCID	NT	BSTART	TRAMP	LCIDK	NTK	BSTARTK	TRAMPK

Card 3. This card is included if and only if the MOISTURE keyword option is used.

MO	ALPHA	BETA	GAMMA	MST			

Card 4. Up to 18 cards may be input. This input is terminated at the next keyword ("*") card.

G*i*	BETA*i*	K*i*	BETAK*i*				

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	BULK	PCF	EF	TREF	A	B
Type	A	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
BULK	Elastic bulk modulus
PCF	Tensile pressure elimination flag for solid elements only. If set to unity, tensile pressures are set to zero.
EF	Elastic flag: EQ.0: The layer is viscoelastic. EQ.1: The layer is elastic.
TREF	Reference temperature for shift function (must be greater than zero)
A	Coefficient for the Arrhenius and the Williams-Landel-Ferry shift functions
B	Coefficient for the Williams-Landel-Ferry shift function

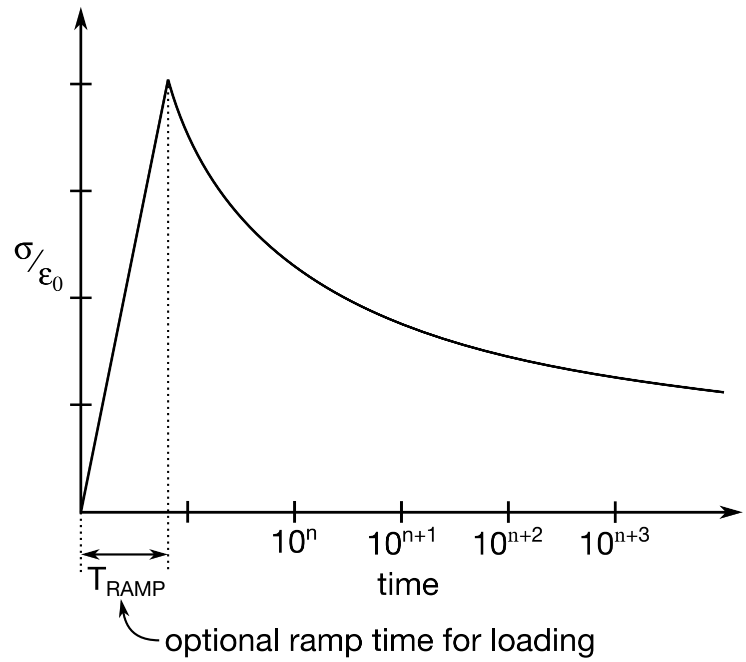


Figure M76-1. Relaxation curves for deviatoric behavior and bulk behavior. The ordinate of LCID is the deviatoric stress divided by 2 times the constant value of deviatoric strain where the stress and strain are in the direction of the prescribed strain, or in non-directional terms, the effective stress divided by 3 times the effective strain. LCIDK defines the mean stress divided by the constant value of volumetric strain imposed in a hydrostatic stress relaxation experiment as a function of time. For best results, the points defined in the curve should be equally spaced on the logarithmic scale. *Note the values for the abscissa are input as time, not log(time).* Furthermore, the 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.

Relaxation Curve Card. Leave blank if the *Prony Series Cards* are used below. Also, leave blank if an elastic layer is defined in a laminated shell.

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

VARIABLE	DESCRIPTION
LCID	Load curve ID for deviatoric relaxation behavior. If LCID is given, constants G_i, and β_i are determined using a least squares fit. See Figure M76-1 for an example relaxation curve.
NT	Number of terms in shear fit. If zero, the default is 6. Fewer than NT terms will be used if the fit produces one or more negative shear moduli. Currently, the maximum number is set to 18.
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 10 times $\beta_{3,}$ 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 relaxation behavior. If LCIDK is given, constants K_i, and β_{ki} are determined via a least squares fit. See Figure M76-1 for an example relaxation curve.
NTK	Number of terms desired in bulk fit. If zero, the default is 6. Currently, the maximum number is set to 18.
BSTARTK	In the fit, β_{k1} is set to zero, β_{k2} is set to BSTARTK, β_{k3} is 10 times β_{k2}, β_{k4} is 100 times $\beta_{k3,}$ and so on. If zero, BSTARTK is determined by an iterative trial and error scheme.
TRAMPK	Optional ramp time for bulk loading.

Moisture Card. Additional card for the MOISTURE keyword option.

Card 3	1	2	3	4	5	6	7	8
Variable	MO	ALPHA	BETA	GAMMA	MST			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
MO	Initial moisture, M_0. Defaults to zero.
ALPHA	Specifies α as a function of moisture. GT.0.0: Specifies a curve ID. LT.0.0: Specifies the negative of a constant value.

VARIABLE	DESCRIPTION
BETA	Specifies β as a function of moisture. GT.0.0: Specifies a curve ID. LT.0.0: Specifies the negative of a constant value.
GAMMA	Specifies γ as a function of moisture. GT.0.0: Specifies a curve ID. LT.0.0: Specifies the negative of a constant value.
MST	Moisture, M. If the moisture is 0.0, the moisture option is disabled. GT.0.0: Specifies a curve ID giving moisture as a function of time. LT.0.0: Specifies the negative of a constant value of moisture.

Prony Series cards. Card Format for viscoelastic constants. Up to 18 cards may be input. If fewer than 18 cards are used, the next keyword ("*") card terminates this input. 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: insert zero if a term is not included. If an elastic layer is defined you only need to define G*i* and K*i* (note in an elastic layer only one card is needed)

Card 4	1	2	3	4	5	6	7	8
Variable	G*i*	BETA*i*	K*i*	BETAK*i*				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
G*i*	Optional shear relaxation modulus for the i^{th} term
BETA*i*	Optional shear decay constant for the i^{th} term
K*i*	Optional bulk relaxation modulus for the i^{th} term
BETAK*i*	Optional bulk decay constant for the i^{th} 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 18 terms from the Prony series:

$$g(t) = \sum_{m=1}^{N} G_m e^{-\beta_m t}$$

We characterize this in the input by shear moduli, G_i, and decay constants, β_i. An arbitrary number of terms, up to 18, may be used when applying the viscoelastic model.

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

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

The Arrhenius and Williams-Landel-Ferry (WLF) shift functions account for the effects of the temperature on the stress relaxation. A scaled time, t',

$$t' = \int_0^t \Phi(T)dt$$

is used in the relaxation function instead of the physical time. The Arrhenius shift function is

$$\Phi(T) = \exp\left[-A\left(\frac{1}{T} - \frac{1}{T_{\mathrm{REF}}}\right)\right]$$

and the Williams-Landel-Ferry shift function is

$$\Phi(T) = \exp\left(-A\frac{T - T_{\mathrm{REF}}}{B + T - T_{\mathrm{REF}}}\right)$$

If all three values (TREF, A, and B) are nonzero, the WLF function is used; the Arrhenius function is used if B is zero; and no scaling is applied if all three values are zero.

The moisture model allows the scaling of the material properties as a function of the moisture content of the material. The shear and bulk moduli are scaled by α, the decay constants are scaled by β, and a moisture strain, $\gamma(M)[M - M_O]$ is introduced analogous to the thermal strain.

*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 Summary:

Card 1. This card is required.

MID	RO	PR	N	NV	G	SIGF	REF

Card 2. Include this card if PR < 0.

TBHYS	LCBI	LCPL	WBI	WPL	D1	D2	D3

Card 3a. Include this card if N > 0.

SGL	SW	ST	LCID1	DATA	LCID2	BSTART	TRAMP

Card 3b. Include this card if N = 0.

C10	C01	C11	C20	C02	C30	THERML	

Card 4. Include up to 12 of this card. The next keyword ("*") card terminates this input. Note that VFLAG is only included in the first card of this set.

G*i*	BETA*i*	G*j*	SIGF*j*	VFLAG			

Data Card Definitions:

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density

VARIABLE	DESCRIPTION
PR	Poisson's ratio. If set to a negative number, the Poisson's ratio is the absolute value, and Card 2 is included for extra parameters. Setting to 0.5 activates a *U-P* formulation for implicit analysis; see Remark 3 of *MAT_027 (the Mooney-Rivlin rubber model).
N	Number of hyperelastic constants to solve for from LCID1 or combinations of LCID1, LCBI, and LCPL: EQ.0: Set hyperelastic constants directly. 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 used in fitting curve LCID2. If zero, the default is 6. Currently, 12 is the maximum number. We recommend values less than 12, possibly 3 – 5, since each term used adds significantly to the cost. Exercise caution when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once you have achieved a satisfactory fit, we recommend inputting the coefficients written into the output file for future runs.
G	Shear modulus for frequency-independent damping. Frequency-independent damping is based on a spring and slider in series. The critical stress for the slider mechanism is SIGF, defined below. For the best results, the value of G should be 250 - 1000 times greater than SIGF.
SIGF	Limit stress for frequency-independent frictional damping
REF	Use reference geometry to initialize the stress tensor. *INITIAL_FOAM_REFERENCE_GEOMETRY defines the reference geometry. EQ.0.0: Off EQ.1.0: On

Hysteresis Card. Additional card included when PR < 0.

Card 2	1	2	3	4	5	6	7	8
Variable	TBHYS	LCBI	LCPL	WBI	WPL	D1	D2	D3
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
TBHYS	Table ID for hysteresis, which can be positive or negative; see Remarks 1 and 2. This field only applies to solid elements.
LCBI	Load curve ID giving force as a function of displacement for the biaxial test used for parameter fitting. Make sure N > 0 on Card 1 if setting this parameter. See Remark 3.
LCPL	Load curve ID giving force as a function of displacement for the planar test used for parameter fitting. Make sure N > 0 on Card 1 if setting this parameter. See Remark 3.
WBI	Weight factor giving the relative influence of the biaxial test data in the fitting of material parameters. A value of 1.0 means that the biaxial test data is of equal importance as the uniaxial test data. Make sure N > 0 on Card 1 if setting this parameter. See Remark 3.
WPL	Weight factor giving the relative influence of planar test data in the fitting of material parameters. A value of 1.0 means that the planar test data is of equal importance as the uniaxial test data. Make sure N > 0 on Card 1 if setting this parameter. See Remark 3.
D1	Compression compliance constant. If this parameter is greater than zero, then LS-DYNA does not use the value of PR set on Card 1 for Poisson’s ratio.
D2	Compression compliance constant
D3	Compression compliance constant

Card 3 for N > 0. For N > 0, LS-DYNA computes a least squares fit from the uniaxial or combined data.

Card 3a	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

VARIABLE	DESCRIPTION
SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness
LCID1	Load curve ID giving the force as a function of actual change in the gauge length. If SGL, SW, and ST are set to unity (1.0), curve LCID1 is also engineering stress as a function of engineering strain. Curve should have both negative (compressive) and positive (tensile) values.
DATA	Type of experimental data (only active if LCBI, LCPL, WBI, and WPL are all zero on Card 2 or Card 2 is not activated): EQ.0.0: Uniaxial data (only option for this model)
LCID2	Load curve ID of the deviatoric stress relaxation curve, neglecting the long term deviatoric stress. If LCID2 is specified, constants G_i and β_i are determined internally using a least squares fit. See Figure M76-1 for an example relaxation curve. The ordinate of the curve is the viscoelastic deviatoric stress divided by 2 times the constant value of deviatoric strain where the stress and strain are in the direction of the prescribed strain, or in non-directional terms, the effective stress divided by 3 times the effective strain.
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 10 times β_3, and so on. If zero, BSTART is determined by an iterative trial and error scheme.
TRAMP	Optional ramp time for loading

Card 3 for N = 0. Set the hyperelastic material parameters directly.

Card 3b	1	2	3	4	5	6	7	8
Variable	C10	C01	C11	C20	C02	C30	THERML	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
C10	C_{10}
C01	C_{01}
C11	C_{11}
C20	C_{20}
C02	C_{02}
C30	C_{30}
THERML	Flag for the thermal option. If THERML > 0.0, then G, SIGF, C10 and C01 must *all* specify curve IDs (zero is not permitted) that define the values as functions of temperature. If THERML < 0.0, then G, SIGF, C10 and C01, C11, C20, C02, and C30 must *all* specify curve IDs (zero is not permitted) that define the values as functions of temperature. A 'flat' curve may be used to define a constant value that does not change with temperature. This thermal option is available only for solid elements.

Optional Viscoelastic Constants & Frictional Damping Constant Cards. Up to 12 cards may be input. The next keyword ("*") card terminates this input.

Card 4	1	2	3	4	5	6	7	8
Variable	G*i*	BETA*i*	G*j*	SIGF*j*	VFLAG			
Type	F	F	F	F	F			

VARIABLE	DESCRIPTION
G*i*	Optional shear relaxation modulus for the i^{th} term. Not used if LCID2 is given.

VARIABLE	DESCRIPTION
BETAi	Optional decay constant of the i^{th} term. Not used if LCID2 is given.
Gj	Optional shear modulus for frequency independent damping represented as the j^{th} spring and slider in series in parallel to the rest of the stress contributions.
SIGFj	Limit stress for frequency independent, frictional, damping represented as the j^{th} spring and slider in series in parallel to the rest of the stress contributions.
VFLAG	Flag for the viscoelasticity formulation. This field appears only in the first Card 4 line. EQ.0: Standard viscoelasticity formulation (default) EQ.1: Viscoelasticity formulation using the instantaneous elastic stress (only applicable to solid elements).

Background:

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

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

$$J_1 = I_1 I_3^{-1/3}$$

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

To prevent volumetric work from contributing to the hydrostatic work, the first and second invariants are modified as shown. If D1 is positive, then

$$W_H(J) = \sum_{i=1}^{3} \frac{(J-1)^{2i}}{D_i} \; .$$

Otherwise, it is

$$W_H(J) = \frac{K}{2}(J-1)^2$$

with K being the linear bulk modulus determined from the corresponding linear shear modulus $G = 2(C_{10} + C_{01})$ and Poisson's ratio. Historically this model has been used for incompressible behavior, but it is also valid for compressible data. This procedure is described in more detail by Sussman and Bathe [1987]. The second Piola-Kirchhoff and Cauchy stress tensors are obtained from the strain energy functional as

$$\boldsymbol{S} = \frac{\partial W}{\partial \boldsymbol{E}}, \qquad \boldsymbol{\sigma}_W = \frac{1}{J}\boldsymbol{F}\boldsymbol{S}\boldsymbol{F}^T,$$

where $\boldsymbol{E}$ is the Green strain tensor and $\boldsymbol{F}$ is the deformation gradient. We use the subscript W here to denote the contribution from the strain energy potential, and with no other contributions the resulting Cauchy stress is simply

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}_W.$$

Rate effects are taken into account through linear viscoelasticity by adding a sequence of stress contributions

$$\boldsymbol{\sigma}_V = \sum_{i=1}^{n} \boldsymbol{\sigma}_V^i$$

where each term is known as a *Prony* term. Each such stress component $\boldsymbol{\sigma}_V^i$ evolves with deformation and time as

$$\left(\boldsymbol{\sigma}_V^i\right)^\nabla = 2G_i\left(\boldsymbol{D} - e^{-\beta_i(t-t_0)}\beta_i\boldsymbol{\varepsilon}_V^i\right), \qquad \left(\boldsymbol{\varepsilon}_V^i\right)^\nabla = e^{\beta_i(t-t_0)}\boldsymbol{D}.$$

Here ∇ denotes the Jaumann rate. $\boldsymbol{D}$ is the rate-of-deformation tensor, t is time and t_0 is an arbitrary time point. Each term has an internal strain $\boldsymbol{\varepsilon}_V^i$ associated with itself, which incorporates the memory properties a viscoelastic material typically possesses. This stress is added to the stress tensor determined from the strain energy functional, so that

$$\boldsymbol{\sigma} = \boldsymbol{\sigma}_W + \boldsymbol{\sigma}_V.$$

This model is effectively a Maxwell fluid which consists of dampers and springs in series. An arbitrary number of such Prony terms can be input, each characterized by the shear modulus, G_i, and relaxation coefficient, β_i. To avoid a constant shear modulus from this viscoelastic formulation, a term in the series is included only when $\beta_i > 0$.

For the sake of understanding the influence these terms have on the rate effects of viscoelasticity, let's investigate the model in a situation with no spin and constant rate-of-deformation with $\boldsymbol{D} \neq \boldsymbol{0}$. This means that the Jaumann rate is simply differentiation with time, and we can look at the implications a specific term has. To make some physical sense of things, we deal with both the no hyperelastic material present and the hyperelastic material present cases. For the latter we assume an elastic shear modulus, G, for the hyperelastic material.

1. **Constant strain rate, $\boldsymbol{D}$.** For the special case of constant strain rate, $\boldsymbol{D}$, we have the following expression for the stress rate

   $$\dot{\boldsymbol{\sigma}}_V^i = 2G_i e^{-\beta_i t}\boldsymbol{D},$$

   so each term contributes with an instantaneous shear stiffness of G_i that decays with time at a rate determined by β_i. If we define the relaxation time as

   $$\tau_i = 1/\beta_i,$$

we see that the term will not contribute much to the response when $t > 5\tau_i$. So with several Prony terms with different relaxation properties, the overall visco-elastic stiffness decays roughly with steps of G_i in time spans of τ_i. This information can be used for determining the material data by making clever use of tensile tests at different strain rates. Looking at the corresponding stress contribution from each term

$$\sigma_V^i = 2\frac{G_i}{\beta_i}(1 - e^{-\beta_i t})\boldsymbol{D}$$

we see that the stress stabilizes at a nonzero level $2\frac{G_i}{\beta_i}\boldsymbol{D}$ as time goes to infinity. See Figure M77-1.

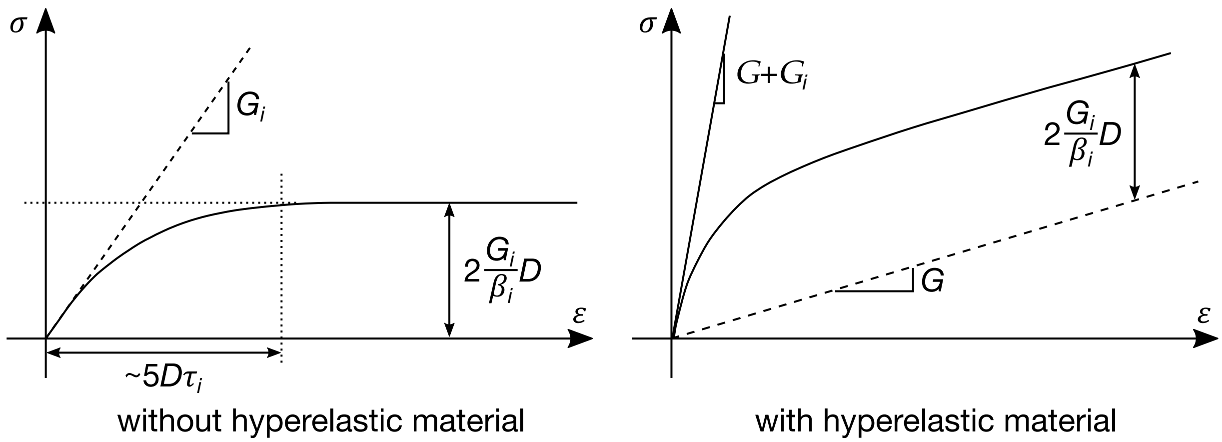


Figure M77-1. Material response with a constant strain rate

2. **Relaxation.** To see its effect on stress relaxation, we assume the material has deformed with a constant rate-of-deformation $\boldsymbol{D}_0 \neq \boldsymbol{0}$ between time 0 and t_0, and then continues with another constant rate-of-deformation $\boldsymbol{D}$ (which we allow to be zero) after time t_0 (see Figure M77-2). The expression for the stress is

$$\sigma_V^i = e^{-\beta_i(t-t_0)}\sigma_0^i + 2\frac{G_i}{\beta_i}(1 - e^{-\beta_i(t-t_0)})\boldsymbol{D},$$

where σ_0^i is the stress level that was reached at time t_0. Stress relaxation occurs when $\boldsymbol{D} = \boldsymbol{0}$ for which we see that the stress decays (or relaxes) to zero at a rate determined by β_i. When a hyperelastic material is included, the stress is relaxed to the hyperelastic stress, illustrated by a dashed line in the figure. As before, when combining many terms with different relaxation properties, the stress relaxes in steps of σ_0^i in time spans of τ_i and essentially determines the shape of the relaxation curve. This can also be used as a basis for estimating material parameters.

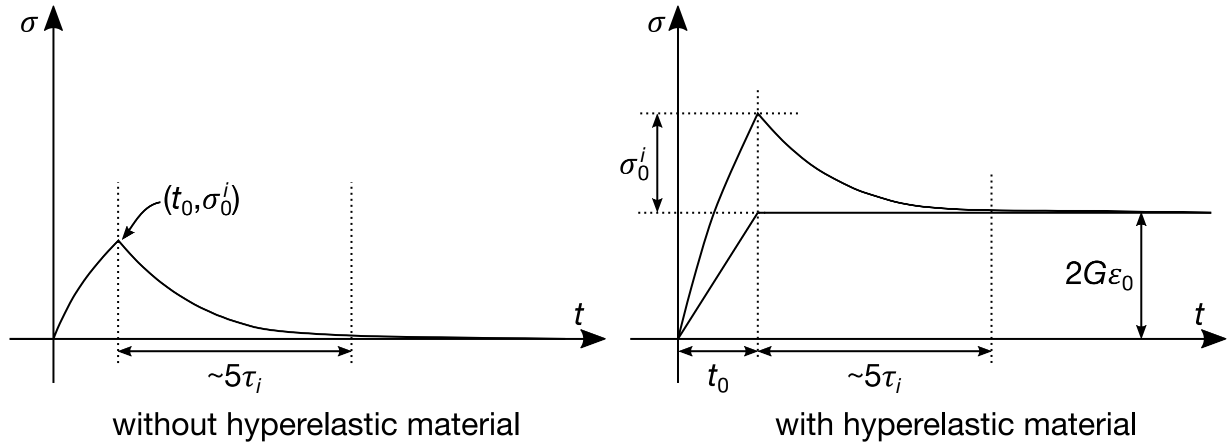


Figure M77-2. Stress relaxation curves

3. **Creep.** For creep, we assume the same situation but instead of prescribing the strain rate, $\boldsymbol{D}$, we enforce the stress, $\boldsymbol{\sigma}$, to be constant after time t_0. The expression for the creep strain, $\boldsymbol{\varepsilon}_c$, in the non-presence of a hyperelastic material becomes

$$\varepsilon_c = \frac{\beta_i}{2G_i}(t - t_0)\sigma_0^i,$$

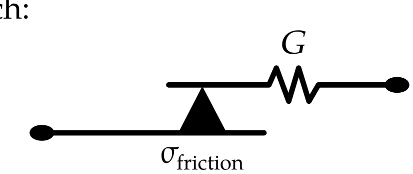
which indicates that the creep strain evolves linearly with time (see Figure M77-3). This is a rather non-physical behavior, but in the presence of a hyperelastic material the creep evolves as

$$\varepsilon_c = \frac{1}{2G_i}\ln\left\{\frac{G + G_i}{G + G_i e^{-\beta_i(t-t_0)}}\right\}\sigma_0^i$$

and saturates as one would expect to a constant value. With many such terms, the creep evolves in a quantitatively different manner, but the qualitative behavior is to be understood as described.

The Mooney-Rivlin rubber model (model 27) is obtained by specifying $N = 1$. Despite the differences in formulations, 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.

Frequency independent damping is obtained by having a spring and slider in series as shown in the following sketch:



Several springs and sliders in series can be defined that are put in parallel to the rest of the stress contributions of this material model.

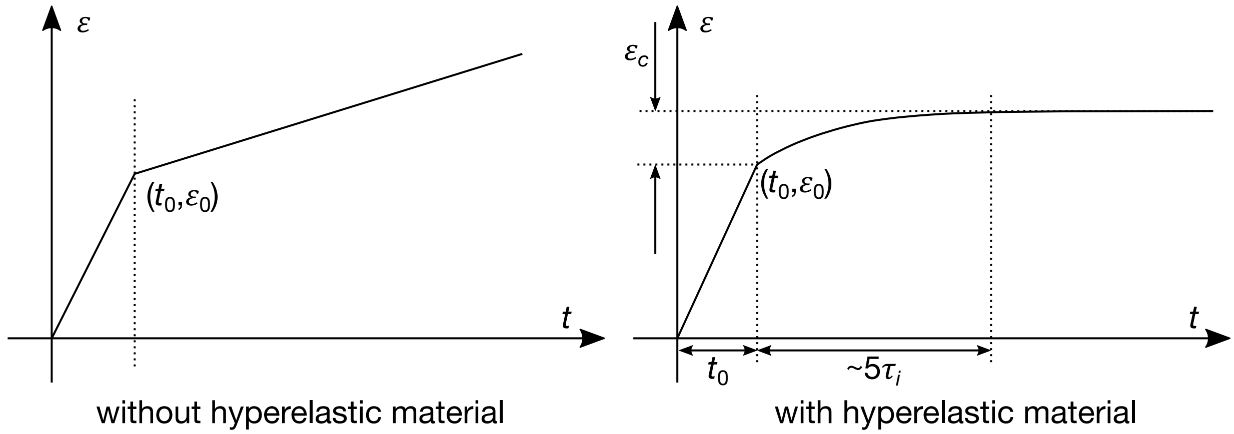


Figure M77-3. Creep curves

Remarks:

1. **Hysteresis (TBHYS > 0).** *If a positive table ID for hysteresis is defined*, then TBHYS is a table having curves that are functions of strain-energy density. Let W_{dev} be the current value of the deviatoric strain energy density as calculated above. Furthermore, let $\overline{W}_{\mathrm{dev}}$ be the peak strain energy density reached up to this point in time. It is then assumed that the resulting stress is reduced by a damage factor according to

$$\mathbf{S} = D(W_{\mathrm{dev}}, \overline{W}_{\mathrm{dev}})\frac{\partial W_{\mathrm{dev}}}{\partial \mathbf{E}} + \frac{\partial W_{\mathrm{vol}}}{\partial \mathbf{E}} ,$$

where $D(W_{\mathrm{dev}}, \overline{W}_{\mathrm{dev}})$ is the damage factor which is input as the table, TBHYS. This table consists of curves giving stress reduction (between 0 and 1) as a function of W_{dev} indexed by $\overline{W}_{\mathrm{dev}}$.

Each $\overline{W}_{\mathrm{dev}}$ curve must be valid for strain energy densities between 0 and $\overline{W}_{\mathrm{dev}}$. It is *recommended* that each curve be monotonically increasing, and it is *required* that each curve equals 1 when $W_{\mathrm{dev}} > \overline{W}_{\mathrm{dev}}$. Additionally, *DEFINE_TABLE *requires* that each curve have the same beginning and end point and, furthermore, that they not cross except at the boundaries, although they are not required to cross.

This table can be roughly estimated from a uniaxial quasistatic compression test as follows (see Figure for an illustration of the different curves):

a) Load the specimen to a maximum displacement $\bar{d}$ and measure the force as function of displacement, $f_{\mathrm{load}}(d)$.

b) Unload the specimen again measuring the force as a function of displacement, $f_{\mathrm{unload}}(d)$.

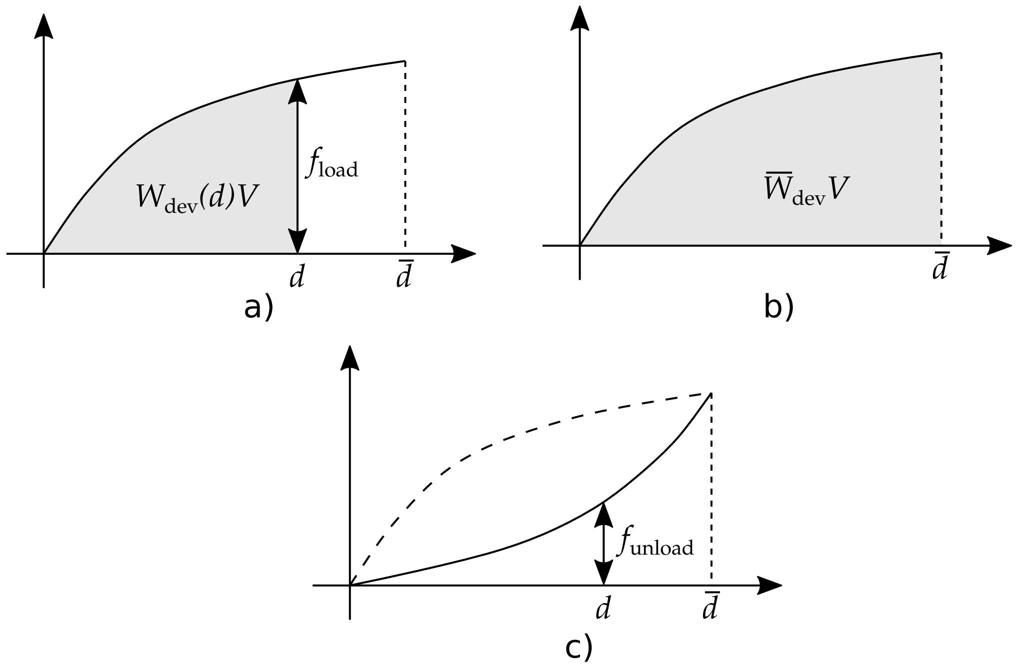


Figure M77-4. Illustration of curves needed from experiments to obtain $D(W_{\mathrm{dev}}, \overline{W}_{\mathrm{dev}})$. a) indicates the response during a uniaxial quasistatic compression test from which you can find $W_{\mathrm{dev}}(d)$ (area under the curve). Each test is associated with a maximum displacement and thus a peak strain energy, $\overline{W}_{\mathrm{dev}}$ (area under the curve in b)). c) indicates the unloading curve during the test. Inverting $W_{\mathrm{dev}}(d)$ allows you to find $D(W_{\mathrm{dev}}, \overline{W}_{\mathrm{dev}})$ from the loading and unloading curves for a value of $\overline{W}_{\mathrm{dev}}$.

c) The strain energy density is, then, given as a function of the loaded displacement as

$$W_{\mathrm{dev}}(d) = \frac{1}{V}\int_0^d f_{\mathrm{load}}(s)ds \quad .$$

   i) The peak energy, which is used to index the data set, is given by

$$\overline{W}_{\mathrm{dev}} = W_{\mathrm{dev}}(\bar{d}) \quad .$$

   ii) From this energy curve we can also determine the inverse, $d(W_{\mathrm{dev}})$. Using this inverse the load curve for LS-DYNA is then given by:

$$D(W_{\mathrm{dev}}, \overline{W}_{\mathrm{dev}}) = \frac{f_{\mathrm{unload}}[d(W_{\mathrm{dev}})]}{f_{\mathrm{load}}[d(W_{\mathrm{dev}})]} \quad .$$

d) This procedure is repeated for different values of $\bar{d}$ (or equivalently $\overline{W}_{\mathrm{dev}}$).

2. **Hysteresis (TBHYS < 0).** If a negative table ID for hysteresis is defined, then all of the above holds with the difference being that the load curves comprising table, |TBHYS|, must give the strain-energy density, W_{dev}, as a function of the

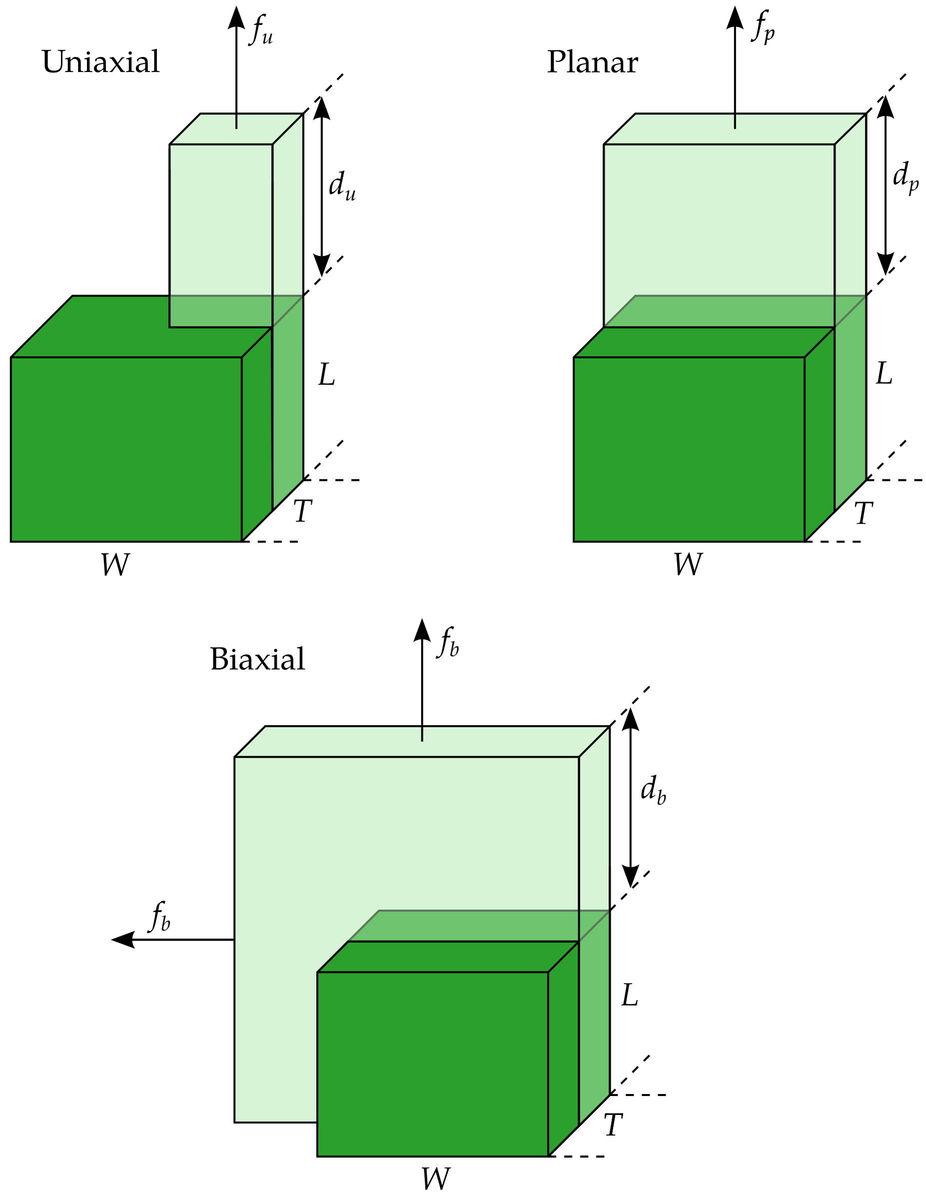


Figure M77-5. Tests for parameter fitting

stress reduction factor. *This scheme guarantees that all curves have the same beginning point, 0, and the same end point, 1.* For negative TBHYS the user provides $W_{\text{dev}}(D, \overline{W}_{\text{dev}})$ instead of $D(W_{\text{dev}}, \overline{W}_{\text{dev}})$. In practice, this case corresponds to swapping the load curve axes.

3. **Parameter fitting.** For general fitting of material parameters we refer to Figure M77-5. If at least one of LCBI with a positive WBI (w_b) or LCPL with a positive WPL (w_p) is set, parameters determined by N on Card 1 are fitted using a nonlinear least square optimization problem. We assume that LCID1 corresponds to a load curve giving f_u as a function of d_u, while LCBI and LCPL are load curves giving f_b as a function of d_b and f_p as a function of d_p, respectively. To obtain the test data, load a specimen of dimensions $L \times W \times T$ as shown in

the figure. The displacements must increase in the curves, and both compressive and tensile data is allowed. Let g_u, g_b and g_p be the simulated forces for the displacement data given, then the material parameters are determined to minimize the potential

$$h = \sum_{d_u} \left(1 - \frac{g_u}{f_u}\right)^2 + w_b \sum_{d_b} \left(1 - \frac{g_b}{f_b}\right)^2 + w_p \sum_{d_p} \left(1 - \frac{g_p}{f_p}\right)^2 \ .$$

The sums are supposed to be over the data points provided for each test. Note that the weight factors can be used to determine the relative influence of each test. Each term in the sums corresponds to the relative force error for the corresponding data point, this to obtain a better fit for smaller strains.

*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 Summary:

Card 1. This card is required.

MID	RO	PR	N	NV	G	SIGF	REF

Card 2. Include this card when PR < 0.

TBHYS	LCBI	LCPL	WBI	WPL	D1	D2	D3

Card 3a. Include this card if N > 0.

SGL	SW	ST	LCID1	DATA	LCID2	BSTART	TRAMP

Card 3b.1. Include this card if N = 0 or -1.

MU1	MU2	MU3	MU4	MU5	MU6	MU7	MU8

Card 3b.2. Include this card if N = 0 or -1.

ALPHA1	ALPHA2	ALPHA3	ALPHA4	ALPHA5	ALPHA6	ALPHA7	ALPHA8

Card 4. Include up to 12 of this card. This input ends with the next keyword ("*") card.

G*i*	BETA*i*	VFLAG					

Data Card Definitions:

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

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).

VARIABLE	DESCRIPTION
RO	Mass density
PR	Poisson's ratio. If set to a negative number, the Poisson's ratio is the absolute value, and Card 2 is included for extra parameters.
N	Order of fit to curve LCID1 or combinations of LCID1, LCBI, and LCPL for the Ogden model (currently < 9, 2 generally works okay). LS-DYNA prints the constants generated during the fit to d3hsp. To save the cost of performing the nonlinear fit in future runs, directly input the constants from this fit. You can visually evaluate the goodness of the fit by plotting data in the output file curveplot*. To do this with LS-PrePost, click *XYplot* → *Add* to read the curveplot* file. EQ.0: Allows you to specify the material parameters directly with Cards 3b.1 and 3b.2 EQ.-1: Same as N = 0 but invokes a thermal option: parameters MU*i* and ALPHA*i* are read as load curves IDs and thereby define these parameters as functions of temperature. It is available only for solid elements. VFLAG must be 0.
NV	Number of Prony series terms for fitting curve LCID2. If zero, the default is 6. Currently, 12 is the maximum number. We recommend values less than 12, possibly 3 – 5, since each term used adds significantly to the cost. Exercise caution when taking the results from the fit. Preferably, all generated coefficients should be positive. Negative values may lead to unstable results. Once you have achieved a satisfactory fit, we recommend inputting the coefficients written into the output file for future runs.
G	Shear modulus for frequency independent damping. Frequency independent damping is based on a spring and slider in series. The critical stress for the slider mechanism is SIGF defined below. For the best results, the value of G should be 250 - 1000 times greater than SIGF.
SIGF	Limit stress for frequency independent frictional damping
REF	Use reference geometry to initialize the stress tensor. *INITIAL_-FOAM_REFERENCE_GEOMETRY defines the reference geometry. EQ.0.0: Off

VARIABLE	DESCRIPTION
	EQ.1.0: On

Hysteresis Card. Additional card included when PR < 0.

Card 2	1	2	3	4	5	6	7	8
Variable	TBHYS	LCBI	LCPL	WBI	WPL	D1	D2	D3
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
TBHYS	Table ID for hysteresis, could be positive or negative; see Remarks 1 and 2 in the manual page for *MAT_HYPERELASTIC_RUBBER. This field only applies to solid elements.
LCBI	Load curve ID giving force as a function of displacement for the biaxial test used in parameter fitting. Make sure N > 0 on Card 1 if setting this parameter. See Remark 3 in the manual page for *MAT_HYPERELASTIC_RUBBER.
LCPL	Load curve ID giving force as a function of displacement for the planar test used in parameter fitting. Make sure N > 0 on Card 1 if setting this parameter. See Remark 3 in the manual page for *MAT_HYPERELASTIC_RUBBER.
WBI	Weight factor giving the relative influence of the biaxial test data in the fitting of material parameters, a value of 1.0 means that it is of equal importance as the uniaxial test data. Make sure N > 0 on Card 1 if setting this parameter. See Remark 3 in the manual page for *MAT_HYPERELASTIC_RUBBER.
WPL	Weight factor giving the relative influence of the planar test data in the fitting of material parameters, a value of 1.0 means that it is of equal importance as the uniaxial test data. Make sure N > 0 on Card 1 if setting this parameter. See Remark 3 in the manual page for *MAT_HYPERELASTIC_RUBBER.
D1	Compression compliance constant. If this parameter is greater than zero, then LS-DYNA does not use the value of PR set on Card 1 for Poisson’s ratio.
D2	Compression compliance constant

VARIABLE	DESCRIPTION
D3	Compression compliance constant

Least Squares Card. For N > 0, a least squares fit to curve LCID1 or LCID1/LCBI/LCPL is computed.

Card 3a	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

VARIABLE	DESCRIPTION
SGL	Specimen gauge length
SW	Specimen width
ST	Specimen thickness
LCID1	Load curve ID giving the force as a function of actual change in the gauge length. If SGL, SW, and ST are set to unity (1.0), then curve LCID1 is also engineering stress as a function of engineering strain. Curve should have both negative (compressive) and positive (tensile) values.
DATA	Type of experimental data (only active if LCBI, LCPL, WBI, and WPL are all zero on Card 2 or Card 2 is not activated): EQ.1.0: Uniaxial data (default) EQ.2.0: Biaxial data EQ.3.0: Pure shear data
LCID2	Load curve ID of the deviatoric stress relaxation curve, neglecting the long term deviatoric stress. If LCID2 is given, constants G_i and β_i are determined using a least squares fit. See M76-1 for an example relaxation curve. The ordinate of the curve is the viscoelastic deviatoric stress divided by the quantity 2 times the constant value of deviatoric strain where the stress and strain are in the direction of the prescribed strain. If in non-directional terms, it is the effective stress divided by the quantity 3 times the effective strain.

VARIABLE	DESCRIPTION
BSTART	In the fit, β_i is set to zero, β_2 is set to BSTART, β_3 is 10 times β_2, β_4 is 10 times β_3, and so on. If zero, BSTART is determined by an iterative trial and error scheme.
TRAMP	Optional ramp time for loading

Material Parameters Card. Include for N = 0 or N = -1 to set the material parameters directly.

Card 3b.1	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

Material Parameters Card. Include for N = 0 or N = -1 to set the material parameters directly.

Card 3b.2	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

VARIABLE	DESCRIPTION
MU*i*	μ_i, the i^{th} shear modulus (N = 0). i varies up to 8. For N = -1, each MU*i* is a load curve ID for specifying the i^{th} shear modulus as a function of temperature, that is, $\mu_i(T)$. If a curve ID is zero, then the corresponding shear modulus is a constant with value zero.
ALPHA*i*	α_i, the i^{th} exponent (N = 0). i varies up to 8. For N = -1, each ALPHA*i* is a load curve ID for specifying the i^{th} exponent as a function of temperature, that is, $\alpha_i(T)$. If a curve IDs is zero, then the corresponding exponent is a constant with value zero.

Optional Viscoelastic Constants Cards. Up to 12 cards may be input. The next keyword ("*") card terminates this input if fewer than 12 cards are used.

Card 4	1	2	3	4	5	6	7	8
Variable	G*i*	BETA*i*	VFLAG					
Type	F	F	I					
Default	none	none	0					

VARIABLE	DESCRIPTION
G*i*	Optional shear relaxation modulus for the i^{th} term. Not used if LCID2 is given.
BETA*i*	Optional decay constant if i^{th} term. Not used if LCID2 is given.
VFLAG	Flag for the viscoelasticity formulation. This appears only on the first line defining G*i*, BETA*i*, and VFLAG. If VFLAG = 0, the standard viscoelasticity formulation is used (the default), and if VFLAG = 1 (only applicable to solid elements), the viscoelasticity formulation using the instantaneous elastic stress is used.

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 that is a function of the relative volume, J, [Ogden 1984]:

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

The asterisk (*) indicates that the volumetric effects have been eliminated from the principal stretches, λ_j^*. The number of terms, n, may vary from 1 to 8 inclusive. If D1 is positive, then

$$W_H(J) = \sum\nolimits_{i=1}^{3}\frac{(J-1)^{2i}}{D_i}$$

whereas otherwise it is

$$W_H(J) = K(J-1-\ln J)$$

with K being the linear bulk modulus determined from the corresponding linear shear modulus $G = \frac{1}{2}\sum_{j=1}^{n}\mu_j\alpha_j$ and Poisson's ratio. Although this material is commonly used for incompressible rubber behavior, the theory is valid for compressible data as well.

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,

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

This model is effectively a Maxwell fluid which consists of a dampers and springs in series. We characterize this in the input by shear moduli, G_i, and decay constants, β_i. The viscoelastic behavior is optional, and an arbitrary number of terms may be used. In order to avoid a constant shear modulus from this viscoelastic formulation, a term in the series is included only when $\beta_i > 0$.

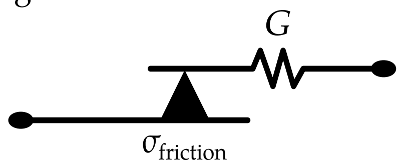
For VFLAG = 1, the viscoelastic term is

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

where σ_{kl}^E is the instantaneous stress evaluated from the internal energy functional. The coefficients in the Prony series therefore correspond to normalized relaxation moduli instead of elastic moduli.

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

The frequency independent damping is obtained by the having a spring and slider in series as shown in the following sketch:



*MAT_SOIL_CONCRETE

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

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	G	K	LCPV	LCYP	LCFP	LCRP
Type	A	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 or label must be specified (see *PART).
RO	Mass density
G	Shear modulus
K	Bulk modulus
LCPV	Load curve ID for pressure as a function of volumetric strain. The pressure as a function of volumetric strain curve is defined in compression only. The sign convention requires that both pressure and compressive strain be defined as positive values where the compressive strain is taken as the negative value of the natural logarithm of the relative volume.
LCYP	Load curve ID for yield as a function of pressure: GT.0: von Mises stress as a function of pressure, LT.0: Second stress invariant, J_2, as a function of pressure. This curve must be defined.

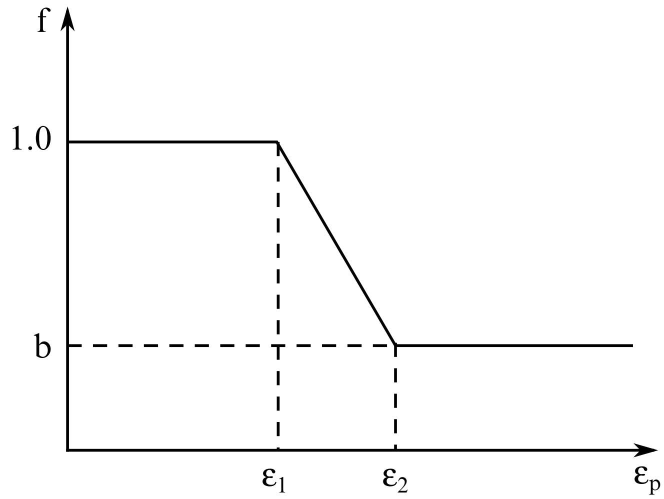


Figure M78-1. Strength reduction factor.

VARIABLE	DESCRIPTION
LCFP	Load curve ID for plastic strain at which fracture begins as a function of pressure. LCFP must be defined if B > 0.0.
LCRP	Load curve ID for plastic strain at which residual strength is reached as a function of pressure. LCRP 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 M78-1.
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 its 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

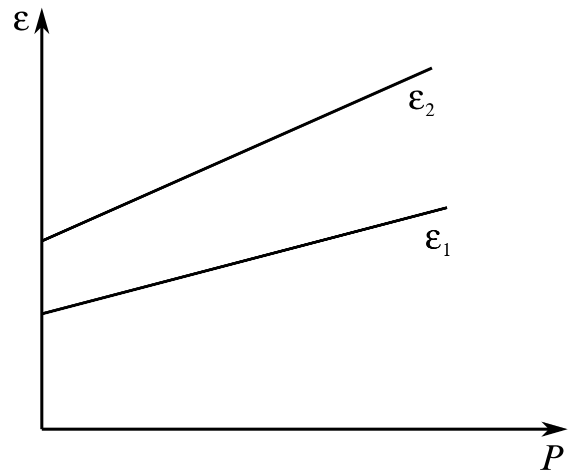


Figure M78-2. Cracking strain versus pressure.

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 as a function of pressure, and the second invariant, J_2, is defined in terms of the deviatoric stress tensor as:

$$J_2 = \frac{1}{2} S_{ij} S_{ij} \ .$$

If LCYP is negative, then the yield function becomes:

$$\phi = J_2 - F(p) \ .$$

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 M78-1.

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 functions of pressure that are defined by load curves, see Figure M78-2. The values on the curves are strain as a function of pressure 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 M78-3.

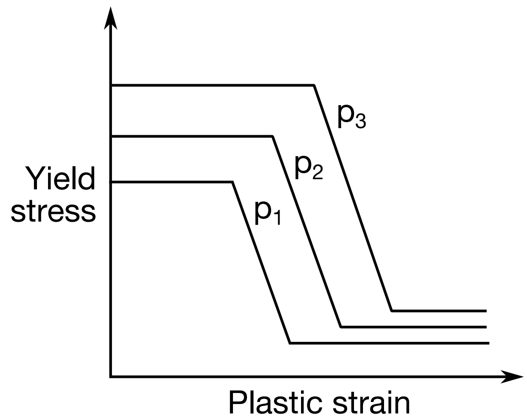


Figure M78-3. Yield stress as a function of plastic strain.

*MAT_HYSTERETIC_SOIL

This is Material Type 79. For this material, you supply a shear stress-strain curve. LS-DYNA converts this curve into a nested surface model with up to twenty superposed "layers" of elastic-perfectly-plastic material, each with its own elastic modulus and yield stress. The hysteretic behavior follows from the yielding of the different "layers" at different stresses and follows the so-called "Masing" rules. See Remarks below.

Card 1	1	2	3	4	5	6	7	8
Variable	MID	R0	K0	P0	B	A0	A1	A2
Type	A	F	F	F	F	F	F	F

Card 2	1	2	3	4	5	6	7	8
Variable	DF	RP	LCID	SFLC	DIL_A	DIL_B	DIL_C	DIL_D
Type	F	F	F	F	F	F	F	F

Card 3	1	2	3	4	5	6	7	8
Variable	GAM1	GAM2	GAM3	GAM4	GAM5	LCD	LCSR	PINIT
Type	F	F	F	F	F	I	I	I

Card 4	1	2	3	4	5	6	7	8
Variable	TAU1	TAU2	TAU3	TAU4	TAU5	FLAG5		
Type	F	F	F	F	F			

Include this card if FLAG5 = 1.

Card 5	1	2	3	4	5	6	7	8
Variable	SIGTH	SIGR	CHI	TPINIT				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
K0	Bulk modulus at the reference pressure. See Remark 1.
P0	Cut-off/datum pressure. P0 is irrelevant if $B = A1 = A2 = 0$. Otherwise, P0 must be < 0 (meaning tensile); a very small negative value is acceptable. Below this pressure, stiffness and strength go to zero. This is also the "zero" pressure for pressure-varying properties. See Remark 3.
B	Exponent for the pressure-sensitive elastic moduli, b. B must be in the range $0 \leq B < 1$. We do not recommend values too close to 1 because the pressure becomes indeterminate. See Remark 1.
A0	Yield function constant a_0 (default = 1.0); see Remark 5.
A1	Yield function constant a_1(default = 0.0); see Remark 5.
A2	Yield function constant a_2 (default = 0.0); see Remark 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; see Remarks 1, 2, and 5.
LCID	Load curve ID defining shear stress as a function of shear strain. Up to 20 points may be specified in the load curve. See *DEFINE_-CURVE and Remarks 4 and 7.
SFLC	Scale factor to apply to shear stress in LCID

VARIABLE	DESCRIPTION
DIL_A	Dilation parameter A, see Remark 11.
DIL_B	Dilation parameter B, see Remark 11.
DIL_C	Dilation parameter C, see Remark 11.
DIL_D	Dilation parameter D, see Remark 11.
GAM1	γ_1, shear strain (ignored if LCID is nonzero)
GAM2	γ_2, shear strain (ignored if LCID is nonzero)
GAM3	γ_3, shear strain (ignored if LCID is nonzero)
GAM4	γ_4, shear strain (ignored if LCID is nonzero)
GAM5	γ_5, shear strain (ignored if LCID is nonzero)
LCD	Optional load curve ID defining the damping ratio of hysteresis at different strain amplitudes (overrides Masing rules for unload/reload). The x-axis is the shear strain, and the y-axis is the damping ratio (such as 0.05 for 5% damping). The strains (x-axis values) of curve LCD must be identical to those of curve LCID. See Remark 15.
LCSR	Load curve ID defining plastic strain rate scaling effect on yield stress. See *DEFINE_CURVE. The x-axis is the plastic strain rate; the y-axis is the yield enhancement factor. See Remark 12.
PINIT	Flag for pressure sensitivity. Positive values apply to both B (elastic stiffness scaling) and the A0, A1, and A2 (strength scaling) equations. Negative values apply only to B, while the A0, A1, and A2 equations use the current pressure like PINIT = 0. See TPINIT below for changing the time PINIT applies, and see Remarks 9 and 10. |PINIT|.EQ.0: Use current pressure (will vary during the analysis). |PINIT|.EQ.1: Use pressure from the initial stress state. |PINIT|.EQ.2: Use initial “plane stress” pressure $(\sigma_v + \sigma_h)/2$. |PINIT|.EQ.3: User (compressive) initial vertical stress.
TAU1	τ_1, shear stress at γ_1(ignored if LCID is nonzero)

VARIABLE	DESCRIPTION
TAU2	τ_2, shear stress at γ_2 (ignored if LCID is nonzero)
TAU3	τ_3, shear stress at γ_3 (ignored if LCID is nonzero)
TAU4	τ_4, shear stress at γ_4 (ignored if LCID is nonzero)
TAU5	τ_5, shear stress at γ_5 (ignored if LCID is nonzero)
FLAG5	If FLAG5 = 1, optional Card 5 will be read.
SIGTH	Threshold shear stress ratio for cyclic degradation, see Remark 13.
SIGR	Residual shear stress ratio for cyclic degradation, see Remark 13.
CHI	Cyclic degradation parameter, see Remark 13.
TPINIT	Time at which PINIT applies. See Remark 10.

Remarks:

1. **Elastic moduli.** The elastic moduli G and K are pressure sensitive:

$$G(p) = \frac{G_0 (p - p_0)^b}{(p_{\mathrm{ref}} - p_0)^b}$$
$$K(p) = \frac{K_0 (p - p_0)^b}{(p_{\mathrm{ref}} - p_0)^b}$$

In the above K_0 is the input value K0, p is the current pressure, p_0 is the cut-off or datum pressure given by input value P0 (must be zero or negative), p_{ref} is the reference pressure given by the input value RP, b is the input value B, and G_0 is the initial shear modulus at small shear strain:

$$G_0 = \mathrm{SFLC} \times \frac{\tau_1}{\gamma_1} \ .$$

In the above (γ_1, τ_1), is the first (nonzero) point in LCID. G_0 is also the total of the shear moduli of all the nested layers; see Remark 6. For limitations on the value of B, see Remark 9.

2. **Volumetric response.** The following equation gives the pressure in compression:

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

Here V is the relative volume, the ratio between the original and current volume. p_{ref} and b are the input values RP and B, respectively. This formula results in an

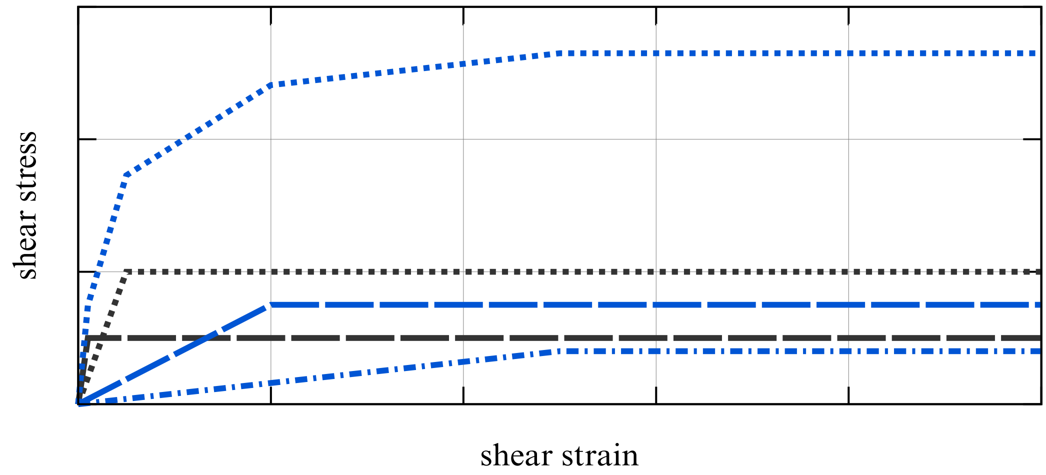


Figure M79-1. Family of stress-strain curves. The curve labeled total represents LCID × SFLC. The other curves represent one "layer" in the material model.

instantaneous bulk modulus that is proportional to p^b and whose value at the reference pressure equals $K_0/(1-b)$.

3. **Tensile cut-off.** If p falls below p_0 (i.e., becomes more tensile than input value P0), the shear stresses are set to zero, and the pressure is set to p_0. Thus, the material has no stiffness or strength when the pressure is more tensile than p_0.

4. **Shear stress-strain curve.** LCID and SFLC define a curve giving shear stress (τ) as a function of shear strain (γ). The shear strains are the x-axis values in LCID. The shear stresses are the y-axis values in LCID multiplied by SFLC. Starting from version R14, LCID may contain up to 20 points (in versions up to R13, the limit was 10 points). 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.

5. **Pressure-sensitivity of the shear response.** The curve LCID applies at the reference pressure (input value RP); at other pressures, the curve is scaled by

$$\frac{\tau(p,\gamma)}{\tau(p_{\mathrm{ref}},\gamma)} = \sqrt{\frac{[a_0 + a_1(p-p_0) + a_2(p-p_0)^2]}{[a_0 + a_1(p_{\mathrm{ref}}-p_0) + a_2(p_{\mathrm{ref}}-p_0)^2]}}$$

The constants a_0, a_1, and a_2 govern the pressure sensitivity of the yield stress. Only the ratios between these values are important - the absolute stress values are taken from the stress-strain curve scaled, as shown above.

6. **Nested yield surface approach.** LS-DYNA automatically converts the shear stress-strain curve (with points (γ_1,τ_1), (γ_2,τ_2),…, (γ_N,τ_N)) into a series of N elastic-perfectly-plastic curves such that $\sum(\tau_i,(\gamma)) = \tau(\gamma)$, as shown in

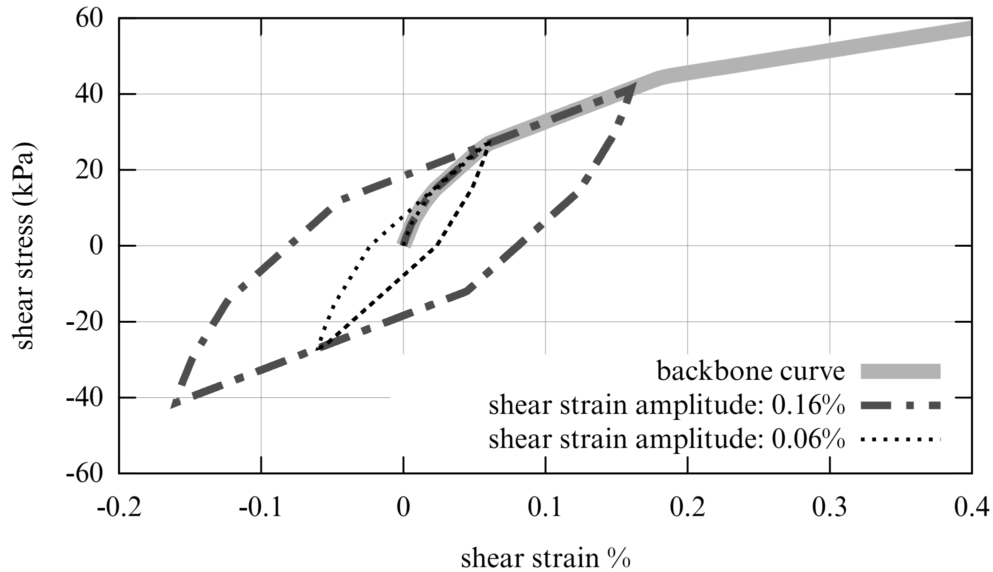


Figure M79-2. Small and large strain cycles superposed on the input curve

Figure M79-1. Each elastic-perfectly-plastic curve represents one "layer" in the material model. Deviatoric stresses are stored and calculated separately for each layer. The total deviatoric stress is the sum of the deviatoric stresses in each layer. This method generates hysteretic (energy-absorbing) stress-strain curves in response to any strain cycle of amplitude greater than the lowest yield strain of any layer. The example in Figure M79-2 shows the response to small and large strain cycles superposed on the input curve (thick line labeled backbone curve).

7. **Definition of shear strain and shear stress.** Different definitions of "shear strain" and "shear stress" are possible when applied to three-dimensional stress states. *MAT_079 uses the following definitions. Input shear stress τ (from LCID multiplied by SFLC) and shear strain γ (from LCID) are treated by the material model as:

$$\tau = 0.5 \times \mathrm{Von\ \ Mises\ \ Stress} = \sqrt{(3\sigma' : \sigma'/8)}$$

$$\gamma = 1.5 \times \mathrm{Von\ \ Mises\ \ Strain} = \sqrt{(3\varepsilon' : \varepsilon'/2)}$$

where σ' and ε' are the deviatoric stress and strain tensors respectively. For a particular stress or strain state (defined by the relationship among the three principal stresses or strains), a scaling factor may be needed to convert between the definitions given above and the shear stress or strain that an engineer would expect. The *MAT_079 definitions of shear stress and shear strain are derived from triaxial testing in which one principal stress is applied while the other two principal stresses are equal to a confining stress which is held constant. In other words, the principal stresses have the form $(a + q, a, a)$, and the shear stress, as defined above, is $0.5q$. If instead you wish the input curve to represent a test in which a pure shear strain is applied over a hydrostatic pressure, such as a shear-

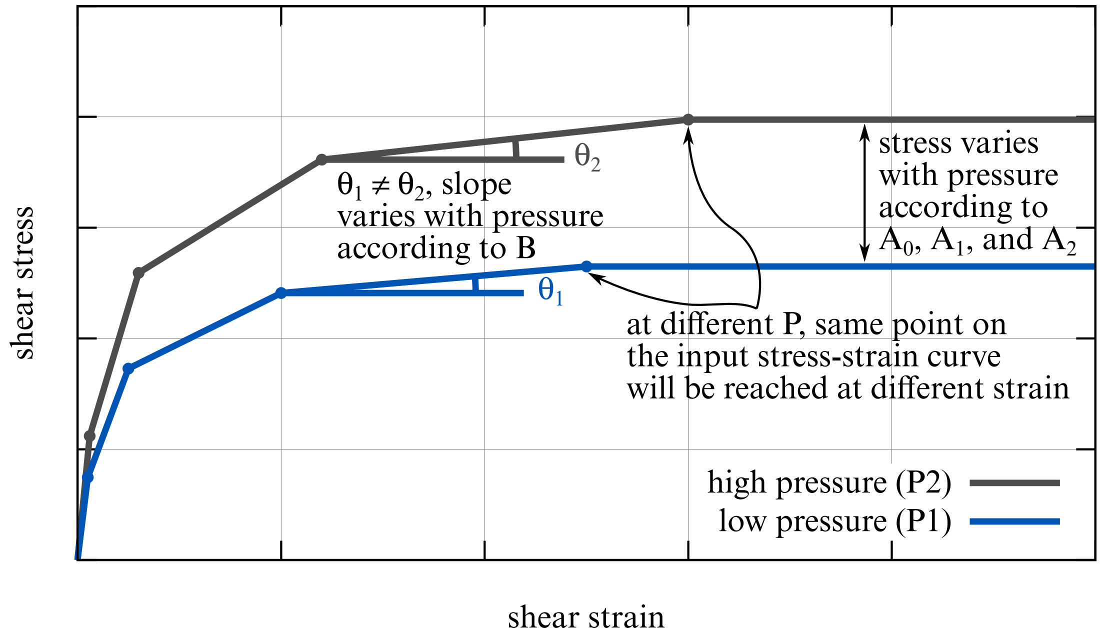


Figure M79-3. Sensitivity of curves to pressure

box test, then we recommend scaling both the x-axis and the y-axis of the curve LCID by 0.866. This factor assumes principal stresses of the form $(p + t, p - t, p)$ where t is the applied shear stress, and similarly for the principal strains.

8. **More about pressure sensitivity.** The yield stresses of the layers, and hence the stress at each point on the shear stress-strain input curve, vary with pressure according to constants A0, A1, and A2. The elastic moduli, and hence also the slope of each section of the shear stress-strain curve, vary with pressure according to constant B. These effects combine to modify the shear stress-strain curve according to pressure, as shown in Figure M79-3.

9. **PINIT.** Pressure sensitivity can make the solution sensitive to numerical noise. In cases where the expected pressure changes are small compared to the initial stress state, using pressure from the initial stress state instead of current pressure as the basis for the pressure sensitivity (option PINIT) may be preferable. This causes the bulk modulus and shear stress-strain curve to be calculated once for each element at the analysis's start and remain fixed thereafter. Positive settings of PINIT affect both stiffness scaling (calculated using B) and strength scaling (calculated using A0, A1, and A2). If using PINIT options 2 ("plane stress" pressure) or 3 (vertical stress), these quantities substitute for pressure p in the equations above. Input values of p_{ref} and p_0 should then also be "plane stress" pressure or vertical stress, respectively. Negative settings of PINIT have these effects only on stiffness scaling (B), while the strength scaling is re-calculated every time step from the current pressure as for PINIT = 0. If PINIT is nonzero, B is allowed to be as high as 1.0 (stiffness proportional to initial pressure); otherwise, we do not recommend values of B higher than about 0.5.

10. **TPINIT.** TPINIT is relevant only when PINIT is nonzero. When TPINIT = 0.0 (the default), PINIT acts at the start of the analysis. The pressures in each element on the first cycle (for instance, due to *INITIAL_STRESS_... cards) determine the pressure-sensitive stiffness and strength parameters which remain constant for the duration of the analysis. If TPINIT is nonzero, the stiffness and strength properties vary dynamically with pressure (same as PINIT = 0) until time TPINIT when they become frozen based on the pressure that exists at time TPINIT. For example, this feature can be used to build up stresses under gravity loading before applying PINIT. If using dynamic relaxation and TPINIT > 0.0, then PINIT acts at time TPINIT in the transient phase.

11. **Dilatancy.** Parameters DIL_A, DIL_B, DIL_C, and DIL_D control the compaction and dilatancy in sandy soils due to shearing motion. Using this feature with pore water pressure (see *CONTROL_PORE_FLUID) can model liquefaction. However, note that the compaction/dilatancy algorithm used in this material model is very unsophisticated compared to recently published research findings.

    The dilatancy is expressed as a volume strain, ε_v:

$$\begin{aligned}
\varepsilon_\mathrm{v} &= \varepsilon_\mathrm{r} + \varepsilon_\mathrm{g} \\
\varepsilon_\mathrm{r} &= \mathrm{DIL_A}(\Gamma)^{\mathrm{DIL_B}} \\
\varepsilon_\mathrm{g} &= \frac{\int (d\gamma_{xz}^2 + d\gamma_{yz}^2)^{1/2}}{\mathrm{DIL_C} + \mathrm{DIL_D} \times \int (d\gamma_{xz}^2 + d\gamma_{yz}^2)^{1/2}} \\
\Gamma &= (\gamma_{xz}^2 + \gamma_{yz}^2)^{1/2} \\
\gamma_{xz} &= 2\varepsilon_{xz} \\
\gamma_{yz} &= 2\varepsilon_{yz}
\end{aligned}$$

    ε_r describes soil's dilation due to the magnitude of the shear strains; this is caused by the soil particles having to climb over each other to develop shear strain. ε_g describes the compaction of the soil due to the collapse of weak areas and voids caused by continuous shear straining.

    Recommended inputs for sandy soil when modeling dilatancy are:

DIL_A	DIL_B	DIL_C	DIL_D
10	1.6	100	10

    DIL_A and DIL_B may cause instabilities in some models.

12. **Strain rate sensitivity.** Scaling the yield stress of each layer by a "rate enhancement factor" accounts for strain rate effects (see optional input field LCSR). This factor is a function of the plastic strain rate in that layer. The stress-strain curve defined by LCID and SFLC is for quasi-static loading. The rate enhancement factor (on the y-axis) is input as a function of plastic strain rate (on the x-axis) in

curve LCSR. All rate enhancement factors must be equal to or larger than 1.0. Because the rate enhancement factor applies to the strength but not the stiffness and is calculated separately for each layer, situations in which not all the layers are yielding cause an overall enhancement factor between 1.0 and the value in LCSR.

13. **Cyclic degradation.** See optional input fields SIGR, CHI, and SIGTH. Reducing the size of all yield surfaces proportionally based on the accumulation of the damage strain accounts for cyclic degradation. The following equation determines the strength reduction factor, f:

$$f = 1 - (1 - \Sigma_R)\left(1 - e^{\frac{-\chi\gamma_d}{1-\Sigma_R}}\right)$$

where Σ is the shear stress ratio (defined as current shear stress divided by shear strength at the current pressure); Σ_R is the residual shear strength ratio SIGR; χ is the input parameter CHI; and γ_d is the damage strain, defined as the summation of absolute incremental changes in Von Mises strain that accumulate whenever Σ exceeds the threshold shear stress ratio SIGTH.

14. **Saturated soil.** When modeling saturated soil, we do not recommend attempting to represent the additional stiffness of the pore water by increasing the bulk modulus on the *MAT card (this method is sometimes termed a “total stress” model). Using that method causes the pressure calculated by the material model to represent the total pressure (the pore water pressure plus the “effective pressure” which is the component of pressure associated with contact between the soil grains). Pressure-sensitive properties, such as shear strength, then depend unrealistically on total pressure, whereas in real-life soils, they depend on effective pressure. To obtain the latter behavior, model the pore pressure effects with *CONTROL_PORE_FLUID and *BOUNDARY_PORE_FLUID and set the properties on the *MAT card to represent the effective stress properties.

15. **Non-Masing damping.** See optional input field LCD. Hysteresis damping arises from the energy absorbed during each stress-strain cycle, meaning the area enclosed by the hysteresis loops, such as those shown in Figure M79-2. The nested yield surface approach of this material model governs the shape of the hysteresis loops, providing a level of damping known as “Masing damping” that depends on the shape of the input shear stress-strain curve and the cyclic strain amplitude. Masing damping often overestimates the actual hysteresis damping shown by soils in cyclic tests, particularly at high cyclic shear strains. To counteract this, specify input curve LCD to define “non-Masing damping.” In this material model, non-Masing damping acts like a damage model, progressively reducing the properties of the separate nested layers as the strain increases, making the hysteresis loops thinner. Thus, this feature only reduces the damping compared to the default Masing damping. It cannot increase the damping. Furthermore, the gap between Masing and non-Masing damping

must increase monotonically with increasing shear strain. The amount of hysteresis damping will not follow an input LCD if it does not obey these rules. In this case, LS-DYNA writes a warning and a table that includes the Masing damping for each point of the input stress-strain curve to the message file. LS-DYNA outputs the table to provide information for adjusting the damping ratios in LCD to meet the above requirements.

*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 1	1	2	3	4	5	6	7	8
Variable	MID	RO	GAMY	TAUY	ALPHA	R	BULK	
Type	A	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
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:

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} + \alpha \left|\frac{\tau}{\tau_y}\right|^r \qquad \text{for } \gamma \geq 0$$

$$\frac{\gamma}{\gamma_y} = \frac{\tau}{\tau_y} - \alpha \left|\frac{\tau}{\tau_y}\right|^r \qquad \text{for } \gamma < 0$$

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

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

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

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

The Ramberg-Osgood equations are inherently one-dimensional and are assumed to apply to shear components. To generalize this theory to the multidimensional case, it is assumed that each component of the deviatoric stress and deviatoric tensorial strain is independently related by the one-dimensional stress-strain equations. A projection is used to map the result back into deviatoric stress space if required. The volumetric behavior is elastic, and, therefore, the pressure p is found by

$$p = -K\varepsilon_v$$

where ε_v is the volumetric strain.

MAT_PLASTICITY_WITH_DAMAGE_*{OPTION}

This manual entry applies to *both* types 81 and 82. Materials 81 and 82 model an elasto-visco-plastic material with user-defined *isotropic* stress versus strain curves, which, themselves, may be strain-rate dependent. This model accounts for the effects of damage prior to rupture based on an effective plastic-strain measure. Additionally, failure can be triggered when the time step drops below some specified value. Adding an orthotropic damage option will invoke material type 82. Since type 82 must track directional strains it is, computationally, more expensive.

Available options include:

<BLANK>

ORTHO

ORTHO_RCDC

ORTHO_RCDC1980

STOCHASTIC

The keyword card can appear in the following ways:

*MAT_PLASTICITY_WITH_DAMAGE or *MAT_081

*MAT_PLASTICITY_WITH_DAMAGE_ORTHO or *MAT_082

*MAT_PLASTICITY_WITH_DAMAGE_ORTHO_RCDC or *MAT_082_RCDC

*MAT_PLASTICITY_WITH_DAMAGE_ORTHO_RCDC1980 or *MAT_082_RCDC1980

*MAT_PLASTICITY_WITH_DAMAGE_STOCHASTIC or *MAT_081_STOCHASTIC

The ORTHO option invokes an orthotropic damage model, an extension that was first added as for modelling failure in aluminum panels. Directional damage begins after a defined failure strain is reached in tension and continues to evolve until a tensile rupture strain is reached in either one of the two orthogonal directions. After rupture is detected at *all* integration points, the element is deleted.

The ORTHO_RCDC option invokes the damage model developed by Wilkins [Wilkins, et al. 1977]. The ORTHO_RCDC1980 option invokes a damage model based on strain invariants as developed by Wilkins [Wilkins, et al. 1980]. A nonlocal formulation, which requires additional storage, is used if a characteristic length is defined. The RCDC option, which was added at the request of Toyota, works well in predicting failure in cast aluminum; see Yamasaki, et al., [2006].

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	PR	SIGY	ETAN	EPPF	TDEL
Type	A	F	F	F	F	F	F	F
Default	none	none	none	none	none	0.0	10^{12}	0.0

Card 2	1	2	3	4	5	6	7	8
Variable	C	P	LCSS	LCSR	EPPFR	VP	LCDM	NUMINT
Type	F	F	F	F	F	F	F	I
Default	0	0	0	0	10^{14}	0	0	0

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

Ortho RCDC Card. Additional card for keyword options ORTHO_RCDC and ORTHO_-RCDC1980.

Card 5	1	2	3	4	5	6	7	8
Variable	ALPHA	BETA	GAMMA	D0	B	LAMBDA	DS	L
Type	F	F	F	F	F	F	F	F
Default	0	0	0	0	0	0	0	0

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus
PR	Poisson's ratio
SIGY	Yield stress
ETAN	Tangent modulus, ignored if (LCSS > 0) is defined.
EPPF	$\varepsilon^p_{\text{failure}}$, effective plastic strain at which material softening begins
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.** When LCSS is a load curve ID, it is taken as defining effective stress as a function of effective plastic strain. If defined, EPS1 - EPS8 and ES1 - ES8 are ignored. **Tabular Data.** The table ID defines for each strain rate value a load curve ID giving the stress as a function effective plastic strain for that rate; see Figure M24-1. When the strain rate falls below the minimum value, the stress as a function of effective plastic strain curve for the lowest value of strain rate is used. Likewise, when the strain rate exceeds the maximum value the stress as a function of effective plastic strain curve for the highest value of strain rate

VARIABLE	DESCRIPTION
	is used. C, P, LCSR, EPS1 - EPS8, and ES1 - ES8 are ignored if a table ID is defined. **Logarithmically Defined Tables.** If the *first* value in the table is negative, LS-DYNA assumes that all the table values represent the natural logarithm of a strain rate. Since the tables are internally discretized to equally space the table values, it makes good sense from an accuracy standpoint that the table values represent the natural log of strain rate when the lowest strain rate and highest strain rate differ by several orders of magnitude.
LCSR	Load curve ID defining strain rate scaling effect on yield stress
EPPFR	$\varepsilon^p_{\text{rupture}}$, effective plastic strain at which material ruptures
VP	Formulation for rate effects: EQ.0.0: Scale yield stress (default) EQ.1.0: Viscoplastic formulation
LCDM	Optional curve ID defining nonlinear damage curve. If this curve is specified, either EPPF or EPPFR must also be input. If LCDM, EPPF, and EPPFR are all nonzero, then EPPFR is ignored.
NUMINT	Number of through thickness integration points which must fail before a shell element is deleted. (If zero, all points must fail.) The default of all integration points is not recommended since shells undergoing large strain are often not deleted due to nodal fiber rotations which limit strains at active integration points after most points have failed. Better results are obtained if NUMINT is set to 1 or a number less than one half of the number of through thickness points. For example, if four through thickness points are used, NUMINT should not exceed 2, even for fully integrated shells which have 16 integration points.
EPS1 - EPS8	Effective plastic strain values (optional if SIGY is defined). At least 2 points should be defined.
ES1 - ES8	Corresponding yield stress values to EPS1 - EPS8
ALPHA	Parameter α for the Rc-Dc model
BETA	Parameter β for the Rc-Dc model
GAMMA	Parameter γ for the Rc-Dc model

VARIABLE	DESCRIPTION
D0	Parameter D_0 for the Rc-Dc model
B	Parameter b for the Rc-Dc model
LAMBDA	Parameter λ for the Rc-Dc model
DS	Parameter D_s for the Rc-Dc model
L	Optional characteristic element length for this material. If zero, nodal values of the damage function are used to compute the damage gradient. See discussion below.

Remarks:

1. 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 M24-1 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:

   a) 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, $\sigma_y^s(\varepsilon_{\text{eff}}^p)$, which is typically given by a load curve ID, and the initial yield stress, SIGY, multiplied by the Cowper-Symonds rate term as follows:

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

   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, $\sigma_y^s(\varepsilon_{\text{eff}}^p)$, must be defined by a load curve:

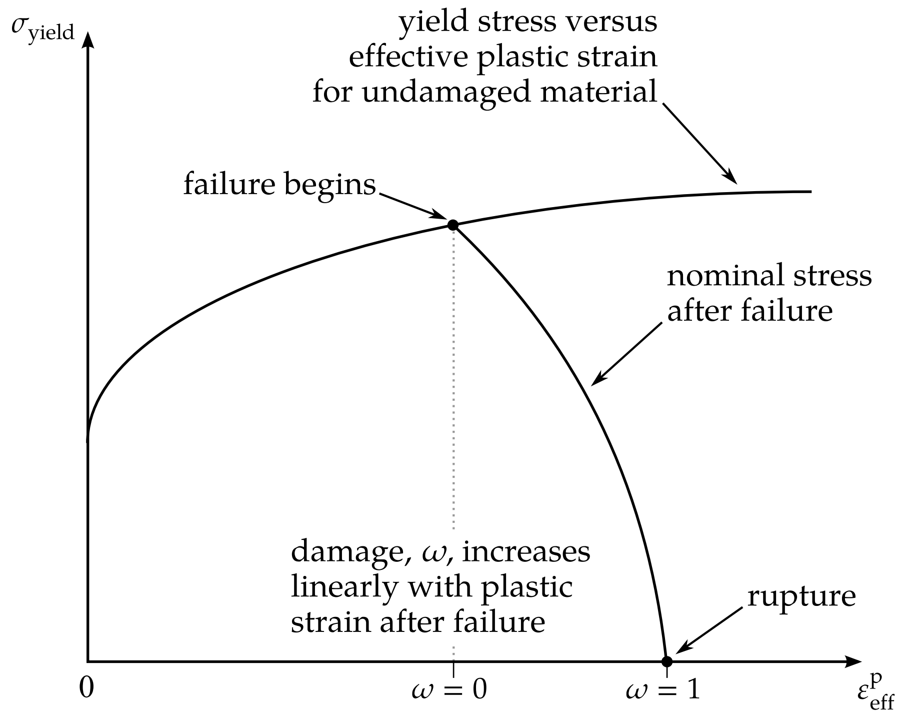


Figure M81-1. Stress strain behavior when damage is included

$$\sigma_y\left(\varepsilon^p_{\text{eff}}, \dot{\varepsilon}^p_{\text{eff}}\right) = \sigma^s_y\left(\varepsilon^p_{\text{eff}}\right)\left[1 + \left(\frac{\dot{\varepsilon}^p_{\text{eff}}}{C}\right)^{1/p}\right]$$

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

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

c) If different stress as a function of strain curves can be provided for various strain rates, a table (LCSS) can be used. Then the table input in *DEFINE_TABLE is expected; see Figure M24-1.

2. **Damage.** The constitutive properties for the damaged material are obtained from the undamaged material properties. The amount of damage is represented 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

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

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

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

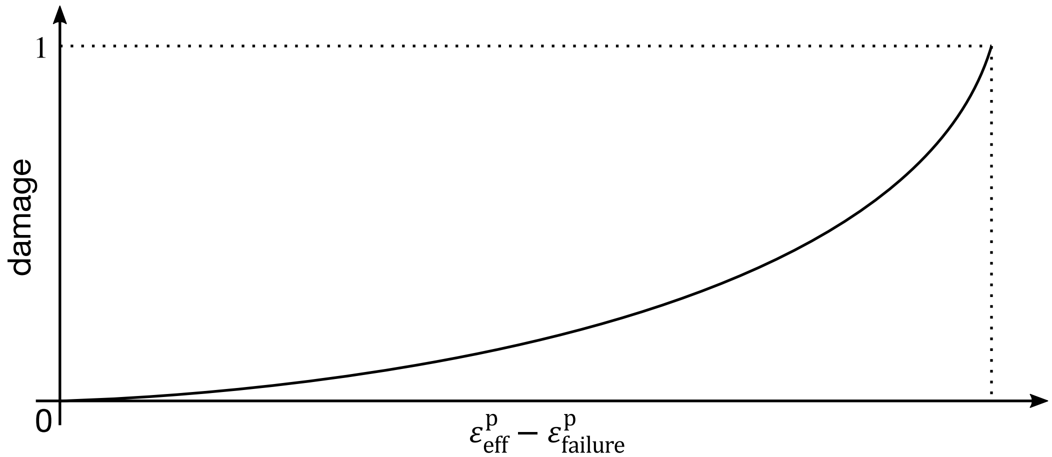


Figure M81-2. A nonlinear damage curve is optional. Note that the origin of the curve is at (0,0). The nonlinear damage curve is useful for controlling the softening behavior after the failure strain EPPF is reached.

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

$$\omega = \frac{A_{\mathrm{loss}}}{A}$$

such that

$$0 \le \omega \le 1.$$

In this model, unless LCDM is defined, damage is defined in terms of effective plastic strain after the failure strain is exceeded as follows:

$$\omega = \frac{\varepsilon^p_{\mathrm{eff}} - \varepsilon^p_{\mathrm{failure}}}{\varepsilon^p_{\mathrm{rupture}} - \varepsilon^p_{\mathrm{failure}}}, \qquad \varepsilon^p_{\mathrm{failure}} \le \varepsilon^p_{\mathrm{eff}} \le \varepsilon^p_{\mathrm{rupture}}$$

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

3. **Rc-Dc Model.** The damage, D, for the Rc-Dc model is given by:

$$D = \int \omega_1 \omega_2 d\varepsilon^p$$

where ε^p is the effective plastic strain,

$$\omega_1 = \left(\frac{1}{1 - \gamma\sigma_m}\right)^{\alpha}$$

is a triaxial stress weighting term and

$$\omega_2 = (2 - A_D)^{\beta}$$

is an asymmetric strain weighting term. In the above σ_m is the mean stress. For A_D we use

$$A_D = \min\left(\left|\frac{\sigma_2}{\sigma_3}\right|, \left|\frac{\sigma_3}{\sigma_2}\right|\right) \ ,$$

where σ_i are the principal stresses and $\sigma_1 > \sigma_2 > \sigma_3$. Fracture is initiated when the accumulation of damage is

$$\frac{D}{D_c} > 1 \ ,$$

where D_c is the critical damage given by

$$D_c = D_0\left(1 + b|\nabla D|^\lambda\right) \ .$$

A fracture fraction,

$$F = \frac{D - D_c}{D_s} \ ,$$

defines the degradations of the material by the Rc-Dc model.

For the Rc-Dc model the gradient of damage needs to be estimated. The damage is connected to the integration points, and, thus, the computation of the gradient requires some manipulation of the LS-DYNA source code. Provided that the damage is connected to nodes, it can be seen as a standard bilinear field and the gradient is easily obtained. To enable this, the damage at the integration points are transferred to the nodes as follows. Let E_n be the set of elements sharing node n, $|E_n|$ be the number of elements in that set, P_e be the set of integration points in element e and $|P_e|$ be the number of points in that set. The average damage $\overline{D}_e$ in element e is computed as

$$\overline{D}_e = \frac{\sum_{p \in P_e} D_p}{|P_e|}$$

where D_p is the damage in integration point p. Finally, the damage value in node n is estimated as

$$D_n = \frac{\sum_{e \in E_n} \overline{D}_e}{|E_n|}.$$

This computation is performed in each time step and requires additional storage. Currently we use three times the total number of nodes in the model for this calculation, but this could be reduced by a considerable factor if necessary.

There is an Rc-Dc option for the Gurson dilatational-plastic model. In the implementation of this model, the norm of the gradient is computed differently. Let E_f^l be the set of elements from within a distance l of element, f, not including the element itself, and let $\left|E_f^l\right|$ be the number of elements in that set. The norm of the gradient of damage is estimated roughly as

$$\|\nabla D\|_f \approx \frac{1}{\left|E_f^l\right|} \sum_{e \in E_f^l} \frac{\left|D_e - D_f\right|}{d_{ef}}$$

where d_{ef} is the distance between element f and e.

The reason for taking the first approach is that it should be a better approximation of the gradient; it can for one integration point in each element be seen as a weak gradient of an elementwise constant field. The memory consumption as well as computational work should not be much higher than for the other approach.

The RCDC1980 model is identical to the RCDC model except the expression for $A_D$is in terms of the principal stress deviators and takes the form

$$A_D = \max\left(\left|\frac{S_2}{S_3}\right|, \left|\frac{S_2}{S_1}\right|\right)$$

4. **STOCHASTIC Option.** The STOCHASTIC option allows spatially varying yield and failure behavior. See *DEFINE_STOCHASTIC_VARIATION for additional information.

5. **Material Histories.** *DEFINE_MATERIAL_HISTORIES can be used to output the instability, plastic strain rate, and damage, following

**DEFINE_MATERIAL_HISTORIES Properties*

Label	Attributes				Description
Instability	-	-	-	-	Failure indicator $\varepsilon^p_{\mathrm{eff}}/\varepsilon^p_{\mathrm{fail}}$, see EPPF
Plastic Strain Rate	-	-	-	-	Effective plastic strain rate $\dot{\varepsilon}^p_{\mathrm{eff}}$
Damage	-	-	-	-	Damage ω

MAT_FU_CHANG_FOAM_*{OPTION}

This is Material Type 83.

Available options include:

DAMAGE_DECAY

LOG_LOG_INTERPOLATION

PATH_DEPENDENT

Rate effects can be modeled in low and medium density foams; see Figure M83-1. Hysteretic unloading behavior in this model is a function of the rate sensitivity with the most rate sensitive foams providing the largest hysteresis and vice versa. The unified constitutive equations for foam materials by Chang [1995] provide 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 from a 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.

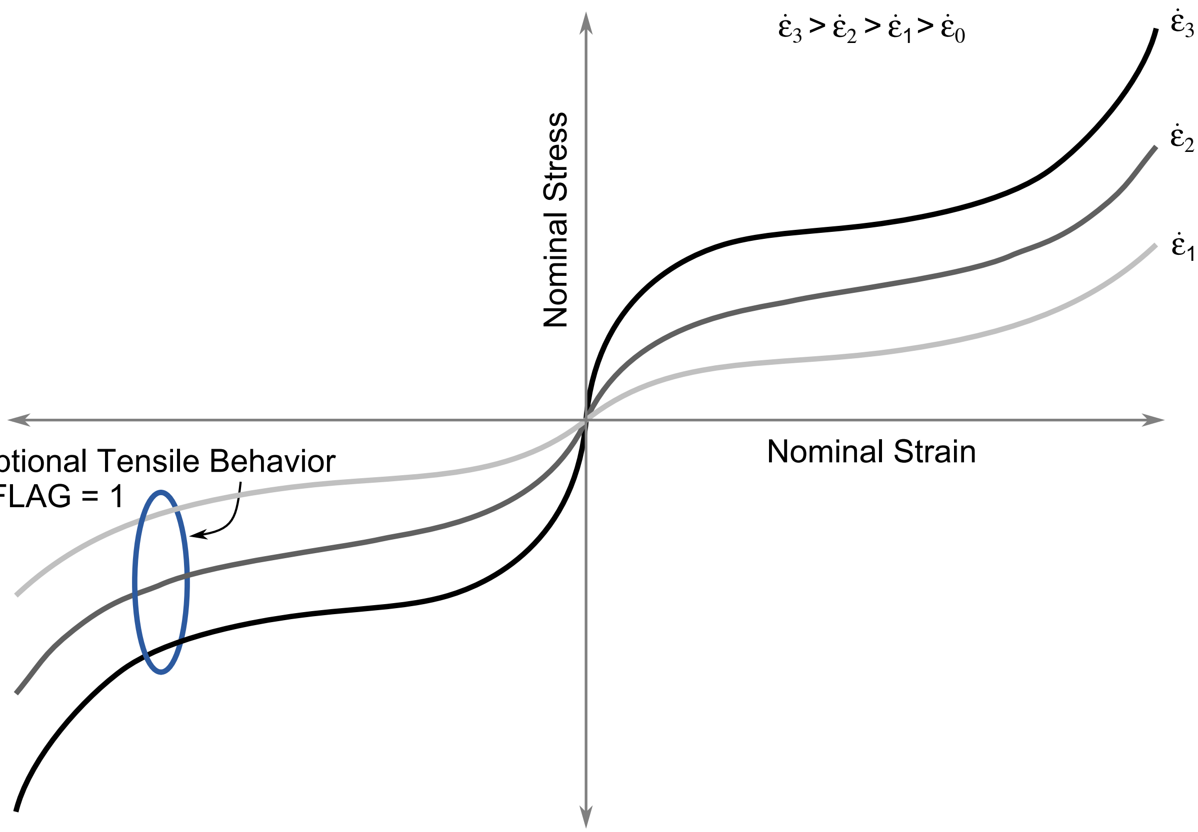


Figure M83-1. Rate effects in the nominal stress versus engineering strain curves, which are used to model rate effects in Fu Chang's foam model.

The unloading response was generalized by Kolling, Hirth, Erhart and Du Bois [2006] to allow the Mullin's effect to be modeled, meaning after the first loading and unloading, further reloading occurs on the unloading curve. If it is desired to reload on the loading curves with the new generalized unloading, the DAMAGE_DECAY option is available which allows the reloading to quickly return to the loading curve as the damage parameter decays back to zero in tension and compression.

Keyword option PATH_DEPENDENT invokes an alternative formulation of the Fu Chang model. In contrast to the original approach (total update of Cauchy stresses), it uses an incremental update of the second Piola-Kirchoff stresses with respect to the Green-Lagrange strains. If this keyword option is used, TBID should be defined as a 3D table for nominal stress, giving nominal stress as a function of volumetric change $1 - J$ (TABLE_3D), strain rate (TABLE), and nominal strain (CURVE). This formulation enables modeling a path-dependent response as sometimes observed in compressive loading followed by shear deformation or vice versa, for instance. Also, with this keyword option FMATRX = 2 should be set on *CONTROL_SOLID.

Card Summary:

Card 1. This card is required.

MID	RO	E	KCON	TC	FAIL	DAMP	TBID

Card 2. This card is required.

BVFLAG	SFLAG	RFLAG	TFLAG	PVID	SRAF	REF	HU

Card 3a. This card is included if the DAMAGE_DECAY keyword option is used.

MINR	MAXR	SHAPE	BETAT	BETAC			

Card 3b. This card is included if the DAMAGE_DECAY keyword option is *not* used.

D0	N0	N1	N2	N3	C0	C1	C2

Card 4. This card is included if the DAMAGE_DECAY keyword option is *not* used.

C3	C4	C5	AIJ	SIJ	MINR	MAXR	SHAPE

Card 5. This card is optional.

EXPON	RIULD						

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	E	KCON	TC	FAIL	DAMP	TBID
Type	A	F	F	F	F	F	F	F
Default	none	none	none	none	10^{20}	none	0.05	none

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
E	Young's modulus
KCON	Optional Young's modulus used to compute the sound speed. This will influence the time step, contact forces, hourglass stabilization forces, and the numerical damping (DAMP). If TBID ≠ 0, the time step is based on a stiffness of $\max(\text{KCON}, \text{E}, \text{ max. current slope of the stress-strain curve})$ The "max .current slope" is taken from the three principal directions. The tensile (negative) portion of the stress-strain curves is included in this evaluation if TFLAG = 1.
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. EQ.2.0: The element is eroded.
DAMP	Viscous coefficient to model damping effects (0.05 < recommended value < 0.50; default is 0.05)
TBID	Table ID (see *DEFINE_TABLE) for nominal stress as a function of strain data at a given strain rate. If the table ID is provided, Cards 3 and 4 may be left blank and the input curves will be used

VARIABLE	DESCRIPTION
	directly in the model. The Table ID can be positive or negative (see Remark 6 below). If TBID < 0, enter |TBID| on the *DEFINE_-TABLE keyword. For keyword option PATH_DEPENDENT, TBID defines a 3D table for nominal stress, giving it as a function of volumetric change $1-J$ (TABLE_3D), strain rate (TABLE), and nominal strain (CURVE).

Card 2	1	2	3	4	5	6	7	8
Variable	BVFLAG	SFLAG	RFLAG	TFLAG	PVID	SRAF	REF	HU
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
BVFLAG	Toggle to turn bulk viscosity off or on (see Remark 1): LT.1.0: No bulk viscosity (recommended) GE.1.0: Bulk viscosity active.
SFLAG	Strain rate flag (see Remark 2 below): EQ.0.0: True constant strain rate EQ.1.0: Engineering strain rate
RFLAG	Strain rate evaluation flag (see Remark 3): 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 (follows E) 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 as a function of volumetric strain. See Remark 4.
SRAF	Strain rate averaging flag (see Remark 5): LT.0.0: Use exponential moving average. EQ.0.0: Use weighted running average. GT.0.0.AND.LE.0.9999: Filter window for averaging strain rates, suppressing the time step dependence of the operation. EQ.1.0: Average the last twelve values. GE.1.0001: SRAF − 1.0 is a filter window for averaging strain rates, suppressing the time step dependence of the operation.
REF	Use reference geometry to initialize the stress tensor. The reference geometry is defined by the keyword: *INITIAL_FOAM_REFERENCE_GEOMETRY. EQ.0.0: Off EQ.1.0: On
HU	Hysteretic unloading factor between 0.0 and 1.0. See Remark 6 and Figure M83-4.

DAMAGE_DECAY Card. Card 3 for DAMAGE_DECAY keyword option.

Card 3a	1	2	3	4	5	6	7	8
Variable	MINR	MAXR	SHAPE	BETAT	BETAC			
Type	F	F	F	F	F			
Default	0.0	0.0	0.0	0.0	0.0			

VARIABLE	DESCRIPTION
MINR	Minimum strain rate of interest
MAXR	Maximum strain rate of interest

VARIABLE	DESCRIPTION
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor HU. Values less than one reduce energy dissipation and greater than one increase dissipation; see Figure M83-4.
BETAT	Decay constant for damage in tension. The damage decays after loading ceases according to $e^{-\mathrm{BETAT}\times t}$.
BETAC	Decay constant for damage in compression. The damage decays after loading ceases according to $e^{-\mathrm{BETAC}\times t}$.

Material Constants Card. Card 3 for keyword option *NOT* set to DAMAGE_DECAY.

Card 3b	1	2	3	4	5	6	7	8
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

VARIABLE	DESCRIPTION
D0	Material constant; see Material Formulation.
N0	Material constant; see Material Formulation.
N1	Material constant; see Material Formulation.
N2	Material constant; see Material Formulation.
N3	Material constant; see Material Formulation.
C0	Material constant; see Material Formulation.
C1	Material constant; see Material Formulation.
C2	Material constant; see Material Formulation.
C3	Material constant; see Material Formulation.
C4	Material constant; see Material Formulation.

VARIABLE	DESCRIPTION
C5	Material constant; see Material Formulation.
AIJ	Material constant; see Material Formulation.
SIJ	Material constant; see Material Formulation.

Material Constants Card. Card 4 for keyword option *NOT* set to DAMAGE_DECAY.

Card 4	1	2	3	4	5	6	7	8
Variable	C3	C4	C5	AIJ	SIJ	MINR	MAXR	SHAPE
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
C3	Material constant; see Material Formulation.
C4	Material constant; see Material Formulation.
C5	Material constant; see Material Formulation.
AIJ	Material constant; see Material Formulation.
SIJ	Material constant; see Material Formulation.
MINR	Minimum strain rate of interest
MAXR	Maximum strain rate of interest
SHAPE	Shape factor for unloading. Active for nonzero values of the hysteretic unloading factor HU. Values less than one reduces the energy dissipation and greater than one increases dissipation; see Figure M83-2.

Unloading Card. This card is optional.

Card 5	1	2	3	4	5	6	7	8
Variable	EXPON	RIULD						
Type	F	F						
Default	1.0	0.0						

VARIABLE	DESCRIPTION
EXPON	Exponent for unloading. Active for nonzero values of the hysteretic unloading factor HU. Default is 1.0
RIULD	Flag for rate-independent unloading, see Remark 6. EQ.0.0: Off EQ.1.0: On

Material Formulation:

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

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

and the strain rate becomes

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

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

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

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

$$\mathbf{E}_t^N(\tau) = \mathbf{E}^N(t-\tau) \ ,$$

where τ is the history parameter:

$$\mathbf{E}_t^N(\tau = \infty) \Leftrightarrow \text{the virgin material} \ .$$

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

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

Hence, the postulated constitutive equation becomes:

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

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

For a special case,

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

we may write

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

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

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

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

$$\begin{aligned}
\dot{S}_{ij} &= \left[c_1\left(a_{ij}R - c_2 S_{ij}\right)P + c_3 W^{n_1}\left(\|\dot{\mathbf{E}}^N\|\right)^{n_2} I_{ij}\right]R \\
R &= 1 + c_4\left[\frac{\|\dot{\mathbf{E}}^N\|}{c_5} - 1\right]^{n_3} \\
P &= \boldsymbol{\sigma}:\dot{\mathbf{E}}^N \\
W &= \int \boldsymbol{\sigma}:(d\mathbf{E})
\end{aligned}$$

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

$$\begin{aligned}
\|\boldsymbol{\sigma}\| &= \left(\sigma_{ij}\sigma_{ij}\right)^{\frac{1}{2}} \\
\|\dot{\mathbf{E}}\| &= \left(\dot{E}_{ij}\dot{E}_{ij}\right)^{\frac{1}{2}} \\
\|\dot{\mathbf{E}}^N\| &= \left(\dot{E}^N{}_{ij}\dot{E}^N{}_{ij}\right)^{\frac{1}{2}}
\end{aligned}$$

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.

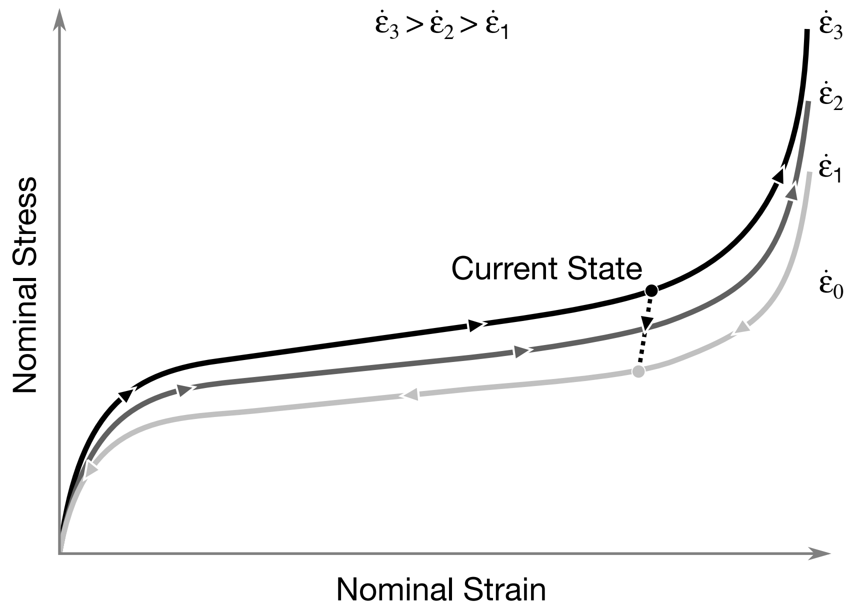


Figure M83-2. HU = 0, TBID > 0

Additional Remarks:

1. **Bulk viscosity.** The bulk viscosity, which generates a rate-dependent pressure, may cause an unexpected volumetric response. Consequently, it is optional with this model.

2. **Constant velocity loading.** 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. See the SFLAG field.

3. **Strain rates with multiaxial loading**. 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. See the RFLAG field.

4. **Triaxial loading.** 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. See the PVID field.

5. **Strain rate averaging.** Four different options are available. The default, SRAF = 0.0, uses a weighted running average with a weight of 1/12 on the current strain rate. With the second option, SRAF = 1.0, the last twelve strain rates are averaged. The third option, SRAF < 0, uses an exponential moving average

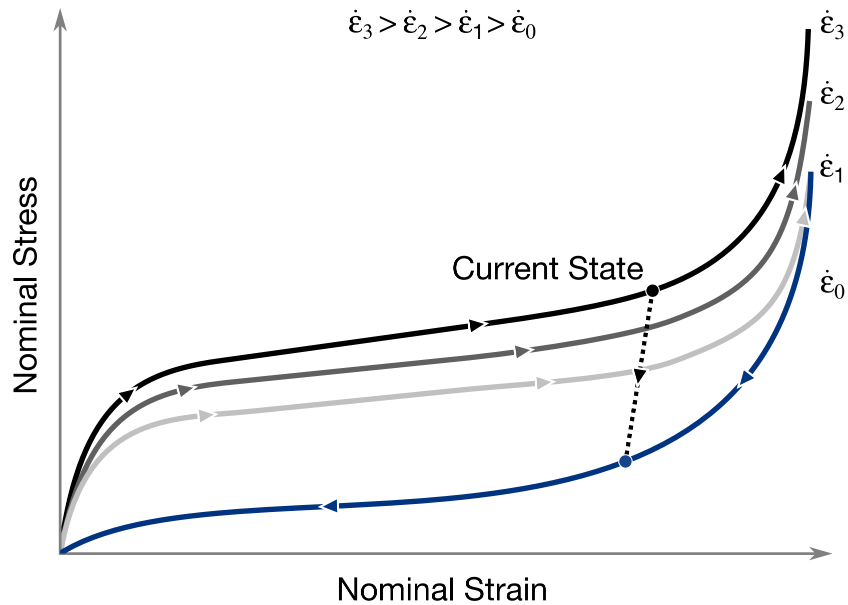


Figure M83-3. HU = 0, TBID < 0

with factor |SRAF| representing the degree of weighting decrease ($-1 \leq \text{SRAF} < 0$). The averaged strain rate at time t_n is obtained by:

$$\dot{\varepsilon}_n^{\text{averaged}} = |\text{SRAF}|\dot{\varepsilon}_n + (1 - |\text{SRAF}|)\dot{\varepsilon}_{n-1}^{\text{averaged}}$$

To suppress time step dependence, you can select a filter window for averaging strain rates. Depending on units and the wanted filter size, you can input either $0.0 < \text{SRAF} \leq 0.999$ for which SRAF becomes the filter size or $\text{SRAF} \geq 1.0001$ for which $\text{SRAF} - 1.0$ becomes the filter size. This rather awkward way of inputting the filter size is for allowing *any* filter size to accurate precision, including a size of 1.0.

6. **Unloading response options.** Several options are available to control the unloading response for MAT_083:

   a) *HU = 0 and TBID > 0.* See Figure M83-2.

      This is the old way. In this case, the unloading response will follow the curve with the lowest strain rate and is rate-independent. The curve with the lowest strain rate value (typically zero) in TBID should correspond to the unloading path of the material as measured in a quasistatic test. The quasistatic loading path then corresponds to a realistic (small) value of the strain rate.

   b) *HU = 0 and TBID < 0.* See Figure M83-3.

      In this case, the curve with the lowest strain rate value (typically zero) in TBID must correspond to the unloading path of the material as measured in a quasistatic test. The quasistatic loading path then corresponds to a

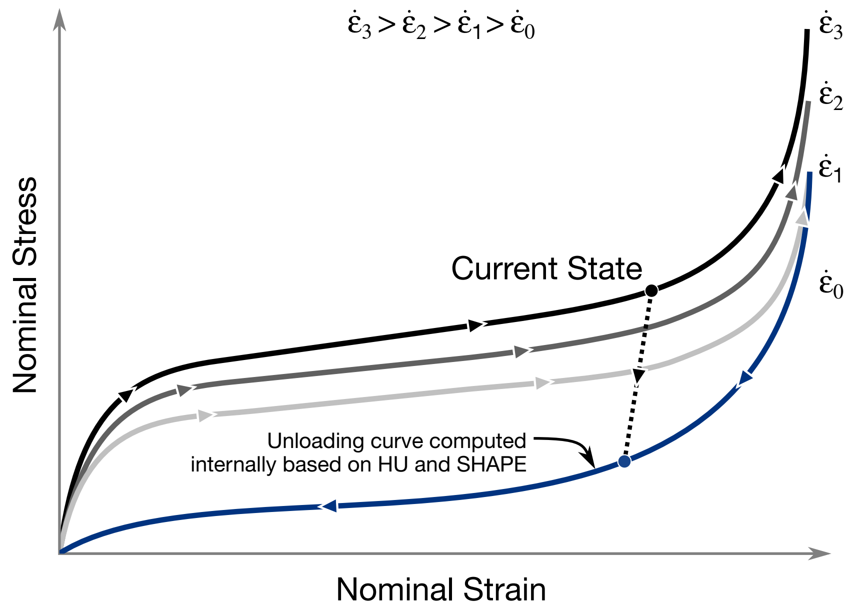


Figure M83-4. HU > 0, TBID > 0

realistic (small) value of the strain rate. At least three curves should be used in the table: one for unloading, one for quasistatic, and one or more for dynamic response. The quasistatic loading and unloading path (thus the first two curves of the table) should form a closed loop. The unloading response is given by a damage formulation for the principal stresses as follows:

$$\sigma_i = (1 - d)\sigma_i$$

The damage parameter, d, is computed internally in such a way that the unloading path under uniaxial tension and compression is fitted exactly in the simulation. The unloading response is rate dependent in this case. In some cases, this rate dependence for loading *and* unloading can lead to noisy results. To reduce that noise, it is possible to switch to rate independent unloading with RIULD = 1.

The internal computation of d using the first two curves of the table only works well if they are both nicely shaped and smooth, and no extreme final slope is present under compression, which is often hard to fulfill. Therefore, it is preferable to use the next option, HU > 0 with TBID > 0, instead.

c) *HU > 0 and TBID > 0.* See Figure M83-4.

No unloading curve should be provided in the table and the curve with the lowest strain rate value in TBID should correspond to the loading path of the material as measured in a quasistatic test. At least two curves should be used in the table: one for quasistatic and one or more for dynamic response. In this case the unloading response is given by a damage formulation for the principal stresses as follows:

$$\sigma_i = (1-d)\sigma_i$$

$$d = (1-HU)\left[1-\left(\frac{W_{\text{cur}}}{W_{\max}}\right)^{\text{SHAPE}}\right]^{\text{EXPON}}$$

where W_{cur} corresponds to the current value of the hyperelastic energy per unit undeformed volume. The unloading response is rate dependent in this case. In some cases, this rate dependence for loading *and* unloading can lead to noisy results. To reduce that noise, it is possible to switch to rate-independent unloading with RIULD = 1.

The LOG_LOG_INTERPOLATION option uses log-log interpolation for table TBID in the strain rate direction.

*MAT_WINFRITH_CONCRETE

This is Material Type 84 with optional rate effects. The Winfrith concrete model is a smeared crack (sometimes known as pseudo crack), smeared rebar model. We implemented this model for the 8-node single integration point solid element (ELFORM = 1 on *SECTION_SOLID) and the 4-node single integration point tetrahedral element (ELFORM = 10). We recommend using a double precision executable for simulations that include this material model. Single precision may produce unstable results.

Broadhouse and Neilson [1987] and Broadhouse [1995] developed this model over many years, and experiments have validated it. Much of the input documentation given here comes directly from the report by Broadhouse. In releases R15 onwards, further developments by Arup are available by setting the input parameter RATE to 8; see Cards 5 through 7.

Rebar may be defined using the keyword *MAT_WINFRITH_CONCRETE_REINFORCEMENT, which appears in the following section, or may be modeled with beam elements fixed to the concrete using *CONSTRAINED_BEAM_IN_SOLID.

Card Summary:

Card 1. This card is required.

MID	RO	TM	PR	UCS	UTS	FE	ASIZE

Card 2. This card is required.

E	YS	EH	UELONG	RATE	CONM	CONL	CONT

Card 3. This card is required but may be left blank.

EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8

Card 4. This card is required but may be left blank.

P1	P2	P3	P4	P5	P6	P7	P8

Card 5. Include this card when RATE = 8.

MAXSHR	LCYMT	LCFTT	LCFCT			LCTST	LCCMP

Card 6. Include this card when RATE = 8. It may be left blank.

CRFAC	COD1	TENPWR	TENRSD	LCFIB	RO_G	ZSURF	LCFTIM

Card 7. Include this card when RATE = 8. It may be left blank.

OTTO	DILATD	DILRAT	DEGRAD	TFAC8	TLOSSC	CDSF	

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	MID	RO	TM	PR	UCS	UTS	FE	ASIZE
Type	A	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
MID	Material identification. A unique number or label must be specified (see *PART).
RO	Mass density
TM	Initial tangent (Young's) modulus of concrete, E_s. See Remarks 2 and 3.
PR	Poisson's ratio, ν. See Remarks 2 and 3.
UCS	Uniaxial compressive strength (see Remarks 2 and 3)
UTS	Uniaxial tensile strength (see Remarks 2, 6 and 11)
FE	The meaning of FE depends on the value of RATE (see Remark 8): RATE.EQ.0: Fracture energy (energy per unit area dissipated in opening the crack). RATE.GT.0: Crack width at which the crack-normal tensile stress goes to zero.
ASIZE	Aggregate size, depending on the value of RATE. RATE.LE.1: Aggregate radius in model length units. RATE.GE.2: Aggregate diameter in meters. The formula for shear stress carried across cracks with aggregate interlock uses this field; see Remark 11.

Card 2	1	2	3	4	5	6	7	8
Variable	E	YS	EH	UELONG	RATE	CONM	CONL	CONT
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
E	Young's modulus of rebar
YS	Yield stress of rebar
EH	Hardening modulus of rebar
UELONG	Ultimate elongation before rebar fails.
RATE	Material model option (see Remarks 8 and 13): EQ.0.0: Original Broadhouse implementation with strain rate effects included. WARNING: This option does not guarantee energy conservation. EQ.1.0: Original Broadhouse implementation with strain rate effects turned off. EQ.2.0: Like RATE = 1 but includes an improved crack algorithm. It is superseded by RATE = 8. EQ.8.0: Improved crack algorithm plus additional inputs on Cards 5 through 7 (recommended).
CONM	Units (conversion) flag: GT.0.0: Factor to convert model mass units to kg EQ.-1.0: Mass, length, and time units in the model are lbf × sec2/in, inch, and sec. EQ.-2.0: Mass, length, and time units in the model are g, cm, and microsec. EQ.-3.0: Mass, length, and time units in the model are g, mm, and msec. EQ.-4.0: Mass, length, and time units in the model are metric ton, mm, and sec. EQ.-5.0: Mass, length, and time units in the model are kg, mm, and msec.

VARIABLE	DESCRIPTION
CONL	Length units conversion factor: CONM.GT.0: CONL is the conversion factor from model length units to meters (for instance, CONL = 0.001 for millimeters). CONM.LE.0: CONL is ignored.
CONT	Time units conversion factor: CONM.GT.0: CONT is the conversion factor from time units to seconds (for example, CONT = 0.001 for milliseconds). CONM.LE.0: CONT is ignored.

Card 3	1	2	3	4	5	6	7	8
Variable	EPS1	EPS2	EPS3	EPS4	EPS5	EPS6	EPS7	EPS8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EPS1, EPS2, …	Volumetric strain values (natural logarithmic values); see Remark 3. If this card is not left blank, a minimum of 2 values must be defined and a maximum of 8 values are allowed.

Card 4	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
P1, P2, …	Pressures corresponding to the volumetric strain values given on Card 3. See Remark 3.

Additional card for RATE = 8.

Card 5	1	2	3	4	5	6	7	8
Variable	MAXSHR	LCYMT	LCFTT	LCFCT			LCTST	LCCMP
Type	F	I	I	I			I	I
Default	↓	none	none	none			none	none

VARIABLE	DESCRIPTION
MAXSHR	Maximum shear stress that can be carried across a crack under conditions of zero normal stress on the crack and zero crack opening displacement. The default value is 1.161 times UTS;` see Remark 11.
LCYMT	Optional load curve ID governing the variation of elastic stiffness with temperature. The x-axis is temperature, and the y-axis is a nondimensional factor on elastic modulus TM.
LCFTT	Optional load curve ID governing the variation of tensile strength with temperature. The x-axis is temperature, and the y-axis is a nondimensional factor on tensile strength UTS. See Remark 9.
LCFCT	Optional load curve ID governing the variation of compressive strength with temperature. The x-axis is temperature, and the y-axis is a nondimensional factor on compressive strength UCS.
LCTST	Optional load curve ID governing the post-cracking tensile response. See Remark 8. The x-axis is crack-opening displacement (length units), and the y-axis is a nondimensional factor on tensile strength UTS. If LCTST is defined, it overrides FE on Card 1. The first point should be (0, 1).
LCCMP	Optional load curve ID governing post-yield compression/shear response. The x-axis is plastic strain,and the y-axis is a nondimensional factor that scales UCS. See Remark 14.

Additional card for RATE = 8.

Card 6	1	2	3	4	5	6	7	8
Variable	CRFAC	COD1	TENPWR	TENRSD	LCFIB	RO_G	ZSURF	LCFTIM
Type	F	F	F	F	I	F	F	I
Default	0.0	0.0	1.0	0.01	none	0.0	0.0	none

VARIABLE	DESCRIPTION
CRFAC	Scale the tensile strength of uncracked elements by (1 – CRFAC) if an adjacent element has cracked. See Remark 9.
COD1	Crack opening displacement (length units) of the adjacent element at which the full value of CRFAC applies. For crack opening displacements smaller than COD1, linear interpolation is applied. See Remark 9.
TENPWR	Power law term governing tensile strength when at least one principal stress is compressive. See Remark 9.
TENRSD	Residual factor term governing tensile strength when at least one principal stress is compressive. See Remark 9.
LCFIB	Optional load curve ID for fiber-reinforced concrete. The x-axis of the curve is crack-opening displacement (length units), and the y-axis is additional tensile stress due to the presence of the fibers. See Remark 10.
RO_G	A nonzero RO_G invokes the option for water pressure to be applied within cracks. The value of RO_G is water density times acceleration due to gravity. See Remark 22.
ZSURF	Global z-coordinate of the water surface, used for calculating water pressure in cracks. See Remark 22.
LCFTIM	Optional load curve ID giving scaling factor on tensile strength (UTS) as a function of time. See Remarks 9 and 21.

Additional card for RATE = 8.

Card 7	1	2	3	4	5	6	7	8
Variable	OTTO	DILATD	DILRAT	DEGRAD	TFAC8	TLOSSC	CDSF	
Type	I	F	F	F	F	F	F	
Default	1	0.0	0.0	0.0	0.9	1.0	8.0	

VARIABLE	DESCRIPTION
OTTO	Option for automatic calculation of the Ottosen yield surface constants (see Remark 13): EQ.1: fib Model Code 2010, normal weight concrete EQ.2: fib Model Code 2010, lightweight concrete EQ.3: Same as RATE = 0, 1 or 2 (Broadhouse model)
DILATD	Maximum dilation displacement (in model length units) due to crack sliding or yielding
DILRAT	Initial dilation ratio
DEGRAD	Lower limit on the factor by which the compressive strength of cracked elements is scaled (see Remark 15): EQ.0: No reduction of compressive strength GT.0: Equation from Eurocode 2 with lower limit = DEGRAD
TFAC8	Nondimensional modification factor applied to any tensile principal stresses when calculating the Ottosen yield function; see Remark 16.
TLOSSC	Nondimensional parameter controlling loss of tensile strength in crushed elements; see Remark 9.
CDSF	Nondimensional ductility factor for confined concrete. CDSF controls scaling of the x-axis of LCCMP; see Remark 14.

Remarks:

1. **Minimum input recommendations.** All of the input parameters on Card 1 should be defined, together with RATE and the unit conversion factors CONM, CONL, and CONT on Card 2 (CONL and CONT may be left blank if CONM is negative). If yielding or failure in compression or shear is anticipated, we recommend RATE = 8. For RATE = 8, we recommend defining LCCMP. All the other input parameters on Cards 5, 6, and 7 may be left blank because the defaults are intended to provide a reasonably realistic response.

2. **Basic properties.** The elastic properties are defined by Young's modulus TM and Poisson's Ratio PR. UCS is the compressive strength under uniaxial stress conditions as measured by a standard cylinder compression test. Note that the strength obtained from standard cube tests is typically 15-25% greater than the uniaxial compressive strength. UTS, the tensile strength, may be estimated from tables or equations in codes and standards such as Eurocode 2 or ACI-318. It is important that a realistic tensile strength (as opposed to an artificially low "design" value) is input, for reasons explained in Remark 13.

3. **Volumetric response.** Cards 3 and 4 enable providing the volumetric response curve. In this curve, pressure is positive in compression. Volumetric strain is defined as the natural log of the relative volume and is negative in compression. The tabulated data must be provided in order of increasing compression, with no initial zero point.

   If omitting the volume compaction curve, i.e., if Cards 3 and 4 are left blank, LS-DYNA uses the scaled curve in Table M84-1. p_1 in the curve is the pressure at uniaxial compressive failure:

$$p_1 = \frac{\text{UCS}}{3} ,$$

   K (referenced in the Table below) is the bulk unloading modulus computed from:

$$K = \frac{E_s}{3(1 - 2v)} .$$

   Here E_s is the input tangent modulus for concrete (input parameter TM), and v is Poisson's ratio.

Volumetric Strain	Pressure
$-p_1/K$	$1.00p_1$
-0.002	$1.50p_1$
-0.004	$3.00p_1$
-0.010	$4.80p_1$

−0.020	$6.00p_1$
−0.030	$7.50p_1$
−0.041	$9.45p_1$
−0.051	$11.55p_1$
−0.062	$14.25p_1$
−0.094	$25.05p_1$

Table M84-1. Default pressure as a function of volumetric strain curve for concrete if the curve is not defined.

4. **Binary crack output database.** The Winfrith concrete model can generate an additional binary output database containing information on crack locations, directions, and widths. Generating the crack database requires modifyinh the LS-DYNA execution line by adding the following:

   q=**crf**

   where **crf** is the desired name of the crack database, such as q = **d3crack**. LS-DYNA writes the output at the same times as the d3plot database.

   LS-PrePost can display the cracks on the deformed mesh plots. To do so, read the d3plot database into LS-PrePost and select *File* → *Open* → *Crack* from the top menu bar. Or, open the crack database by adding the following to the LS-PrePost execution line:

   q=**crf**

   where **crf** is the name of the crack database.

   By default, LS-PrePost shows all the cracks in visible elements. Setting a minimum crack width for displayed cracks eliminates narrow cracks from the display. To do this, choose *Settings* → *Post Settings* → *Concrete Crack Width*. From the top menu bar of LS-PrePost, choosing *Misc* → *Model Info* reveals the number of cracked elements and the maximum crack width in a given plot state.

5. **Crack summary output file.** Including *DATABASE_BINARY_D3CRACK in the input deck causes LS-DYNA to write an ASCII output file named aea_crack. This command does not have any bearing on the aforementioned binary crack database.

6. **Crack plane directions.** The crack algorithm uses a non-rotating approach. Once cracks have initiated, their directions remain fixed relative to the element's local axis system. Up to three mutually perpendicular cracks can form in each element. The first crack is initiated on a plane normal to the maximum tensile principal stress when that principal stress reaches the tensile strength (input parameter UTS; see Remark 9 for RATE = 8). A second crack can initiate on any

plane normal to the first crack and does so if the tensile stress acting perpendicular to that plane reaches the tensile strength. After creating two cracks, the only possible plane for the third crack is the one mutually perpendicular to the first two cracks. The third crack initiates if the tensile stress acting perpendicular to that plane reaches the tensile strength.

7. **Limitation of non-rotating cracks.** The algorithm prevents the tensile stress from exceeding UTS only in directions normal to actual or potential crack planes. It is possible to observe tensile principal stresses greater than UTS in the results if the loading directions rotate after a crack has formed. This result is a limitation of the non-rotating crack approach.

8. **Crack tensile response.** We model cracks with the "smeared crack" approach, meaning that the stress-strain relationships model the presence of a crack instead of the mesh breaking apart. To reduce the sensitivity of results to mesh size, these relationships are formulated using displacement instead of strain as the abscissa. Displacement, δ, is calculated from strain, ε, and the initial element volume, V_0, as follows:

$$\delta = L_0 \varepsilon$$
$$L_0 = V_0^{1/3}$$

After the initiation of a crack, the tensile stress decays with increasing crack opening displacement. For RATE = 1 and 2, the decay follows a linear relationship with the tensile strength reaching zero at a crack opening displacement equal to the input parameter FE. For RATE = 0, the decay follows a bilinear relationship. LS-DYNA automatically scales the displacement axis of this bilinear relationship based on the input parameter FE which represents the fracture energy.

For RATE = 8 the default is to use FE in the same way as RATE = 1 and 2, but this may optionally be overridden using the load curve LCTST. If defined, the first point of LCTST should be (0, 1), i.e., at zero crack opening displacement, the uniaxial tensile strength is equal to unity times UTS. It is expected, but not essential, that the y-axis values drop to zero at some finite x-axis value, meaning the tensile strength drops to zero at a finite crack opening displacement. See also Remark 10 regarding fiber-reinforced concrete.

9. **Modifications to tensile strength (RATE = 8).** When RATE = 8, the tensile strength of a given element may be different from UTS for the following reasons:

   a) If compressive principal stresses are also present, the tensile strength is scaled by a factor k_{TC} defined as follows:

$$k_{TC} = \text{TENRSD} + (1 - \text{TENRSD})\left(1 - f^{\text{TENPWR}}\right)$$
$$f = {}^{-\sigma_{\text{comp}}}/_{\text{UCS}} \text{ subject to } 0 \le f \le 1$$

Here, σ_{comp} is the most compressive principal stress, and TENRSD and TENPWR are defined on Card 6. The default values provide a linear reduction of tensile strength from UTS to 0.01 × UTS as the most compressive principal stress increases from zero to UCS.

b) If any neighboring elements are cracked (where "neighboring" means elements that share at least one node with the uncracked element being considered), the tensile strength is scaled by a crack propagation factor k_{CP} intended to represent the effect of stress concentrations near a crack tip:

$$k_{CP} = 1 - \text{CRFAC}\left(\min(1.0, \frac{\delta_{\text{crack,max}}}{\text{COD1}})\right)$$

Here, $\delta_{\text{crack,max}}$ is the greatest crack opening displacement in any neighboring element, and CRFAC and COD1 are input parameters on Card 6.

c) If the element has yielded in compression (see Remark 13), it is assumed that the damage to bonds within the material caused by crushing rapidly eliminates the tensile strength. To represent this effect, the tensile strength is scaled by a factor calculated as follows:

$$k_Y = \max\left[0.0, {}^{L_0\varepsilon_p}\big/_{(\text{TLOSSC} \times \delta_{\sigma=0})}\right]$$

In the above equation, L_0 is the initial element size as defined in Remark 8, ε_p is the plastic strain associated with yielding on the Ottosen yield surface, TLOSSC is a nondimensional input parameter defined on Card 7, and $\delta_{\sigma=0}$ is the crack opening displacement at which the tensile stress reduces to zero. By default, $\delta_{\sigma=0}$ is equal to FE, but if LCTST is defined, it is the x-axis value at which the y-axis value falls to zero.

d) If LCFTIM is defined (see Remark 21), the current y-axis value of LCFTIM is a scaling factor k_t. Otherwise, $k_t = 1$.

e) If LCFTT is defined (see Card 5), a temperature-dependent scaling factor k_T is applied. Otherwise, $k_T = 1$.

f) The initial tensile strength of an element, f_t, is calculated from UTS and the above factors as follows:

$$f_t = k_{TC}k_{CP}k_Yk_tk_T \times \text{UTS}$$

10. **Fiber-reinforced concrete.** Steel fibers increase the ductility of concrete because they resist the opening of cracks. In order for cracks to widen, the fibers that span across the crack must be pulled out or stretched. This effect may be modeled using the load curve LCFIB on Card 6. The x-axis is crack opening displacement in length units. The y-axis is additional tensile stress resisting further opening of the crack. This additional tensile strength is not subject to the reduction factors described in Remark 9, which are appropriate only for the concrete

itself and not for the effect of the fibers. For this reason, we recommend using LCFIB instead of combining the influence of the fibers with the tensile response of the concrete into the curve LCTST. We recommend that the first point of LCFIB should be (0, 0) so that the fibers do not influence the overall initial tensile strength.

11. **Shear transfer across cracks (RATE = 2 and 8).** When RATE = 2 and 8, the following equations model the aggregate-interlock that allows cracked concrete to carry shear loading. The equations were proposed by Vecchio & Collins (1986) and subsequently adopted into Norwegian standard NS3473. The maximum shear stress, $\tau_{\max}$, that the crack plane can carry depends on the compressive stress on the crack, σ_c, for a closed crack or on the crack opening width, w, for an open crack:

$$\tau_{\max} = 0.18\tau_{\rm rm} + 1.64\sigma_c - 0.82\frac{\sigma_c^2}{\tau_{\rm rm}}$$

$$\tau_{\rm rm} = \frac{2f_{t,sh}}{0.31 + \dfrac{0.024w}{(\text{ASIZE} + 0.016)}}$$

ASIZE is the aggregate diameter in meters defined on Card 1. For this purpose, CONL is ignored, and the input value of ASIZE should be in meters even if the model units are not meters.

In the above equations, $f_{t,sh}$ is a tensile strength that depends on RATE and, if RATE = 8, on MAXSHR on Card 5. If RATE = 8 and MAXSHR = 0.0 (recommended), $f_{t,sh}$ is equal to the tensile strength (i.e., UTS defined on Card 1) in accordance with Vecchio & Collins. The above equations then give $\tau_{\max} =$ 1.16UTS when σ_c and w are both zero. For RATE = 8, if MAXSHR is nonzero, $f_{t,sh}$ is automatically set such that $\tau_{\max} =$ MAXSHR when σ_c and w are both zero.

If RATE = 2, MAXSHR is unavailable. In this case, $f_{t,sh} = \max(\text{UTS}, 0.086\text{UCS})$, giving $\tau_{\max} = \max(1.16\text{UTS}, 0.1\text{UCS})$ when σ_c and w are both zero. The 0.1UCS does not comply with the recommendations of Vecchio & Collins; this is one of the reasons why RATE = 2 is not recommended.

12. **Compression response: general comments.** The compression response of concrete varies according to the stress state. Under uniaxial conditions, such as occur in a cylinder test, concrete exhibits a brittle response, failing rapidly after reaching its compressive strength. Under confined conditions, which occur in reinforced concrete structures when the reinforcement cage resists expansion of the concrete in the directions perpendicular to the main compressive load, the stress state in the concrete consists of one large compressive stress in the loading direction and, typically, two smaller compressive stresses ("confining stresses") in the perpendicular directions. The influence of the confining stresses is two-fold: firstly, the compressive strength in the loading direction is increased from

the uniaxial value to an enhanced value (denoted here as σ_c and σ_{cc}, respectively); and secondly, the compressive response becomes more ductile, meaning that the rate of softening with strain is reduced and the strain to failure is increased.

13. **Compression response: yield surface.** The Ottosen yield surface governs yielding under compressive stress states. This surface captures the influence of confining stresses on the compressive strength for all settings of RATE. The following equation defines the Ottosen yield surface:

$$\alpha \frac{J_2}{\sigma_c^2} + \lambda \frac{\sqrt{J_2}}{\sigma_c} + \beta \frac{I_1}{\sigma_c} - 1 = 0 \ ,$$

where

$$\lambda = c_1 \cos\left[\frac{1}{3}\arccos(c_2 \cos(3\theta))\right] \ .$$

In the above I_1, J_2 and J_3 are the first, second and third stress invariants, σ_c is the uniaxial compressive strength (input parameter UCS for RATE = 0, 1 and 2, and see Remark 14 for RATE = 8), and θ is the Lode Angle. α, β, c_1, and c_2 are calibration constants that LS-DYNA internally calculates to fit the yield surface through four reference stress states which are chosen automatically.

Two of the reference stress states are uniaxial compression and uniaxial tension, defined by input parameters UCS and UTS. This calibration has a counterintuitive side effect whereby the input value of tensile strength (UTS) affects the whole yield surface, including stress states in which no tensile stresses are present. For this reason, it is important to use values for UTS and UCS that are in similar proportions to each other as for real concrete. This applies to all settings of RATE.

Two further reference stress states are needed for calibration. These differ according to the setting of RATE. RATE = 0 and 1 are as described by Broadhouse. RATE = 8 offers a choice via the input parameter OTTO, with the default being to adopt the recommendations of fib Model Code for Concrete Structures 2010 ("MC2010") Section 5.1.6. The choice of reference stress state influences the degree to which small confining stresses increase the compressive strength, with the MC2010 method giving a smaller increase than the Broadhouse method.

For RATE = 2, the yield surface is calibrated using the same method as RATE = 0 and 1 except that an enhanced tensile strength, namely max(1.25UTS, 0.1UCS), is used instead of the actual tensile strength UTS for purpose of calibrating the yield surface. Using an enhannced tensile strength was done in order to reduce the counter-intuitive influence of tensile strength on all-compressive stress states, but it does not accord with recommendations in the literature. For this reason, RATE = 8 is preferred over RATE = 2.

14. **Compression Response Post-Yield.** For RATE = 0, 1 and 2, the post-yield response is perfectly-plastic, which fails to capture the brittle response under uniaxial stress conditions and is not representative of real concrete. These settings of RATE are unsuitable for assessing failure under compressive stress states.

    For RATE = 8, the compressive stress-strain relation under uniaxial stress conditions is controlled by the loadcurve LCCMP, which should be calibrated by the user to obtain the desired brittle response:

    $$\sigma_c = \mathrm{LCCMP}(\varepsilon_{p,\mathrm{uniaxial}}) \times \mathrm{UCS}$$

    The uniaxial-equivalent plastic strain, $\varepsilon_{p,\mathrm{uniaxial}}$, is calculated from the equation below so as to provide increased ductility under confined conditions compared to uniaxial conditions by stretching the load curve LCCMP along the x-axis:

    $$\varepsilon_{p,\mathrm{uniaxial}} = \sum \left[\frac{d\varepsilon_p}{1 + \mathrm{CDSF}(\sigma_{cc}/\sigma_c - 1)} \right]$$

    Here, the actual plastic strain increments are denoted by $d\varepsilon_p$, CDSF is an input parameter on Card 7, and σ_{cc} is the confined compressive strength defined by most compressive principal stress at the point on the Ottosen yield surface corresponding to the current stress state.

15. **Influence of cracking on compressive strength.** By default, cracking of an element has no influence on its compressive strength. In practice, the compressive strength parallel to an open crack is reduced to some degree. This may be modelled using the optional input parameter DEGRAD which invokes the following equations based on Eurocode 2:

    $$\sigma_c = k_{cr}\sigma_{c,\mathrm{uncracked}}$$
    $$k_{cr} = {}^{1}\!/_{(0.8 + 170\varepsilon_{cr})}$$

    where $\varepsilon_{cr} = \max(\delta_{cr1}, \delta_{cr1}, \delta_{cr1}) / \mathrm{Vol}^{1/3}$ subject to DEGRAD $\le k_{cr} \le 1.0$. Here, $\delta_{cr1,2,3}$ are the high-tide crack opening displacements.

16. **Input parameter TFAC8.** For RATE = 8, yielding (governed by the Ottosen yield surface) is treated as a separate deformation mechanism from cracking. Since the Ottosen surface covers the whole stress space including where one or more principal stress is tensile, and since the yield surface is calibrated to pass through the point of uniaxial tensile failure at a stress equal to UTS, either or both deformation mechanisms could potentially occur under conditions where only cracking is expected. The parameter TFAC8 modifies the Ottosen yield surface such that the activation of cracking rather than yielding becomes unambiguous under these circumstances. It does this by calculating the yield surface from principal stresses where any tensile values are artificially scaled down by TFAC8. Thus, for compressive-only stress states, the yield surface calibration is

unaffected by TFAC8, while the expected cracking occurs under tensile stress states. The default value of 0.9 is recommended.

17. **Output history variables.** The meaning of "plastic strain" in output files differs depending on the setting of RATE. Use NEIPH on *DATABASE_EXTENT_BINARY to request extra history variables with NEIPH on *DATABASE_EXTENT_BINARY. The meanings of these also depend on RATE. The following tables give the meanings of a selection of these variables. In the tables, Crack 1, Crack 2, and Crack 3 refer to the first, second, and third cracks, respectively, to form in the element.

Plastic Strain or Extra History Variable	Description for RATE = 0
Plastic strain	Volumetric plastic strain, see compaction curve defined on Cards 3 and 4.
1	Maximum current value of the crack status flag across all three cracks; see Remark 18.
2	Energy absorbed by crack formation
3-5	High-tide value of the non-dimensional crack opening parameter for Cracks 1, 2, and 3. This value is capped at 5.16. See Remark 19.
36-38	Crack status flags for Cracks 1, 2, and 3. See Remark 18.
45-47	Current crack opening displacement for Cracks 1, 2, and 3. See Remark 19.
48-50	Initiation time for Cracks 1, 2, and 3

Plastic Strain or Extra History Variable	Description for RATE = 1
Plastic strain	Volumetric plastic strain, see compaction curve defined on Cards 3 and 4.
1	Maximum current value of the crack status flag across all three cracks. See Remark 18.
30-32	High-tide opening displacement for Cracks 1, 2, and 3, capped at a displacement equal to input parameter FE. See Remark 19.
36-38	Crack status flags for Cracks 1, 2, and 3. See Remark 18.
45-47	Current crack opening displacement for Cracks 1, 2, and 3. See Remark 19.
48-50	Initiation time for Cracks 1, 2, and 3

Plastic Strain or Extra History Variable	Description for RATE = 2 and 8
Plastic strain	(Starting from R15): "Damage deformation" in length units, see Remark 20.
1	Number of cracks that have formed (0, 1, 2, or 3).
2	Plastic strain due to yielding on Ottosen surface (RATE = 8 only)
3-5	High-tide opening displacement for Cracks 1, 2, and 3 (not capped). See Remark 19.
27	(Starting from R15): Volumetric plastic strain, see compaction curve defined on Cards 3 and 4.
36-38	Crack status flags for Cracks 1, 2, and 3. See Remark 18.
45-47	Current crack opening displacement for Cracks 1, 2, and 3. See Remark 19.

18. **Crack status flags.** The crack status flags referred to in the Extra History Variables have the following meanings:

    EQ.0: No crack has formed.

    EQ.1: The crack is opening and on the descending branch of the stress-displacement relationship, such that the tensile strength has not yet reached zero.

    EQ.2: The crack has partially closed (unloading/reloading branch).

    EQ.3: The crack has fully opened such that the tensile strength has reached zero.

19. **Extra history variables related to crack opening displacement.** The *current* crack opening displacement may be output for all settings of RATE in Extra History Variables 45 through 47. It can rise and fall during an analysis as cracks open and close due to changes in loading. These values are not capped, meaning that they reflect the total width of cracks within the element even if the crack opens further after the tensile strength has reached zero.

    The *high-tide* crack opening displacement is the maximum width that has occurred up to that point in the analysis. It differs from the current value in cases where the cracks open and then fully or partially close. This output parameter is available for RATE = 2 and 8. For RATE = 0 and RATE = 1, high-tide output parameters are available, but we cap them at the value where the tensile strength reaches zero. Although cracks can continue to open further with zero resistance, the capped output parameters do not reflect the additional opening. The capped output parameters are useful for assessing how much of the tensile capacity has been lost but not for assessing crack widths. For RATE = 1, the high-tide output is the crack displacement capped at a value equal to the input parameter FE. For RATE = 0, the high-tide output is in a non-dimensional form and is capped at a value of 5.16 which is the point at which the tensile strength reaches zero.

20. **Damage deformation.** Starting from R15, the parameter output in place of plastic strain for RATE = 2 and 8, δ_{dam}, is defined as follows:

$$\delta_{\text{dam}} = \varepsilon_p \text{Vol}_0^{1/3} + \delta_{\text{crack1}} + \delta_{\text{crack2}} + \delta_{\text{crack3}}$$

In this equation, $\delta_{\text{crack1,2,3}}$ are the high-tide crack opening displacements, ε_p is the plastic strain associated with yielding on the Ottosen surface and Vol_0 is the initial element volume.

21. **Low "design" values of tensile strength.** Users may wish to check that the performance of a structure is not reliant on the tensile strength of concrete, but this should *not* be done by setting an artificially low value for UTS. Doing so would distort the yield surface as explained in Remark 13 and may also cause cracks to form at random angles due to small tensile stresses occurring dynamically during application of the load. Instead, we recommend scaling down the tensile strength as a function of time, starting from a realistic value given by UTS which will be used to calibrate the yield surface, and then reducing to the desired low "design" value after loads have been applied. This may be achieved using the load curve LCFTIM. The ordinate of LCFTIM is a scaling factor applied to UTS. The abscissa is time. The first point of LCFTIM should be (0, 1).

22. **Water pressure in cracks.** When water seeps into cracks in underwater structures, the water pressure acts to push the crack surfaces apart, balancing the effect of pressure on the structure's outer surfaces which would tend to push the crack together. The input parameters RO_G and ZSURF model the effect of water in any cracks that form by applying an additional compressive stress normal to the crack, irrespective of whether or not there is a path for the water to reach the cracked element from the outer surface of the structure. To use this feature, the model should be oriented such that the global z-coordinate is vertically upwards. The additional compressive stress, σ_{water}, ramps up from zero to its full value as the crack opening displacement increases from zero to FE, as follows:

$$\sigma_{\text{water}} = \rho g (z_0 - z_{\text{el}}) \times \min(1.0, \delta_{\max}/\text{FE})$$

Here, ρg is the input parameter RO_G, z_0 is the input parameter ZSURF, z_{el} is the z-coordinate of the element center, $\delta_{\max}$ is the maximum opening displacement of the crack so far during the analysis, and FE is the input parameter on Card 1.

References:

[1] Ottosen N.S., “A failure criterion for concrete”. Journal of the Engineering Mechanics Division 103(4):527-35, 1977.

[2] Sturt, R., Montalbini, G. & Jung, H-I., “Developments in *MAT_WINFRITH_CONCRETE and Application to Modelling Tunnel Linings”, 14th European LS-DYNA Conference, 2023.

[3] Taerwe L, Matthys S., “Fib model code for concrete structures”, CEB-FIP, 2010.

[4] Vecchio, F.J. & Collins, M.P., “The Modified Compression-Field Theory for Reinforced Concrete Elements Subjected to Shear”, ACI J 83(2) (1986) pp219-231, 1986.

*MAT_WINFRITH_CONCRETE_REINFORCEMENT

This is *MAT_084_REINF for rebar reinforcement supplemental to concrete defined using Material type 84. Reinforcement may be defined in specific groups of elements, but it is usually more convenient to define a two-dimensional material in a specified layer of a specified part. 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.

Card Summary:

Card 1a. Reinforcement is defined in specific groups of elements.

EID1	EID2	INC	XR	YR	ZR		

Card 1b. Reinforcement is defined in two-dimensional layers by part ID. This option is active when the first entry is left blank.

	PID	AXIS	COOR	RQA	RQB		

Data Card Definitions:

Option 1. Reinforcement quantities in element groups

Card 1a	1	2	3	4	5	6	7	8
Variable	EID1	EID2	INC	XR	YR	ZR		
Type	I	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)

Option 2. Two dimensional layers by part ID. Option 2 is active when first entry is left blank.

Card 1b	1	2	3	4	5	6	7	8
Variable		PID	AXIS	COOR	RQA	RQB		
Type	blank	I	I	F	F	F		

VARIABLE	DESCRIPTION
PID	Part ID of reinforced elements. If PID = 0, the reinforcement is applied to all parts which use the Winfrith concrete model.
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 x and z, respectively. EQ.3: A and B are parallel to global x and y, respectively.
COOR	Coordinate location of layer: AXIS.EQ.1: x-coordinate AXIS.EQ.2: y-coordinate AXIS.EQ.3: z-coordinate
RQA	Reinforcement quantity (A).
RQB	Reinforcement quantity (B).

Remarks:

1. **Reinforcement Quantity.** 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.