*SECTION

In this section, the element formulation, integration rule, nodal thicknesses, and cross sectional properties are defined. All section identifiers (SECIDs) defined in this section must be unique, i.e., if a number is used as a section ID for a beam element then this number cannot be used again as a section ID for a solid element. The keyword cards in this section are defined in alphabetical order:

*SECTION_ALE1D

*SECTION_ALE2D

*SECTION_AUTODYN

*SECTION_BEAM

*SECTION_BEAM_AISC

*SECTION_DISCRETE

*SECTION_FPD

*SECTION_ISPG

*SECTION_POINT_SOURCE

*SECTION_POINT_SOURCE_MIXTURE

*SECTION_SEATBELT

SECTION_SHELL_*{OPTION}

SECTION_SOLID_*{OPTION}

**SECTION_SOLID_PERI*

SECTION_SPH_*{OPTION}

*SECTION_TSHELL

The location and order of these cards in the input file are arbitrary.

An additional option **TITLE** may be appended to all the ***SECTION** keywords. If this option is used then an addition line is read for each section in 80a format which can be used to describe the section. At present, the title serves no purpose other than to perhaps lend clarity to input decks.

*SECTION_ALE1D

Purpose: Define section properties for 1D ALE elements

Card Sets. For each ALE1D section add one pair of Cards 1 and 2. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ALEFORM	AET	ELFORM				
Type	I/A	I	I	I				
Default	none	none	0	none				

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

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
ALEFORM	ALE formulation: EQ.11: Multi-Material ALE formulation
AET	Ambient Element Type: EQ.4: Pressure inflow
ELFORM	Element formulation: EQ.7: plane strain EQ.8: axisymmetric (per radian) EQ.-8: spherical (per unit of solid angle)

VARIABLE	DESCRIPTION
THICK	Nodal thickness. See Remark 1.

Remarks:

1. **Nodal Thickness.** *SECTION_ALE1D is using the common *SECTION_BEAM reader which expects two thickness values. The ALE 1D will simply take the average of these two values as the beam thickness. The thickness, however, is not used for ELFORM = -8, but the reader routine expects values on the 2nd line.

2. **Output to d3plot.** Unlike normal beams, ALE beams have history variables for more than one material. Since ALE beams only have one integration point, we use BEAMIP in *DATABASE_EXTENT_BINARY in a similar fashion to NEIPH for solids to tell LS-DYNA how much data needs to be exported to the d3plot file. Normally BEAMIP just tells LS-DYNA how many integration points for which data is output. By doing this, we can access for each ALE group in an element, 6 stresses, 1 plastic strain and a number of history variables that depend on the material and equation of state for the ALE group. Note that each increase in the value of BEAMIP leads to an increase of 5 data slots in the output file.

   Because this is an overloaded use of BEAMIP, LS-PrePost labels will not match the data being output. The data being output will be under beam integration points (bintpt) which can be accessed with Post → FriComp → Beam. The interpretation of the data will be described with examples below.

   For all cases of BEAMIP including BEAMIP = 0, d3plot will include 5 volume averaged items that will be under the first integration point. These five values will be under the first integration point in LS-PrePost and are described in the table below. Their real meaning is in the column labeled Variable Description.

LS-PrePost Label	Variable Description	ALE Group
axial stress	pressure	averaged
rs-shear stress	density	averaged
tr-shear stress	compression ratio	averaged
plastic strain	internal energy/vol	averaged
axial strain	ALE group ID	

Now suppose that we have 2 ALE groups, one with 8 history variables and one with 3 history variables. If we want all the output data, the number of data variables that we need output to d3plot would be

$$5 + \text{number of ALE groups} \times 7 + \sum_{i=1}^{n} \text{history variables of ALE group } i$$

since the 5 averaged variables listed above are always output, each ALE group has 7 values for the stress and effective plastic strain, and each ALE group has history variables. In our current example, this would $5 + 2 \times 7 + 8 + 3 = 30$. Since each integration points has 5 data slots as understood by LS-PrePost, then 6 integration points of data are needed, so BEAMIP should be set to 5 (note that this is because BEAMIP = 0 causes one integration point worth of data to be output with this implementation). If it is not set to 5, then d3plot will only include the data up to the number of integration points being output. The description for the values starting with the second integration point are listed in the table below.

Integration Point in LS-PrePost	LS-PrePost Label	Variable Description	ALE Group
2	axial stress	*xx*-stress	1
2	*rs*-shear stress	*yy*-stress	1
2	*tr*-shear stress	*zz*-stress	1
2	plastic strain	*xy*-stress	1
2	axial strain	*yz*-stress	1
3	axial stress	*zx*-stress	1
3	*rs*-shear stress	effective plastic strain	1
3	*tr*-shear stress	history variable # 1	1
3	plastic strain	history variable # 2	1
3	axial strain	history variable # 3	1
4	axial stress	history variable # 4	1
4	*rs*-shear stress	history variable # 5	1
4	*tr*-shear stress	history variable # 6	1

Integration Point in LS-PrePost	LS-PrePost Label	Variable Description	ALE Group
4	plastic strain	history variable # 7	1
4	axial strain	history variable # 8	1
5	axial stress	*xx*-stress	2
5	*rs*-shear stress	*yy*-stress	2
5	*tr*-shear stress	*zz*-stress	2
5	plastic strain	*xy*-stress	2
5	axial strain	*yz*-stress	2
6	axial stress	*zx*-stress	2
6	*rs*-shear stress	effective plastic strain	2
6	*tr*-shear stress	history variable # 1	2
6	plastic strain	history variable # 2	2
6	axial strain	history variable # 3	2

*SECTION_ALE2D

Purpose: Define section properties for 2D ALE elements. This supersedes the old way of defining section properties for 2D ALE elements using *SECTION_SHELL.

Section Cards. For each ALE2D section include a card. This input terminates at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ALEFORM	AET	ELFORM				
Type	I/A	I	I	I				
Default	none	none	0	none				

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
ALEFORM	ALE formulation: EQ.11: Multi-Material ALE formulation.
AET	Part type flag: EQ.0: This is a regular or non-ambient part (default) EQ.4: Reservoir or ambient type part EQ.5: Reservoir or ambient type part, but only used together with *LOAD_BLAST_ENHANCED (LBE). It defines this part as an "ambient receptor part" for the transient blast load supplied by a corresponding LBE KW (see *LOAD_BLAST_ENHANCED, available only for ALEFORM = 11).
ELFORM	Element formulation: EQ.13: Plane strain (xy-plane) EQ.14: Axisymmetric solid (xy-plane, y-axis of symmetry) – area weighted (see Remark 2)

Remarks:

1. **Lagrangian-ALE coupling.** For coupling between 2D Lagrangian elements and 2D ALE elements, use *CONSTRAINED_LAGRANGE_IN_SOLID rather than *CONTACT_2D_AUTOMATIC_SURFACE_IN_CONTINUUM.

2. **Axisymmetric analysis.** For an axisymmetric analysis, ELFORM for *SECTION_ALE2D can only be set to 14 (area-weighted). In the same analysis, axisymmetric Lagrangian elements are not restricted to an area-weighted formulation. In other words, shell formulation 14 or 15 are permitted for Lagrangian shells, and beam formulation 8 is permitted for Lagrangian beams. Coupling forces between the axisymmetric ALE elements and axisymmetric Lagrangian elements are automatically adjusted as needed.

*SECTION_BEAM

Purpose: Define cross-sectional properties for beam, truss, discrete beam, and cable elements.

Card Summary:

Card Sets. For each BEAM section in the model add one set of Cards 1 and 2 (maybe additionally Card 3 for ELFORM = 12) cards. This input ends at the next keyword ("*") card.

Card 1. This card is required.

SECID	ELFORM	SHRF	QR/IRID	CST	SCOOR	NSM	NAUPD

Card 2a. Include this card when ELFORM is 1 or 11.

TS1	TS2	TT1	TT2	NSLOC	NTLOC	ITORM	

Card 2b. Include this card when ELFORM is 2, 3, or 12 and the first 7 characters of the card spell out "SECTION."

STYPE	D1	D2	D3	D4	D5	D6	ITORM

Card 2b.1. This card is optional.. It is only read when ELFORM = 2, the preceding card is Card 2b, and the first column includes the character "OPTCARD."

OPTC	ASA						

Card 2c. Include this card when ELFORM is 2, 12, or 13 and the first 7 characters of the card *do not* spell out "SECTION."

A	ISS	ITT	J	SA	IST	ITORM	

Card 2c.1. Include this card when ELFORM equals 12 and the preceding card is Card 2c.

YS	ZS	IYR	IZR	IRR	IW	IWR	

Card 2d. Include this card when ELFORM is 3 and the first 7 characters of the card *do not* spell "SECTION."

A	RAMPT	STRESS					

Card 2e. Include this card for ELFORM equal to 4 or 5.

TS1	TS2	TT1	TT2				

Card 2f. Include this card for ELFORM equal to 6 for any material other than material type 146.

VOL	INER	CID	CA	OFFSET	RRCON	SRCON	TRCON

Card 2g. Include this card for ELFORM equal to 6 for material type 146.

VOL	INER	CID	DOFN1	DOFN2			

Card 2h. Include this card for ELFORM equal to 7 or 8.

TS1	TS2						

Card 2i. Include this card for ELFORM equal to 9.

TS1	TS2	TT1	TT2	PRINT		ITOFF	

Card 2j. Include this card for ELFORM equal to 14.

PR	IOVPR	IPRSTR					

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	SHRF	QR/IRID	CST	SCOOR	NSM	NAUPD
Type	I/A	I	F	F	F	F	F	I
Default	none	1	1.0	2.0	0.0	0.0	0.0	0

VARIABLE	DESCRIPTION
SECID	Section ID. The *PART card references SECID. A unique number or label must be specified.
ELFORM	Element formulation options: EQ.1: Hughes-Liu with cross section integration (default) EQ.2: Belytschko-Schwer resultant beam (resultant) EQ.3: Truss (resultant) EQ.4: Belytschko-Schwer full cross-section integration

VARIABLE	DESCRIPTION
	EQ.5: Belytschko-Schwer tubular beam with cross-section integration EQ.6: Discrete beam/cable EQ.7: 2D plane strain shell element (xy-plane) EQ.8: 2D axisymmetric volume weighted shell element (xy-plane, y-axis of symmetry) EQ.9: Spot weld beam; see *MAT_SPOTWELD. EQ.11: Integrated warped beam. See Remark 3. EQ.12: Resultant warped beam EQ.13: Small displacement, linear Timoshenko beam with exact stiffness. See Remark 5. EQ.14: Elbow integrated tubular beam element. A user defined integration rule with tubular cross section (9) must be used. Note that the 2D and 3D element types must not be mixed, and different types of 2D elements must not be used together. For example, the plane strain element type must not be used with the axisymmetric element type. In 3D the different beam elements types, that is, 1-6 and 9 can be freely mixed together.
SHRF	Shear factor. This factor is not needed for truss, resultant beam, discrete beam, and cable elements. The recommended value for rectangular sections is 5/6, the default is 1.0.
QR/IRID	Quadrature rule or rule number for a user-defined rule for integrated beams. For rectangular cross sections, Gauss or Lobatto quadrature will be employed, while for circular cross sections, integration points are evenly spaced on the perimeter of the surface. See Remark 9 regarding beam formulations 7 and 8. EQ.1.0: One integration point EQ.2.0: 2×2 Gauss or 4-point circular quadrature (default beam) EQ.3.0: 3×3 Gauss or 9-point circular quadrature EQ.4.0: 3×3 Lobatto or 9-point circular quadrature EQ.5.0: 4×4 Gauss or 16-point circular quadrature EQ.-n: |n| is the number of the user-defined rule. IRID integration rule n is defined using *INTEGRATION_BEAM

VARIABLE	DESCRIPTION
	card.
CST	Cross section type, not needed for truss, resultant beam, discrete beam, and cable elements: EQ.0.0: Rectangular, EQ.1.0: Tubular (circular only), EQ.2.0: Arbitrary (user defined integration rule).
SCOOR	Affects the discrete beam formulation (see Remarks 6 and 14) and the update of the local coordinate system of the discrete beam element. This field does not apply to cable elements. The force and moment resultants in the output databases are output in the local coordinate system. See Remark 8 for more on the local coordinate system update. EQ.-14.0: Like -13.0, but with improved rotational stability EQ.-13.0: Like -3.0, but with correction for beam rotation EQ.-12.0: Like -2.0, but with correction for beam rotation EQ.-3.0: Beam node 1; the angular velocity of node 1 rotates triad. EQ.-2.0: Beam node 1: The angular velocity of node 1 rotates triad, but the *r*-axis is adjusted to lie along the line between the two beam nodal points. This option is not recommended for zero length discrete beams. EQ.-1.0: Beam node 1; the angular velocity of node 1 rotates triad. EQ.0.0: Centered between beam nodes 1 and 2. The triad rotates with the average angular velocity of nodes 1 and 2. EQ.+1.0: Beam node 2: The angular velocity of node 2 rotates the triad. EQ.+2.0: Beam node 2: The angular velocity of node 2 rotates the triad, but the *r*-axis is adjusted to lie along the line between the two beam nodal points. This option is not recommended for zero length discrete beams. EQ.+3.0: Beam node 2; the angular velocity of node 2 rotates triad. EQ.+12.0: Like +2.0, but with correction for beam rotation

VARIABLE	DESCRIPTION
	EQ.+13.0: Like +3.0, but with correction for beam rotation
	EQ.+14.0: Like +13.0 but with improved rotational stability
NSM	Nonstructural mass per unit length. This option applies to beam types 1-5 and does not apply to discrete, 2D, and spot weld beams.
NAUPD	Neutral axis update option. See Remark 11.
	EQ.0: Not used
	EQ.1: Update the neutral axis when damage or failure occurs at one or more integration points.

Integrated Beam Card (types 1 and 11). This card is Card 2 for ELFORM set to either type 1 or 11.

Card 2a	1	2	3	4	5	6	7	8
Variable	TS1	TS2	TT1	TT2	NSLOC	NTLOC	ITORM	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
TS1	Beam thickness (CST = 0.0, 2.0) or outer diameter (CST = 1.0) in s-direction at node n_1. The thickness defined on *ELEMENT_BEAM_THICKNESS overrides the definition given here.
TS2	Beam thickness (CST = 0.0, 2.0) or outer diameter (CST = 1.0) in s-direction at node n_2.
TT1	Beam thickness (CST = 0.0, 2.0) or inner diameter (CST = 1.0) in t-direction at node n_1.
TT2	Beam thickness (CST = 0.0, 2.0) or inner diameter (CST = 1.0) in t-direction at node n_2.
NSLOC	Location of reference surface normal to s-axis for Hughes-Liu beam elements only. See Remark 4.
	EQ.1.0: Side at $s = 1.0$
	EQ.0.0: Center
	EQ.-1.0: Side at $s = -1.0$

VARIABLE	DESCRIPTION
NTLOC	Location of reference surface normal to t-axis for Hughes-Liu beam elements only. See Remark 4. EQ.1.0: Side at $t = 1.0$ EQ.0.0: Center EQ.-1.0: Side at $t = -1.0$
ITORM	Flag for improved representation of torsional and rotational modes for beam type 1 for eigenvalue analyses only. EQ.0.0: Not active. LS-DYNA uses the torsional and rotational inertia from the structural analysis. This may result in too large eigenvalues related to torsional and rotational modes. EQ.1.0: LS-DYNA more accurately recomputes the mass related to torsional modes. EQ.2.0: LS-DYNA more accurately recomputes the mass related to torsional and rotational modes.

Resultant Beam with Shape Card (types 2, 3, and 12). Include this card as Card 2 when ELFORM equals 2, 3, or 12 and the first 7 characters of the card spell out "SECTION."

Card 2b	1	2	3	4	5	6	7	8
Variable	STYPE	D1	D2	D3	D4	D5	D6	ITORM
Type	A10	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
STYPE	Section type (A format) of resultant beam (see Figure 41-1): EQ.SECTION_01: I-Shape EQ.SECTION_02: Channel EQ.SECTION_03: L-shape EQ.SECTION_04: T-shape EQ.SECTION_05: Tubular box EQ.SECTION_06: Z-Shape EQ.SECTION_07: Trapezoidal EQ.SECTION_12: Cross EQ.SECTION_13: H-Shape EQ.SECTION_14: T-Shape 2 EQ.SECTION_15: I-Shape 3 EQ.SECTION_16: Channel 2 EQ.SECTION_17: Channel 3 EQ.SECTION_18: T-Shape 3

VARIABLE	DESCRIPTION
	EQ.SECTION_08: Circular EQ.SECTION_09: Tubular EQ.SECTION_10: I-Shape 2 EQ.SECTION_11: Solid Box EQ.SECTION_19: Box-Shape 2 EQ.SECTION_20: Hexagon EQ.SECTION_21: Hat-Shape EQ.SECTION_22: Hat-Shape 2
D1-D6	Input parameters for section option using STYPE above. See Figures 41-1 through 41-22.
ITORM	Flag for improved representation of torsional and rotational modes for beam type 2 for eigenvalue analyses only. EQ.0.0: Not active. LS-DYNA uses the torsional and rotational inertia from the structural analysis. This may result in too large eigenvalues related to torsional and rotational modes. EQ.1.0: LS-DYNA more accurately recomputes the mass related to torsional modes. EQ.2.0: LS-DYNA more accurately recomputes the mass related to torsional and rotational modes.

Resultant Beam with Optional Shape Card (type 2 only). This card is optional. It is read when ELFORM = 2, the preceding card is Card 2b, and the first column includes the string "OPTCARD."

Card 2b.1	1	2	3	4	5	6	7	8
Variable	OPTC	ASA						
Type	A	F						

VARIABLE	DESCRIPTION
OPTC	Include string "OPTCARD" to activate this card.
ASA	Automized shear area computation for section types defined with STYPE on Card 2b. EQ.0.0: Not active. The shear area is set equal to the cross-sectional area. EQ.1.0: Approximate the shear area based on an energy consideration. It currently applies to Tubular (SECTION_09)

VARIABLE	DESCRIPTION
	and Solid Box (SECTION_11) section types.

Resultant Beam Card 1 (types 2, 12, and 13). Include this card as Card 2 when ELFORM equals 2, 12, or 13 and the first 7 characters of the card *do not* spell "SECTION."

Card 2c	1	2	3	4	5	6	7	8
Variable	A	ISS	ITT	J	SA	IST	ITORM	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
A	Cross-sectional area. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here.
ISS	I_{ss}, area moment of inertia about local s-axis. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here.
ITT	I_{tt}, area moment of inertia about local t-axis. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here.
J	J, torsional constant. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here. If J is zero, then J is reset to the sum of ISS + ITT as an approximation for warped beam.
SA	Shear area. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here.
IST	I_{st}, product area moment of inertia with respect to the local s- and t-axes. This is only nonzero for asymmetric cross sections and it can take positive and negative values; for example, it is negative for SECTION_03.
ITORM	Flag for improved representation of torsional and rotational modes for beam types 2 and 13 for eigenvalue analyses only. EQ.0.0: Not active. LS-DYNA uses the torsional and rotational inertia from the structural analysis. This may result in too large eigenvalues related to torsional and rotational modes. EQ.1.0: LS-DYNA more accurately recomputes the mass related to torsional modes.

VARIABLE	DESCRIPTION
	EQ.2.0: LS-DYNA more accurately recomputes the mass related to torsional and rotational modes.

Resultant Beam Card 2 (type 12 only). Include this card if ELFORM equals 12 and the preceding card is Card 2c.

Card 2c.1	1	2	3	4	5	6	7	8
Variable	YS	ZS	IYR	IZR	IRR	IW	IWR	
Type	F	F	F	F	F	F	F	

VARIABLE	DESCRIPTION
YS	s coordinate of shear center of cross-section. The coordinate system is located at the centroid.
ZS	t coordinate of shear center of cross-section. The coordinate system is located at the centroid.
IYR	$\int_A sr^2 dA$, where $r^2 = s^2 + t^2$
IZR	$\int_A tr^2 dA$, where $r^2 = s^2 + t^2$
IRR	$\int_A r^4 dA$, where $r^2 = s^2 + t^2$
IW	Warping constant. $\int_A \omega^2 dA$, where ω is the sectorial area.
IWR	$\int_A \omega r^2 dA$

Resultant Beam Card (type 3). Include this card for ELFORM equal to 3.

Card 2d	1	2	3	4	5	6	7	8
Variable	A	RAMPT	STRESS					
Type	F	F	F					

VARIABLE	DESCRIPTION
A	Cross-sectional area. The definition on *ELEMENT_BEAM_THICKNESS overrides the value defined here.
RAMPT	Optional ramp-up time for dynamic relaxation. At the end of the ramp-up time, a uniform stress, STRESS, exists in the truss element. This option does not work for hyperelastic materials.
STRESS	Optional initial stress for dynamic relaxation. At the end of dynamic relaxation, a uniform stress equal to this value should exist in the truss element.

Integrated Beam Card (types 4 and 5). Include this card when ELFORM equals 4 or 5.

Card 2e	1	2	3	4	5	6	7	8
Variable	TS1	TS2	TT1	TT2				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
TS1	Beam thickness (CST = 0.0, 2.0) or outer diameter (CST = 1.0) in s-direction at node n_1. The thickness defined on *ELEMENT_BEAM_THICKNESS overrides the definition given here.
TS2	Beam thickness (CST = 0.0, 2.0) or outer diameter (CST = 1.0) in s-direction at node n_2.
TT1	Beam thickness (CST = 0.0, 2.0) or inner diameter (CST = 1.0) in t-direction at node n_1.
TT2	Beam thickness (CST = 0.0, 2.0) or inner diameter (CST = 1.0) in t-direction at node n_2.

Discrete Beam Card (type 6). Include this card when ELFORM equals 6 for any material *other* than material type 146.

Card 2f	1	2	3	4	5	6	7	8
Variable	VOL	INER	CID	CA	OFFSET	RRCON	SRCON	TRCON
Type	F	F/I	F	F	F	F	F	F

VARIABLE	DESCRIPTION
VOL	Volume of discrete beam and scalar (MAT_146) beam. Used in calculating mass. If VOL = 0 for cable elements, volume is calculated as the product of cable length and cable area. If the mass density of the material model for the discrete beam is set to unity, the magnitude of the lumped mass can be defined here instead. This lumped mass is partitioned equally to the two nodes of the element. See Remark 12.
INER	Mass moment of inertia for the six degrees of freedom discrete beam and scalar (*MAT_146) beam. This parameter does not apply to cable elements. This lumped inertia is partitioned equally to the two nodes of the element. See Remarks 12 and 13. GT.0.0: Inertia partitioned to the nodes. EQ.-1.0: The element inertia is computed as a solid sphere of volume VOL. This inertia, which is reported to d3hsp, is then partitioned to the nodes. EQ.-2.0: The element inertia is computed such that the rotational time step matches the translational. It is currently only available for *MAT_196. The computed element inertias are written to the message file(s).
CID	Coordinate system ID for orientation (material types 66-69, 93, 95, 97), see *DEFINE_COORDINATE_*OPTION*. If CID = 0, a default coordinate system is defined in the global system or on the third node of the beam, which is used for orientation. This option is not defined for material types that act between two nodal points, such as cable elements. The coordinate system rotates with the discrete beam, see SCOOR above.
CA	Cable area. See material type 71, *MAT_CABLE_DISCRETE_BEAM. For a crushable beam using *MAT_119 with IFLAG = 2, CA is the cross-sectional area ratio between end 2 and end 1. A crushable beam of uniform cross section has CA = 1, the default value.
OFFSET	Optional offset for cable. See material type 71, *MAT_CABLE_DISCRETE_BEAM.
RRCON	*r*-rotational constraint for the local coordinate system (see Remark 7) EQ.0.0: Coordinate ID rotates about *r*-axis with nodes.

VARIABLE	DESCRIPTION
	EQ.1.0: Rotation is constrained about the *r*-axis
SRCON	*s*-rotational constraint for the local coordinate system (see Remark 7) EQ.0.0: Coordinate ID rotates about *s*-axis with nodes. EQ.1.0: Rotation is constrained about the *s*-axis
TRCON	*t*-rotational constraint for the local coordinate system (see Remark 7) EQ.0.0: Coordinate ID rotates about *t*-axis with nodes. EQ.1.0: Rotation is constrained about the *t*-axis.

Discrete Beam Card (type 6, mat 146). Include this card when ELFORM equals 6 for material type 146.

Card 2g	1	2	3	4	5	6	7	8
Variable	VOL	INER	CID	DOFN1	DOFN2			
Type	F	F/I	F	F	F			

VARIABLE	DESCRIPTION
VOL	Volume of discrete beam and scalar (MAT_146) beam. Used in calculating mass. If VOL = 0 for cable elements, volume is calculated as the product of cable length and cable area. If the mass density of the material model for the discrete beam is set to unity, the magnitude of the lumped mass can be defined here instead. This lumped mass is partitioned equally to the two nodes of the element. See Remark 12.
INER	Mass moment of inertia for the six degrees of freedom discrete beam and scalar (MAT_146) beam. This parameter does not apply to cable elements. This lumped inertia is partitioned equally to the two nodes of the element. See Remarks 12 and 13. GT.0.0: Inertia partitioned to the nodes. EQ.-1.0: The element inertia is computed as a solid sphere of volume VOL. This inertia, which is reported to d3hsp, is then partitioned to the nodes.

VARIABLE	DESCRIPTION
	EQ.-2.0: The element inertia is computed such that the rotational time step matches the translational. It is currently only available for *MAT_196. The computed element inertias are written to the message file(s).
CID	Coordinate system ID for orientation, material type 146, see *DEFINE_COORDINATE_SYSTEM. If CID = 0, a default coordinate system is defined in the global system.
DOFN1	Active degree of freedom at node 1, a number between 1 and 6 where 1 is x-translation and 4 is x-rotation.
DOFN2	Active degree of freedom at node 2, a number between 1 and 6.

2D Shell Card (types 7 and 8). Include this card when ELFORM equals 7 or 8.

Card 2h	1	2	3	4	5	6	7	8
Variable	TS1	TS2						
Type	F	F						

VARIABLE	DESCRIPTION
TS1	Thickness at node n_1.
TS2	Thickness at node n_2.

Spot Weld Card (type 9). Include this card when ELFORM equals 9.

Card 2i	1	2	3	4	5	6	7	8
Variable	TS1	TS2	TT1	TT2	PRINT		ITOFF	
Type	F	F	F	F	F		F	

VARIABLE	DESCRIPTION
TS1	Beam thickness (CST = 0.0, 2.0) or outer diameter (CST = 1.0) in s-direction at node n_1. Note that the thickness defined on the *ELEMENT_BEAM_THICKNESS card overrides the definition given here.

VARIABLE	DESCRIPTION
TS2	Beam thickness (CST = 0.0, 2.0) or outer diameter (CST = 1.0) in s-direction at node n_2.
TT1	Beam thickness (CST = 0.0, 2.0) or inner diameter (CST = 1.0) in t-direction at node n_1.
TT2	Beam thickness (CST = 0.0, 2.0) or inner diameter (CST = 1.0) in t-direction at node n_2.
PRINT	Output spot force resultant from spot welds. EQ.0.0: Data is output to swforc file. EQ.1.0: Output is suppressed.
ITOFF	Option to specify torsional behavior for spot weld beams. EQ.0.0: Torsional stiffness is active. EQ.1.0: Torsional stiffness is zero (free to twist).

Integrated Beam Card (types 14). Include this card when ELFORM equals 14.

Card 2j	1	2	3	4	5	6	7	8
Variable	PR	IOVPR	IPRSTR					
Type	F	F	F					

VARIABLE	DESCRIPTION
PR	Pressure inside ELBOW elements that belong to the section. The pressure acts as a stiffener and will reduce the ovalization of the pipe. Pressure acting on the inside wall is taken as positive.
IOVPR	Print flag for the ELBOW ovalization degrees of freedom. EQ.1.0: An ASCII file named elbwov is created and filled with the ovalization. Default no file is created.
IPRSTR	Flag for adding stress due to pressure PR into the material routine. EQ.0: No stress is added to the material. In this case the pressure only acts as a stiffener for the tube. EQ.1: The pressure PR is used to calculate additional axial and circumferential stresses due to the applied pressure PR.

VARIABLE	DESCRIPTION
	The stress added is given by: $$\sigma_{\text{axial}} = \text{PR} \times \frac{D}{4T}, \qquad \sigma_{\text{circ}} = \text{pr} \times \frac{D}{2T}$$ for a straight pipe, and $$\sigma_{\text{axial}} = \text{PR} \times \frac{D}{4T}, \qquad \sigma_{\text{circ}} = \text{PR} \times \frac{D}{4T} \frac{[2R + r\cos(\theta)]}{[R + r\cos(\theta)]}$$ for a curved pipe. D is the pipe diameter, T is the thickness, R is the curvature of the bend, r is the pipe radius (mean) and θ is an angle pointing out a point on the pipe. This option also includes the stiffening effect.

Remarks:

1. **Implicit time integrator.** For implicit calculations all the beam element choices are implemented.

2. **Local coordinate system rotation.** The local coordinate system rotates as the nodal points that define the beam rotate. In some cases this may lead to unexpected results if the nodes undergo significant rotational motions. In the definition of the local coordinate system using *DEFINE_COORDINATE_NODES, if the option to update the system each cycle is active then this updated system is used. This latter technique seems to be more stable in some applications.

3. **Integrated warped beam.** The integrated warped beam (type 11) is a 7 degrees of freedom beam that must be used with an integration rule of the open standard cross sections, see *INTEGRATION_BEAM. To incorporate the additional degrees of freedom corresponding to the twist rates, one scalar node (*NODE_SCALAR) should be declared for each node attached to a warped beam. This degree of freedom is associated with the beam element using the warpage option on the *ELEMENT_BEAM card.

4. **Beam offsets.** Beam offsets are sometimes necessary for correctly modeling beams that act compositely with other elements such as shells or other beams. A beam offset extends from the beam's n_1-to-n_2 axis to the reference axis of the beam. The beam reference axis lies at the origin of the local s- and t-axes. This origin is located at the center of the cross-section footprint for beam formulations 1 and 11, but it is located at the cross-section centroid for beam formulation 2. Note that for cross-sections that are not doubly symmetric, such as a T-section, the center of the cross-section footprint and the centroid of the cross-section do not coincide. The offset in the positive s-direction is

$$\text{s-offset} = -0.5 \times \text{NSLOC} \times (\text{beam cross-section dimension in } s\text{-direction}) .$$

Similarly, the offset in the positive t-direction is

$$\text{t-offset} = -0.5 \times \text{NTLOC} \times (\text{beam cross-section dimension in } t\text{-direction}) .$$

If IRID is used to point to an integration rule with ICST > 0, then offsets must be defined using SREF and TREF on the *INTEGRATION_BEAM card as they will override NSLOC and NTLOC even if SREF = 0 or TREF = 0. See also *ELEMENT_BEAM_OFFSET for an alternate approach to defining beam offsets.

5. **Three-dimensional Timoshenko beam.** Element type 13 is a three-dimensional Timoshenko resultant-based beam element with two nodes for small displacement, linear isotropic elasticity. The stiffness matrix is identical to the residual stiffness formulation used in the Belytschko-Schwer element (type 2). This element only works with *MAT_ELASTIC. It uses the reference geometry to calculate the element stiffness and calculates the element forces by multiplying the element stiffness by the displacements. Offsets work but they are fixed for all time like the reference geometry.

6. **SCOOR.** If the magnitude of SCOOR is less than or equal to unity, then zero length discrete beams are assumed with infinitesimal separation between the nodes in the deformed state. For large separations or nonzero length beams set |SCOOR| to 2, 3, 12, 13 or 14, in which case true beam-like behavior is invoked to provide equilibrating torques to offset any force couples that arise due to translational stiffness or translational damping. For $|\text{SCOOR}| \leq 13$ the equilibrating torque is divided equally between Node 1 and Node 2. In some cases, this can lead to unstable rotational behavior and non-physical energy growth; $|\text{SCOOR}| = 14$ fixes that by applying all of the equilibrating torque onto the node that carries the axis system, i.e., Node 1 for SCOOR = −14 and Node 2 for SCOOR = +14. Also, rigid body rotation is measured, and the spring strain modified so that rotation does not create strain. A flaw in this strain modification was found in the implementation of |SCOOR| = 2 and 3; the improved formulation is activated by setting |SCOOR| = 12, 13, or 14. The original options were left in place to allow legacy data to run without change. See also Remark 14.

7. **Disabling nodal rotations.** RRCON, SRCON, and TRCON are optional and apply only to non-cable discrete beams. If set to 1, RRCON, SRCON, and TRCON will prevent nodal *rotations* about the local *r*-, *s*-, and *t*-axes, respectively, from affecting the update of the local coordinate system. These three parameters have no influence on how nodal translations may affect the local coordinate system update.

8. **Note about local coordinate updates and FLAG.** If CID is nonzero for a discrete beam and the coordinate system identified by CID uses *DEFINE_COORDINATE_NODES with FLAG = 1, the beam local system is updated based on the current orientation of the three nodes identified in *DEFINE_COORDINATE_NODES. In this case, local coordinate system updates per SCOOR types 0, ±1, ±3, ±13 and ±14 are inactive while for SCOOR types ±2 and ±12 a final adjustment is made to the local coordinate system so that the local r-axis lies along the n_1 to n_2 axis of the beam. An optional output database (*DATABASE_DISBOUT) will report relative displacements, rotations, and force resultants of discrete beams, all in the local coordinate system. See also Remark 13.

9. **Beam forms 7 and 8.** Beam formulations 7 and 8 are two-dimensional shell elements. For these two formulations, variable QR/IRID is the number of through thickness integration points for the shell. Output for these integration points is controlled by the variable BEAMIP in *DATABASE_EXTENT_BINARY.

10. **Beam form 1.** The integration points for beam formulation 1 are located at mid-length and are arranged in the cross-section according to the specified integration rule. Stresses are reported at integration points according to the variable BEAMIP in *DATABASE_EXTENT_BINARY.

11. **NAUPD.** Integrated beams have integration points arranged about the neutral axis. Therefore, axial stress at each point will generate both an axial force and a moment at each node. The moment is proportional to the distance from the neutral axis. For a pure axial load, the moments calculated from the integration points cancel and only the force remains. However, if damage or failure occurs at one or more integration points, then the moments will no longer cancel and an axial load will produce moments at the nodes. To prevent this default behavior, set NAUPD to 1 to activate the neutral axis update option. With this option, the neutral axis will be updated such that partially failed elements will continue to generate balanced moments during axial loading. This option applies to beam forms 1, 9, 11, and 14 when used with material types 3, 98, 100, 124, and 158.

12. **Discrete beam volume and inertia.** The translational time step size for the type 6 beam depends on the volume, mass density, and the translational stiffness values. Similarly, the rotational time step size for the type 6 beam depends on the lumped inertia and the rotational stiffness values. Therefore, it is important to define the volume and inertia with the VOL and INER parameters. Defining the volume and inertia is also essential for mass scaling if the type 6 beam controls the time step size. To avoid instabilities, it is recommended to always set the volume and inertia parameters to reasonable non-zero values and/or having model-dependent properties; if the inertia set is smaller than that of an equivalent solid sphere, LS-DYNA issues a warning. If VOL (or INER) is not set, the

element mass (or inertia) is computed as the harmonic mean of half the nodal masses (or inertias) for the time step estimation.

13. **Discrete beam rotational time step.** For discrete beams of *nonzero* length with |SCOOR| values of 2, 3, 13, and 14, the translational shear and rotational stiffnesses are coupled. More specifically, the off-axis rotational stiffness (for some deformation modes) increases by an amount proportional to the beam length times the shear stiffness. Thus, nonzero shear stiffness decreases the rotational time step. To counteract this effect, additional rotational inertia is added to the beam. This additional inertia is primarily a function of the beam length such that the total element rotational inertia is as listed in the following table:

|SCOOR|	INER	Element inertia
0, 1, or 12	> 0	INER
0, 1, or 12	-1	I_{sphere}
0, 1, or 12	-2	$\frac{k_r}{k_t} m_e$
2, 3, 13, or 14	> 0	$\text{INER} + \max\left(m_e L^2, 16 m_e \sqrt{\text{VOL} \times L}\right)$
2, 3, 13, or 14	-1	$I_{\text{sphere}} + \max\left(m_e L^2, 16 m_e \sqrt{\text{VOL} \times L}\right)$
2, 3, 13, or 14	-2	$\left(\frac{k_\text{r}}{k_\text{t}} + \frac{0.95^2}{1-0.95^2}\frac{L^2}{4}\right) m_\text{e}$

Here, $m_\text{e} = \text{RHO} \times \text{VOL}$ is the element mass, L is the element length, k_r and k_t are the maximum rotational and translational stiffnesses, and

$$I_{\text{sphere}} = \frac{2}{5} m_\text{e} \left(\frac{3}{4\pi}\text{VOL}\right)^{\frac{2}{3}}$$

is the rotational inertia of a sphere of volume VOL.

INER < 0 is meant to help set a reasonable rotational inertia when this inertia is unknown. Otherwise, if the rotational inertias of the nodes are known, INER should be set to the sum of these inertias.

14. **Further notes on SCOOR for nonzero-length discrete beams.** This remark does not apply to cable elements (see *MAT_071). Unintended rotation of the local axes during the analysis can lead to unexpected or confusing results. To avoid such issues and ensure that the local axes retain their expected directions throughout the analysis, we recommend the following setups:

a) For applications in which only an axial force is required, meaning a force that always acts along the line defined by the two nodes of the beam no matter how that line rotates during the analysis, the following set-up is recommended:

- The material card should contain only *r*-axis translational properties.
- SCOOR can be set to any of 2, -2, 12, or -12 with identical results.
- CID can be left blank or zero.

b) For applications intended to generate force in directions other than purely axial, and where at least one end of the discrete beam is connected to a relatively stiff component (for example, a suspension bush connecting two metal parts), fix the local axes in the stiff component as follows:

- Create a *DEFINE_COORDINATE_NODES based on three nodes in the stiff component and set FLAG = 1 (see Remark 8).
- Reference the *DEFINE_COORDINATE_NODES using CID on *SECTION_BEAM.
- Set SCOOR = 13.

c) For applications intended to generate force in directions other than purely axial, and where no suitable stiff component exists to carry the local axes (or if avoiding the need to create coordinate systems), the following process is recommended. Considering the other elements connected to Node 1 and Node 2 of the beam, estimate whether Node 1 or Node 2 will provide greater resistance to rotation in response to torques generated by the beam. For example, if Node 1 is connected to solid elements (which do not generate any rotational stiffness at their nodes), while Node 2 is connected to shell elements, choose Node 2. On the other hand, if Node 1 is connected to rigid elements, choose Node 1. Then set SCOOR = -14 if Node 1 was chosen; set SCOOR = 14 if Node 2 was chosen. These settings fix the beam's local axes in the most rotationally stable of its two nodes.

Note also that shell elements do not generate rotational stiffness for the "drilling" degree of freedom of their nodes. Creating a small rigid body or nodal rigid body to provide a rotationally stiff base for the node that carries the local axes may be helpful.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$$$$   *SECTION_BEAM
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a Belytschko-Schwer resultant beam (elform = 2) with the following
$  properties.  This beam models the connection/stiffening beams of a medium
$  size roadside sign.
```

```
$
$    cross-sectional area:                           a =     515.6 mm2
$    2nd moment of area about s-axis:              iss =  99,660.0 mm4
$    2nd moment of area about t-axis:              iss =  70,500.0 mm4
$    2nd polar moment of area about beam axis:   j = 170,000.0 mm4
$
*SECTION_BEAM
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform      shrf   qr/irid       cst
       111         2
$
$        a       iss       itt         j        sa
     515.6   99660.0   70500.0  170000.0
$
*SECTION_BEAM_TITLE
    Main beam member
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform      shrf   qr/irid       cst
       111         2
$
$        a       iss       itt         j        sa
     515.6   99660.0   70500.0  170000.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

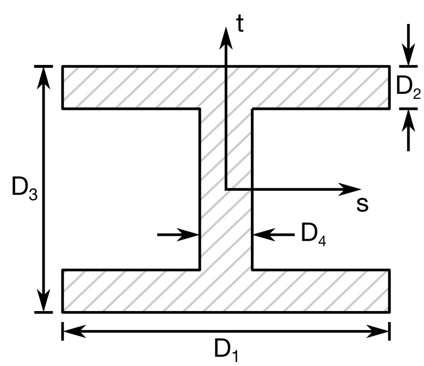


Figure 41-1. SECTION_01 ⇒ I-Shape

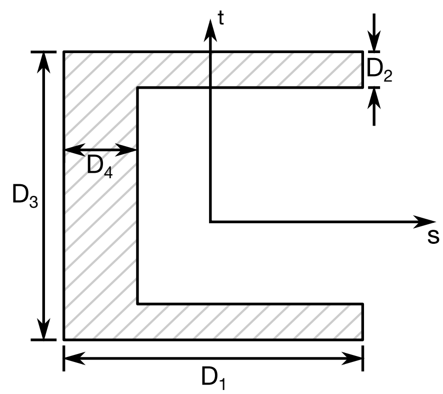
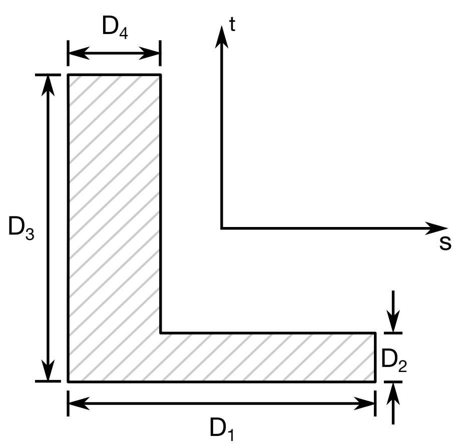


Figure 41-2. SECTION_02 ⇒ Channel



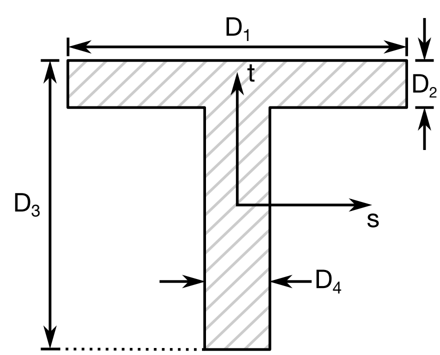


Figure 41-3. SECTION_03 ⇒ L-Shape

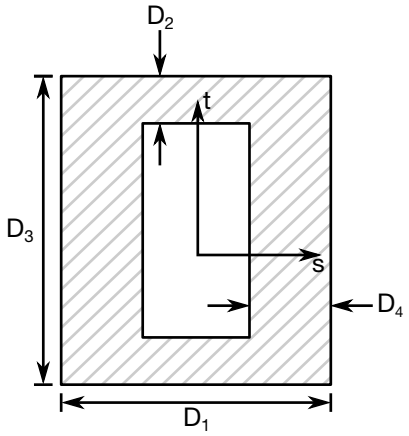


Figure 41-4. SECTION_04 ⇒ T-Shape

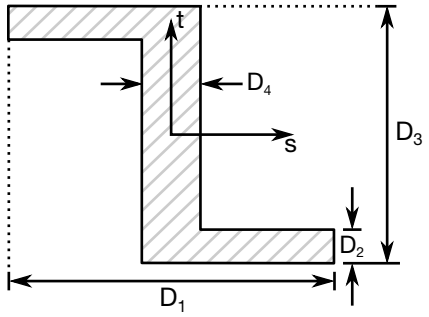
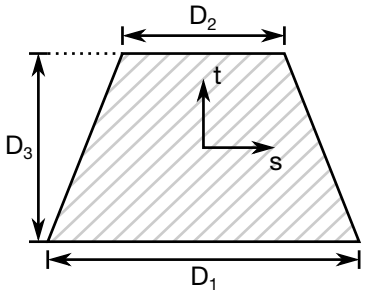


Figure 41-5. SECTION_05 ⇒ Box-Shape

Figure 41-6. SECTION_06 ⇒ Z-Shape



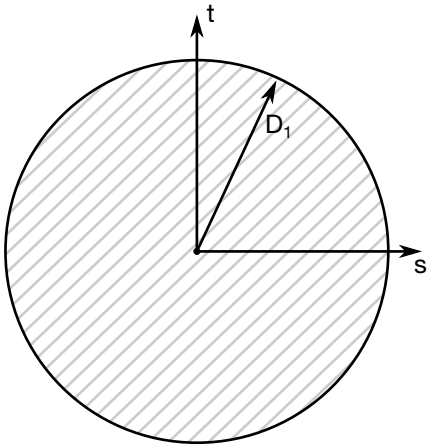
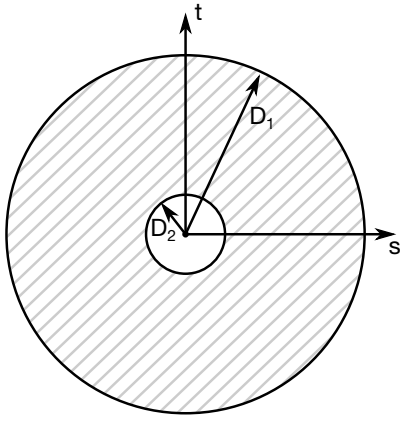


Figure 41-7. SECTION_07 ⇒ Trapezoidal

Figure 41-8. SECTION_08 ⇒ Circular



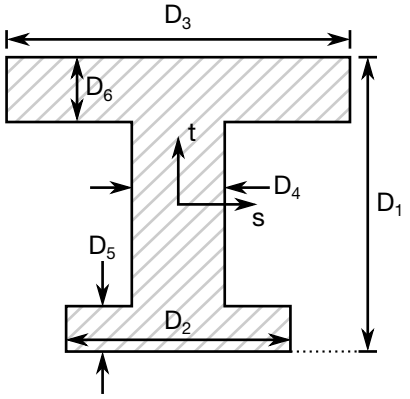


Figure 41-9. SECTION_09 ⇒ Tubular

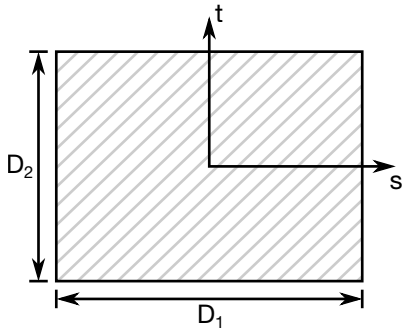


Figure 41-10. SECTION_10 ⇒ I-Shape 2

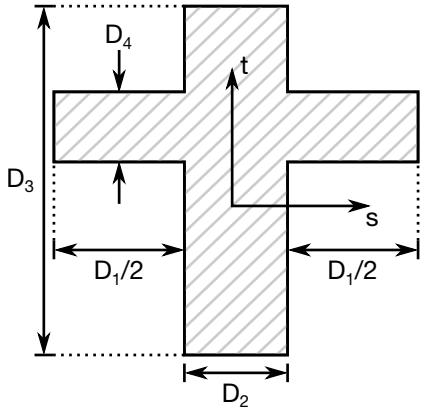


Figure 41-11. SECTION_11 ⇒ Solid Box

Figure 41-12. SECTION_12 ⇒ Cross

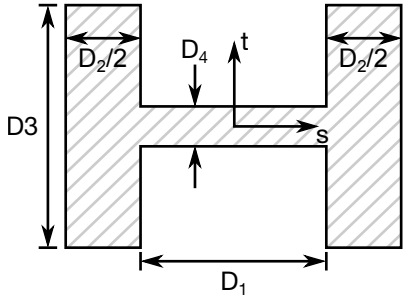


Figure 41-13. SECTION_13 ⇒ H-Shape

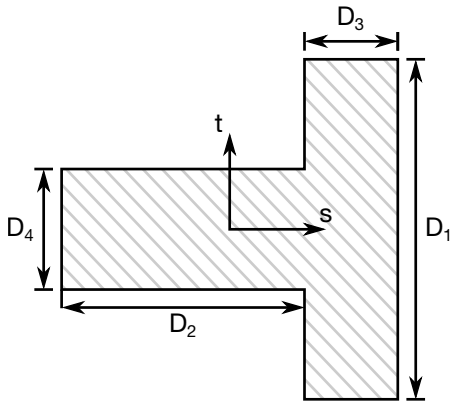


Figure 41-14. SECTION_14 ⇒ T-Shape 2

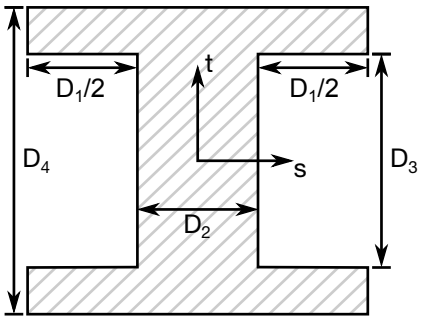


Figure 41-15. SECTION_15 ⇒ I-Shape 3

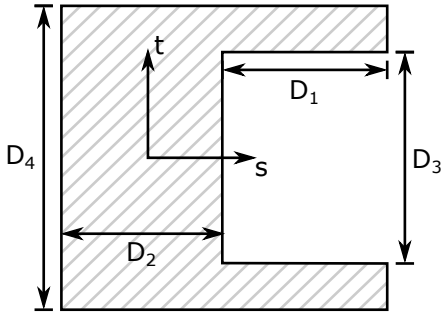


Figure 41-16. SECTION_16 ⇒ Channel 2

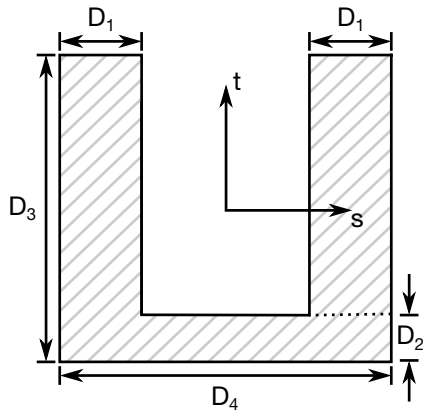


Figure 41-17. SECTION_17 ⇒ Channel 3

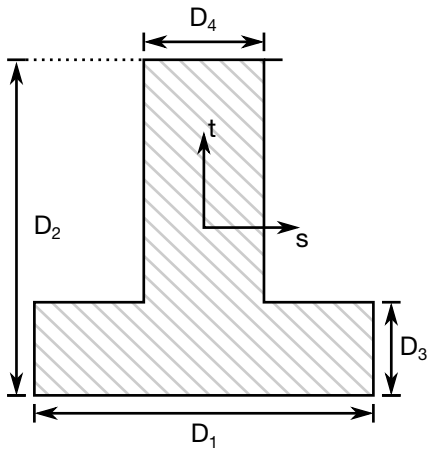


Figure 41-18. SECTION_18 ⇒ T-Shape 3

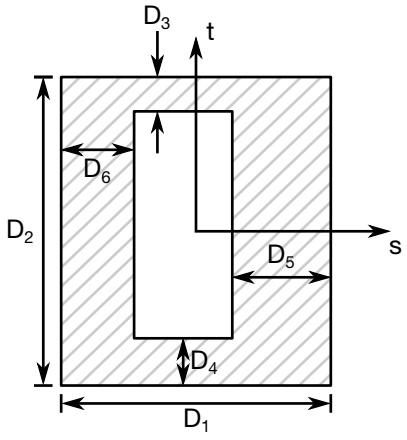


Figure 41-19. SECTION_19 ⇒ Box-Shape 2

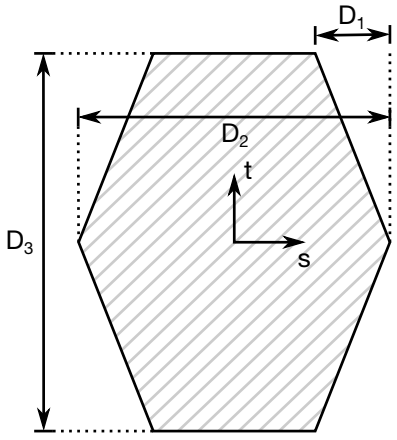


Figure 41-20. SECTION_20 ⇒ Hexagon

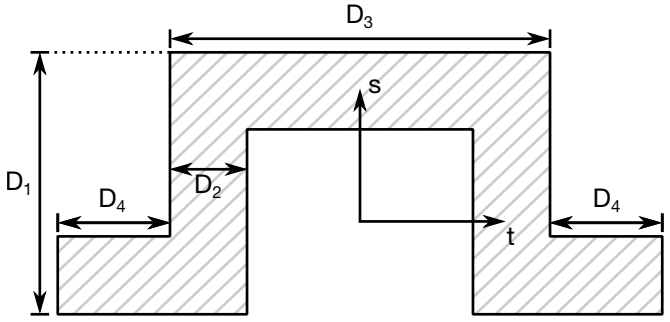


Figure 41-21. SECTION_21 ⇒ Hat Shape

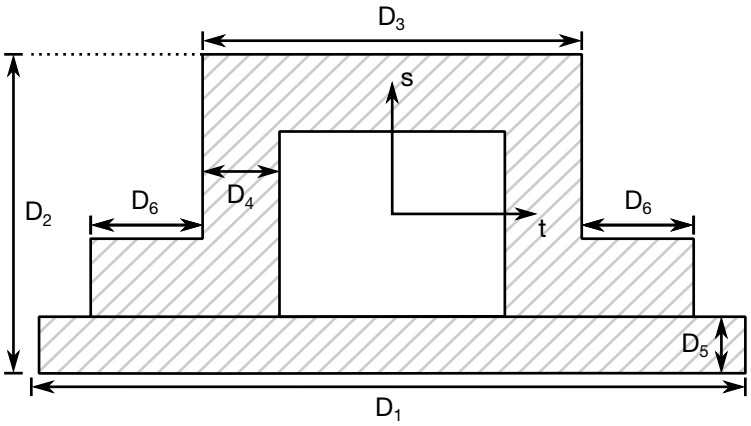


Figure 41-22. SECTION_22 ⇒ Hat Shape 2

*SECTION_BEAM_AISC

Purpose: Define cross-sectional properties for beams and trusses using section labels from the AISC Steel Construction Manual, 2005, 13th Edition, as published in the AISC Shapes Database V13.1.1

Card Summary:

Card Sets. For each BEAM_AISC section include one pair of Cards 1 and 2. This input ends at the next keyword ("*") card.

Card 1. This card is required.

SECID	LABEL

Card 2a. Card 2 for ELFORM equal to 1 or 11.

ELFORM	SHRF	NSM	LFAC	NSLOC	NTLOC	K	

Card 2b. Card 2 for ELFORM equal to 2 or 12.

ELFORM	SHRF	NSM	LFAC				

Card 2c. Card 2 for ELFORM equal to 3.

ELFORM	LFAC	RAMPT	STRESS				

Card 2d. Card 2 for ELFORM equal to 4 or 5.

ELFORM	SHRF	NSM	LFAC	K			

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	LABEL						
Type	I	A70						

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
LABEL	AISC section label

Integrated Beam Card (types 1 and 11). Card 2 for ELFORM equal to 1 or 11.

Card 2a	1	2	3	4	5	6	7	8
Variable	ELFORM	SHRF	NSM	LFAC	NSLOC	NTLOC	K	
Type	I	F	F	F	F	F	I	

Resultant Beam Card (types 2 and 12). Card 2 for ELFORM equal to 2 or 12.

Card 2b	1	2	3	4	5	6	7	8
Variable	ELFORM	SHRF	NSM	LFAC				
Type	I	F	F	F				

Truss Beam Card (type 3). Card 2 for ELFORM equal to 3.

Card 2c	1	2	3	4	5	6	7	8
Variable	ELFORM	LFAC	RAMPT	STRESS				
Type	I	F	F	F				

Integrated Beam Card (types 4 and 5). Card 2 for ELFORM equal to 4 or 5.

Card 2d	1	2	3	4	5	6	7	8
Variable	ELFORM	SHRF	NSM	LFAC	K			
Type	I	F	F	F	F			

VARIABLE	DESCRIPTION
ELFORM	Element formulation (see *SECTION_BEAM). Only types 1–5,11,12 are allowed
SHRF	Shear factor (see *SECTION_BEAM)
NSM	Non-structural mass per unit length

VARIABLE	DESCRIPTION
LFAC	GT.0.0: Length scale factor to convert dimensions from standard units If LFAC < 0, then a predefined length factor for specific model units is used: EQ.-1.0: ft EQ.-2.0: m EQ.-3.0: in EQ.-4.0: mm EQ.-5.0: cm
NSLOC	Location of reference surface (see *SECTION_BEAM)
NTLOC	Location of reference surface (see *SECTION_BEAM)
K	Integration refinement parameter (see *INTEGRATION_BEAM)
RAMPT	Optional ramp-up time (see *SECTION_BEAM)
STRESS	Optional initial stress (see *SECTION_BEAM)

Remarks:

This keyword uses the dimensions of the standard AISC beams sections — as defined by the section label — to define *SECTION_BEAM and *INTEGRATION_BEAM cards with the appropriate parameters.

The AISC section label may be specified either as the shape designation as seen in the AISC Steel Construction Manual, 2005, or the designation according to the AISC Naming Convention for Structural Steel Products for Use in Electronic Data Interchange (EDI), 2001. As per the EDI convention, the section labels are to be case-sensitive and space sensitive, i.e. “W36X150” is acceptable but “W36 x 150” is not. Labels can be specified in terms of either the U.S. Customary units (in) or metric units (mm), which will determine the length units for the section dimensions. The parameter LFAC may be used as a multiplier to convert the dimensions to other lengths units. For user convenience, predefined conversion factors are provided for specific choices of the model length unit.

AISC requires the following legal notice specifying that LS-DYNA users may not extract AISC shapes data in any manner from LS-DYNA to create commercial software. Users are certainly allowed to create LS-DYNA input models for commercial purposes using this card.

AISC FLOW-DOWN LICENSE TERMS (TERMS OF USE)

This application contains software from the American Institute of Steel Construction, Inc. of Chicago, Illinois d/b/a AISC ("AISC"). The software from AISC (the "AISC Shapes Database") enables this application to provide dimensions and properties of structural steel shapes and to perform other functions. You may use AISC Data only by means of the intended End User functions of this application software. You agree that you will use AISC Data for non-commercial use only, meaning that it will not be used to develop or create revenue-producing software. You agree not to assign, copy, transfer or transmit the AISC Shapes Database or any AISC Data to any third party.
YOU AGREE NOT TO USE OR EXPLOIT AISC DATA OR THE AISC SHAPES DATA-BASE EXCEPT AS EXPRESSLY PERMITTED HEREIN.

*SECTION_DISCRETE

Purpose: Defined spring and damper elements for translation and rotation. These definitions must correspond with the material type selection for the elements, that is, *MAT_SPRING_... and *MAT_DAMPER_...

Card Sets. For each DISCRETE section include a pair of Cards 1 and 2. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	DRO	KD	V0	CL	FD		
Type	I/A	I	F	F	F	F		

Card 2	1	2	3	4	5	6	7	8
Variable	CDL	TDL						
Type	F	F						

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
DRO	Displacement/Rotation Option: EQ.0: the material describes a translational spring/damper, EQ.1: the material describes a torsional spring/damper.
KD	Dynamic magnification factor, k_d. See Remarks 1 and 2 below.
V0	Test velocity, V_0. See Remarks 1 and 2 below.
CL	Clearance. See Remark 3 below.
FD	Failure deflection (twist for DRO = 1). Negative for compression, positive for tension.
CDL	Deflection (twist for DRO = 1) limit in compression. See Remark 4 below.

VARIABLE	DESCRIPTION
TDL	Deflection (twist for DRO = 1) limit in tension. See Remark 4 below.

Remarks:

1. **Optional Constants.** The constants from KD to TDL are optional and do not need to be defined.

2. **Forces and Amplification Factors.** If k_d is nonzero, the forces computed from the spring elements are assumed to be the static values and are scaled by an amplification factor to obtain the dynamic value:

$$F_{\text{dynamic}} = \left(1. + k_d \frac{V}{V_0}\right) F_{\text{static}} \ ,$$

where

V = absolute value of the relative velocity between the nodes.

V_0 = dynamic test velocity.

For example, if it is known that a component shows a dynamic crush force at 15 m/s equal to 2.5 times the static crush force, use $k_d = 1.5$ and $V_0 = 15$.

3. **Clearance.** Here, "clearance" defines a compressive displacement which the spring sustains before beginning the force-displacement relation given by the load curve defined in the material selection. If a non-zero clearance is defined, the spring is compressive only.

4. **Deflection Limits.** The deflection limit in compression and tension is restricted in its application to no more than one spring per node subject to this limit and to deformable bodies only. For example, in the former case, if three springs are in series, either the center spring or the two end springs may be subject to a limit, but not all three. When the limiting deflection is reached, momentum conservation calculations are performed, and a common acceleration is computed in the appropriate direction. An error termination will occur if a rigid body node is used in a spring definition where deflection is limited.

   Constrained boundary conditions on the *NODE cards and the BOUNDARY_SPC cards must not be used for nodes of springs with deflection limits.

5. **Implicit.** Discrete elements can be included in implicit applications.

Examples:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *SECTION_DISCRETE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Note: These examples are in kg, mm, ms, kN units.
$
$  A translational spring (dro = 0) is defined to have a failure deflection
$  of 25.4 mm (fd = 25.4).  The spring has no dynamic effects or
$  deflection limits, thus, those parameters are not set.
$
*SECTION_DISCRETE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid       dro        kd        v0        cl        fd
       104         0                                    25.4
$
$      cdl       tdl

$
$
$  Define a translational spring that is known to have a dynamic crush force
$  equal to 2.5 times the static force at a 15 mm/ms deflection rate.
$  Additionally, the spring is known to be physically constrained to deflect
$  a maximum of 12.5 mm in both tension and compression.
$
*SECTION_DISCRETE
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid       dro        kd        v0        cl        fd
       107         0       1.5      15.0
$
$      cdl       tdl
      12.5      12.5
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

*SECTION_FPD

Purpose: Define section properties for incompressible smoothed particle Galerkin (ISPG) element.

> **NOTE:** This keyword does not work with *ISPG or *MAT_ISPG keywords. It cannot be used in an ISPG input deck included with *INCLUDE_ISPG. The element formulations on this keyword are compatible with *MAT_319.

ISPG has been implemented as a 3D element formulation (ELFORM = 49) in *SECTION_FPD for fully implicit dynamic analysis. The ISPG solver automatically converts the nodes from the user's FEM model into the ISPG nodes.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM						
Type	I/A	I						

Card 2	1	2	3	4	5	6	7	8
Variable	DX	DY	DZ					
Type	F	F	F					
Default	1.5	1.5	1.5					

Card 3	1	2	3	4	5	6	7	8
Variable			TSTART	DT_IMP	RCFORCH	DTSCL		
Type			F	F	F	F		
Default			0.0	optional	optional	0.1		

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
ELFORM	Element formulation options. EQ.49: Incompressible smoothed particle Galerkin formulation
DX, DY, DZ	Normalized dilation parameters of the kernel function in the x, y and z directions. The normalized dilation parameters of the kernel function are introduced to provide the smoothness and compact support properties on the construction of the mesh-free shape functions. Values between 1.4 and 1.8 are recommended. The nodal support size of particles are automatically adjusted with the material's deformation but cannot be decreased. An Updated Lagrangian (UL) kernel is used.
TSTART	Starting time for the fully implicit ISPG iterations. Before TSTART, only 10 ISPG iterations are done in each thermal-structural implicit step with one-way coupling (no forces from the fluid are returned to the structure) to guarantee that the fluid moves with the solid boundaries. After TSTART, the ISPG algorithm performs full iterations in the two-way coupled thermal-structural implicit analysis. This option is very useful for cases where the structural simulation time is very long (e.g., in seconds or minutes) while the reflow process to a steady state is very short. With this option, the full ISPG iterations start from TSTART, saving some computational resources.
DT_IMP	Reset the implicit structural time step size to DT_IMP after TSTART. Because the solder reflow process is very fast, a small implicit structural time step size is needed. Generally, the value of DT_IMP should be around 10 to 50 times the ISPG time step size to guarantee convergence of the solution if the gravity-driven simulation is deployed. This field is optional.
RCFORCH	Define the z coordinate of the cutting plane for evaluating the reaction forces between *all* the solders and the structure. If RCFORCH is nonzero, two files are generated in the working folders: fpdcpl_u.txt and fpdcpl_l.txt. The first file contains the reaction force history of the solders that are above the cutting plane, while the second file contains the reaction force history of those below the cutting plane.

VARIABLE	DESCRIPTION
DTSCL	The time step size scaling factor for ISPG iteration. We recommend a value between 0.1 and 0.5. Large DTSCL may cause contact detection issues.

*SECTION_IGA_SHELL

Purpose: Define section properties for isogeometric shell elements.

Card Summary:

Card Sets. For each isogeometric shell section include one set of data cards. This input ends at the next keyword ("*") card.

Card 1. This card is required.

SECID	ELFORM	SHRF	NIP	IRL	QR / IRID	ICOMP	

Card 2. This card is required.

T				NLOC			

Card 3. Include ceil(NIP/8) cards if ICOMP = 1.

B1	B2	B3	B4	B5	B6	B7	B8

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	SHRF	NIP	IRL	QR / IRID	ICOMP	
Type	I/A	I	F	F	I	F	I	
Default	none	0	1.0	2.0	0	0.0	0	

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
ELFORM	Element formulation: EQ.0: Reissner-Mindlin with fibers at the control points EQ.1: Kirchhoff-Love with fibers at the control points EQ.2: Kirchhoff-Love with fibers at the integration points EQ.3: Reissner-Mindlin with fibers at the integration points

VARIABLE	DESCRIPTION
	EQ.5: Thick shell with thickness stretch based on the ELFORM = 0. See Remark 1.
	EQ.6: Thick shell with thickness stretch based on ELFORM = 3. See Remark 1.
SHRF	Shear correction factor which scales the transverse shear stress; see Remark 2.
NIP	Number of through thickness integration points (up to 10 points). See Remark 3.
IRL	Lamina integration rule: EQ.0: Reduced Gauss-Legendre EQ.1: Gauss-Legendre EQ.2: Patchwise reduced Gauss-Legendre (for biquadratic NURBS only) EQ.3: Generalized Gaussian Quadrature rule. It only works with a uniform knot vector with UNIR/S = 1 or 2 in *IGA_2D_NURBS_XYZ.
QR/IRID	Fiber quadrature rule or fiber integration rule ID (see *INTEGRATION_SHELL): LT.0.0: Absolute value is specified rule number. EQ.0.0: Gauss-Legendre/Gauss-Lobatto (up to 10 points) EQ.1.0: Trapezoidal, not recommended for accuracy reasons
ICOMP	Flag for anisotropic layered composite material model (see Remark 4): EQ.1: A material angle in degrees is defined for each through thickness integration point. Thus, each layer has one integration point.

Card 2	1	2	3	4	5	6	7	8
Variable	T				NLOC			
Type	F				F			
Default	0.0				0.0			

VARIABLE	DESCRIPTION
T	Shell thickness
NLOC	Location of reference surface; see Remark 5.

Material Orientation Angle Cards. Include ceil(NIP/8) cards if ICOMP = 1 in order to define material orientation angle with respect to the baseline orientation at the fiber integration points.

Card 3	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
B*i*	Material angle at the i^{th} fiber integration point

Remarks:

1. **Thick Shell Formulations.** Element formulations 5 and 6 permit linear strains to develop in the thickness direction in a manner similar to the type 25 and 26 finite element shells (see *SECTION_SHELL). Thus solid materials with damage models that depend on the full stress tensor are compatible with these elements. For instance, these elements can be used with ductile fracture models that require accurate pressures for the nucleation and growth of voids.

2. **Shear Correction Factor.** Reissner-Mindlin shell formulations (FORM = 0 or 3) are based on a first order shear deformation theory that yields constant transverse shear strains. This approximation violates the condition of zero traction on the top and bottom surfaces of the shell. The shear correction factor is an

attempt to compensate for this error. A suggested value is 5/6 for isotropic materials. This value is incorrect for sandwich or laminated shells; consequently, laminated/sandwich shell theory is now an option in some of the constitutive models, that is, material types 22, 54, and 55.

3. **Fiber Integration Rule.** Either Gauss-Legendre (default) or Gauss-Lobatto integration can be used to integrate along the fiber. The flag for Gauss-Lobatto integration is set with INTGRD in *CONTROL_SHELL.

   If NIP is 0 or 1 and the *MAT_SIMPLIFIED_JOHNSON_COOK model is used, then a resultant plasticity formulation is activated. NIP is always set to 1 if a constitutive model based on resultants is used.

4. **Material Orientation.** This option applies to material types 21, 22, 23, 33, 33_96, 34, 36, 40, 41-50, 54, 55, 58, 59, 103, 103_P, 104, 108, 116, 122, 133, 135, 135_PLC, 136, 157, 158, 190, 219, 226, 233, 234, 235, 242, 243, 261, and 262. See also Remark 5 under *MAT_034 for additional information specific to fiber directions for fabrics.

5. **Shell Offset.** If NLOC $\neq 0$, the offset distance from the mid-surface to the reference surface of the shell in the direction of the shell normal vector is computed as

$$\text{offset} = -0.5 \times \text{NLOC} \times (\text{average shell thickness}).$$

   Except for Mortar contacts, this offset is not considered in the contact subroutines unless CNTCO is set to 1 or 3 in *CONTROL_SHELL.

*SECTION_IGA_SOLID

Purpose: Define section properties for isogeometric solid elements.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	IR					
Type	I/A	I	I					
Default	none	0	0					

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
ELFORM	Element formulation: EQ.0: Standard
IR	Integration rule: EQ.0: Reduced Gauss-Legendre EQ.1: Gauss-Legendre EQ.2: Patchwise reduced Gauss-Legendre (for biquadratic NURBS only) EQ.3: Generalized Gaussian Quadrature rule. It only works with a uniform knot vector with UNIR/S/T = 1 or 2 in *IGA_3D_NURBS_XYZ.

*SECTION_ISPG

Purpose: Set section properties for incompressible smoothed particle Galerkin (ISPG) elements.

> **NOTE:** This keyword may only be used in an ISPG input deck included with *INCLUDE_ISPG.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM						
Type	I/A	I						

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
ELFORM	Element formulation options. EQ.49: 3D and pseudo 2D ISPG formulation. For pseudo 2D simulations,*ISPG_BOUNDARY_SYMMETRY must specify the symmetric boundary condition. EQ.50: Planar 2D ISPG formulation. One layer of elements should be defined in the thickness direction. *ISPG_-BOUNDARY_SYMMETRY must set the symmetric boundary condition. EQ.51: Axisymmetric 2D ISPG formulation. Notice that (r,z) for the axisymmetric definition must be (x,z). Because remeshing is based on 3D remeshing, the model needs one layer of elements in the y-direction with the same thickness. *ISPG_BOUNDARY_SYMMETRY defines the symmetric boundary condition. The surface with CO-ORD1 defined in *ISPG_BOUNDARY_SYMMETRY gives the axisymmetric surface (see Figure 41-23).

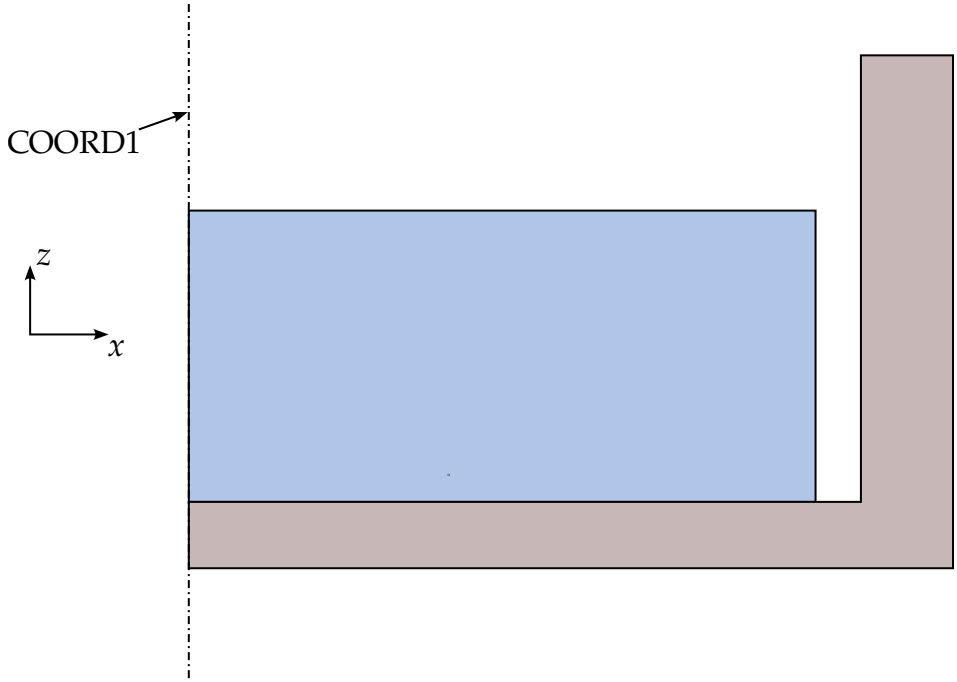


Figure 41-23. Example of an axisymmetric geometry

*SECTION_POINT_SOURCE

Purpose: This command provides the inlet boundary condition for single gas in flow (inflation potential) using a set of point source(s). It also provides the inflator orifice geometry information. It requires 3 curves defining the inlet condition for the inflator gas coming into the tank or an airbag as input ($\overline{T}_{\text{gas corrected}}(t)$, $v_r(t)$, and vel(t)). Please see also the *ALE_TANK_TEST card for additional information.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	LCIDT	LCIDVR	LCIDVEL	NIDLC1	NIDLC2	NIDLC3	
Type	I/A	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	

Source Node Cards. Include one card for each source node. This input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	NODEID	VECID	ORIFA					
Type	I	I	F					
Default	0	0	0.0					

VARIABLE	DESCRIPTION
SECID	Section ID. A unique number or label must be specified.
LCIDT	Temperature load curve ID
LCIDVR	Relative volume load curve ID
LCIDVEL	Inlet flow velocity load curve ID
NIDLC1	The 1st node ID defining a local coordinate (See Remark 2).
NIDLC2	The 2nd node ID defining a local coordinate (See Remark 2).

VARIABLE	DESCRIPTION
NIDLC3	The 3rd node ID defining a local coordinate (See Remark 2).
NODEID	The node ID(s) defining the point source(s).
VECID	The vector ID defining the direction of flow at each point source.
ORIFA	The orifice area at each point source.

Remarks:

1. **Airbag Inflator Tank Test.** In an airbag inflator tank test, the tank pressure data is measured. This pressure is used to derive $\dot{m}(t)$ and the estimated $\overline{T}_{\text{gas}}(t)$, usually using a lumped-parameter method, a system of conservation equations and EOS. Subsequently $\dot{m}(t)$ and $\overline{T}_{\text{gas}}(t)$ (stagnation temperature) are used as input to obtain $\overline{T}_{\text{gas\_corrected}}(t)$ (static temperature), $v_r(t)$, and vel(t). These 3 curves are then used to describe inflator gas inlet condition (see *ALE_TANK_TEST for more information).

2. **Local Coordinate System.** In a car crash model, the inflator housing may get displaced during the impact. The 3 node IDs defines the local reference coordinate system to which the point sources are attached. These 3 reference nodes may be located on a rigid body which can translate and rotate as the inflator moves during the impact. This allows for the point sources to move in time. These reference nodes may be used as the point sources themselves.

3. **ALE Tank Test Keyword.** If the *ALE_TANK_TEST card is present, please see the Remarks under that card.

Example:

Consider a tank test model which consists of the inflator gas (PID 1) and the air inside the tank (PID 2). The 3 load curves define the thermodynamic and kinetic condition of the incoming gas. The nodes define the center of the orifice, and the vector the direction of flow at each orifice.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
*PART
inflator gas
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         1         1         1         0         0         0         0         0
*SECTION_POINT_SOURCE
$    SECID     LCIDT  LCIDVOLR   LCIDVEL NIDLCOOR1 NIDLCOOR2 NIDLCOOR3
         1         3         4         5         0         0         0
$   NODEID    VECTID      AREA
     24485         3    15.066
```

```
       ...
     24557         3    15.066
*PART
air inside the tank
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         2         2         2         0         0         0         0         0 *SEC-
TION_SOLID
$    SECID    ELFORM       AET
         2        11         0
*ALE_MULTI-MATERIAL_GROUP
$      SID   SIDTYPE
         1         1
         2         1
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
```

*SECTION_POINT_SOURCE_MIXTURE

Purpose: This command provides (a) an element formulation for a solid ALE part of the type similar to ELFORM = 11 of *SECTION_SOLID, and (b) the inlet gas injection boundary condition for multiple-gas mixture in-flow using a set of point source(s). It also provides the inflator orifice geometry information. This must be used in combination with the *MAT_GAS_MIXTURE and/or *INITIAL_GAS_MIXTURE card (see Remark 1).

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	LCIDT	Not Used	LCIDVEL	NIDLC1	NIDLC2	NIDLC3	IDIR
Type	I/A	I		I	I	I	I	I
Default	none	none		0	none	none	none	0

Card 2	1	2	3	4	5	6	7	8
Variable	LCMD1	LCMD2	LCMD3	LCMD4	LCMD5	LCMD6	LCMD7	LCMD8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Source Node Cards. Include one card for each source node. This input ends at the next keyword ("*") card.

Card 3	1	2	3	4	5	6	7	8
Variable	NODEID	VECID	ORIFA					
Type	I	I	F					
Default	none	none	0.0					

VARIABLE	DESCRIPTION
SECID	Section ID. A unique number or label must be specified.

VARIABLE	DESCRIPTION
LCIDT	Inflator gas mixture average stagnation temperature load curve ID (all gases of the mixture are assumed to have the same average temperature).
LCIDVEL	User-defined inflator gas mixture average velocity load curve ID. If LCIDVEL = 0 or blank, LSDYNA will estimate the inlet gas velocity.
NIDLC001	The 1st node ID defining a local coordinate (see Remark 2).
NIDLC002	The 2nd node ID defining a local coordinate (see Remark 2).
NIDLC003	The 3rd node ID defining a local coordinate (see Remark 2).
IDIR	A flag for constraining the nodal velocity of the nodes of the ALE element containing a point source. EQ.0: ALE nodes behind the point source (relative position of nodes based on the vector direction of flow of point source) will have zero velocity (default). EQ.1: all ALE nodes will have velocity distributed based on energy conservation. This option seems to be more robust in airbag modeling (see Remark 5).
LCMD*n*	The mass flow rate load curve ID of the n^{th} gas in the mixture. See Remark 3.
NODEID	The node ID(s) defining the point sources (see Remark 7).
VECID	The vector ID defining the direction of flow at each point source.
ORIFA	The orifice area at each point source.

Remarks:

1. **Keyword Applications.** This command is used to define a part that acts as the ideal gas mixture injection source. The associated ALE material (gas mixture) may not be present at time zero but can be introduced (injected) into an existing ALE domain. For airbag applications, the input from control volume analysis, inlet mass flow rate, $\dot{m}(t)$, and, inlet stagnation gas temperature, $\overline{T}_{\text{gas}}(t)$ may be used as direct input for ALE analysis. If available, the user may input a load curve for the gas mixture average inlet velocity. If not, LS-DYNA will estimate the inlet gas velocity.

2. **Local Coordinate System.** In a car crash model, the inflator housing may get displaced during the impact. The 3 node IDs defines the local reference coordinate system to which the point sources are attached. These 3 reference nodes may be located on a rigid body which can translate and rotate as the inflator moves during the impact. This allows for the point sources to move in time. These reference nodes may be used as the point sources themselves.

3. **Mass Flow Rate.** The gas mixture is assumed to have a uniform temperature ($\overline{T} \approx T_i$) and inlet velocity. However, the species in the mixture may each have a different inlet mass flow rate.

4. **Model.** A brief review of the concept used is presented. The total energy, e_t, is the sum of internal (e_i) and kinetic ($V^2/2$) energies (per unit mass).

$$e_T = e_i + \frac{V^2}{2}$$
$$C_V T_{stag} = C_V T + \frac{V^2}{2}$$
$$T_{stag} = T + \frac{V^2}{2C_V}$$

The distinction between stagnation and static temperatures is shown above. C_V is the constant-volume heat capacity. The gas mixture average internal energy per unit mass in terms of mixture species contribution is

$$e_i = \bar{C}_V \overline{T} = \sum_i \left(\frac{\rho_i}{\rho_{mixture}}\right) C_{V_i} T_i = \left[\sum_i \left(\frac{\rho_i}{\rho_{mixture}}\right) C_{V_i}\right] \overline{T}$$
$$\bar{C}_V = \left[\sum_i \left(\frac{\rho_i}{\rho_{mixture}}\right) C_{V_i}\right]$$

Since we approximate $\overline{T} \approx T_i$, then gas mixture average static temperature is related to the mixture average internal energy per unit mass as following

$$\overline{T} = \frac{e_i}{\left[\sum_i \left(\frac{\rho_i}{\rho_{mixture}}\right) C_{V_i}\right]}$$

Note that the "i" subscript under "e" denotes "internal" energy, while the other "i" subscripts denote the "i^{th}" species in the gas mixture. The total mixture pressure is the sum of the partial pressures of the individual species.

$$\bar{p} = \sum_i p_i$$

The ideal gas EOS applies to each individual species (by default)

$$P_i = \rho_i (C_{P_i} - C_{V_i}) T_i$$

5. **Energy Conservation.** Generally, conservation of both momentum and kinetic (KE) at the same time is impossible. Typically, internal energy (IE) is conserved

and KE may not be. This may result in some KE loss (hence, total energy loss). For many analyses this is tolerable, but for an airbag, a reduction of the inflating potential of the inflator gas may result.

In *MAT_GAS_MIXTURE computation, any kinetic energy not accounted for during advection is stored in the internal energy. Therefore, no kinetic energy is lost, and the total energy of the element is conserved over the advection step. This simple, ad hoc approach is not rigorously derived for the whole system based on first principles. Therefore, it is not guaranteed to apply universally to all scenarios. The user must validate the model with data.

6. **EOS.** Since ideal gas is assumed, there is no need to define the EOS for the gases in the mixture.

7. **Point Source Location.** In general, it is best to locate a point source near the center of an ALE element. Associated with each point source is an area and a vector indicating flow direction. Each point source should occupy one ALE element by itself, and there should be at least two empty ALE elements between any two point sources. A point source should be located at least three elements away from the free surface of an ALE mesh for stability.

Example 1:

Consider a tank test model without coupling which consists of:

-a background mesh with air (PID 1 = gas 1) initially inside that mesh (tank space), and
-the inflator gas mixture (PID 2 consisting of inflator gases 2, 3, and 4).

The mixture is represented by one AMMGID and the air by another AMMGID.

The tank internal space is simply modeled with an Eulerian mesh of the same volume. The tank itself is not modeled thus no coupling is required. The inflator gases fill up this space mixing with the air initially inside the tank.

The background air (gas 1) is included in the gas mixture definition in this case because that air will participate in the mixing process. Only include in the mixture those gases that actually undergo mixing (gases 1, 2, 3 and 4). Note that for an airbag model, the "outside" air should not be included in the mixture (it should be defined independently) since it does not participate in the mixing inside the airbag. This is shown in the next example.

The nodes define the center of the orifices, and the vectors define the directions of flow at these orifices.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*PART
Tank background mesh, initially filled with air, allows gas mixture to flow in.
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         1         1         1         0         0         0         0         0
*SECTION_SOLID
$    SECID    ELFORM       AET
         1        11         0
$ The next card defines the properties of the gas species in the mixture.
*MAT_GAS_MIXTURE
$      MID
         1
$      Cv1       Cv2       Cv3       Cv4       Cv5       Cv6       Cv7       Cv8
    654.47    482.00   2038.30    774.64       0.0       0.0       0.0       0.0
$      Cp1       Cp2       Cp3       Cp4       Cp5       Cp6       Cp7       Cp8
    941.32    666.67   2500.00   1071.40       0.0       0.0       0.0       0.0

$ The next card specifies that gas 1 (background air) occupies PID 1 at time 0.
*INTIAL_GAS_MIXTURE
$      SID     STYPE    AMMGID     TEMP0
         1         1         1    293.00
$     RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7      RHO8
   1.20E-9       0.0       0.0       0.0       0.0       0.0       0.0       0.0
*PART
The gas mixture (inlet) definition (no initial mesh required for this PID)
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         2         2         1         0         0         0         0         0
*SECTION_POINT_SOURCE_MIXTURE
$    SECID     LCIDT   NOTUSED   LCIDVEL NIDLCOOR1 NIDLCOOR2 NIDLCOOR3      IDIR
         2         1         0         5         0         0         0         0
$  LCMDOT1   LCMDOT2   LCMDOT3   LCMDOT4   LCMDOT5   LCMDOT6   LCMDOT7   LCMDOT8
         0         2         3         4         0         0         0         0
$   NODEID    VECTID      AREA
     24485         1      25.0
       ...
     24557         1      25.0
*ALE_MULTI-MATERIAL_GROUP
$      SID   SIDTYPE
         1         1
         2         1
*DEFINE_VECTOR
$   VECTID     XTAIL     YTAIL     ZTAIL     XHEAD     YHEAD     ZHEAD
         1       0.0       0.0       0.0       0.0       1.0       0.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

Example 2:

Consider an airbag inflation model which consists of:

-a background Eulerian mesh for air initially outside the airbag (PID 1)
-the inflator gas mixture (PID 2 consisting of inflator gases 1, 2, and 3).

The mixture is represented by one AMMGID and the air by another AMMGID.

The background air (PID 1) is NOT included in the gas mixture definition in this case because that air will NOT participate in the mixing process. Only include in the mixture those gases that actually undergo mixing (gases 1, 2, and 3). Gases 1, 2, and 3 in this example correspond to gases 2, 3, and 4 in example 1. Compare the air properties in PID

1 here to that of example 1. Note that the *INITIAL_GAS_MIXTURE card is not required to initialize the background mesh in this case.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*PART
Tank background mesh, initially filled with air, allows gas mixture to flow in.
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         1         1         1         0         0         0         0         0
*SECTION_SOLID
$    SECID    ELFORM       AET
         1        11         0
*MAT_NULL
$      MID       RHO      PCUT        MU     TEROD     CEROD        YM        PR
         1   1.20E-9   -1.0E-6       0.0       0.0       0.0       0.0       0.0
*EOS_IDEAL_GAS
$    EOSID       CV0       CP0     COEF1     COEF2        T0    RELVOL0
         1    654.47    941.32       0.0       0.0    293.00       1.0
$ The next card defines the properties of the gas species in the mixture.
*PART
The gas mixture (inlet) definition (no initial mesh required for this PID)
$      PID     SECID       MID     EOSID      HGID      GRAV    ADPOPT      TMID
         2         2         2         0         0         0         0         0 *SEC-
TION_POINT_SOURCE_MIXTURE
$    SECID     LCIDT   NOTUSED   LCIDVEL NIDLCOOR1 NIDLCOOR2 NIDLCOOR3      IDIR
         2         1         0         5         0         0         0         0
$  LCMDOT1   LCMDOT2   LCMDOT3   LCMDOT4   LCMDOT5   LCMDOT6   LCMDOT7   LCMDOT8
         2         3         4         0         0         0         0         0
$   NODEID    VECTID      AREA
     24485         1      25.0
       ...
     24557         1      25.0
*MAT_GAS_MIXTURE
$      MID
         2
$       Cv1       Cv2       Cv3       Cv4       Cv5       Cv6       Cv7       Cv8
    482.00   2038.30    774.64       0.0       0.0       0.0       0.0
$       Cp1       Cp2       Cp3       Cp4       Cp5       Cp6       Cp7       Cp8
    666.67   2500.00   1071.40       0.0       0.0       0.0       0.0
$ The next card specifies that gas 1 (background air) occupies PID 1 at time 0.
*ALE_MULTI-MATERIAL_GROUP
$      SID   SIDTYPE
         1         1
         2         1
*DEFINE_VECTOR
$   VECTID     XTAIL     YTAIL     ZTAIL     XHEAD     YHEAD     ZHEAD
         1       0.0       0.0       0.0       0.0       1.0       0.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

*SECTION_SEATBELT

Purpose: Define section properties for the seat belt elements. This card is required for the *PART Section. Currently, only the ID is required.

Seatbelt Section Cards. Include one card for each SEATBELT section. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	AREA	THICK					
Type	I/A	F	F					
Default	none	0.01	↓					

VARIABLE	DESCRIPTION
SECID	Section ID. A unique number or label must be specified.
AREA	Optional area of cross-section used in the calculation of contact stiffness, which is proportional to the cross-section area.
THICK	Optional contact thickness which can be overwritten by a nonzero SAST defined in *CONTACT. If not defined, a value proportional to element length is used as the contact thickness.

Remarks:

Seatbelt elements are implemented for both explicit and implicit calculations.

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *SECTION_SEATBELT
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a seat belt section that is referenced by part 10.  Nothing
$  more than the sid is required.
$
*SECTION_SEATBELT
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid
       111
$
$
*PART
```

```
Seatbelt material
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid       sid       mid     eosid      hgid    adpopt
        10       111       220
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

***SECTION_SHELL**_*{OPTION}*

Available options include:

<BLANK>

EFG

THERMAL

XFEM

MISC

Purpose: Define section properties for shell elements.

Card Summary:

Card Sets. For each shell section, of a type matching the keyword's options, include one set of data cards. This input ends at the next keyword ("*") card.

Card 1. This card is required.

SECID	ELFORM	SHRF	NIP	PROPT	QR / IRID	ICOMP	SETYP

Card 2. This card is required.

T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET

Card 3. Additional cards for ICOMP = 1. Include the minimum number of cards necessary to input NIP values: 8 values per card ⇒ number of cards = ceil(NIP/8) where ceil(x) = the smallest integer greater than x.

B1	B2	B3	B4	B5	B6	B7	B8

Card 4a. Include this card if using the EFG keyword option.

DX	DY	ISPLINE	IDILA	IEBT	IDIM		

Card 4b. Include this card if using the THERMAL keyword option.

ITHELFM							

Card 4c. Include this card if using the XFEM keyword option.

CMID	BASELM	DOMINT	FAILCR	PROPCR	FS	LS/FS1	NC/CL

Card 4d. Include this card if using the MISC keyword option.

THKSCL							

Card 5. Additional card for ELFORM = 101, 102, 103, 104 or 105.

NIPP	NXDOF	IUNF	IHGF	ITAJ	LMC	NHSV	ILOC

Card 5.1. Additional card for ELFORM = 101, 102, 103, 104 or 105. Define NIPP of this card.

XI	ETA	WGT					

Card 5.2. Additional card for ELFORM = 101, 102, 103, 104 or 105. Include the minimum number of cards necessary to input LMC values: 8 values per card ⇒ number of cards = ceil(LMC/8) where ceil(x) = the smallest integer greater than x.

P1	P2	P3	P4	P5	P6	P7	P8

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	SHRF	NIP	PROPT	QR / IRID	ICOMP	SETYP
Type	I/A	I	F	F	F	F	I	I
Default	none	none	1.0	2	0.0	0.0	0	1

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
ELFORM	Element formulation options (see Remarks 1 and 3 below): EQ.1: Hughes-Liu EQ.2: Belytschko-Tsay EQ.3: BCIZ triangular shell EQ.4: C0 triangular shell EQ.5: Belytschko-Tsay membrane

VARIABLE	DESCRIPTION
	EQ.6: Selectively reduced Hughes-Liu
	EQ.7: Selectively reduced, co-rotational Hughes-Liu
	EQ.8: Belytschko-Leviathan shell
	EQ.9: Fully integrated Belytschko-Tsay membrane
	EQ.10: Belytschko-Wong-Chiang
	EQ.11: Fast (co-rotational) Hughes-Liu
	EQ.12: Plane stress (*xy*-plane)
	EQ.13: Plane strain (*xy*-plane)
	EQ.14: Axisymmetric solid (*xy*-plane, *y*-axis of symmetry) - area weighted (see Remark 11)
	EQ.15: Axisymmetric solid (*xy*-plane, *y*-axis of symmetry) - volume weighted
	EQ.16: Fully integrated shell element (very fast). See Remark 14.
	EQ.-16: Fully integrated shell element modified for higher accuracy. See Remark 15.
	EQ.17: Fully integrated DKT, triangular shell element. See Remark 10.
	EQ.18: Fully integrated linear DK quadrilateral/triangular shell. See Remarks 2 and 3.
	EQ.20: Fully integrated linear assumed strain C0 shell. See Remark 3.
	EQ.21: Fully integrated linear assumed strain C0 shell (5 DOF)
	EQ.22: Linear shear panel element. 3 DOF per node. See Remark 4.
	EQ.23: 8-node quadratic quadrilateral shell (see IRQUAD in *CONTROL_SHELL)
	EQ.24: 6-node quadratic triangular shell
	EQ.25: Belytschko-Tsay shell with thickness stretch
	EQ.26: Fully integrated shell with thickness stretch
	EQ.27: C0 triangular shell with thickness stretch
	EQ.29: Cohesive shell element for edge-to-edge connection of shells. See Remark 13.

VARIABLE	DESCRIPTION
EQ.-29:	Cohesive shell element for edge-to-edge connection of shells (more suitable for pure shear). See Remark 13.
EQ.30:	Fast fully integrated element with 2 in-plane integration points based on ELFORM 16 (see Remark 14)
EQ.41:	Mesh-free (EFG) shell local approach (more suitable for crashworthiness analysis).
EQ.42:	Mesh-free (EFG) shell global approach (more suitable for metal forming analysis).
EQ.43:	Mesh-free (EFG) plane strain formulation (xy-plane)
EQ.44:	Mesh-free (EFG) axisymmetric solid formulation (xy-plane, y-axis of symmetry)
EQ.46:	Cohesive element for two-dimensional plane strain, plane stress, and area-weighted axisymmetric problems (use with type 14 shells). See Remark 17.
EQ.47:	Cohesive element for two-dimensional volume-weighted axisymmetric problems (use with type 15 shells). See Remark 17.
EQ.52:	Plane strain (xy-plane) XFEM, base element type 13 with full integration. See Remark 9.
EQ.54:	Shell XFEM, base element type defined by BASELM (default 2). See Remark 9.
EQ.55:	8-node singular plane strain (xy-plane) finite element. See Remark 12.
EQ.98:	Interpolation shell
EQ.99:	Simplified linear element for time-domain vibration studies. See Remark 5.
EQ.101:	User defined shell
EQ.102:	User defined shell
EQ.103:	User defined shell
EQ.104:	User defined shell
EQ.105:	User defined shell
EQ.201:	Isogeometric shells with NURBS. See *ELEMENT_SHELL_NURBS_PATCH.
GE.1000:	Generalized shell element formulation (user defined). See *DEFINE_ELEMENT_GENERALIZE_SHELL.

VARIABLE	DESCRIPTION
	Note that the 2D and 3D element types must not be mixed, and different types of 2D elements must not be used together. For example, two-dimensional axisymmetric calculations can use either element types 14 or 15, but these element types must not be mixed together. Likewise, the plane strain element type must not be used with either the plane stress element or the axisymmetric element types. In three dimensions, the different shell element types, i.e., 1-11 and 16, can be freely mixed together.
SHRF	Shear correction factor which scales the transverse shear stress. The shell formulations in LS-DYNA, with the exception of the BCIZ and DK elements, are based on a first order shear deformation theory that yields constant transverse shear strains which violates the condition of zero traction on the top and bottom surfaces of the shell. The shear correction factor is an attempt to compensate for this error. A suggested value is 5/6 for isotropic materials. This value is incorrect for sandwich or laminated shells. Consequently, laminated/sandwich shell theory is now an option in some of the constitutive models, that is, material types 22, 54, and 55.
NIP	Number of through thickness integration points. Either Gauss (default) or Lobatto integration can be used. INTGRD in *CONTROL_SHELL specifies the integration rule. The tables Gaussian Quadrature Points and Lobatto Quadrature Points below tabulate the location of the Gauss and Lobatto integration points. EQ.0.0: Set to 2 integration points for shell elements. EQ.1.0: 1 point (no bending) EQ.2.0: 2 points EQ.3.0: 3 points EQ.4.0: 4 points EQ.5.0: 5 points EQ.6.0: 6 points EQ.7.0: 7 points EQ.8.0: 8 points EQ.9.0: 9 points EQ.10.0: 10 points GT.10.0: Trapezoidal or user defined rule

VARIABLE	DESCRIPTION
	Through thickness integration for two-dimensional elements (options 12-15, 43, 44, 52 and 55 above) and cohesive shells (option ±29) is not meaningful; consequently, the default is 1 integration point. Fully integrated two-dimensional elements are available for options 13 and 15 (but not 12 and 14) by setting NIP equal to a value of 4, corresponding to a 2 by 2 Gaussian quadrature. For element type 55, NIP can be 4, 9 or 16. If NIP is 0 or 1 and the *MAT_SIMPLIFIED_JOHNSON_COOK model is used, then a resultant plasticity formulation is activated. NIP is always set to 1 if a constitutive model based on resultants is used.
PROPT	Printout option (***NOT ACTIVE***): EQ.1.0: Average resultants and fiber lengths EQ.2.0: Resultants at plan points and fiber lengths EQ.3.0: Resultants, stresses at all points, fiber lengths
QR/IRID	Quadrature rule or integration rule ID, see *INTEGRATION_SHELL: LT.0.0: Absolute value is specified rule number. EQ.0.0: Gauss/Lobatto (up to 10 points are permitted) EQ.1.0: Trapezoidal, not recommended for accuracy reasons
ICOMP	Flag for orthotropic/anisotropic layered composite material model. This option applies to material types 21, 22, 23, 33, 33_96, 34, 36, 40, 41-50, 54, 55, 58, 59, 103, 103_P, 104, 108, 116, 122, 133, 135, 135_PLC, 136, 157, 158, 190, 219, 226, 233, 234, 235, 242, and 243. For these material types, see *PART_COMPOSITE as an alternative to *SECTION_SHELL. Note: Please refer to *Remark 5* under *MAT_034 for additional information specific to fiber directions for fabrics. EQ.1: A material angle in degrees is defined for each through thickness integration point. Thus, each layer has one integration point.
SETYP	Not used (obsolete)

Card 2	1	2	3	4	5	6	7	8
Variable	T1	T2	T3	T4	NLOC	MAREA	IDOF	EDGSET
Type	F	F	F	F	F	F	F	I
Default	0.0	T1	T1	T1	0.0	0.0	0.0	↓

VARIABLE	DESCRIPTION
T1	Shell thickness at node n_1, unless the thickness is defined on the *ELEMENT_SHELL_*OPTION* card.
T2	Shell thickness at node n_2, see comment for T1 above.
T3	Shell thickness at node n_3, see comment for T1 above.
T4	Shell thickness at node n_4, see comment for T1 above.
NLOC	Location of reference surface (shell mid-thickness) for three-dimensional shell elements. If nonzero, the offset distance from the plane of the nodal points to the reference surface of the shell in the direction of the shell normal vector is a value, $\text{offset} = -0.50 \times \text{NLOC} \times (\text{average shell thickness})$. Alternatively, the offset can be specified with the OFFSET keyword option of *ELEMENT_SHELL. Except for Mortar contacts, the contact subroutines do not consider this offset unless CNTCO = 1 or 3 in *CONTROL_SHELL. For Mortar contacts, NLOC or OFFSET determines the location of the contact surface regardless of the value of CNTCO. EQ.1.0: Nodes are located at top surface of shell. EQ.0.0: Nodes are located at mid-thickness of shell (default). EQ.-1.0: Nodes are located at bottom surface of shell. For nonzero offset distances, the time step size is reduced to prevent instabilities. See NLOCDT in *CONTROL_SHELL.
MAREA	Non-structural mass per unit area. This is additional mass which comes from materials, such as carpeting. This mass is not directly included in the time step calculation. An often more convenient alternative for defining distributed mass is with *ELEMENT_MASS_PART, which allows additional non-structural mass to be

VARIABLE	DESCRIPTION
	distributed by an area weighted distribution to all nodes of a given part ID.
IDOF	Treatment of through thickness strain (see Remark 7).
	LT.0: Same as IDOF = 3 but the contact pressure is averaged over a time –IDOF to reduce noise and thus improve stability.
	EQ.1: The thickness field is continuous across the element edges for metal forming applications. This option applies to element types 25, 26 and 27.
	EQ.2: The thickness field is discontinuous across the element edges. This is necessary for crashworthiness simulations due to shell intersections, sharp included angles, and non-smooth deformations. This option applies to and is the default for element types 25, 26 and 27.
	EQ.3: The thickness strain is governed by stress from contact and pressure loads, meaning that the strain is adjusted for the through thickness stress to equilibrate the contact and load pressure. Please note that the pressure is assumed positive, meaning acting towards the shell surface. Thus, vacuum loads cannot be used with this option. This option applies to element types 2, 4, and ±16.
	EQ.11: Same as IDOF = 1 but the through thickness strain is simplified in the kinematics in order to reduce locking effects which were observed for considerable thickness changes. This option applies to element type 25 only.
	EQ.12: Same as IDOF = 2 but the through thickness strain is simplified in the kinematics in order to reduce locking effects which were observed for considerable thickness changes. This option applies to element type 25 only.
EDGSET	Edge node set required for shell type seatbelts. See Remark 8. See Figure 19-17 in *ELEMENT_SEATBELT for additional clarification.

Angle Cards. Additional cards for ICOMP = 1. Include the minimum number of cards necessary to input NIP values: 8 values per card ⇒ number of cards = ceil(NIP/8) where ceil(x) = the smallest integer greater than x.

Card 3	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
Bi	β_i, material angle at the i^{th} integration point

EFG Card. Additional card for EFG keyword option. See *CONTROL_EFG.

Card 4a	1	2	3	4	5	6	7	8
Variable	DX	DY	ISPLINE	IDILA	IEBT	IDIM		
Type	F	F	I	I	I	I		
Default	1.1	1.1	0	0	-1 or 1	2 or 1		

VARIABLE	DESCRIPTION
DX, DY	Normalized dilation parameters of the kernel function in X and Y directions. The normalized dilation parameters of the kernel function are introduced to provide the smoothness and compact support properties on the construction of the mesh-free shape functions. Values between 1.0 and 2.0 are recommended. Values smaller than 1.0 are not allowed. Larger values will increase the computation time and will sometimes result in a divergence problem.
ISPLINE	Replace the choice for the EFG kernel functions definition in *CONTROL_EFG which permits defining different ISPLINE in different sections.
IDILA	Replace the choice for the normalized dilation parameter definition in *CONTROL_EFG which permits defining different IDILA in different sections.

VARIABLE	DESCRIPTION
IEBT	Essential boundary condition treatment EQ.1: Full transformation (default for ELFORM = 42) EQ.-1: Without full transformation (default for ELFORM = 41) EQ.3: Coupled FEM/EFG EQ.7: Maximum entropy approximation
IDIM	For mesh-free shell local approach (ELFORM = 41) EQ.1: First-kind local boundary condition method EQ.2: Gauss integration (default) For mesh-free shell global approach (ELFORM = 42) EQ.1: First-kind local boundary condition method (default) EQ.2: Second-kind local boundary condition method

Thermal Card. Additional Card for THERMAL keyword option.

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

VARIABLE	DESCRIPTION
ITHELFM	Thermal shell formulation EQ.0: Default is governed by THSHEL on *CONTROL_SHELL EQ.1: Thick thermal shell EQ.2: Thin thermal shell

XFEM Card. Additional card for XFEM keyword option. See Remark 9.

Card 4c	1	2	3	4	5	6	7	8
Variable	CMID	BASELM	DOMINT	FAILCR	PROPCR	FS	LS/FS1	NC/CL
Type	I	I	I	I	I	F	F	I/F
Default	none	none	0	1	0	0.0	0.0	none

VARIABLE	DESCRIPTION
CMID	Cohesive material ID. *MAT_185 is available for both brittle and ductile fracture, and *MAT_240 is available for ductile material. See Remark 18.
BASELM	Base element type for XFEM (type 13 for 2D, types 2, 16 or 17 for shell)
DOMINT	Option for domain integration in XFEM: EQ.0: Phantom element integration (default) EQ.1: Subdomain integration with triangular local boundary integration (available in 2D only)
FAILCR	Option for different failure criteria: EQ.0: Failure criterion from *MAT_ADD_EROSION / GISSMO model EQ.1: Maximum tensile stress (failure value given in cohesive law) EQ.2: Maximum shear stress (failure value given in cohesive law) EQ.-1: Effective plastic strain (EPS) EQ.-2: Crack length dependent EPS. See Remark 16. EQ.-4: Stress triaxiality based failure plastic strain
PROPCR	PROPCR is interpreted digit-wise: $\text{PROPCR} = [IP] = P + 10 \times I$ P determines the crack propagation direction: P.EQ.0: First principal strain direction if FAILCR < 0 (default for ductile fracture), first principal stress if FAILCR = 1, or

VARIABLE	DESCRIPTION
	maximum shear stress if FAILCR = 2. P = 0 is the only option for brittle fracture (FAILCR > 0). P.EQ.2: Center of effective plastic strain P.EQ.3: Directional center of effective plastic strain *I* determines crack initiation: I.EQ.0: Crack initiates at boundary (default) I.EQ.3: Crack initiates anywhere
FS	Failure value for FAILCR = -1 or -2: FAILCR.EQ.-1: Failure strain/failure critical value FAILCR.EQ.-2: Initial failure plastic strain Curve ID or Table ID for stress triaxiality based failure plastic strain for FAILCR = -4.
LS	Length scale for strain regularization (FAILCR = -1 only)
FS1	Final failure plastic strain (FAILCR = -2 only)
NC	Number of cracks allowed in the part (FAILCR ≠ -2)
CL	Crack length at which the failure strain is FS1 (FAILCR = -2 only)

Miscellaneous Settings Card. Additional card for MISC keyword option.

Card 4d	1	2	3	4	5	6	7	8
Variable	THKSCL							
Type	F							
Default	1.0							

VARIABLE	DESCRIPTION
THKSCL	Thickness scale factor. Shell thicknesses for all elements of this section including those with thickness specified using *ELEMENT_SHELL_THICKNESS are scaled by THKSCL.

User Defined Element Card. Additional card for ELFORM = 101,102,103,104 or 105. See Appendix C

Card 5	1	2	3	4	5	6	7	8
Variable	NIPP	NXDOF	IUNF	IHGF	ITAJ	LMC	NHSV	ILOC
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

User Defined Element Integration Point Cards. Additional cards for ELFORM = 101, 102, 103, 104 or 105. Define NIPP cards according to the following format. See Appendix C.

Card 5.1	1	2	3	4	5	6	7	8
Variable	XI	ETA	WGT					
Type	F	F	F					
Default	none	none	none					

User Defined Element Property Cards. Include the minimum number of cards necessary to input LMC values: 8 values per card $\Rightarrow$ number of cards = ceil(LMC/8) where ceil(x) = the smallest integer greater than x. See Appendix C.

Card 5.2	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

VARIABLE	DESCRIPTION
NIPP	Number of in-plane integration points for user-defined shell (0 if resultant/discrete element)

VARIABLE	DESCRIPTION
NXDOF	Number of extra degrees of freedom per node for user-defined shell
IUNF	Flag for using nodal fiber vectors in user-defined shell: EQ.0: Nodal fiber vectors are not used. EQ.1: Nodal fiber vectors are used.
IHGF	Flag for using hourglass stabilization (NIPP is greater than 0) EQ.0: Hourglass stabilization is not used. EQ.1: LS-DYNA hourglass stabilization is used. EQ.2: User-defined hourglass stabilization is used. EQ.3: Same as 2, but the resultant material tangent moduli are passed
ITAJ	Flag for setting up finite element matrices (NIPP is greater than 0) EQ.0: Set up matrices with respect to isoparametric domain EQ.1: Set up matrices with respect to physical domain
LMC	Number of property parameters
NHSV	Number of history variables
ILOC	Coordinate system option: EQ.0: Pass all variables in LS-DYNA local coordinate system EQ.1: Pass all variables in global coordinate system
XI	First isoparametric coordinate
ETA	Second isoparametric coordinate
WGT	Isoparametric weight
P*i*	i^{th} user defined element property

Gaussian Quadrature Points

Point	1 Point	2 Points	3 Points	4 Points	5 Points
#1	.0	-.5773503	.0	-.8611363	.0
#2		+.5773503	-.7745967	-.3399810	-.9061798
#3			+.7745967	+.3399810	-.5384693
#4				+.8622363	+.5384693
#5					+.9061798
Point	6 Points	7 Points	8 Points	9 Points	10 Points
#1	-.9324695	-.9491080	-.9602899	-.9681602	-.9739066
#2	-.6612094	-.7415312	-.7966665	-.8360311	-.8650634
#3	-.2386192	-.4058452	-.5255324	-.6133714	-.6794096
#4	+.2386192	.0	-.1834346	-.3242534	-.4333954
#5	+.6612094	+.4058452	+.1834346	.0	-.1488743
#6	+.9324695	+.7415312	+.5255324	+.3242534	+.1488743
#7		+.9491080	+.7966665	+.6133714	+.4333954
#8			+.9602899	+.8360311	+.6794096
#9				+.9681602	+.8650634
#10					+.9739066

Location of through thickness Gauss integration points. The coordinate is referenced to the shell midsurface at location 0. The inner surface of the shell is at -1 and the outer surface is at +1.

Lobatto Quadrature Points

Point	1 Point	2 Points	3 Points	4 Points	5 Points
#1			0.0	-1.0	0.0
#2			-1.0	-0.4472136	-1.0
#3			+1.0	+0.4472136	-0.6546537
#4				+1.0	+0.6546537
#5					+1.0
Point	6 Points	7 Points	8 Points	9 Points	10 Points
#1	-1.0	-1.0	-1.0	-1.0	-1.0
#2	-0.7650553	-0.8302239	-0.8717401	-0.8997580	-0.9195339
#3	-0.2852315	-0.4688488	-0.5917002	-0.6771863	-0.7387739
#4	+0.2852315	0.0	-0.2092992	-0.3631175	-0.4779249
#5	+0.7650553	+0.4688488	+0.2092992	0.0	-0.1652790
#6	+1.0	+0.8302239	+0.5917002	+0.3631175	+0.1652790
#7		+1.0	+0.8717401	+0.6771863	+0.4779249
#8			+1.0	+0.8997580	+0.7387739
#9				+1.0	+0.9195339
#10					+1.0

Location of through thickness Lobatto integration points. The coordinate is referenced to the shell midsurface at location 0. The inner surface of the shell is at -1 and the outer surface is at +1.

Remarks:

1. **Formulations.** The default shell formulation is 2 unless overridden by THEORY in *CONTROL_SHELL. ELFORM in *SECTION_SHELL overrides THEORY.

   For implicit calculations the following element formulations are implemented: 2, 5, 6, 10, 12,12, 13, 14, 15, 16, -16, 17, 18, 20, 21, 22, 23, 24, 25, 26, 27, 29, 41, 42, 55.

   If another element formulation is requested for an implicit analysis, LS-DYNA will substitute one of the above in place of the one chosen.

2. **Problem types for the type 18 element.** The type 18 element is only for linear static and normal modes. It can also be used for linear springback in sheet metal stamping.

3. **Description of linear element types 18 and 20.** The linear elements consist of an assembly of membrane and plate elements. They have six degrees of freedom per node and can, therefore, be connected to beams or used in complex shell surface intersections. These elements possess the required zero energy rigid body modes and have exact constant strain and curvature representation, that is, they pass all the first order patch tests. In addition, the elements have behavior approaching linear bending (cubic displacement) in the plate-bending configuration.

   a) The membrane component is based on an 8-node/6-node isoparametric mother element which incorporates nodal in-plane rotations through cubic displacement constraints of the sides [Taylor 1987; Wilson 2000].

   b) The plate component of element 18 is based on the Discrete Kirchhoff Quadrilateral (DKQ) [Batoz 1982]. Because the Kirchhoff assumption is enforced, the DKQ is transverse-shear rigid and can only be used for thin shells. No transverse shear stress information is available. The triangle is based on a degeneration of the DKQ. This element sometimes gives slightly lower eigenvalues when compared with element type 20.

   c) The plate component of element 20 is based on the 8-node serendipity element. At the mid-side, the parallel rotations and transverse displacements are constrained, and the normal rotations are condensed to yield a 4-node element. The element is based on thick plate theory and is recommended for thick and thin plates.

   d) The quadrilateral elements contain a warpage correction using rigid links.

   e) The membrane component of element 18 has a zero-energy mode associated with in-plane rotations. This is automatically suppressed in a non-flat shell by the plate stiffness of the adjacent elements. In contrast, element 20 has no spurious zero energy modes.

4. **Linear shear element (22).** The linear shear panel element resists tangential in plane shearing along the four edges and can only be used with the elastic material constants of *MAT_ELASTIC. Membrane forces and out-of-plane loads are not resisted.

5. **Simplified element for time domain vibrations (99).** Element type 99 is intended for vibration studies carried out in the time domain. These models may have very large numbers of elements and may be run for relatively long durations. The purpose of this element is to achieve substantial CPU savings. This

is achieved by imposing strict limitations on the range of applicability, thereby simplifying the calculations:

a) Elements must be rectangular; all edges must parallel to the global *x*-, *y*- or *z*-axis;

b) Small displacement, small strain, negligible rigid body rotation;

c) Elastic material only.

If these conditions are satisfied, the performance of the element is similar to the fully integrated shell (ELFORM = 16) but at less CPU cost than the default Belytschko-Tsay shell element (ELFORM = 2). Single element torsion and in-plane bending modes are included; meshing guidelines are the same as for fully integrated shell elements.

No damping is included in the element formulation (such as volumetric damping). It is strongly recommended that damping be applied with keywords such as *DAMPING_PART_MASS or *DAMPING_FREQUENCY_RANGE.

6. **2D formulations.** For 2D formulations (12-15, 43, 44, 46, 47, 52, and 55), nodes must lie in the global *xy*-plane, so the *z*-coordinate must be zero. Furthermore, the element normal should be in positive *z* direction. For axisymmetric element formulations, the global *y*-axis is taken as the axis of symmetry, and all nodes must have *x*-coordinate values greater than or equal to 0.

   Shell thickness values on Card 2 are ignored by formulations 13, 14, 15, 43, 44, 52, and 55. For formulation 14 input values of loads, lumped masses, discrete element stiffnesses, etc. in axisymmetric models are interpreted as values per unit circumference (that is per unit length in the circumferential direction) whereas for formulation 15 they are interpreted per radian. Output of forces for shell formulation 15 are in units of force per radian in the output files, such as bndout, nodfor, secforc, spcforc, and rcforc. The units of forces output for shell formulation 14 are, at present, inconsistent. For defining contact in 2D simulations, see the entry for the *CONTACT_2D keyword.

7. **Shells with thickness stretch.** Shell element formulation 25 and 26 are the fully integrated shell element based on the Belytschko-Tsay element but with two additional degrees of freedom allowing for a linear variation of strain through the thickness. By specifying IDOF = 1, the thickness field is continuous across the element edges implying that there can be no complex intersections since this would lock up the structure. It assumes a relatively flat surface and is intended primarily for sheets in metal forming. By default, the thickness field is decoupled between elements which makes the element suited for crash.

8. **Seat belts.** You must input a set of nodes along one of the transverse edges of a seatbelt. If there is no retractor associated with a belt, the node set can be on either edge. If the retractor exists, the edge should be on the retractor side and input in the same sequence as the retractor's node set. Therefore, each seat belt must have its own section definition and its own part.

9. **Fracture.** XFEM 2D and shell formulations are recommended for brittle or semi-brittle fracture with pre-crack (see *BOUNDARY_PRECRACK) or with geometry imperfection such as a notch or a hole, and for ductile fracture analysis with regularized effective plastic strain criterion or nonlocal continuum damage material models (provided with GISSMO model).

10. **Discrete Kirchoff Theory shell (17).** Shell element formulation 17 (DKT) is based on Discrete Kirchhoff Theory. It neglects out-of-plane shear strain energy and is thus valid only for thin plates where shear strain energy is negligible compared to bending energy.

11. **Limitations of area-weighted shells (14).** The exact stiffness matrix for the area-weighted shell formulation type 14 is nonsymmetric. The nonsymmetric terms are dropped for computational efficiency, making this formulation unsuitable for implicit linear analysis and eigenvalue analysis. For implicit modal computations element type 14 must be switched to type 15. It may, however, be used effectively for implicit nonlinear analysis.

    For explicit dynamics, viscous hourglass limits high frequency noise that may otherwise lead to nonphysical element distortion when nodes are on or near the axis of symmetry. Viscosity can be added to type 6 or 7 hourglass control by using VDC > 0 under the *HOURGLASS keyword. Alternatively, type 1 hourglass control is viscous.

12. **Eight node singular shell (55).** The eight node singular element for fracture analysis is based on the eight node quadratic quadrilateral element. There are two ways to include the singularity around the crack tip:

    a) Move the two mid-nodes on the edges connected to the crack tip to the quarter location and obtain a strain singularity of $1/\sqrt{r}$ along the edges with quarter-point mid-nodes and of weaker than one-half order elsewhere.

    b) Collapse the three nodes on one side of a quadrilateral element to the crack tip (but the three nodes remain independent) and move the two neighboring mid-nodes to the quarter location to obtain a strain singularity of exact one-half order $1/\sqrt{r}$ everywhere around the crack tip..

    This element uses 2 × 2, 3 × 3, or 4 × 4 quadrature (default 2 × 2) and is available in implicit analysis only. The reference coordinates of the quadrature points are

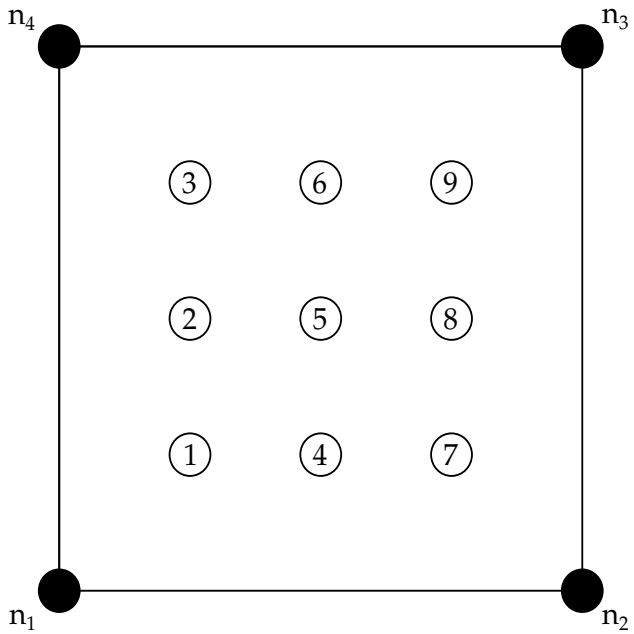


Figure 41-24. Integration point locations for a type 55 element with a 3 × 3 quadrature scheme

as in the Gaussian Quadrature Points table but have a different order (#2, #1 and #3) in one direction for the 3 × 3 quadrature scheme. The global coordinates of the quadrature points depend on the connectivity of the element, as shown in Figure 41-24.

To easily access the stress data, the stress components at all the integration points are output to the ELOUT file.

13. **Cohesive shell (+/-29).** Element type +/-29 is a cohesive element that models cohesive interfaces between shell element edges. The element takes bending forces into account and uses drilling force stabilization.

Consider two shell elements in the same plane with nodes m_1, m_2, m_3, m_4, n_1, n_2, n_3, and n_4 such that the (m_3, m_4) edge and the (n_1, n_2) edge are connected through a cohesive shell having nodes m_4, m_3, n_2, and n_1; see Figure 41-25. The initial area of the cohesive element may be zero, in which case density is defined in terms of the length of the single connecting edge.

Element type +/-29 works similarly to solid element type 20. For example, extruding the two non-cohesive shells in their respective normal directions defines two 8 node solids. The cohesive mid-surface is located between the opposing faces of the solids, and tractions are calculated in four mid-surface points using differences of displacements between the opposing faces, giving rise to nodal forces and moments in the cohesive shell nodes m_4, m_3, n_2, and n_1. Note that, regardless of the value of INTFAIL, a 2 x 2 Newton-Cotes quadrature is used in the force integrals. Additional details can be found in the Theory Manual.

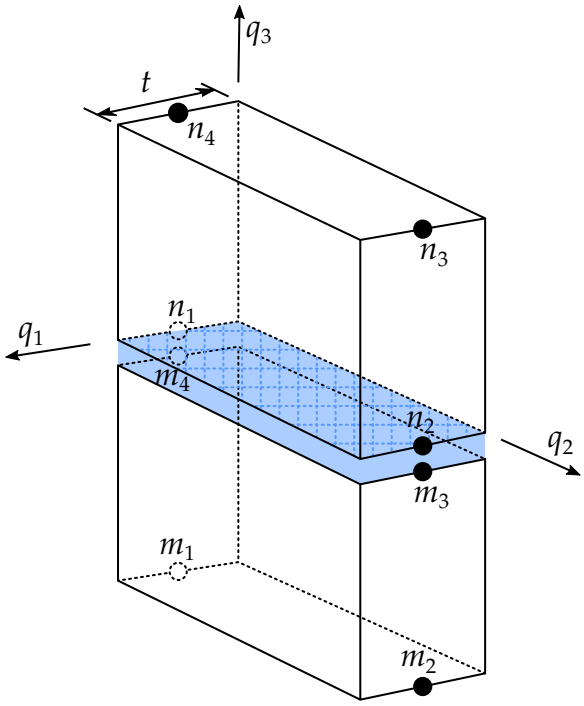


Figure 41-25. Cohesive interface coordinate system for element type +/-29

Difference between type 29 and -29: In both formulations, the cohesive coordinate system direction q_2 is defined by the midpoints of the (n_1, m_4) and (n_2, m_3) edges. Type 29 defines the cohesive midsurface normal q_3 using the midpoints on the far side edges (m_1, m_2) and (n_3, n_4), while type -29 defines q_1 to be the mean of the neighboring element normals. Thus, in pure out-of-plane shear, type 29 will initially have pure tangential traction that turns into a normal traction as the separation increases, while type -29 will only have tangential traction.

Cohesive shell formulations +/-29 output three tractions having units of force per unit area into the d3plot database rather than the usual six stress components. The out-of-plane shear traction replaces the x-stress (mode III), the in-plane shear traction replaces the y-stress (mode II), and the traction in the normal direction replaces the z-stress (mode I). The tractions on the four midsurface points on the cohesive interface can be saved in d3plot by setting MAXINT to -1 on *DATABASE_EXTENT_BINARY or *DATABASE_D3PART.

14. **Switching between type 16 and type 30 shells.** We implemented two methods to globally switch between types 16 and 30 without changing *SECTION_SHELL. For the first method, use the command line option "shell=16to30" or "shell=30to16" to change from type 16 to 30 or from type 30 to 16. See Execution Syntax in the Getting Started section for details about command line options. For the second method, set flag ISWSHL on *CONTROL_SHELL to 1 to convert from 16 to 30 or to 2 to convert from 30 to 16.

15. **Accurate fully-integrated shell (-16).** Accuracy issues have been observed in shell formulation 16 under large deformations/rotations over a single time step. Formulation -16 is a version of formulation 16 which has been enhanced to address those accuracy issues. Formulation 16 is unchanged to maintain back compatibility and, although element -16 is supported in explicit mode, it is primarily intended for implicit time integration. A strongly objective version can be activated by combining ELFORM = -16 and IACC = 1 on *CONTROL_ACCURACY, thereby ensuring that arbitrarily large rigid body rotation in a single step will transform stresses correctly and not generate any spurious strains.

16. **XFEM ductile fracture.** This feature is supported by joint research among Honda, JSOL and LST. For FAILCR = -2, the failure strain is given by:

$$\mathrm{EPS} = \mathrm{FS} + (\mathrm{FS1} - \mathrm{FS}) \times \min(L/\mathrm{CL}, 1.0)$$

17. **Cohesive formulations 46 and 47.** We intend cohesive formulations 46 and 47 for cohesive elements with nonzero initial thickness. Instabilities may occur if the element has zero initial thickness.

    Like for cohesive solids, LS-DYNA outputs three tractions having units of force per unit area for plane strain and axisymmetric cohesive formulations 46 and 47 into the d3plot database rather than the usual six stress components. The in-plane shear traction along the 1-2 edge replaces the x-stress, the orthogonal in plane shear traction replaces the y-stress, and the traction in the normal direction replaces the z-stress.

18. **Cohesive material law for XFEM.** For the XFEM formulation, modeling the fracture process zone around the crack tip is simplified with a cohesive zone model. The initially rigid cohesive material law provides the kinetic relationship (traction force as a function of crack opening displacement) for the cohesive crack surface inserted in the element. The work done by the traction represents the energy release rate in brittle fracture or plastic work in ductile fracture to create a real crack surface where the traction force becomes zero. When a nonlocal continuum damage constitutive model is used for ductile fracture, the cohesive law is not needed since the stresses in the failed element become zero due to the damage induced stress softening.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *SECTION_SHELL
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a shell section that specifies the following:
$     elform = 10  Belytschko-Wong-Chiang shell element formulation.
$        nip = 3   Three through the shell thickness integration points.
$    t1 - t4 = 2.0 A shell thickness of 2 mm at all nodes.
```

```
$
*SECTION_SHELL
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      sid    elform      shrf       nip     propt   qr/irid     icomp
         1        10              3.0000
$
$       t1        t2        t3        t4      nloc
       2.0       2.0       2.0       2.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

*SECTION_SOLID_{OPTION}

Available options include:

<BLANK>

EFG

SPG

MISC

Purpose: Define section properties for solid continuum and fluid elements.

Card Summary:

Card Sets. For each unique solid section, include one set of data cards. The EFG and SPG options cannot appear in the same model. This input ends at the following keyword ("*") card.

Card 1. This card is required.

SECID	ELFORM	AET				COHOFF	GASKETT

Card 2a.1. This card is included if the EFG keyword option is used.

DX	DY	DZ	ISPLINE	IDILA	IEBT	IDIM	TOLDEF

Card 2a.2. This card is read only if the EFG keyword option is used. It is optional.

IPS	STIME	IKEN	SF	CMID	IBR	DS	ECUT

Card 2b.1. This card is included if the SPG keyword option is used.

DX	DY	DZ	ISPLINE	KERNEL		SMSTEP	MSC

Card 2b.2. This card is read only if the SPG keyword option is used. It is optional.

IDAM	FS	STRETCH	ITB	MSFAC	ISC	BOXID	PDAMP

Card 2c. This card is read only if the MISC keyword option is used. It is optional.

COHTHK							

Card 3. This card is included if ELFORM = 101, 102, 103, 104, or 105.

NIP	NXDOF	IHGF	ITAJ	LMC	NHSV	XNOD	

Card 4. This card is included if ELFORM = 101, 102, 103, 104, or 105. Include NIP of this card.

XI	ETA	ZETA	WGT				

Card 5. This card is included if ELFORM = 101, 102, 103, 104, or 105. Include ceil(LMC/8) of this card.

P1	P2	P3	P4	P5	P6	P7	P8

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	AET				COHOFF	GASKETT
Type	I/A	I	I				F	F

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
ELFORM	Element formulation options. Remark 2 enumerates the element formulations available for implicit calculations:
	EQ.0: 1 point corotational for *MAT_MODIFIED_HONEY-COMB (see Remark 3)
	EQ.1: Constant stress solid element: default element type. By specifying hourglass type 10 with this element, a Cosserat Point Element is invoked; see *CONTROL_-HOURGLASS.
	EQ.-1: 8 point hexahedron intended for elements with poor aspect ratios, efficient formulation (see Remark 13)
	EQ.-2: 8 point hexahedron intended for elements with poor aspect ratios, accurate formulation (see Remark 13)
	EQ.2: 8 point hexahedron (see Remark 5)
	EQ.3: Fully integrated quadratic 8 node element with nodal rotations
	EQ.4: S/R quadratic tetrahedron element with nodal

VARIABLE	DESCRIPTION
	rotations
EQ.5:	1 point ALE
EQ.6:	1 point Eulerian
EQ.7:	1 point Eulerian ambient
EQ.8:	Acoustic
EQ.9:	1 point corotational for *MAT_MODIFIED_HONEY-COMB (see Remark 3)
EQ.10:	1 point tetrahedron (see Remark 1)
EQ.11:	1 point ALE multi-material element (see Remark 4)
EQ.12:	1 point integration with single material and void
EQ.13:	1 point nodal pressure tetrahedron (see Remark 12)
EQ.14:	8 point acoustic
EQ.15:	2 point pentahedron element (see Remark 1)
EQ.16:	4 or 5 point 10-noded tetrahedron (see Remark 11). By specifying hourglass type 10 with this element, a Cosserat Point Element is invoked; see *CONTROL_-HOURGLASS.
EQ.17:	10-noded composite tetrahedron (see Remark 11)
EQ.-18:	8 point enhanced strain solid element with 13 incompatible modes (see Remarks 5 and 17)
EQ.18:	9 point enhanced strain solid element with 12 incompatible modes (implicit only; see Remarks 5 and 17)
EQ.-19:	8-noded, 4 point cohesive element with or without offset for use with both solids and shells (see Remarks 1 and 7)
EQ.19:	8-noded, 4 point cohesive element without offset for use with solids (see Remarks 1 and 7)
EQ.20:	8-noded, 4 point cohesive element with offsets for use with shells (see Remarks 1, 7, and 9)
EQ.-21:	6-noded, 1 point pentahedron cohesive element with or without offset for use with solids and shells (see Remarks 1 and 8)
EQ.21:	6-noded, 1 point pentahedron cohesive element without offset for use with solids (see Remarks 1 and 8)

VARIABLE	DESCRIPTION
	EQ.22: 6-noded, 1 point pentahedron cohesive element with offsets for use with shells (see Remarks 1, 8, and 9)
	EQ.23: 20-node solid formulation
	EQ.24: 27-noded, fully integrated S/R quadratic solid element (see Remark 16)
	EQ.25: 21-noded, quadratic pentahedron (see Remark 16)
	EQ.26: 15-noded, quadratic tetrahedron (see Remark 16)
	EQ.27: 20-noded, cubic tetrahedron (see Remark 16)
	EQ.28: 40-noded, cubic pentahedron (see Remark 16)
	EQ.29: 64-noded, cubic hexahedron (see Remark 16)
	EQ.41: Mesh-free (EFG) solid formulation (see Remark 19)
	EQ.42: Adaptive 4-noded mesh-free (EFG) solid formulation (see Remark 19)
	EQ.43: Mesh-free enriched finite element
	EQ.45: Tied mesh-free enriched finite element
	EQ.47: Smoothed Particle Galerkin (SPG) method (see Remark 14)
	EQ.60: 1 point tetrahedron (see Remark 15)
	EQ.62: 8 point brick with incompatible modes by assumed strain (see Remarks 5 and 18)
	EQ.98: Interpolation solid
	EQ.99: Simplified linear element for time-domain vibration studies (See Remark 6)
	EQ.101: User-defined solid
	EQ.102: User-defined solid
	EQ.103: User-defined solid
	EQ.104: User-defined solid
	EQ.105: User-defined solid
	EQ.115: 1 point pentahedron element with hourglass control
	GE.201: Isogeometric solids with NURBS (see *ELEMENT_SOLID_NURBS_PATCH)
	GE.1000: Generalized user-defined solid element formulation (see *DEFINE_ELEMENT_GENERALIZED_SOLID)

VARIABLE	DESCRIPTION
AET	Ambient element type (can be defined for ELFORM 7, 11, and 12): EQ.0: Non-ambient EQ.1: Temperature (not currently available) EQ.2: Pressure and temperature (not currently available) EQ.3: Pressure outflow (obsolete) EQ.4: Pressure inflow/outflow (default for ELFORM 7) EQ.5: Receptor for blast load (See *LOAD_BLAST_ENHANCED. Available only for ELFORM = 11.)
COHOFF	Applies to cohesive solid elements 20 and 22. COHOFF specifies the relative location of the cohesive layer. It must be a number between -1 and 1. A value of -1 will place it on the bottom face of the cohesive element, while a value of +1 will place it on the top face. This parameter is preferably used when the cohesive element connects shells with different thicknesses. In this case, the cohesive layer should not be located exactly between the bottom and top layer, which is the default location (COHOFF = 0).
GASKETT	Gasket thickness for converting ELFORM 19, 20, 21, and 22 to gasket elements and use with *MAT_COHESIVE_GASKET.

EFG Card. Additional card for the EFG keyword option. See *CONTROL_EFG.

Card 2a.1	1	2	3	4	5	6	7	8
Variable	DX	DY	DZ	ISPLINE	IDILA	IEBT	IDIM	TOLDEF
Type	F	F	F	I	I	I	I	F
Default	1.01	1.01	1.01	0	0	3	2	0.01

VARIABLE	DESCRIPTION
DX, DY, DZ	Normalized dilation parameters of the kernel function in x, y, and z-directions. The normalized dilation parameters of the kernel function are introduced to provide the smoothness and compact support properties on the construction of the mesh-free shape functions. Values between 1.0 and 1.5 are recommended. Values

VARIABLE	DESCRIPTION
	smaller than 1.0 are not allowed. Larger values will increase the computation time and sometimes result in a divergence problem.
ISPLINE	Replace the choice for the EFG kernel functions definition in *CONTROL_EFG, which allows for different spline functions in different sections. EQ.0: Cubic spline function (default) EQ.1: Quadratic spline function EQ.2: Cubic spline function with a circular shape
IDILA	Replace the choice for the normalized dilation parameter definition in *CONTROL_EFG. This allows users to define different IDILA in different sections. EQ.0: Maximum distance based on the background elements. EQ.1: Maximum distance based on surrounding nodes.
IEBT	Essential boundary condition treatment (see Remarks 20 and 21): EQ.1: Full transformation method EQ.-1: (W/o transformation) EQ.2: Mixed transformation method EQ.3: Coupled FEM/EFG method (default) EQ.4: Fast transformation method EQ.-4: (W/o transformation) EQ.5: Fluid particle method for EOS and *MAT_ELASTIC_FLUID materials, currently supports only 4-noded background elements. EQ.7: Maximum entropy approximation
IDIM	Domain integration method (see Remark 22): EQ.1: Local boundary integration EQ.2: Two-point Gauss integration (default) EQ.3: Improved Gauss integration for IEBT = 4 or -4 EQ.-1: Stabilized EFG integration method (apply to 6-noded cell, 8-noded cell, or combination of these two) EQ.-2: EFG fracture method (apply to 4-noded cell and SMP only). The EFG fracture method is obsolete – see Remark

VARIABLE	DESCRIPTION
	24.
TOLDEF	Deformation tolerance for the activation of adaptive EFG Semi-Lagrangian and Eulerian kernel. See Remark 23. EQ.0.0: Lagrangian kernel GT.0.0: Semi-Lagrangian kernel LT.0.0: Eulerian kernel

Optional EFG Card. Additional optional card for the EFG keyword option. See *CONTROL_EFG.

Card 2a.2	1	2	3	4	5	6	7	8
Variable	IPS	STIME	IKEN	SF	CMID	IBR	DS	ECUT
Type	I	F	I	I	I	I	F	F
Default	0	10^{20}	0	0.0	0	1	1.01	0.1

VARIABLE	DESCRIPTION
IPS	Pressure smoothing flag: EQ.0: No pressure smoothing (default) EQ.1: Moving-least squared pressure recovery (material response is slightly more compressible when turning on this function). Only the adaptive 4-noded mesh-free (EFG) solid formulation (ELFORM = 42) is supported.
STIME	Time to switch from stabilized EFG to standard EFG formulation. Obsolete - see Remark 24.
IKEN	EQ.0: Moving-least-square approximation (default, recommended) EQ.1: Maximum entropy approximation Obsolete - see Remark 24.
SF	Failure strain. It is recommended as an additional condition for crack initiation under slow loading besides the stress-based

VARIABLE	DESCRIPTION
	cohesive law. Obsolete - see Remark 24.
CMID	Cohesive material ID for EFG fracture analysis (only Mode I crack is considered, and only *MAT_COHESIVE_TH is available). Obsolete - see Remark 24.
IBR	EQ.1: No branching allowed. EQ.2: Branching is allowed. Obsolete - see Remark 24.
DS	Normalized support defined for computing the displacement jump in fracture analysis. Obsolete - see Remark 24.
ECUT	Define the minimum distance to the node that a crack surface can cut to the edge. Obsolete - see Remark 24.

SPG Card. Additional card for the SPG keyword option.

Card 2b.1	1	2	3	4	5	6	7	8
Variable	DX	DY	DZ	ISPLINE	KERNEL		SMSTEP	MSC
Type	F	F	F	I	I		I	F
Default	↓	↓	↓	0	0		↓	0

VARIABLE	DESCRIPTION
DX, DY, DZ	Normalized dilation parameters of the kernel function in x, y, and z directions, respectively. The normalized dilation parameters of the kernel function provide the smoothness and locality on the construction of the mesh-free shape functions. Values between 1.4 and 1.8 are recommended. Values smaller than 1.0 are not allowed. Larger values will increase the computation time and sometimes result in divergence of the solution. The default value depends on the choice of KERNEL:

KERNEL	0	1	2
Default	1.6	1.8	1.5

VARIABLE	DESCRIPTION
ISPLINE	Type of kernel function: EQ.0: Cubic spline function with cubical support (default) EQ.1: Quadratic spline function with cubical support EQ.2: Cubic spline function with spherical support
KERNEL	Type of kernel support update scheme: EQ.0: Updated Lagrangian, failure or no failure analysis, tension dominant problem EQ.1: Eulerian, failure analysis, global extreme deformation EQ.2: Pseudo Lagrangian, failure analysis, local extreme deformation
SMSTEP	Interval of time steps to conduct displacement smoothing. The default value depends on the choice of KERNEL:
MSC	Smoothing scheme for momentum consistent SPG only (ITB = 3 on Card 2b.2): EQ.0: Regular smoothing scheme EQ.1: New smoothing scheme for very low-speed deformation, with better controls of the low energy modes than the regular smoothing scheme

KERNEL	0	1	2
Default	15	5	30

Optional SPG Card. Additional optional card for the SPG keyword option.

Card 2b.2	1	2	3	4	5	6	7	8
Variable	IDAM	FS	STRETCH	ITB	MSFAC	ISC	BOXID	PDAMP
Type	I	F	F	I	F	I	I	F
Default	1	↓	10^{10}	↓	↓	0	0	-0.001

VARIABLE	DESCRIPTION
IDAM	Option of bond failure mechanism EQ.1: Effective plastic strain (phenomenological strain damage, default) EQ.2: Maximum principal stress EQ.3: Maximum shear strain EQ.4: Minimum principal strain (input must be positive) EQ.5: Effective plastic strain and maximum shear strain EQ.7: Anisotropic damage for honeycomb modeled with *MAT_126 only. We recommend only using this with ITB = 3. This feature is available starting with R13. EQ.11: Pre-damage model for brittle material failure (with crack propagation). It includes both bond failure and stress degradation. It is available as of R13. We recommend using this with ITB = 3. EQ.13: Pre-damage model for ductile material failure. It includes stress degradation but not bond failure. It is available as of R13. We recommend using this with ITB = 3.
FS	Critical value of the quantity indicated by IDAM for triggering bond failure. The default value is 10^{10}, meaning no failure analysis in general. However, starting with R14.0, if an SPG part has a material law with *MAT_ADD_EROSION, bond failure will be checked for using the failure criterion defined in *MAT_ADD_EROSION, even when FS = 0.0. FS on the material cards overwrites this value for *MAT_003 and *MAT_024. When FS is defined on the data cards for these materials, bond failure will occur when it is reached, and stress will be set to zero according to material law. If FS is defined here only, bond failure will occur without the stress being set to zero.
STRETCH	Critical relative deformation (stretching or compression ratio) between the two nodes forming the bond for bond failure
ITB	Option of stabilization: EQ.1: Fluid particle approximation (accurate but slow), used with KERNEL = 0 or 1

VARIABLE	DESCRIPTION

EQ.2: Simplified fluid particle approximation (efficient and robust), used with KERNEL = 2

EQ.3: Momentum consistent SPG (MC-SPG) formulation (we recommend the latest beta version or R14). MC-SPG can be applied for large deformation, tension-dominant problems. For coupled thermal-mechanical problems, MC-SPG is the only option. KERNEL = 1 is recommended for MC-SPG.

The default for ITB depends on the value of KERNEL:

KERNEL	0	1	2
Default	1	1	2

MSFAC: For momentum consistent SPG invoked with ITB = 3 only, quadrature factor for surface nodes to suppress shear locking in thin structures. We recommend using the latest beta version or R14. The default for a regular solid structure is 1.00 while for a thin structure is 0.75.

ISC: Self-contact indicator:

EQ.0: No self-contact between the bond-failed particles in the same part.

EQ.1: Self-contact is defined between the bond-failed particles in the same part. The penalty factor in the self-contact is between 0.01 to 0.1 × Young's modulus. This option is available for SMP only.

BOXID: ID of a box defining the active SPG region. Outside this region, the particles are not included in the SPG calculation. See *DEFINE_BOX.

PDAMP: Particle-to-particle damping coefficient. It is used for momentum consistent SPG (ITB = 3) *only*. The recommended range of values is -0.01 to -0.001. A positive value is not recommended.

Optional MISC Card. Additional optional card for the MISC keyword option.

Card 2c	1	2	3	4	5	6	7	8
Variable	COHTHK							
Type	F							
Default	optional							

VARIABLE	DESCRIPTION
COHTHK	Cohesive thickness. This value supersedes the THICK value from *MAT_240 or *MAT_ADD_COHESIVE, allowing a section- or part-wise definition of cohesive thickness.

User-Defined Element Card. Additional card for ELFORM = 101, 102, 103, 104, or 105. See Appendix C.

Card 3	1	2	3	4	5	6	7	8
Variable	NIP	NXDOF	IHGF	ITAJ	LMC	NHSV	XNOD	
Type	I	I	I	I	I	I	I	
Default	0	0	0	0	0	0	0	

VARIABLE	DESCRIPTION
NIP	Number of integration points for user-defined solid (0 if resultant element)
NXDOF	Number of extra degrees-of-freedom per node for user-defined solid
IHGF	Flag for using hourglass stabilization (NIP > 0) EQ.0: Do not use hourglass stabilization. EQ.1: Use LS-DYNA hourglass stabilization. EQ.2: Use user-defined hourglass stabilization. EQ.3: Same as 2, but pass the resultant material tangent moduli.

VARIABLE	DESCRIPTION
ITAJ	Flag for setting up finite element matrices (NIP > 0) EQ.0: Set up matrices with respect to the isoparametric domain EQ.1: Set up matrices with respect to the physical domain
LMC	Number of property parameters
NHSV	Number of history variables
XNOD	Controls how the mass of extra degrees-of-freedom is interpreted when defined using user subroutine. EQ.0: The mass is to be defined per degree-of-freedom. EQ.1: The mass is to be defined as a nodal mass, where each scalar node has three extra degrees-of-freedom. See Extra Degrees-Of-Freedom in Appendix C.

Integration Point Card. Additional card for ELFORM = 101, 102, 103, 104, or 105. Add NIP cards adhering to the format below. Because the default value for NIP is 0, these cards are read only for user-defined elements. See Appendix C.

Card 4	1	2	3	4	5	6	7	8
Variable	XI	ETA	ZETA	WGT				
Type	F	F	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
XI	First isoparametric coordinate
ETA	Second isoparametric coordinate
ZETA	Third isoparametric coordinate
WGT	Isoparametric weight

Property Parameter Cards. Additional card for ELFORM = 101, 102, 103, 104, or 105. Add LMC property parameters by packing eight parameters per card. See Appendix C.

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
Default	0.	0.	0.	0.	0.	0.	0.	0.

VARIABLE	DESCRIPTION
P*i*	i^{th} property parameter

Remarks:

1. **ESORT to Stabilize Degenerate Solids.** The ESORT variable of the *CONTROL_SOLID keyword can be set to automatically convert degenerate tetrahedrons and degenerate pentahedrons into more suitable solid element formulations. The sorting is performed internally and is transparent to the user. See *CONTROL_SOLID for details.

2. **Implicit Analysis.** For implicit calculations, the following element choices are implemented:

   EQ.1: Constant stress solid element

   EQ.-1: 8 point hexahedron for poor aspect ratios, efficient formulation

   EQ.-2: 8 point hexahedron for poor aspect ratios, accurate formulation

   EQ.2: 8 point hexahedron

   EQ.3: Fully integrated 8 node solid with rotational DOFs

   EQ.4: Fully integrated S/R 4 node tetrahedron with rotational DOFs

   EQ.10: 1 point tetrahedron

   EQ.13: 1 point nodal pressure tetrahedron

   EQ.15: 2 point pentahedron element

   EQ.16: 5 point 10-noded tetrahedron

   EQ.17: 10-noded composite tetrahedron

   EQ.-18: 8 point enhanced strain solid element (13 incompatible modes)

EQ.18: 9 point enhanced strain solid element (12 incompatible modes)

EQ.±19: 8-noded, 4 point cohesive element

EQ.20: 8-noded, 4 point cohesive element with offsets for use with shells

EQ.±21: 6-noded, 1 point pentahedron cohesive element

EQ.22: 6-noded, 1 point pentahedron cohesive element with offsets for use with shells

EQ.23: 20-node solid formulation

EQ.24: 27-node solid formulation

EQ.25: 21-noded, quadratic pentahedron

EQ.26: 15-noded, quadratic tetrahedron

EQ.27: 20-noded, cubic tetrahedron

EQ.28: 40-noded, cubic pentahedron

EQ.29: 64-noded, cubic hexahedron

EQ.41: Mesh-free (EFG) solid formulation

EQ.42: 4-noded mesh-free (EFG) solid formulation

EQ.43: Mesh-free enriched finite element

EQ.62: 8 point brick with incompatible modes by assumed strain

If another element formulation is requested, LS-DYNA will substitute, when possible, one of the above in place of the one chosen. The type 1 element, constant stress, is generally much more accurate than the type 2 element, the selectively reduced integrated element for implicit problems.

3. **Elements for Modified Honeycomb Material.** Element formulations 0 and 9 behave essentially as nonlinear springs to permit severe distortions sometimes seen in honeycomb materials. They are applicable only to *MAT_MODIFIED_HONEYCOMB, but as of R14, *MAT_ADD_EROSION can also be enabled. For formulation 0, the local coordinate system follows the element rotation, whereas, for formulation 9, the local coordinate system is based on axes passing through the centroids of the element faces. Formulation 0 is preferred for severe shear deformation where the barrier is fixed in space. If the barrier is attached to a moving body, which can rotate, then formulation 9 is usually preferred.

4. **ALE Element (ELFORM = 11) Shapes.** We designed the ALE solver assuming hexahedral meshes of ALE elements. The ALE solver treats tetrahedral elements as degenerate hexahedrons. Thus, the advection calculation may be inaccurate and cause LS-DYNA to crash. We recommend only using ALE meshes with exclusively hexahedral elements.

5. **8/9 Point Hexahedrons: Types 2, 18, and 62.** Solid formulation 2 often employs a B-bar method and thereby assumes that pressure is constant throughout the element to avoid pressure locking during nearly incompressible flow. But when using material model 2, 21-23, 30, 38, 54, 57, 59, 63, 83, 91-92, 126, 143, 176-179, 181 (compressible option), 183, 189, 213, 215, 218, 221, 227, 249, 261-262, 266, 269, 274-275 or 295, solid formulation 2 employs full integration to treat compressible behavior better. IHYPER=±10 on *MAT_USER_DEFINED_MATERIAL_MODELS will also invoke full integration of solid form 2 when using the corresponding hyperelastic user material. If the element aspect ratios are poor, shear locking will lead to an excessively stiff response of solid form 2. Given poor aspect ratios, a better choice is the one point solid element (formulation 1), the 8 point solid formulation -1, or the 8 point solid formulation -2 (see Remark 13). Elements with formulations -1 and -2 are insensitive to a lousy aspect ratio but become very stiff if skewed or distorted. The enhanced assumed strain solid formulations -18 (8 point) and 18 (9 point) work well for elements with poor aspect ratios and can handle some distortion. Highly distorted elements lead to excessive stiffness for element formulations -18 and 18. Element form 62 is also an enhanced assumed strain element with 8 nodes that is not susceptible to volumetric or shear locking.

6. **Element Type 99 for Vibration.** Element type 99 is intended for vibration studies in the time domain using explicit time integration only. This model type may have a large number of elements and run for a relatively long duration. The purpose of this element is to achieve substantial CPU savings. This is achieved by imposing strict limitations on the range of applicability, thereby simplifying the calculations:

   a) Elements must be cubed; all edges must parallel to the global *x*-, *y*- or *z*-axis;

   b) Small displacement, small strain, negligible rigid body rotation;

   c) Elastic material only

   If these conditions are satisfied, the performance of the element is similar to the fully integrated S/R solid (ELFORM = 2) but at less CPU cost than the default solid element (ELFORM = 1). Single-element bending and torsion modes are included, so meshing guidelines are the same as for fully integrated solids, e.g., relatively thin structures can be modeled with a single solid element through the thickness if required. Typically, the CPU requirement per element-cycle is roughly two-thirds that of the default solid element.

   No damping is included in the element formulation (for example, volumetric damping). It is strongly recommended that damping be applied, using keywords including *DAMPING_PART_MASS or *DAMPING_FREQUENCY_RANGE_DEFORM.

7. **8-Node Cohesive Element: Type ±19.** Element types ±19 are cohesive elements. The tractions on the midsurface, defined as the midpoints between the nodal pairs 1-5, 2-6, 3-7, and 4-8, are functions of the differences of the displacements between nodal pairs interpolated to the four integration points. Element type 19 only includes translational DOF and performs best if there is no offset between the bottom and top nodal planes (i.e. having zero thickness). As a complement there is element type -19, which also only includes translational DOF, but accounts for an offset between the top and bottom planes. This feature means that element type -19 can be used with both solid and shell elements. For implicit calculations, element types ±19 are internally switched to a version similar to -19, but with a higher degree of accuracy for large step sizes.

The initial volume of the cohesive element may be zero, in which case, the density may be defined in terms of the area of the midsurface. Since the cohesive element represents an infinitely thin interface between other elements, its density can theoretically be zero. However, (especially when the cohesive elements are connected using tied contacts) we recommend setting the density to some nonzero value to avoid problems during model initiation. See parameter COHTIEM on *CONTROL_CONTACT, parameter ICOH on *CONTROL_SOLID, and *MAT_ADD_COHESIVE.

We calculate he tractions in the local coordinate system defined at the centroid of the element; see Figure 41-26. Representing the rotation matrix from the local to the global coordinate system at time t as $\mathbf{R}(t)$, the initial coordinates as $\mathbf{X}$, and the current coordinates as $\mathbf{x}$, the displacements at an integration point are

$$\Delta\mathbf{u} = \mathbf{R}^{\mathrm{T}}(t)\Delta\mathbf{x} - \mathbf{R}^{\mathrm{T}}(0)\Delta\mathbf{X}$$

$$\Delta\mathbf{x} = \sum_{i=1}^{4} N_i(s,t)\Delta\mathbf{x}_{i+4,i}$$

$$\Delta\mathbf{X} = \sum_{i=1}^{4} N_i(s,t)\Delta\mathbf{X}_{i+4,i}$$

The forces are obtained by integrating the tractions over the midsurface and rotating them into the global coordinate system. It is the sum over integration points g = 1,2,3,4.

$$\mathbf{F}_i = \mathbf{R}(t)\sum_{g=1}^{4}\mathbf{T}_g N_i(s_g,t_g)\det(\mathrm{J}_g), \quad \text{for } 1 \le i \le 4, \quad \text{and } \mathbf{F}_{i+4} = -\mathbf{F}_i \ ,$$

where

$\mathbf{T}_g$ = the traction stress in the local coordinate system

N_i = the shape function of the cohesive element at node i

s_g and t_g = the parametric coordinates of the 4 integration points (determined by INTFAIL on the cohesive material card).

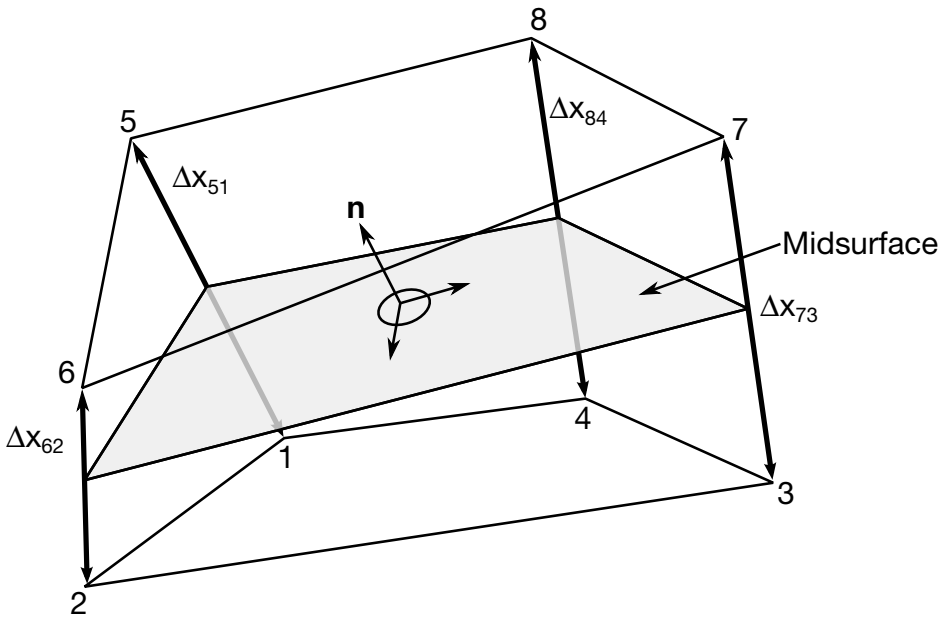


Figure 41-26. Illustration of solid local coordinates.

J_g = the integration point's portion of the determinate of the cohesive element, which is equivalent to the element volume

See Remark 10 regarding output for cohesive solids.

8. **6-Node Cohesive Element: Type ±21.** Element types ±21 are the pentahedral counterparts to element types ±19, with three nodes on the bottom and top surfaces. The tractions on the mid surface, defined as the midpoints between the nodal pairs 1-5, 2-6, and 3-7 are functions of the differences of the displacements between nodal pairs interpolated to one integration point. The ordering of the nodal points in *ELEMENT_SOLID is given by:

6-node (cohesive) pentahedron	N1, N2, N3, N3, N5, N6, N7, N7, 0, 0

Setting ESORT > 0 in *CONTROL_SOLID will automatically sort degenerated cohesive elements type ±19 to cohesive pentahedron elements type ±21. See Remark 10 regarding output for cohesive solids.

Element type 21 is internally switched to this type -21 for implicit calculations.

9. **Cohesive Element with Offsets: Types 20 and 22.** Element type 20 is identical to element 19 but with offsets for use with shells. If COHOFF = 0, the element is assumed to be centered between two layers of shells on the cohesive element's lower (1-2-3-4) and upper (5-6-7-8) surfaces. The offset distances for both shells are one-half the initial thicknesses of the nodal pairs (1-5, 2-6, 3-7, and 4-8), separating the two shells. These offsets are used with the nodal forces to calculate moments that are applied to the shells. Element type 20 in tied contacts will

work correctly with the option TIED_SHELL_EDGE_TO_SURFACE, which transmits moments. Other tied options will leave the rotational degrees-of-freedom unconstrained with the possibility that the rotational kinetic energy will cause a significant growth in the energy ratio.

Element type 22 is the pentahedron counterpart to element type 20, with three nodes on the bottom and top surfaces. The ordering of the nodal points in *ELEMENT_SOLID is identical to element type 21 (see Remark 8). Setting ESORT > 0 in *CONTROL_SOLID will automatically sort degenerated cohesive elements type 20 to cohesive pentahedron elements type 22.

For implicit calculations, element types 20 and 22 are internally switched to versions with a higher degree of accuracy for large step sizes.

See Remark 10 regarding output for cohesive solids.

10. **Cohesive Element Output.** Cohesive solids (formulations ±19, 20, ±21, 22) output three tractions, having units of force per unit area, into the d3plot database rather than the usual six stress components. The in-plane shear traction along the 1-2 edge replaces the x-stress, the orthogonal in-plane shear traction replaces the y-stress, and the traction in the normal direction replaces the z-stress.

11. **10-Node Tetrahedra: Types 16 and 17.** Formulations 16 and 17 are 10-noded tetrahedral formulations. The parameter NIPTETS in *CONTROL_SOLID controls the number of integration points for these formulations. Formulation 17 is generally preferred over formulation 16 in explicit analysis because, unlike formulation 16, the nodal weighting factors are equal, and thus nodal forces from contact and applied pressures are distributed correctly.

    When applying loads to 10-noded tetrahedrons via segments, no load will be applied to the mid-side nodes if the segments contain only corner nodes. When defining contact, it is recommended that *CONTACT_AUTOMATIC_… be used and the contact surface of the 10-noded tetrahedral part be specified by its part ID. In this manner, mid-side nodes receive contact forces.

    If the 10-noded element connectivity is not defined in accordance with Figure 19-31 shown in *ELEMENT_SOLID, the order of the nodes can be quickly changed using a permutation vector specified with *CONTROL_SOLID. If *ELEMENT_SOLID defines 4-noded tetrahedrons, the command *ELEMENT_SOLID_TET4TOTET10 can easily convert them to 10-noded tetrahedrons. Because the characteristic length of a 10-noded tetrahedron is half that of a 4-noded tetrahedron, the time step for the tetrahedrons will be smaller by a factor of 2. Setting the variable TET10S8 to 1 in *CONTROL_OUTPUT causes the full 10-node connectivity to be written to the d3plot and d3part databases.

12. **1-Point Nodal Pressure Tetrahedron: Type 13.** Element type 13 is identical to type 10 but includes additional averaging of nodal pressures, which significantly decreases the chance of volumetric locking. Therefore, it is well suited for applications with incompressible and nearly incompressible material behavior, such as rubber materials or ductile metals with isochoric plastic deformations (e.g., bulk forming). Compared to the standard tetrahedron (type 10), a speed penalty with a maximum of 25% can be observed. For implicit simulations, all material models supported for type 10 are also supported for this element, while for explicit currently material models *MAT_001 (excluding FLUID option), 003, 006, 007, 015, 024, 027, 041-050, 077, 081, 082, 089, 091, 092, 098, 103, 106, 120, 122, 123, 124, 128, 129, 133, 157, 181, 183, 187, 199, 224, 225, 228, 233, 244, 260, 263, 266, 269, 271, 272 and 273 are fully supported. For other materials, this element behaves like the type 10 tetrahedron. Type 13 tetrahedral elements that use two different material models should not share nodes because the nodal pressure averaging will cause spurious energy. An exception to this rule is if the different materials have the same bulk modulus.

13. **8 point Hexahedrons with Poor Aspect Ratio: Types -1 and -2.** Solid formulations -1 and -2 employ an assumed strain approach to avoid the shear locking behavior seen in formulation 2 elements with poor aspect ratios. Formulation -1 is a more computationally efficient implementation of formulation -2. Formulation -1 has a side effect: resistance to a particular deformation mode, similar to an hourglass mode, is weakened. This side effect is not truly hourglassing behavior, so there is no hourglass energy, and the behavior is not affected by hourglass parameters. For material models 57, 83, 181 (compressible option), and 274, formulations -1 and -2 employ full integration, rather than a B-bar method, to better deal with compressible behavior. If IHYPER = ±10 on *MAT_USER_DEFINED_MATERIAL_MODELS, this will also invoke full integration of the hyperelastic user material.

14. **Smoothed Particle Galerkin (SPG) method: Type 47.** With the SPG method, LS-DYNA converts finite element nodes into particles. This method supports 4-noded, 6-noded, and 8-noded *solid* elements. The method is suitable for severe deformation and failure analysis.

15. **1-Point Tetrahedron: Type 60.** Aside from including additional averaging of element volumetric locking, element type 60 is identical to type 10. In contrast to type 13, type 60 averages the normal stress between two adjacent elements, so nodes can be shared between the different materials with different bulk moduli. This element supports explicit analysis but is still under development for implicit simulations.

16. **Higher Order Elements.** Quadratic solids (ELFORM = 24, 25, and 26) and cubic solids (ELFORM = 27, 28, and 29) remain under development.

17. **Incompatible Modes Elements: Types -18 and 18.**

    a) Element type -18 is available for both implicit and explicit analysis. This is an 8-point integrated element, using 9 (Wilson) modes to alleviate Poisson locking in bending and another 4 to alleviate volumetric locking. The element is costly for explicit analysis; an additional factor of between 2 and 5 in simulation time is expected compared to element type 2. The exact number depends on the number of iterations needed to solve the compatibility equation for each element; typically, more iterations are needed for severely deformed elements and/or nonlinear effects in materials. In implicit analysis, the expense of the element is the same as in explicit, but this cost becomes insignificant relative to the global linear algebra. Increased accuracy seems to compensate for the cost. Therefore, the element should perform well in many types of analyses.

    b) Element type 18 is a similar element formulation to -18 but uses 9 integration points instead of 8. Similar to -18, it uses 9 modes to alleviate Poisson locking in bending. Unlike -18, it uses 3 instead of 4 incompatible modes for the volumetric treatment. This element is not available for explicit analysis, and when used, it will be switched to element formulation 1. Both elements 18 and -18 are based on the works of Simo and Rafai (1990) and Simo et al. (1993).

18. **Fully Integrated QBI Element.** Element form 62 is a fully integrated 8 node hexahedral element that uses an assumed strain field to prevent shear and volumetric locking and enhance bending stiffness. In Belytschko and Bindeman (1993), this assumed strain field is called the quintessential bending incompressible or QBI field. Element form 62 is available for implicit and explicit analysis. It is about 10% slower in explicit analyses than the fully integrated element formulation 2.

19. **EFG Solid Elements: Types 41 and 42.** EFG element type 41 is supported for 4-node, 6-node, and 8-node solid elements. Element type 42 is only supported for 4-node tetrahedral meshes, but it is optimized to achieve better computational efficiency than 41. For three-dimensional tetrahedron *r*-adaptive analysis (ADPTYP = 7 in *CONTROL_ADAPTIVE and ADPOPT = 2 in *PART), the initial mesh must be purely comprised of tetrahedrons, and element type 42 should be used.

    The EFG method still uses elements (so-called background elements or cells) to integrate nodal forces and stiffness in implicit analysis. The number of integration points for EFG depends on the EFG formulation and element type. For EFG type 41, the standard EFG formulation (IDIM = 1, 2, and 3) always allocates memory for 8 integration points, but the actual number of points used for the domain integration changes with the type of background element: 8 points for

an 8-node element, 2 for a 6-node element, and 1 for a 4-node element. The stabilized formulation (IDIM = -1) only allocates and uses one integration point. It applies to 6-node elements, 8-node elements, or a mixed mesh of these two types of cells. EFG type 42 has only the standard formulation. It uses and allocates memory for one integration point since it applies to a pure tetrahedral mesh.

20. **Automatic Sorting for EFG Background Mesh.** The current EFG formulation performs automatic sorting for finite element tetrahedral, pentahedron, and hexahedral elements of the background mesh to identify the mesh-free geometry and provide the contact surface definition in the computation.

21. **Essential Boundary Conditions.** The mixed transformation method, the coupled FEM/EFG method, and the fast transformation method were implemented for the EFG 3D solid formulation. These three features were added to improve the efficiency of the imposition of essential boundary conditions and the transfer of actual nodal and generalized nodal values. The mixed transformation method is equivalent to the full transformation method with improved efficiency. The behavior of the coupled FEM/EFG method is between FEM and EFG. The fast transformation method provides the most efficient and robust results.

22. **IDIM.** For compressible materials like foam and soil, IDIM = 1 is recommended. IDIM = 2 (default) is recommended for nearly incompressible materials like metal and rubber.

23. **TOLDEF.** This field is introduced to improve the negative volume problem usually seen during large deformation analysis. For the same analysis, the larger the magnitude of TOLDEF, the earlier the Semi-Lagrangian or Eulerian kernel is introduced into the EFG computation, so more CPU time is expected. A value between 0.0 and 0.1 is suggested for crashworthiness analysis. We suggest the Semi-Lagrangian kernel for solid materials and the Eulerian kernel for fluid and EOS materials.

24. **EFG Fracture.** The EFG fracture method (IDIM = -2) is obsolete. For ductile failure, consider using *SECTION_SOLID_SPG.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$   *SECTION_SOLID
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  A bolt modeled with solids was found to have excessive hourglassing.
$  Thus, the section (sid = 116) associated with the bolt part was used
```

```
$  to specify that a fully integrated Selectively-Reduced solid element
$  formulation be used to totally eliminate the hourglassing (elform = 2).
$
*SECTION_SOLID
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$       sid    elform
        116         2
$
*PART
bolts
$       pid       sid       mid     eosid      hgid    adpopt
         17       116         5
$
$$$$$$$$$$$$$$$$$$$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

*SECTION_SOLID_PERI

Purpose: Define section properties for Peridynamics solid and laminate elements.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM						
Type	I/A	I						

Card 2	1	2	3	4	5	6	7	8
Variable	DR	PTYPE						
Type	F	I						
Default	1.01	1						

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
ELFORM	Element formulation options (see Remarks): EQ.48: Peridynamics element formulation. This formulation works for both peridynamics solids (4, 6 and 8 node elements) and peridynamics laminates (4 node surface elements)
DR	Normalized horizon size, 0.6 – 1.2 is recommended
PTYPE	Peridynamics formulation: EQ.1: Bond based formulation (default)

Remarks:

1. **Peridynamics Solids.** The elements in a Peridynamics solid part should be defined with *ELEMENT_SOLID. These elements can be meshed with a 3D solid mesher. To represent material split, the elements should not share nodes. Thus, shared corners will have coincident nodes, one for each element. The total

number of nodes is then the summation over the number of elements of the number of nodes in each element.

2. **Peridynamics Laminates.** Elements in a Peridynamics laminate part should be specified with *ELEMENT_SOLID_PERI. See *ELEMENT_SOLID_PERI.

***SECTION_SPH**_*{OPTION}*

Available options include:

<BLANK>

ELLIPSE

INTERACTION (See Remark 3)

USER (See Remark 2)

Purpose: Define section properties for SPH particles.

> **NOTE:** This feature is not supported for use in implicit calculations.

Card Sets. For each SPH section add one set of the following cards (depending on the keyword option). This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	CSLH	HMIN	HMAX	SPHINI	DEATH	START	SPHKERN
Type	I/A	F	F	F	F	F	F	I
Default	none	1.2	0.2	2.0	0.0	10^{20}	0.0	0

Ellipse Card. Additional card for ELLIPSE keyword option.

Card 2	1	2	3	4	5	6	7	8
Variable	HXCSLH	HYCSLH	HZCSLH	HXINI	HYINI	HZINI		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.

VARIABLE	DESCRIPTION
CSLH	Constant used to calculate the initial smoothing length of the particles. The default value works for most problems. Values between 1.05 and 1.3 are acceptable. Taking a value less than 1 is inadmissible. Values larger than 1.3 will increase the computational time. The default value is recommended. See Remark 1.
HMIN	Scale factor for the minimum smoothing length. Ignored if FORM = 12 in *CONTROL_SPH. See Remark 1.
HMAX	Scale factor for the maximum smoothing length. Ignored if FORM = 12 in *CONTROL_SPH. See Remark 1.
SPHINI	Optional initial smoothing length (overrides true smoothing length). With this option LS-DYNA will not calculate the smoothing length during initialization, and the field CSLH is ignored.
DEATH	Time imposed SPH approximation is stopped.
START	Time imposed SPH approximation is activated.
SPHKERN	Option for SPH kernel functions (smoothing functions): EQ.0: Cubic spline kernel function (default). EQ.1: Quintic spline kernel function: a higher order smoothing function with a larger support size. It is only available for the 3D case with FORM = 0, 1, 5, 6, 9 and 10 (see *CONTROL_SPH). EQ.2: Quadratic spline kernel function: it helps to relieve the problem of compressive instability and aims for HVI problems. It is only available for the 3D case with FORM = 0, 1, 5, and 6 (see *CONTROL_SPH). EQ.3: Quartic kernel function: this kernel function is very close to cubic spline kernel function but is more stable. It is only available for the 3D case with FORM = 0, 1, 5, and 6 (see *CONTROL_SPH).
HXCSLH	Constant applied for the smoothing length in the x-direction for the ellipse case.
HYCSLH	Constant applied for the smoothing length in the y-direction for the ellipse case.
HZCSLH	Constant applied for the smoothing length in the z-direction for the ellipse case.

VARIABLE	DESCRIPTION
HXINI	Optional initial smoothing length in the x-direction for the ellipse case (overrides true smoothing length)
HYINI	Optional initial smoothing length in the y-direction for the ellipse case (overrides true smoothing length)
HZINI	Optional initial smoothing length in the z-direction for the ellipse case (overrides true smoothing length)

Remarks:

1. **Smoothing Length.** The SPH processor in LS-DYNA employs a variable smoothing length. LS-DYNA computes the initial smoothing length, h_0, for each SPH part by taking the maximum of the minimum distance between every particle and then scaling this value by CSLH. The recommended values of CSLH should be used so that the radius of the support domain covers more than two layers of SPH particles along each direction. Every particle has its own smoothing length which varies in time according to the following equation:

$$\frac{d}{dt}h(t) = h(t)\nabla \cdot \mathbf{v} \ ,$$

where $h(t)$ is the smoothing length, and $\nabla \cdot \mathbf{v}$ is the divergence of the flow. The smoothing length increases as particles separate and reduces as the concentration increases. This scheme is designed to hold constant the number of particles in each neighborhood. In addition to being governed by the above evolution equation, the smoothing length is constrained to be between a user-defined upper and lower value,

$$\text{HMIN} \times h_0 < h(t) < \text{HMAX} \times h_0.$$

Defining a value of 1 for HMIN and 1 for HMAX will result in a constant smoothing length in time and space.

When formulation 12 is employed (FORM = 12 in *CONTROL_SPH), the smoothing length remains constant with HMIN and HMAX set to 1.0 internally.

2. **USER Option.** The USER option allows the definition of customized subroutine for the variation of the smoothing length. A subroutine called *hdot* is defined in the file dyn21.F (Unix/linux) or lsdyna.f (Windows).

3. **Inter-Part Particle Interaction.** There are two fundamental ways that particles from different SPH parts can interact with each other. One way is through "particle approximation" and the other way is through *DEFINE_SPH_TO_SPH_-

COUPLING. When CONT = 0 in *CONTROL_SPH, "particle approximation" is used for all SPH parts in treating inter-part particle interaction. When CONT = 1 in *CONTROL_SPH, inter-part particle interaction by "particle approximation" occurs only for those SPH parts that use *SECTION_SPH_INTERACTION, while any SPH part that does not make use of *SECTION_SPH_INTERACTION will not participate in inter-part particle interaction, *except* as defined using *DEFINE_SPH_TO_SPH_COUPLING.

*SECTION_TSHELL

Purpose: Define section properties for thick shell elements.

Card Sets. For each TSHELL section include a set of the following cards. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	SECID	ELFORM	SHRF	NIP	PROPT	QR	ICOMP	TSHEAR
Type	I/A	I	F	I	I	F	I	I
Default	none	1	1.0	2	1	0.0	0	0

Angle Cards. If ICOMP = 1 specify NIP angles putting 8 on each card. Include as many cards as necessary.

Card 2	1	2	3	4	5	6	7	8
Variable	B1	B2	B3	B4	B5	B6	B7	B8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
SECID	Section ID. SECID is referenced on the *PART card. A unique number or label must be specified.
ELFORM	Element formulation: EQ.1: one point reduced integration (default), EQ.2: selective reduced 2 × 2 in plane integration. EQ.3: assumed strain 2 × 2 in plane integration, see remark below. EQ.5: assumed strain reduced integration with brick materials EQ.6: assumed strain reduced integration with shell materials EQ.7: assumed strain 2 × 2 in plane integration.
SHRF	Shear factor. A value of 5/6 is recommended (see Remark 4).

VARIABLE	DESCRIPTION
NIP	Number of through thickness integration points for the thick shell. See the variable INTGRD in *CONTROL_SHELL for details of the through thickness integration rule. EQ.0: set to 2 integration points.
PROPT	Printout option: EQ.1: average resultants and fiber lengths, EQ.2: resultants at plan points and fiber lengths, EQ.3: resultants, stresses at all points, fiber lengths.
QR	Quadrature rule: LT.0.0: absolute value is specified rule number, EQ.0.0: Gauss (up to ten points are permitted), EQ.1.0: trapezoidal, not recommended for accuracy reasons.
ICOMP	Flag for layered composite material mode: EQ.1: a material angle is defined for each through thickness integration point. For each layer one integration point is used.
TSHEAR	Flag for transverse shear strain or stress distribution (see Remarks 4 and 5): EQ.0: Parabolic, EQ.1: Constant through thickness.
B1	β_1, material angle at first integration point. The same procedure for determining material directions is use for thick shells that is used for the 4 node quadrilateral shell.
B2	β_2, material angle at second integration point
B3	β_3, material angle at third integration point
⋮	⋮
BNIP	β_{NIP}, material angle at NIPth integration point

Remarks:

1. **Thick Shell Element Formulations.** Thick shell elements are bending elements that have 4 nodes on the bottom face and 4 on the top face. Thick shell element formulations 1, 2 and 6 are extruded thin shell elements and use thin shell material models and have an uncoupled stiffness in the z-direction. Thick shell element formulations 3, 5, and 7 are layered brick elements that use 3D brick material models. Thick shell formulations 3 and 5, and 6 are distortion sensitive and should not be used in situations where the elements are badly shaped. A single thick shell element through the thickness will capture bending response, but with thick shell formulation 3, at least two elements through the thickness are recommended to avoid excessive softness.

2. **Formulation 1 Quadrature Quirk.** When using Gaussian quadrature with element formulation 1, the number of integration points is automatically switched to 3 when NIP = 2 and 5 when NIP = 4.

3. **Implicit Time Integration.** Thick shell elements are available for implicit analysis with the exception of thick shell formulation 1. If an element of type 1 is specified in an implicit analysis, it is internally switched to type 2

4. **SHRF Field.** For ELFORM = 1, 2 and 6, the transverse shear stiffness is scaled by the SHRF parameter. Since the strain is assumed to be constant through the thickness, setting SHRF = 5/6 is recommended to obtain the correct shear energy. For ELFORM = 3 and 5, the SHRF parameter is not used, except for material types 33, 36, 133, 135, and 243. For ELFORM = 3, the shear stiffness is assumed constant through the thickness. For ELFORM = 5, 6, and 7, the shear distribution is assumed either parabolic if TSHEAR = 0, or constant if TSHEAR = 1. The parabolic assumption is good when the elements are used in a single layer to model a shell type structure, but the constant option may be better when elements are stacked one on top of the other.

5. **Modeling Composites.** Thick shell elements of all formulations can be used to model layered composites, but element formulations 5 and 6 use assumed strain to capture the complex Poisson's effects and through thickness stress distribution in layered composites. To define the layers of a composite, use QR < 0 to point to *INTEGRATION_SHELL data. Alternatively, the *PART_COMPOSITE_TSHELL keyword offers a simplified way to define the layers.

   When modeling composites, laminated shell theory may be used to correct the transverse shear strain if the shear stiffness varies by layer. Laminated shell theory is activated by setting LAMSHT = 4 or 5 on *CONTROL_SHELL. When laminated shell theory is active, the TSHEAR parameter works with all ELFORM values to select either a parabolic or constant shear stress distribution.