*INITIAL

The keyword *INITIAL provides a way of initializing velocities and detonation points. The keyword control cards in this section are defined in alphabetical order:

*INITIAL_AIRBAG_PARTICLE_POSITION

*INITIAL_ALE_MAPPING

*INITIAL_AXIAL_FORCE_BEAM

*INITIAL_CONTACT_WEAR

*INITIAL_CPG

*INITIAL_CRASHFRONT

*INITIAL_DETONATION

*INITIAL_DETONATION_GEOMETRY

*INITIAL_EOS_ALE

INITIAL_FATIGUE_DAMAGE_RATIO_*{OPTION}

INITIAL_FOAM_REFERENCE_GEOMETRY_*{OPTION}

*INITIAL_GAS_MIXTURE

*INITIAL_HISTORY_NODE

*INITIAL_HYDROSTATIC_ALE

INITIAL_IMPULSE_MINE

INITIAL_INTERNAL_DOF_SOLID_*OPTION

INITIAL_LAG_MAPPING_*{OPTION}

*INITIAL_MOMENTUM

INITIAL_PWP_DEPTH_*{OPTION}

*INITIAL_PWP_NODAL_DATA

*INITIAL_SOLID_VOLUME

*INITIAL_STRAIN_BEAM

*INITIAL_STRAIN_IGA_SHELL

INITIAL_STRAIN_SHELL_*{OPTION}

*INITIAL_STRAIN_SHELL_NURBS_PATCH

INITIAL_STRAIN_SOLID_*{OPTION}

*INITIAL_STRAIN_SOLID_NURBS_PATCH

*INITIAL_STRAIN_TSHELL

*INITIAL_STRESS_BEAM

INITIAL_STRESS_DEPTH_*{OPTION}

*INITIAL_STRESS_DES

*INITIAL_STRESS_IGA_SHELL

*INITIAL_STRESS_SECTION

INITIAL_STRESS_SHELL_*{OPTION}

*INITIAL_STRESS_SHELL_NURBS_PATCH

INITIAL_STRESS_SOLID_*{OPTION}

*INITIAL_STRESS_SOLID_NURBS_PATCH

*INITIAL_STRESS_SPH

*INITIAL_STRESS_TSHELL

INITIAL_TEMPERATURE_*OPTION

*INITIAI_VAPOR_PART

*INITIAL_VEHICLE_KINEMATICS

There are two alternative sets of keywords for setting initial velocities. Cards from one set cannot be combined with cards from the other. **Standard velocity cards:**

*INITIAL_VELOCITY

*INITIAL_VELOCITY_NODE

*INITIAL_VELOCITY_RIGID_BODY

Alternative initial velocity cards supporting initial rotational about arbitrary axes and start times. **Alternative velocity cards:**

*INITIAL_VELOCITY_GENERATION

*INITIAL_VELOCITY_GENERATION_START_TIME

INITIAL_VOID_*OPTION

INITIAL_VOLUME_FRACTION_*{OPTION}

*INITIAL_VOLUME_FRACTION_GEOMETRY

*INITIAL_AIRBAG_PARTICLE_POSITION

Purpose: This card initializes the position of CPM initial air particle to the location specified.

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

Particle Cards. The i^{th} card specifies the location of the i^{th} particle. LS-DYNA expects one card for each particle, if fewer cards are supplied the coordinates will be reused and particles may share the same location at the beginning of the simulation.

Card	1	2	3	4	5	6	7	8
Variable		X	Y	Z				
Type	8x	F	F	F				
Default								

VARIABLE	DESCRIPTION
Bag_ID	Airbag ID defined in *AIRBAG_PARTICLE_ID card
X	*x* coordinate
Y	*y* coordinate
Z	*z* coordinate

*INITIAL_ALE_MAPPING

Purpose: This card initializes the current ALE run with data from the last cycle of a previous ALE run. A mapping file or files must be specified on the command line to provide the data to initialize the run. To map from a single mapping file, use map= on the command line (see Remark 6). To map from multiple mapping files, use map2= on the command line (see Remark 8). To map data histories (not just the last cycle) to a region of selected elements, see *BOUNDARY_ALE_MAPPING.

The following transitions are allowed:

1D → 2D	2D → 2D	3D → 3D
1D → 3D	2D → 3D	3D → 2D

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

Card 2	1	2	3	4	5	6	7	8
Variable	X0	Y0	Z0	VECID	ANGLE	SYM	TBEG	
Type	F	F	F	I	F	I	F	
Default	0.0	0.0	0.0	none	none	0	0.0	

VARIABLE	DESCRIPTION
PID	Part ID or part set ID.
TYP	Type of "PID" (see Remark 1): EQ.0: Part set ID (PSID). EQ.1: Part ID (PID).
AMMSID	Set ID of ALE multi-material groups defined in *SET_MULTI-MATERIAL_GROUP. See Remark 1.

VARIABLE	DESCRIPTION
XO	Origin position in global x-direction. See Remarks 2 and 5.
YO	Origin position in global y-direction. See Remarks 2 and 5.
ZO	Origin position in global z-direction. See Remarks 2 and 5.
VECID	ID of the symmetric axis defined by *DEFINE_VECTOR. See Remarks 3 and 5.
ANGLE	Angle of rotation in degrees around an axis defined by *DEFINE_VECTOR for the 3D to 3D mapping. See Remark 4.
SYM	Treatment of the elements and nodes that are out of the mapping bounds (meaning the coordinates of their projections on the previous mesh are outside this mesh). SYM is a 6-digit parameter. Each digit represents a plane for a box that encloses the previous mesh. These planes are parallel to the previous coordinate system. The value of the digit determines the rule to be applied, so any combination of rules may be used. EQ.00000p: Rule for the X-plane along the lower previous mesh bound EQ.0000p0: Rule for the X-plane along the upper previous mesh bound EQ.000p00: Rule for the Y-plane along the lower previous mesh bound EQ.00p000: Rule for the Y-plane along the upper previous mesh bound EQ.0p0000: Rule for the Z-plane along the lower previous mesh bound EQ.p00000: Rule for the Z-plane along the upper previous mesh bound The value of p defines the rule to be applied to the given plane: p.EQ.0: Do nothing. p.EQ.1: Translational symmetry (direction of translation orthogonal to the box plane) p.EQ.2: Mirror-image symmetry about the box plane p.EQ.3: Continuity of boundary elements and nodes along the box's plane

VARIABLE	DESCRIPTION
TBEG	Time to start the run. It replaces the termination time of the previous run that generated the mapping file if TBEG is larger.

Remarks:

1. **Mapping of ALE multi-material groups.** The routines of this card need to know which mesh will be initialized with the mapping data, and more specifically, which multi-material groups. The first two fields, PID, and TYP, define the mesh. The third field, AMMSID, refers to a multi-material group list ID; see the *SET_MULTI-MATERIAL_GROUP_LIST card. The group list AMMSID should have as many elements as there are groups in the previous calculation (see *ALE_MULTI-MATERIAL_GROUP).

   Example: If the previous model has 3 groups, the current one has 5 groups and the following mapping is wanted.

   Group 1 from the previous run → Group 3 in the current run
   Group 2 from the previous run → Group 5 in the current run
   Group 3 from the previous run → Group 4 in the current run

   The *SET_MULTI-MATERIAL_GROUP_LIST card should be set as follows:

```
*SET_MULTI-MATERIAL_GROUP_LIST
300
3,5,4
```

   In special cases, a group can be replaced by another. If Group 4 in the previous example should be replaced by Group 3, the keyword setup would be modified to have -3 instead of 4. The minus sign is a way for the code to know that the replacing group (-3 replaces 4) is a complement of the group 3:

```
*SET_MULTI-MATERIAL_GROUP_LIST
300
3,5,-3
```

2. **Coordinate system origin.** The location to which the data is mapped is controlled by the origin of the coordinate system (XO, YO, ZO).

3. **Symmetry axis.** For a mapping file created by a previous axisymmetric model, the symmetric axis orientation in the current model is specified by VECID. For a mapping file created by a 3D or 1D spherical model, VECID is read but ignored. For a mapping file created by a 1D plane strain model, the vector specified with VECID orients the beams in the current mesh. For a 3D to 3D mapping the vector is used if the parameter ANGLE is defined (see Remark 4).

4. **Rotating 3D data onto a 3D calculation.** For a mapping from a previous 3D run to a current 3D model the previous 3D data will be rotated about the vector, VECID, through an angle specified in the ANGLE field.

5. **Plain strain and 3D to 2D.** The definitions of X0, Y0, Z0 and VECID change in the case of the following mappings:

   a) plain strain 2D (ELFORM = 13 in *SECTION_ALE2D) to plain strain 2D

   b) plain strain 2D to 3D

   c) 3D to 2D

   While VECID still defines the *y*-axis in the 2D domain, the 3 first parameters in *DEFINE_VECTOR, additionally, define the location of the origin. The 3 last parameters define a position along the *y*-axis. For this case when 2D data is used in a 3D calculation the point X0, Y0, Z0 together with the vector, VECID, define the plane.

6. **Mapping file.** Including the command line argument "map=" will invoke the creation of a mapping file. When the keyword INITIAL_ALE_MAPPING is not in the input deck, but the argument "map=" is present on the command line, the ALE data from the last cycle is written in the mapping file. This file contains the following nodal and element data:

   - nodal coordinates (last step)
   - nodal velocities
   - part IDs
   - element connectivities
   - element centers
   - densities
   - volume fractions
   - stresses
   - plastic strains
   - internal energies
   - bulk viscosities
   - relative volumes

7. **Chained mappings.** To chain mapping operations so that LS-DYNA both reads and writes a mapping file the command line argument "map1=" is necessary. If the keyword INITIAL_ALE_MAPPING is in the input deck and the prompt map= is in the command line, the ALE data is read from the mapping file defined

by map= to initialize the run. Data from the last cycle are written in the mapping file defined by map1=.

8. **Mapping from multiple mapping files.** The command line option map2 = enables mapping from multiple mapping files in the run. The file specified with this command line option needs to be a .txt file that lists all the mapping files from which to map. Note that each specified mapping file should map to a different portion of the domain.

*INITIAL_AXIAL_FORCE_BEAM

Purpose: Initialize the axial force resultants in beam elements that are used to model bolts. This option works for beam type 9 (a Hughes-Liu type beam) with *MAT_SPOTWELD only and for beam type 1 with any material.

Card 1	1	2	3	4	5	6	7	8
Variable	BSID	LCID	SCALE	KBEND				
Type	I	I	F	I				
Default	none	none	1.0	0				

VARIABLE	DESCRIPTION
BSID	Beam set ID
LCID	Load curve ID defining preload force versus time. When the load curve ends or goes to zero, the initialization is assumed to be completed. See Remark 2 below.
SCALE	Scale factor on load curve.
KBEND	Bending stiffness flag EQ.0: Bending stiffness is negligible since all integration points are assigned the same axial stress. EQ.1: Bending stiffness is retained by keeping the axial stress gradient. EQ.2: Same as 1, but also allows for lining up several beams with prescribed axial force. See Remark 3.

Remarks:

1. **Damping.** To achieve convergence during explicit dynamic relaxation, the application of the damping options is very important. If contact is active, contact damping is recommended with a value between 10-20 percent. Additional damping applied using *DAMPING_PART_STIFFNESS also speeds up convergence where a coefficient of 0.10 is effective. If damping is not used, convergence may not be possible.

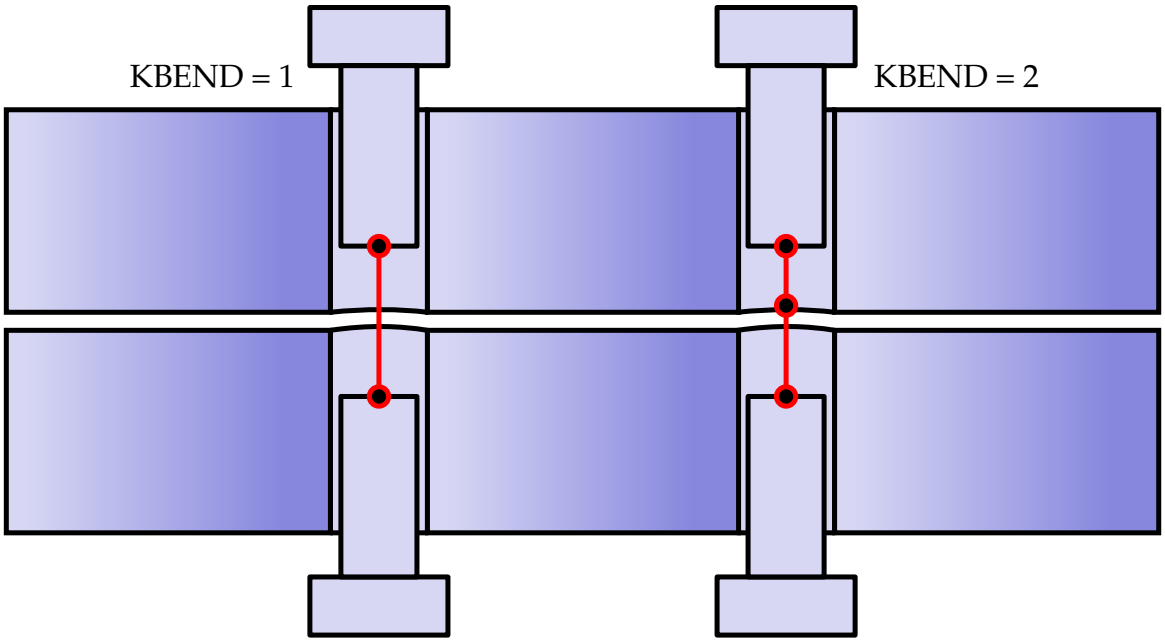

Figure 28-1. Illustration of modeling strategies for different values of KBEND

2. **Ramping.** When defining the load curve, LCID, a ramp starting at the origin should be used to increase the force to the desired value. The time duration of the ramp should produce a quasistatic response. When the end of the load curve is reached, or when the value of the load decreases from its maximum value, the initialization stops. If the load curve begins at the desired force value, that is, no ramp, convergence will take much longer, since the impulsive-like load created by the initial force can excite nearly every frequency in the structural system where force is initialized.

3. **Modeling Strategy.** When preloading bolts, users commonly reserve *one* beam element along the shank/threaded part of each bolt for the *INITIAL_AXIAL_-FORCE_BEAM keyword. The selected beam element must be long enough such that it does not reduce to zero length by the end of preload. The formulation of the beam causing crimping upon loading. Mesh connections, boundary conditions or contact prevent crimping. We illustrate this strategy with the left bolt in Figure 28-1. The beam to be load is red. Note that the loaded beam must be significantly longer than the sum of *(i)* the gap between the two plates and *(ii)* the two gaps between the plates and bolt heads. We use only one beam because otherwise the position of any node connecting two preloaded beam elements cannot be determined from the equations of equilibrium; the positions of intermediate nodes along the axis of the bolt are arbitrary. With KBEND = 2, you can load more than one beam as shown by the right bolt in the figure. LS-DYNA internally imposes additional constraints for intermediate nodes to make the problem well posed. The length reduction is displacement controlled to avoid excessive dynamic effects. Dynamic effects might otherwise occur if the bolt

heads impact the plates to be clamped at high speed. Note that the previously discussed differences apply even if only one beam element is used for the pre-load.

*INITIAL_CONTACT_WEAR

Purpose: Initialize contact wear for simulation of wear processes. Define as many cards as desired. See Remarks.

Note that this card is not supposed to be manually inserted; rather it is supposed to be generated by LS-DYNA during wear processes.

Card 1	1	2	3	4	5	6	7	8
Variable	CID	NID	WDEPTH	NX	NY	NZ	ISEQ	NCYC
Type	I	I	F	F	F	F	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
CID	Contact Interface ID.
NID	Node ID.
WDEPTH	Wear depth, in units of length.
NX, NY, NZ	Direction vector for wear, internally normalized.
ISEQ	Simulation sequence number for the entire process.
NCYC	The wear on this card will be processed NCYC times to modify the worn geometry. This means that *one* LS-DYNA simulation is used to predict the wear for *NCYC* repetitions of the process to save simulation time. This number should be chosen with care; a negative number means that LS-DYNA will not apply this card. See Remarks below.

Remarks:

This card is not supposed to be manually inserted, but is automatically generated by LS-DYNA when simulating wear processes; see *CONTACT_ADD_WEAR and parameters NCYC on *INTERFACE_SPRINGBACK_LSDYNA and SPR/MPR on *CONTACT. A sequence of identical simulations, except for perturbation of the geometry of certain components due to wear, is undertaken. For a given contact interface and node ID, the corresponding node is perturbed by the wear depth in the direction of wear. If the cycle

number NCYC is negative, the geometry has been already processed in LS-PrePost and the card is ignored by LS-DYNA. If a node appears multiple times, the wear from the individual sequences is accumulated.

*INITIAL_CPG

Purpose: Initialize fluid quantities in the CPG domain.

Card 1	1	2	3	4	5	6	7	8
Variable	BAGID	ITYPE	IDDR	PREI	TEMPI			
Type	I	I	I	F	F			
Default	none	none	none	none	none			

Optional card.

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

Initial Gas Fraction Card. Define NGAS cards.

Card 2.1	1	2	3	4	5	6	7	8
Variable	XFRAC*i*							
Type	F							
Default	none							

VARIABLE	DESCRIPTION
BAGID	Airbag ID defined in *AIRBAG_CPG card
ITYPE	Defines the initialization type: EQ.1: Give initial pressure and temperature.

VARIABLE	DESCRIPTION
IDDR	ID of the region where the initial conditions will apply. See *DEFINE_CPG_REGION.
PREI/TEMPI	Initial pressure and temperature in the region defined by IDDR for ITYPE = 1
NGAS	Initial gas fractions for the gases given in the same order as in *AIRBAG_CPG. NGAS defined here cannot be larger than NGAS defined in *AIRBAG_CPG.
XFRAC*i*	Initial gas fractions for the different gases

*INITIAL_CRASHFRONT

Purpose: For some composite materials, the strength of an element is reduced if one of its neighboring elements fails due to damage. The damaged nodes are considered crash-front. In general, all nodes are initially perfect and there are no initial crashfront nodes. However, this card allows the user to initialize crashfront nodes for damage that may occur prior to the simulation, such as during the manufacturing process. Any elements connected to these crashfront nodes are initialized with reduced strength.

The crashfront node set can be initialized using the nodes that are in a set of segments, shell elements, parts, or nodes. This keyword works with material models 17, 54, 55, 58, 169, 261, and 262.

Card 1	1	2	3	4	5	6	7	8
Variable	SID	STYPE						
Type	I	I						
Default	none	0						

VARIABLE	DESCRIPTION
SID	Set ID from which the initial crashfront nodes are defined.
STYPE	ID type of SID: EQ.0: segment set ID, EQ.1: shell element set ID, EQ.2: part set ID, EQ.3: part ID, EQ.4: node set ID.

***INITIAL_DETONATION** *{OPTION}*

Available options include:

<BLANK>

SET

Purpose: Define points to initiate the location of high explosive detonations in part IDs that use *MAT_HIGH_EXPLOSIVE_BURN (*MAT_008). Also, see *CONTROL_EXPLOSIVE_SHADOW. If no *INITIAL_DETONATION is defined, detonation occurs in all the high explosive elements at time = 0.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	X	Y	Z	LT		MMGSET	
Type	I	F	F	F	F		I	
Default	all HE	0.	0.	0.	0.		0	

Acoustic Boundary Card. Additional card for PID = -1.

Card 2	1	2	3	4	5	6	7	8
Variable	PEAK	DECAY	XS	YS	ZS	NID		
Type	F	F	F	F	F	I		

VARIABLE	DESCRIPTION
PID	Part or part set ID of the high explosive to be lit, except in the case where the high explosive is modeled using an ALE formulation, in which case PID is the part, part set, or element set ID of the mesh where the high explosive material to be lit initially resides. However, two other options are available: EQ.-1: An acoustic boundary or *BOUNDARY_USA_SURFACE EQ.0: All high explosive materials are considered. LT.-1: |PID| is the ID of a part set (*SET_PART) GT.0: PID is a part ID unless using the SET keyword option.

VARIABLE	DESCRIPTION
	For the set keyword option, it is an element set (*SET_BEAM for ALE 1D models, *SET_SHELL for ALE 2D models, or *SET_SOLID for ALE 3D models).
X	x-coordinate of detonation point, see Figure 28-2.
Y	y-coordinate of detonation point.
Z	z-coordinate of detonation point.
LT	Lighting time for detonation point. This time is ignored for an acoustic boundary.
MMGSET	ID of *SET_MULTI-MATERIAL_GROUP_LIST selecting the explosive ALE groups to be lit in the mesh defined by PID.
PEAK	Peak pressure, p_o, of the incident pressure pulse. See Remark 3.
DECAY	Decay constant, τ. See Remark 3.
XS	x-coordinate of standoff point; see Figure 28-2.
YS	y-coordinate of standoff point
ZS	z-coordinate of standoff point
NID	Reference node ID near structure

Remarks:

1. **Lighting time.** For solid elements (not acoustic) two options are available. If the control card option *CONTROL_EXPLOSIVE_SHADOW is not used, the lighting time for an explosive element is computed using the distance from the center of the element to the nearest detonation point, L_d; the detonation velocity, D; and the lighting time for the detonator, t_d:

$$t_L = t_d + \frac{L_d}{D}.$$

The detonation velocity for this default option is taken from the element whose lighting time is computed and does not account for the possibilities that the detonation wave may travel through other explosives with different detonation velocities or that the line of sight may pass outside of the explosive material.

If the control card option *CONTROL_EXPLOSIVE_SHADOW is defined, the lighting time is based on the shortest distance through the explosive material. If

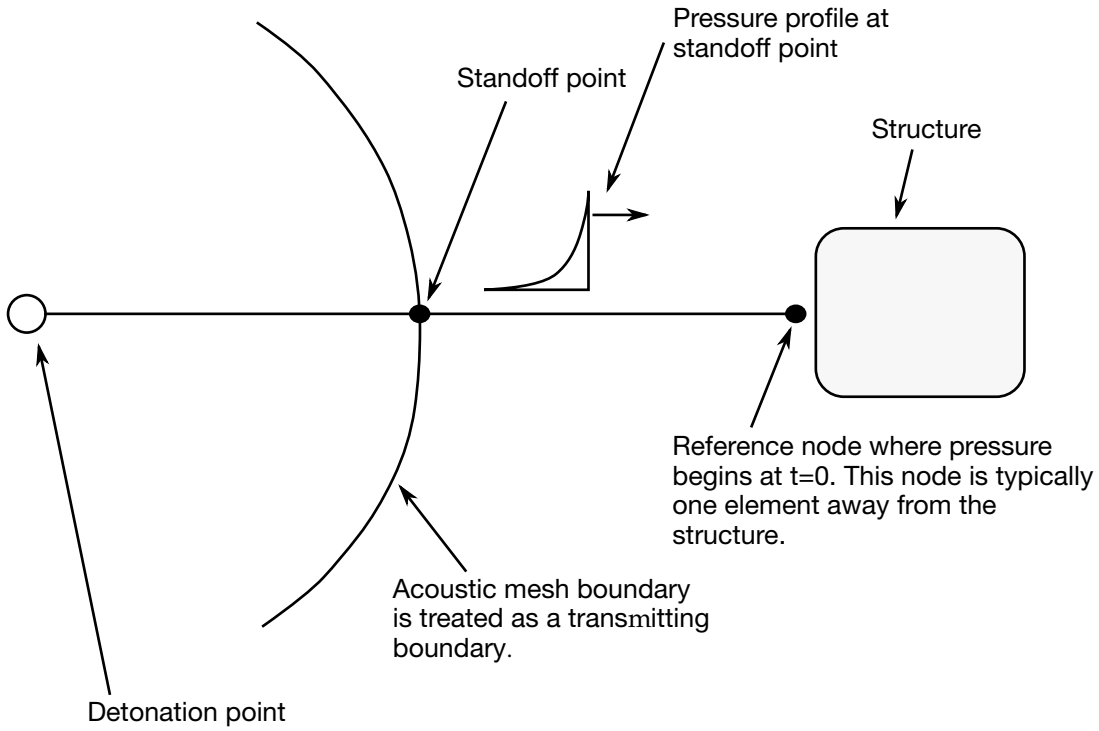

Figure 28-2. Initialization of the initial pressures due to an explosive disturbance is performed in the acoustic media. LS-DYNA automatically determines the acoustic mesh boundary and applies the pressure time history to the boundary. This option is only applicable to the acoustic element formulation; see *SECTION_SOLID.

inert obstacles exist within the explosive material, the lighting time will account for the extra time required for the detonation wave to travel around the obstacles. The lighting times also automatically accounts for variations in the detonation velocity if different explosives are used. No additional input is required for this option, but care must be taken when setting up the input. This option works for two and three-dimensional solid elements. It is recommended that for best results:

a) Keep the explosive mesh as uniform as possible with elements of roughly the same dimensions.

b) Inert obstacle such as wave shapers within the explosive must be somewhat larger than the characteristic element dimension for the automatic tracking to function properly. Generally, a factor of two should suffice. The characteristic element dimension is found by checking all explosive elements for the largest diagonal.

c) The detonation points should be either within or on the boundary of the explosive. Offset points may fail to initiate the explosive. When LT is

nonzero, the detonation point is fixed to the explosive material at $t = 0$ and moves as the explosive material moves prior to detonation.

d) Check the computed lighting times in the post processor LS-PrePost. The lighting times may be displayed at time = 0., state 1, by plotting component 7 (a component normally reserved for plastic strain) for the explosive material. The lighting times are stored as negative numbers. The negative lighting time is replaced by the burn fraction when the element ignites.

2. **Line detonations.** Line detonations may be approximated by using a sufficient number of detonation points to define the line. Too many detonation points may result in a significant initialization cost.

3. **Acoustic boundary.** The pressure as a function of time curve for the acoustic boundary is defined by:

$$p(t) = p_o e^{-\frac{t}{\tau}}.$$

*INITIAL_DETONATION_GEOMETRY

Purpose: Define detonation points that are selected based on a specified geometry. These points ignite high explosives defined by parts that use *MAT_HIGH_EXPLOSIVE_BURN (*MAT_008).

Also see *INITIAL_DETONATION and *CONTROL_EXPLOSIVE_SHADOW.

Card Summary:

Card 1. This card is required.

HEID	HETYP	MMGSET					

Card 2. This card is required.

GEOTYP	LT	DGEO	BOXID				

Card 3a. This card is included if GEOTYP = 1.

VID1							

Card 3b. This card is included if GEOTYP = 2.

VID1	VID2	VID3	VID4				

Card 3c. This card is included if GEOTYP = 3.

VID1	VID2	VID3					

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	HEID	HETYP	MMGSET					
Type	I	I	I					
Default	0	0	0					

VARIABLE	DESCRIPTION
HEID	ID specifying the high explosives to be lit

VARIABLE	DESCRIPTION
HETYP	Type of HEID: EQ.0: Part set ID (*SET_PART) EQ.1: Part (*PART) EQ.2: Element set ID (*SET_SOLID in 3D and *SET_SHELL in 2D)
MMGSET	ID of *SET_MULTI-MATERIAL_GROUP_LIST selecting the explosive ALE groups to be lit

Card 2	1	2	3	4	5	6	7	8
Variable	GEOTYP	LT	DGEO	BOXID				
Type	I	F	F	I				
Default	none	0.	↓	0				

VARIABLE	DESCRIPTION
GEOTYP	Type of geometry formed by the detonation points: EQ.1: Plane EQ.2: Cylindrical or truncated conical surface EQ.3: Spherical or ellipsoidal surface
LT	Lighting time for detonation point
DGEO	Maximum distance from the detonation geometry for determining which HE elements become detonation points. If the element center for the specified HE is less than this distance away from the detonation geometry, the element center becomes a detonation point. If zero or undefined, DGEO becomes the length of the largest specified HE element. Note that this condition can be limited to only elements inside a box specified with BOXID below.
BOXID	ID of a box (see *DEFINE_BOX). If used, the element center becomes a detonation point if it is both within the box and within the distance specified with DGEO relative to the detonation geometry.

Plane Geometry Card. This card is included if GEOTYP = 1.

Card 3a	1	2	3	4	5	6	7	8
Variable	VID1							
Type	I							
Default	0							

VARIABLE	DESCRIPTION
VID1	ID of the vector (see *DEFINE_VECTOR) that gives the plane's normal vector. The tail of VID1 is a point in the plane.

Cylindrical Surface Geometry Card. This card is included if GEOTYP = 2.

Card 3b	1	2	3	4	5	6	7	8
Variable	VID1	VID2	VID3	VID4				
Type	I	I	I	I				
Default	0	0	0	0				

VARIABLE	DESCRIPTION
VID1, …, VID4	IDs of vectors (see *DEFINE_VECTOR) for specifying the orientation and location of a cylinder or truncate cone with either circular or elliptical bases. The cylinder or truncated cone can also be oblique. For all cases, the tail of VID1 gives the center of one of the bases, the length of VID2 gives the length of the cylinder or truncated cone's axis, and the direction of VID2 gives the direction of the cylinder or truncated cone's axis. For a cylinder with circular bases, the length of VID1 is the radius of the circle. The bases lie in planes formed by VID1 and the cross product of VID1 with VID2. VID3 should not be input for this case. See Figure 28-3. For a cylinder with elliptical bases, the lengths of VID1 and VID3 are the lengths of the two axes. The direction of the axis with the length of VID1 is in the direction of VID1. The direction for the axis

VARIABLE	DESCRIPTION
	with the length given by VID3 is in the direction given by the cross product of VID1 with VID2. See Figure 28-4. For a truncated cone, you must specify VID4 in addition to VID1 and VID2. The bottom base determines if the bases are circular or elliptical based on whether you specify VID3 as previously discussed. VID4 gives the direction and length of the first axis of the second base. It does not need to be parallel to VID1. The second base is scaled uniformly from the first base. See Figure 28-5.

Spherical or Ellipsoidal Surface Geometry Card. This card is included if GEOTYP = 3.

Card 3c	1	2	3	4	5	6	7	8
Variable	VID1	VID2	VID3					
Type	I	I	I					
Default	0	0	0					

VARIABLE	DESCRIPTION
VID1, …, VID3	IDs of vectors (see *DEFINE_VECTOR) for specifying the orientation and location of the sphere or ellipsoid. In each case, the center of the surface is given by the tail of VID1. For a sphere, only specify VID1. The length of VID1 is the radius of the sphere. For an ellipsoid with two axes of equal length, specify VID1 and VID2. The length of VID1 is used for the axes of equal length which are in the directions VID1 and the cross product of VID1 with VID2. The length of VID2 gives the length of the third axis which is orthogonal to VID1 and the cross product of VID1 with VID2. See Figure 28-6. For an ellipsoid with all axes of different length, specify VID1, VID2, and VID3. One axis is in the direction of VID1 with the length of VID1. Another axis is in the direction of the cross product of VID1 with VID2 with the length given by the length of VID3. The third axis is orthogonal to the other two axes and has a length equal to the length of VID2. See Figure 28-7.

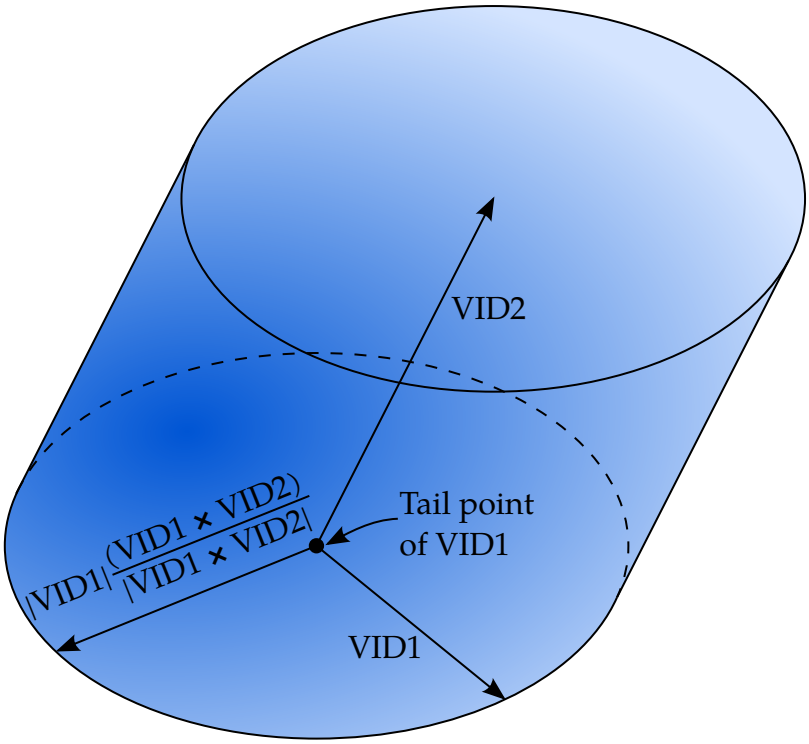

Figure 28-3. Oblique cylinder with circular bases (GEOTYP = 2)

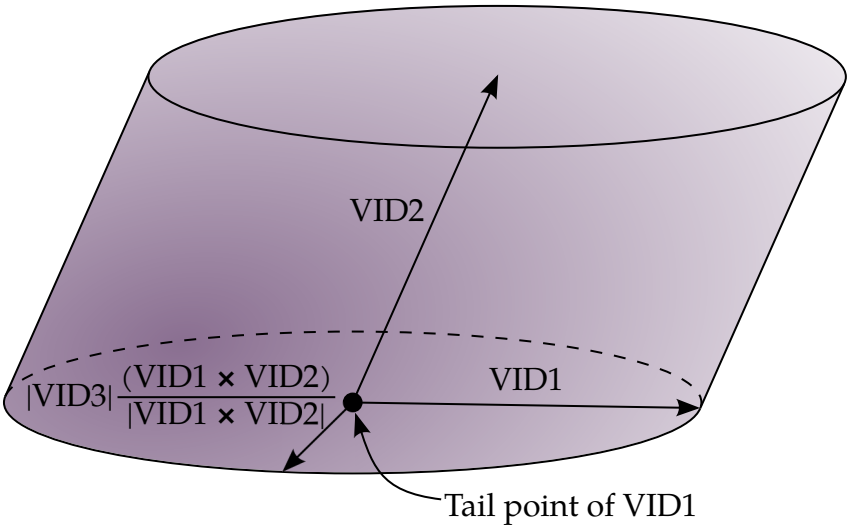

Figure 28-4. Oblique cylinder with elliptical bases (GEOTYP = 2)

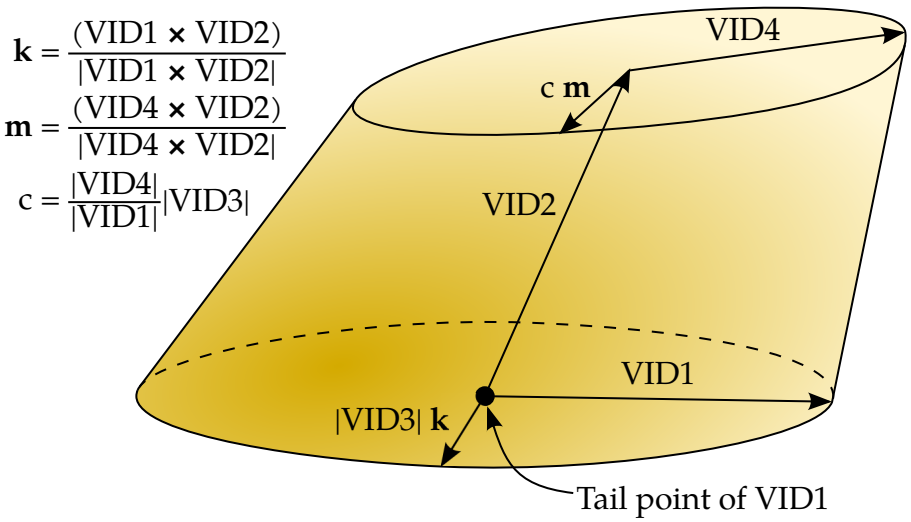

Figure 28-5. Oblique truncated cone with elliptical bases (GEOTYP = 2)

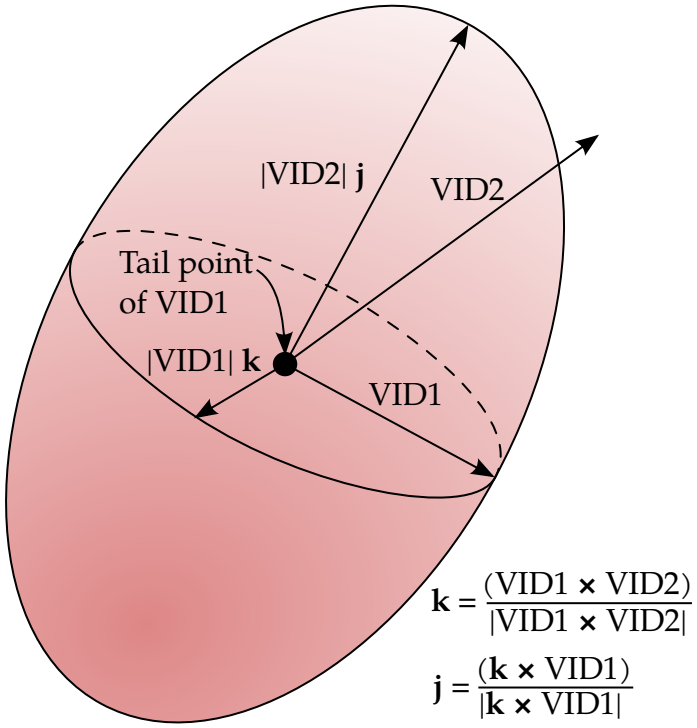

Figure 28-6. Ellipsoid with two equal axes (GEOTYP = 3)

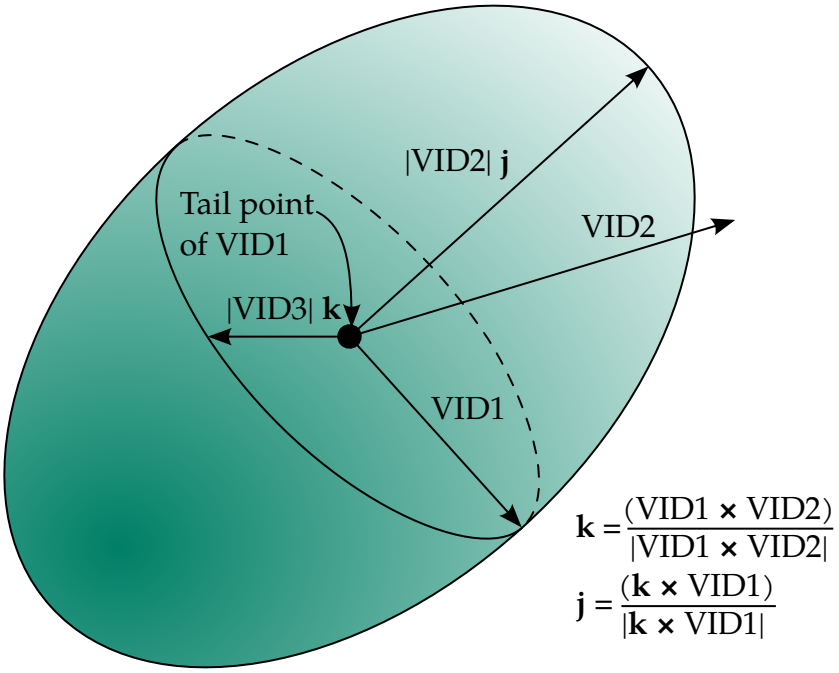

Figure 28-7. Ellipsoid with axes of different length (GEOTYP = 3)

*INITIAL_EOS_ALE

Purpose: This card initializes the pressure in ALE elements that have materials with *EOS.

Card 1	1	2	3	4	5	6	7	8
Variable	ID	TYP	MMG	E0	V0	P0		
Type	I	I	I	F	F	F		
Default	none	none	none	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
ID	Part ID or part set ID or element set ID.
TYP	Type of "ID": EQ.0: part set ID. EQ.1: part ID. EQ.2: element set ID (*SET_BEAM in 1D, *SET_SHELL in 2D, *SET_SOLID in 3D).
MMG	Specifies the multi-material group. GT.0: ALE multi-material group. LT.0: Set ID of ALE multi-material groups defined in *SET_MULTI-MATERIAL_GROUP.
E0	Initial internal energy per reference volume unit (as defined in *EOS). See Remark 1.
V0	Initial relative volume (as defined in *EOS). See Remark 1.
P0	Initial pressure. See Remark 2.

Remarks:

1. **Initialization with Volume and Energy.** For most *EOS, E0 and V0 should be used to initialize the pressure. If only the internal energy is initialized, V0 should be 1.0 (If V0 = 0.0, E0 will not be applied).

2. **Initial Pressure with Derived Volume and Energy.** For *EOS_001, *EOS_004 and *EOS_006, the initial pressure P0 can be input directly. An iterative method will compute the initial internal energy and relative volume. This approach is applied if E0 = 0.0 and V0 = 0.0.

*INITIAL_FATIGUE_DAMAGE_RATIO_{OPTION1}_{OPTION2}

Available options for *OPTION1* include:

<BLANK>

D3FTG

D3PLOT

Available options for *OPTION2* include:

<BLANK>

SCALE_FACTOR

Purpose: This card sets the initial damage ratio for fatigue analysis. The initial damage ratio may come from the previous loading cases. The initial damage ratio can be defined directly by the user or can be extracted from an existing binary database like D3FTG (using the option D3FTG) and D3PLOT (using the option D3PLOT). A scale factor for the damage ratio from each previous loading case can be defined if that loading case repeats using the SCALE_FACTOR keyword option

Card Summary:

Card Sets. Include as many sets of the following cards as needed to include the initial damage ratio from each loading condition (for multiple loading condition cases). This input terminates with the next keyword ("*") card.

Card 1a. Include this card if no keyword option is used (<BLANK>).

PID/SID	PTYP	DRATIO					

Card 1b. Include this card if *OPTION1* is D3FTG and *OPTION2* is unused (<BLANK>).

FILENAME

Card 1c. Include this card if *OPTION1* is D3FTG and *OPTION2* is SCALE_FACTOR.

SF	FILENAME

Card 1d. Include this card if *OPTION1* is D3PLOT and OPTION2 is blank.

FILENAME

Card 1e. Include if *OPTION1* is D3PLOT and *OPTION2* is SCALE_FACTOR.

SF	FILENAME

Card 2. Include this card if the D3PLOT keyword option is used.

NSTATE	NEIPHD	NEIPSD					

Data Card Definitions:

Initial Damage Ratio Card. Card 1 for no option (<BLANK>)

Card 1a	1	2	3	4	5	6	7	8
Variable	PID/SID	PTYP	DRATIO					
Type	I	I	F					
Default	none	0	0.0					

VARIABLE	DESCRIPTION
PID/SID	Part ID or part set ID for which the initial damage ratio is defined
PTYP	Type of PID/PSID: EQ.0: Part ID EQ.1: Part set ID
DRATIO	Initial damage ratio

D3FTG Card. Card 1 for the D3FTG keyword option with *OPTION2* unused (<BLANK>).

Card 1b	1	2	3	4	5	6	7	8
Variable	FILENAME							
Type	C							
Default	d3ftg							

VARIABLE	DESCRIPTION
FILENAME	Path and name of the existing binary database for fatigue information

D3FTG Card. Card 1 for the D3FTG keyword option with *OPTION2* as SCALE_FACTOR

Card 1c	1	2	3	4	5	6	7	8
Variable	SF	FILENAME						
Type	F	C						
Default	1.0	d3ftg						

VARIABLE	DESCRIPTION
SF	Scale factor
FILENAME	Path and name of the existing binary database for fatigue information

D3PLOT Card. Card 1 for the D3PLOT keyword option with *OPTION2* unused.

Card 1d	1	2	3	4	5	6	7	8
Variable	FILENAME							
Type	C							
Default	d3plot							

VARIABLE	DESCRIPTION
FILENAME	Path and name of the existing binary database for fatigue information

D3PLOT Card. Card 1 for the D3PLOT keyword option with *OPTION2* as SCALE_FACTOR

Card 1e	1	2	3	4	5	6	7	8
Variable	SF	FILENAME						
Type	F	C						
Default	1.0	d3plot						

VARIABLE	DESCRIPTION
SF	Scale factor
FILENAME	Path and name of the existing binary database for fatigue information

D3PLOT Card. Card 2 for the D3PLOT keyword option.

Card 2	1	2	3	4	5	6	7	8
Variable	NSTATE	NEIPHD	NEIPSD					
Type	I	I	I					
Default	1	1	1					

VARIABLE	DESCRIPTION
NSTATE	State ID in binary database (e.g. d3plot) for reading damage variables
NEIPHD	ID of additional integration point history variable which saves the damage for solid elements
NEIPSD	ID of additional integration point history variable which saves the damage for shell and thick shell elements

Remarks:

1. **Fatigue Problem Types.** This card works for both time-domain fatigue and frequency-domain fatigue problems.

2. **Transient Preload Cases.** The option D3PLOT can be used for users who want to use damage from transient preload cases (e.g. from GISSMO damage model) as the initial fatigue damage ratio. In this case, LS-DYNA reads the binary database defined in Card 1d or 1e (e.g. d3plot) and extracts the damage variables from the additional integration point history variables based on NSTATE, NEIPHD, and NEIPSD defined in Card 2.

*INITIAL_FIELD_SOLID

Purpose: This keyword is a simplified version of *INITIAL_STRESS_SOLID which can be used with hyperelastic materials. The keyword is used for history variable input. Data is usually in the form of the eigenvalues of diffusion tensor data. These are expressed in the global coordinate system.

> **NOTE:** As of LS-DYNA R5 in all contexts, other than *MAT_TISSUE_DISPERSED, this keyword is deprecated (and disabled). For all other materials this keyword has been superceded by *INITIAL_STRESS_SOLID.

Card Sets: Include as many pairs of Cards 1 and 2 as necessary. This input ends at the next keyword ("*") card.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	NINT	NHISV					
Type	I	I	I					
Default	none	none	0					

Card 2	1	2	3	4	5	6	7	8
Variable	FLD1	FLD2	FLD3	FLD4	FLD5	FLD6	FLD7	FLD8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
EID	Element ID
NINT	Number of integration points (should correspond to the solid element formulation).
NHISV	Number of field variables. If NHISV exceeds the number of integration point field variables required by the constitutive model, only the number required is output; therefore, if in doubt, set NHISV to a large number.

VARIABLE	DESCRIPTION
FLD*n*	Data for the n^{th} field (history) variable. NOTE that *MAT_TISSUE_DISPERSED only use FLD1 to FLD3 since NHISV = 3.

Remarks:

Add as many cards as necessary. The keyword input ends at the next keyword ("*") card. For example for two elements it can look as:

```
*INITIAL_FIELD_SOLID
$EID       NINT     NHISV
    1          1         3
$FLD1      FLD2      FLD3
  0.1       0.8       0.1
$EID       NINT     NHISV
    2          1         3
$FLD1      FLD2      FLD3
  0.3       0.2       0.5
```

*INITIAL_FOAM_REFERENCE_GEOMETRY_{OPTION}

Available options include:

<BLANK>

RAMP

Purpose: Define the reference geometry of a foam for stress initialization. LS-DYNA initializes the stresses using the reference geometry when the REF flag is turned on in the *MAT input for the following hyperelastic material models: 2, 5, 7, 21, 23, 27, 31, 38, 57, 73, 77, 83, 132, 179, 181, 183, and 189. Supported solid elements are the constant stress hexahedron (#1), the fully integrated S/R hexahedron (#2), the tetrahedron (#10), and the pentahedron (#15). This keyword is also supported for plane strain (shell #13) and axisymmetric solid (#14 and #15) elements. With this keyword, dynamic relaxation can be avoided once a deformed configuration is obtained, usually on the first run of a particular problem.

To use this keyword, define the geometry of the low-density foam in a deformed configuration. The stresses depend only on the deformation gradient matrix, F_{ij}:

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

where x_i is the deformed configuration and X_j is the undeformed configuration.

Some shell and solid elements modeling the fabric and foam, respectively, share nodes. The reference geometry specified by either *INITIAL_FOAM_REFERENCE_GEOMETRY or *AIRBAG_REFERENCE_GEOMETRY will be applied to both the shell and solid elements that share these nodes. However, having different reference geometry for shell and solid elements sharing common nodes can be achieved by using *INITIAL_FOAM_REFERENCE_GEOMETRY for solid elements and *AIRBAG_SHELL_REFERENCE_GEOMETRY for shell elements.

Card Summary:

Card 1. This card is included if the RAMP option is used.

NDTRRG							

Card 2. Include as many of this card as needed. The next keyword ("*") card terminates this input.

NID	X	Y	Z			

Data Card Definitions:

RAMP Card. Additional card for the option of RAMP.

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

Include as many cards as necessary. This input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	NID	X		Y		Z				
Type	I	F		F		F				
Default	none	0.		0.		0.				

VARIABLE	DESCRIPTION
NDTRRG	Number of time steps taken for an element to restore its reference geometry, for 3D solid elements only. The definition of NDTRRG allows an element to ramp up to its reference shape in NDTRRG time steps. Currently, LS-DYNA uses only one value of NDTRRG and applies it to all foam materials with reference geometries. If more than one NDTRRG is defined, the one defined last will be used.
NID	Node number
X	x-coordinate in reference configuration
Y	y-coordinate in reference configuration
Z	z-coordinate in reference configuration

*INITIAL_GAS_MIXTURE

Purpose: This command is used to specify (a) which ALE multi-material groups may be present inside an ALE mesh set at time zero, and (b) the corresponding reference gas temperature and density which define the initial thermodynamic state of the gases. The order of the species in the gas mixture corresponds to the order of different gas species defined in the associated *MAT_GAS_MIXTURE card. This card must be used together with a *MAT_GAS_MIXTURE card or, equivalently, a *MAT_ALE_GAS_MIXTURE card.

Card 1	1	2	3	4	5	6	7	8
Variable	SID	STYPE	MMGID	TEMP				
Type	I	I	I	F				
Default	none	0	none	none				

Card 2	1	2	3	4	5	6	7	8
Variable	R01	R02	R03	R04	R05	R06	R07	R08
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
SID	Set ID for initialization. This SID defines the ALE mesh within which certain ALE multi-material group(s) may be present at t = 0.
STYPE	Set type for the SID above: EQ.0: SID is a part set ID EQ.1: SID is a part ID
MMGID	ALE multi-material group ID of the material that may be present at t = 0 in the ALE mesh set defined by SID. For general ALE, it must be AMMGID. For S-ALE, either AMMGID or AMMG name (AMMGNM) could be used here. Please refer to *ALE_STRUC-

VARIABLE	DESCRIPTION
	TURED_MULTI-MATERIALS_GROUP for more details.
TEMP	Initial static temperature of the gas species occupying the ALE mesh. Note that all species in the mixture are assumed to be in thermal equilibrium (having the same T).
RO1-RO8	Initial densities of the ALE material(s), which may occupy some region (or all) of the aforementioned ALE mesh, for up to eight different gas species. The order of the density input corresponds to the order of the materials defined in the associated *MAT_GAS_MIXTURE card.

Remarks:

1. **Example.** Please see the example under the *MAT_GAS_MIXTURE card definition for an application of the *INITIAL_GAS_MIXTURE card.

2. **Initial pressure.** The temperature is assumed to be the initial temperature which together with the gas density, will define the initial pressure of the gas species using the perfect gas law,

$$P|_{t=0} = \rho|_{t=0}(C_P - C_V)T|_{t=0} \ .$$

The user should manually check the initial pressure for consistency.

3. **ALE multi-material groups.** Given an ALE mesh, this mesh may initially be occupied by one or more ALE multi-material groups (AMMG). For example, a background ALE mesh (H1) containing AMMG 1 may be partially filled with AMMG 2 using the volume filling command *INITIAL_VOLUME_FRACTION_GEOMETRY. Then there are two AMMGs to be initialized for this mesh H1. The commands look like the following.

```
$---------------------------------------------------------------------
$ One card is defined for each AMMG that will occupy some elements of a mesh
set
*INITIAL_GAS_MIXTURE
$      SID     STYPE     MMGID        T0
         1         1         1   298.15
$     RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7
RHO8
    1.0E-9
*INITIAL_GAS_MIXTURE
$      SID     STYPE     MMGID        T0
         1         1         2   298.15
$     RHO1      RHO2      RHO3      RHO4      RHO5      RHO6      RHO7
RHO8
    1.2E-9
$---------------------------------------------------------------------
```

***INITIAL_HISTORY_NODE**_{OPTION}_

Available options include:

<BLANK>

SET

Purpose: Initialize certain history variable values on a nodal basis for shells, thick shells, and solids. The value of the history variable at an integration point is then determined by interpolating the nodal values as described in more detail in Remark 1.

Card Sets per NODE. Define as many nodes or node sets in this section as desired. The input is assumed to terminate when a new keyword ("*") card is detected.

Node Card.

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

History Variable Cards. Include NHISV (see Card 1) cards, one card per history variable.

Card 2	1	2	3	4	5	6	7	8
Variable	HINDEX	VAL						
Type	I	F						
Default	none	0.0						

VARIABLE	DESCRIPTION
NID/NSID	Node ID or node set ID, see *SET_NODE_…
NHISV	Number of history variables to be initialized
HINDEX	Define the index in the history variable vector

VARIABLE	DESCRIPTION
VAL	Define the value of the history variable

Remarks:

1. **Interpolation Methods.** For shell elements, the values of the history variables assigned to its nodes are interpolated to the location of the in-plane integration points using the finite element shape functions associated to the individual element formulation. All the through thickness integration points will receive the same value as the one in the shell reference plane.

   For thick shell and solid elements, the values of the history variables assigned to its nodes are interpolated to the location of all the integration points using the finite element shape functions associated to the individual element formulation.

2. **Number of Nodes Required for Initialization.** Only integration points of an element where *at least one of* its associated nodes has a history value set with this keyword will be initialized. The values of the history variables of the uninitialized nodes are assumed to be 0.0.

3. **Keyword Limitations.** Note that initialization with nodes that are shared between two parts should be performed with caution since this keyword does not discriminate based on the part ID and more than one material may be involved. In the case of more than one material for a given node, LS-DYNA will issue an error saying that the history variable cannot be initialized. Also, this keyword should not be used for composite shells because of the material change through the thickness (see Remark 1).

4. **Comparison to *INITIAL_STRESS_SOLID/SHELL.** When defining initial values of history variables, this keyword serves a similar purpose to *INITIAL_STRESS_SOLID/SHELL. There are two big differences. The first is how the history variables are initialized. With this keyword you input the nodal values for the history variables which are then interpolated for the integration points. For *INITIAL_STRESS_SOLID/SHELL, you input the initial values at each integration point for an element. Another difference is that for this keyword only the desired history variables are initialized. For instance, if you want to initialize history variable 14, with INITIAL_STRESS_SHELL/SOLID you would have to initialize history variables 1 through 14, whereas with this keyword you would only initialize history variable 14:

```
*INITIAL_HISTORY_NODE_SET
nsid,1
14,50.0
```

   or like this

```
*INITIAL_HISTORY_STRESS_SHELL_SET
sid,1,1,14
0.0

,,,,,50.0
```

*INITIAL_HYDROSTATIC_ALE

Purpose: When an ALE model contains one or more regular (not reservoir-type) ALE parts (ELFORM = 11 and AET = 0), this command may be used to initialize the hydrostatic pressure field in the regular ALE domain due to gravity. The *LOAD_BODY_(OPTION) keyword must be defined.

Card 1	1	2	3	4	5	6	7	8
Variable	ALESID	STYPE	VECID	GRAV	PBASE			
Type	I	I	I	F	F			
Default	none	0	none	0.0	0.0			

Multi-material Layers Group Cards. Repeat card 2 as many times as the number of AMMG layers present in the model.

Card 2	1	2	3	4	5	6	7	8
Variable	NID	MMGBLO						
Type	I	I						
Default	none	none						

VARIABLE	DESCRIPTION
ALESID	ALESID is a set ID defining the ALE domain/mesh whose hydrostatic pressure field due to gravity is being initialized by this keyword. See Remark 2 and 4.
STYPE	ALESID set type. See Remark 4. EQ.0: Part set ID (PSID), EQ.1: Part ID (PID), EQ.2: Solid set ID (SSID).
VECID	Vector ID of a vector defining the direction of gravity.

VARIABLE	DESCRIPTION
GRAV	Magnitude of the Gravitational acceleration. For example, in metric units the value is usually set to 9.80665 m/s^2.
PBASE	Nominal or reference pressure at the top surface of all fluid layers. By convention, the gravity direction points from the top layer to the bottom layer. Each fluid layer must be represented by an ALE multi-material group ID (AMMGID or MMG). See Remark 1.
NID	Node ID defining the top of an ALE fluid (AMMG) layer.
MMGBLO	AMMG ID of the fluid layer immediately below this NID. Each node is defined in association with one AMMG layer below it. See Remark 3. In case of S-ALE, AMMG name (AMMGNM) could be also used in place of AMMGID. See Remark 5.

Remarks:

1. **Pressure in Multi-Layer Fluids.** For models using multi-layer ALE Fluids the pressure at the top surface of the top fluid layer is set to PBASE and the hydrostatic pressure is computed as following

$$P = P_{\text{base}} + \sum_{i=1}^{N_{\text{layers}}} \rho_i g h_i \, .$$

2. **Limitations on Element Formulation.** This keyword applies only to the regular ALE parts with ELFORM = 11 and AET = 0 on the *SECTION_SOLID and *SECTION_ALE2D cards (not reservoir-type). This keyword cannot be used to initialize reservoir-type ALE parts (AET = 4). Also, ramping functions are not supported, so the loading is done in one step at $t = 0$. For initializing reservoir-type ALE domain, please review the *ALE_AMBIENT_HYDROSTATIC keyword.

3. **Limitation on EOS Model.** The keyword only supports *EOS_GRUNEISEN and *EOS_LINEAR_POLYNOMIAL, but only in the following two cases:

$$c_3 = c_4 = c_5 = c_6 = 0, \qquad E_0 = 0$$
$$c_4 = c_5 > 0, \qquad c_1 = c_2 = c_3 = c_6 = 0, \qquad V_0 = 0.$$

4. **Structured ALE usage.** When used with structured ALE, the PART and PART set options might not make too much sense. This is because all elements inside a structured ALE mesh are assigned to one single PART ID. If we want to prescribe initial hydrostatic pressure for all the elements inside the structured mesh, we can certainly use that PART ID. But if we only want to do that to some elements, we must generate a solid set which contains those structured ALE

elements. This is done by using the *SET_SOLID_GENERAL keyword with SALECPT option and STYPE = 2 (solid element set ID) on this keyword.

5. **AMMG NAME for S-ALE.** For the general ALE solver, you define each AMMG with *ALE_MULTI-MATERIAL_GROUP. In this case, each AMMG can only be referred to by their AMMGID. The AMMGID for each AMMG is based on the order of appearance of the AMMG in the input deck. For the S-ALE solver, you can define the AMMG using *ALE_STRUCTURED_MULTI-MATERIAL_-GROUP instead of *ALE_MULTI-MATERIAL_GROUP. With *ALE_STRUC-TURED_MULTI-MATERIAL_GROUP, you give each AMMG a name with the field AMMGNM. Each AMMG defined with that keyword can then be referred with either its name or its AMMGID (which is again based on order of appear-ance). We recommend using the name as it leads to fewer errors. For instance, if you add or delete AMMGs, then the AMMGIDs may change. Then, you must find all those references and change them accordingly. With the name, you do not need to modify the input deck for unchanged AMMGs.

Example:

Model Summary: Consider a model consisting of 2 ALE parts, air on top of water.
H1 = AMMG1 = Air part above.
H2 = AMMG2 = Water part below.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ (non-ambient) ALE materials (fluids) listed from top to bottom:
$
$ NID AT TOP OF A LAYER SURFACE         ALE MATERIAL LAYER BELOW THIS NODE
$ TOP OF 1st LAYER -------> 1722        ----------------------------------------
$                                       Air above   = PID 1 = H1 = AMMG1 (AET=0)
$ TOP OF 2nd LAYER -------> 1712        ----------------------------------------
$                                       Water below = PID 2 = H2 = AMMG2 (AET=0)
$ BOTTOM --------------------------------------------------------------------
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*INITIAL_HYDROSTATIC_ALE
$   ALESID     STYPE     VECID      GRAV     PBASE
        12         0        11   9.80665  101325.0
$      NID    MMGBLO
      1722         1
      1712         2
*SET_PART_LIST
        12
         1         2
*ALE_MULTI-MATERIAL_GROUP
         1         1
         2         1
*DEFINE_VECTOR
$       VID        XT        YT        ZT        XH        YH        ZH       CID
        11       0.0       1.0       0.0       0.0       0.0       0.0
*DEFINE_CURVE
         9
               0.000               0.000
               0.001               1.000
              10.000               1.000
*LOAD_BODY_Y
```

```
$     LCID        SF    LCIDDR        XC        YC        ZC
         9   9.80665         0       0.0       0.0       0.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

*INITIAL_IMPULSE_MINE

Purpose: Apply initial velocities to the nodes of a three-dimensional structure due to the impulse imparted by the detonation of a buried land mine. This keyword cannot be used with 2D models or with rigid parts. It is also not supported by MPP (see workaround in Remark 3). This feature is based on the empirical model developed by [Tremblay 1998].

Card 1	1	2	3	4	5	6	7	8
Variable	SSID	MTNT	RHOS	DEPTH	AREA	SCALE	not used	UNIT
Type	I	F	F	F	F	F		I
Default	none	0.0	0.0	0.0	0.0	1.0		1
Remarks	1	4						

Card 2	1	2	3	4	5	6	7	8
Variable	X	Y	Z	NIDMC	GVID	TBIRTH	PSID	SEARCH
Type	F	F	F	I	I	F	I	F
Default	0.0	0.0	0.0	0	none	0.0	0	0.0

VARIABLE	DESCRIPTION
SSID	Segment set ID. See *SET_SEGMENT and Remark 1.
MTNT	Equivalent mass of TNT. See Remark 4.
RHOS	Density of overburden soil.
DEPTH	Burial depth from the ground surface to the center of the mine. This value must be positive.
AREA	Cross sectional area of the mine.
SCALE	Scale factor for the impulse.

VARIABLE	DESCRIPTION
UNIT	Unit system. This must match the units used by finite element model. EQ.1: inch, dozen slugs (i.e., lbf × s^2/in), second, PSI (default) EQ.2: meter, kilogram, second, Pascal EQ.3: centimeter, gram, microsecond, megabar EQ.4: millimeter, kilogram, millisecond, GPa EQ.5: millimeter, metric ton, second, MPa EQ.6: millimeter, gram, millisecond, MPa
X, Y, Z	*x*-, *y*-, and *z*- coordinates of mine center.
NIDMC	Optional node ID representing the mine center (see *NODE). If defined then X, Y and Z are ignored.
GVID	Vector ID representing the vertically upward direction, that is, the normal to the ground surface. See *DEFINE_VECTOR.
TBIRTH	Birth time. Impulse is activated at this time.
PSID	Part set ID identifying the parts affected by the mine. See *SET_-PART. If the segment set defined by SSID includes segments of more than one part, PSID may be used to load only segments of identified parts. Otherwise, if PSID is set to zero, the part affected by the mine defaults to the part comprised by the nodes of the segment set.
SEARCH	Limit the search depth into the structure. Initial nodal velocity is distributed from the segment to a depth equal to the SEARCH value. The value must be positive. If set to zero, the search depth is unlimited and extends through the part(s) identified by PSID.

Remarks:

1. **Orientation.** Segment normals should nominally point toward the mine.

2. **Element types.** The segments should belong to 3D thin shell, solid, or thick shell elements. This keyword cannot be used with 2D geometries.

3. **Workaround to use MPP.** Although this feature is unavailable in MPP, a simple workaround for this limitation involves performing the initialization in SMP and then using that initial velocity field in MPP.

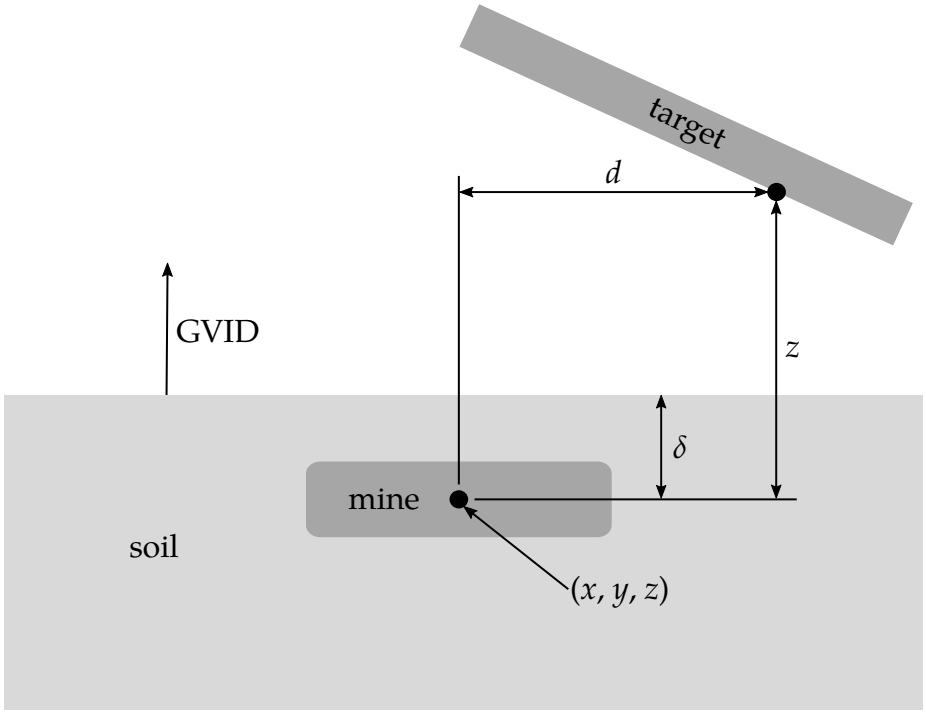

Figure 28-8. Schematic of the buried mine parameters.

4. **Equivalent mass of TNT.** Several methods can be used to approximate the equivalent mass of TNT for a given explosive. One method involves scaling the mass by the ratio of the squares of the Chapman-Jouguet detonation velocities given by:

$$M_{\text{TNT}} = M_e \frac{v_e^2}{v_{\text{TNT}}^2}$$

where M_{TNT} is the equivalent TNT mass and $v_{\text{TNT}}$is the Chapman-Jouguet detonation velocity of TNT. M_e and v_e are, respectively, the mass and C-J velocity of the explosive under investigation. LS-DYNA takes the density of "Standard" TNT to be $1.57 \frac{\text{g}}{\text{cm}^3}$ and v_{TNT} to be $0.693 \frac{\text{cm}}{\mu\text{s}}$

5. **Energy release.** This implementation assumes the energy release (heat of detonation) for 1 kilogram of TNT is 4.516 MJ.

6. **Error bounds.** Prediction of the impulse relies on an empirical approach which involves fitting curves to experimental results. The upper error bound is 1.8 times the predicted value and the lower is the predicted value divided by 1.8. Thus, if the predicted impulse is 10 kN-seconds, then the solution space ranges from 5.6 kN-sec to 18 kN-sec.

7. **Limits of the model's validity.** The computed impulse is valid when the following criteria are met:

$$0.106 \leq \frac{\delta}{z} \leq 1$$
$$6.35 \leq \frac{E/A}{\rho c^2 z} \leq 150$$
$$0.154 \leq \frac{\sqrt{A}}{z} \leq 4.48$$
$$0 \leq \frac{d}{z} \leq 19.3$$

where,

δ = the distance from the mine center to the ground surface (DEPTH)
z = the vertical distance from the mine center to the target point
E = the energy release of the explosive
A = the cross-sectional area of the mine (AREA)
ρ = the soil density (RHOS)
c = the wave speed in the soil
d = the lateral distance from the mine center to the target point.

See Figure 28-8.

References:

Tremblay, J.E., "Impulse on Blast Deflectors from a Landmine Explosion," DRDC Valcartier, DREV-TM-9814, (1998).

INITIAL_INTERNAL_DOF_SOLID_*OPTION

Available Options include:

TYPE3

TYPE4

Purpose: Initialize the internal degrees of freedom for solid element types 3 and 4.

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

Value Cards. Include 12 cards for type 3 and 6 cards for type 4.

Card	1	2	3	4	5	6	7	8
Variable	VALX	VALY	VALZ					
Type	F	F	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
LID	Element ID.
VALX	x component of internal degree of freedom.
VALY	y component of internal degree of freedom.
VALZ	z component of internal degree of freedom.

Remarks:

1. **Internal DOF.** The internal degrees of freedom are specified in terms of the displacements of the corresponding mid-side nodes of the 20 node hex and the 10

node tetrahedron that are the basis of the type 3 and 4 solid elements, respectively.

*INITIAL_LAG_MAPPING_{OPTION}

The available options are

<BLANK>

WRITE

WRITE3DAXI

Purpose: This card initializes a 3D Lagrangian calculation with data from the last cycle of a preceding 2D or 3D Lagrangian calculation.

In its *INITIAL_LAG_MAPPING form (<BLANK> option), this keyword causes data to be read in from a *mapping file*. With the WRITE option active, this card is used to set which parts are written to the mapping file. The mapping file's filename is specified using the "lagmap=" command line argument (see Remarks 1 and 2). The option WRITE3DAXI causes LS-DYNA to output the mapping file for a 3D axisymmetric mesh as if it was generated by a 2D axisymmetric model.

The following transitions are allowed:

2D → 2D	3D → 3D
2D → 3D	3D → 2D

Card 1	1	2	3	4	5	6	7	8
Variable	SETID							
Type	I							

Mesh Mapping Card. Additional card for <BLANK> or WRITE3DAXI keyword option.

Card 2	1	2	3	4	5	6	7	8
Variable	XP	YP	ZP	VECID	ANGLE	NELANGL		
Type	F	F	F	I	F	I		
Default	0.0	0.0	0.0	none	none	0		

VARIABLE	DESCRIPTION
SETID	Part set ID. See Remarks 3 and 4.
XP	*x*-position of a point belonging to the plane from which the 3D mesh is generated (only for a 2D to 3D mapping or 3Daxi to 3D mapping). See Remark 5.
YP	*y*-position of a point belonging to the plane from which the 3D mesh is generated (only for a 2D to 3D mapping or 3Daxi to 3D mapping). See Remark 5.
ZP	*z*-position of a point belonging to the plane from which the 3D mesh is generated (only for a 2D to 3D mapping or 3Daxi to 3D mapping). See Remark 5.
VECID	ID of the rotation axis (symmetric axis for a 2D to 3D mapping or 3Daxi to 3D mapping) defined by *DEFINE_VECTOR. See Remark 5.
ANGLE	Angle of rotation around an axis defined by *DEFINE_VECTOR. See Remark 5.
NELANGL	Mapping parameter. See Remark 5. GT. 0: For a 2D to 3D mapping, number of elements to create in the azimuthal direction for ANGLE EQ.-1: No mesh is generated or projected. EQ.-2: For a 3D to 3D mapping, ANGLE only rotates the data from the mapping file (not the current mesh). EQ.-3: No mesh is generated or projected aside from the boundary nodes of the current mesh being projected onto the boundary faces of the previous mesh.

Remarks:

1. **The Mapping File as Output.** In the absence of a *INITIAL_LAG_MAPPING card, adding a "lagmap=" argument to the command line will cause LS-DYNA to write a mapping file. This file contains the following nodal and element data:
   - nodal coordinates (initial and last steps)
   - nodal velocities
   - nodal temperatures (if *CONTROL_THERMAL_SOLVER is used)
   - part IDs

- element connectivities
- element centers
- densities
- volume fractions
- stresses
- plastic strains
- internal energies
- bulk viscosities
- relative volumes
- strains (only if the strain tensor is flagged for output using the variable STRFLG in *DATABASE_EXTENT_BINARY)
- extra history variables

2. **The Mapping File as Input.** If the keyword INITIAL_LAG_MAPPING is in the input deck and the "lagmap=" argument is in the command line, then Lagrangian data is read from the mapping file defined by "lagmap=" to initialize the run.

3. **Part Sets (Write).** The part set, SETID, defines which parts are involved in the mapping. The WRITE option can be used to write data in the mapping file for *only* the parts specified by the set. If the keyword *INITIAL_LAG_MAPPING_WRITE is not included in the input deck, then *all* Lagrangian parts are written in the mapping file during the last cycle.

4. **Part Sets (Read).** The mapping initializes the data for every node and element defined by SETID within the domain swept by the 2D mesh or the region initially occupied by the previous 3D mesh. For nodes and elements outside of SETID it has no effect.

5. **Embedding.** The first point in the definition of the rotation axis VECID specifies the origin location for the mesh of the previous run in the current 3D space. The 2D to 3D mapping depends on whether or not a 3D mesh has already been defined. The 3D to 3D mapping requires a pre-existing mesh.

   a) *No Mesh Case for a 2D to 3D mapping*. If there is no 3D mesh (no element with parts in SETID), the point (XP,YP,ZP) together with the symmetry axis (VECID) are used to generate a mesh. The point defines the plane in which the 2D mesh is embedded. The 3D mesh is generated by rotating the 2D mesh around the axis. The point (XP,YP,ZP) must not be on the symmetry axis. ANGLE defines the angle of rotation in degrees. The rotation is

counterclockwise when viewed from the axis head. NELANGL is the number of elements to generate in the azimuthal direction.

b) *Pre-existing Mesh Case for a 2D to 3D mapping.* If there is a 3D mesh (elements with parts in SETID), the nodes should be within the domain swept by the initial positions of the 2D mesh. Then, the nodes are projected to new locations based on the positions of the previous run's last mesh. If NELANGL = -1, the nodes should be within the domain swept by the final positions of the 2D mesh and the mesh location is not modified.

c) *Pre-existing Mesh Case for a 3D to 3D mapping.* If there is a 3D mesh (elements with parts in SETID) and NELANGL ≠ -1, the nodes should be in the region initially occupied by the previous 3D mesh. Then, the nodes are mapped to new locations based on the previous run's last mesh positions. If ANGLE is defined, the pre-existing mesh is rotated by ANGLE about VECID. The first point in VECID is still the previous origin location and it can be used to translate the pre-existing mesh. However, if NELANGL = -1, the nodes should be in the region finally occupied by the previous 3D mesh and the mesh location is not modified.

d) *3D Axisymmetric to 3D (or 2D) mapping (keyword option WRITE3DAXI).* In the 3D axisymmetric model (1st step model with only solids and/or shells), the point (XP,YP,ZP) defines the radial plane of the 3D axisymmetric mesh used to generate the mapping file. The point (XP,YP,ZP) must not be on the symmetry axis defined by VECID. It is recommended to locate this point on one of the two radial boundaries of the 3D axisymmetric mesh as the nodes of these boundaries do not move out of their plane in the orthoradial direction. The solid faces and shell edges in the radial plane defined by the point (XP,YP,ZP) become shells and beams, respectively, in the mapping file as if a 2D axisymmetric model is run to output this file. The Remarks 5a) and 5b) are still valid to set up the 3D or 2D model of the 2nd step that reads the mapping file.

*INITIAL_MOMENTUM

Purpose: Define initial momentum to be deposited in solid elements. This keyword crudely simulates an impulsive type of loading.

Card	1	2	3	4	5	6	7	8
Variable	EID	MX	MY	MZ	DEPT			
Type	I	F	F	F	F			
Default	none	0.	0.	0.	0.			

VARIABLE	DESCRIPTION
EID	Element ID
MX	Initial *x*-momentum
MY	Initial *y*-momentum
MZ	Initial *z*-momentum
DEPT	Deposition time

Remarks:

Assuming an 8-noded brick element, the specified momentum is distributed equally to each node of the element, and the change in velocity due to this deposited momentum is added to the nodal velocity at the specified deposition time.

***INITIAL_PWP_DEPTH**_{OPTION}_

The available options include:

<BLANK>

SET

Purpose: Initialize pore water pressure in solid elements where a non-hydrostatic profile is required.

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

VARIABLE	DESCRIPTION
PID	Part ID or Part Set ID for the SET keyword option
LC	Curve of pore water pressure head (length units) as a function of z-coordinate

Remarks:

This feature overrides the automatically calculated hydrostatic pressure profile; the hydrostatic pressure profile is the default behavior for a part with pore fluid. The points in the curve must be ordered with the most negative z-coordinate first – this order looks "upside-down" on the page.

*INITIAL_PWP_NODAL_DATA

Purpose: Initialize nodal pore pressure data. This keyword is written by LS-DYNA to the dynain file if *CONTROL_PORE_FLUID is present, to enable subsequent analyses beginning from a state reached in a previous analysis. It is not expected that users will create or modify this keyword.

Card 1	1	2	3	4	5	6	7	8
Variable	NID	NHISV	PID					
Type	I	I	I					
Default	none	0	0					

History Variable Cards. Define as many cards as necessary to define NHISV variables. For example, if NHISV = 25, then 5 cards are required.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	HISV1		HISV2		HISV3		HISV4		HISV5	
Type	F		F		F		F		F	

VARIABLE	DESCRIPTION
NID	Node ID
NHISV	Number of nodal pore pressure history variables.
PID	Part ID with which this node is associated for purposes of evaluating pore fluid related properties.
HISV*i*	Define NHISV history variables.

*INITIAL_SOLID_VOLUME

Purpose: Recalculate and reset initial volume of solid elements using material models with EOS before analysis if the original nodal position has been moved by nodal projections in contact initialization. This option eliminates calculation of non-physical initial hydrostatic pressure due to the nodal repositioning.

Card Sets.

Part Set Card. Define as many of this card as desired. The input terminates at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
PSID	Part set ID.

*INITIAL_STRAIN_BEAM

Purpose: Initialize displacements of discrete beams for output to DISBOUT (see *DATABASE_DISBOUT). This keyword can be used for multi-stage analyses where the accumulated displacements of discrete beams are needed for post-processing. These displacements do not affect the solution. They are for output only. If a dynain file is requested, this keyword is automatically written to the dynain file provided that (a) the model contains discrete beams and (b) *DATABASE_DISBOUT is set.

Element Cards. Define initial displacements for as many beam elements as desired. This input terminates when the next keyword ("*") card is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	RDISP	SDISP	TDISP	RROT	SROT	TROT	
Type	I	F	F	F	F	F	F	
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
EID	Element ID (must be a discrete beam, i.e., ELFORM = 6 on *SECTION_BEAM).
RDISP	The initial displacement along the local *r*-axis
SDISP	The initial displacement along the local *s*-axis
TDISP	The initial displacement along the local *t*-axis
RROT	The initial rotation about the local *r*-axis
SROT	The initial rotation about the local *s*-axis
TROT	The initial rotation about the local *t*-axis

*INITIAL_STRAIN_IGA_SHELL

Purpose: Initialize strain tensor and thicknesses at in-plane integration points for isogeometric shell elements, defined via *IGA_SHELL. This keyword is primarily for multi-stage metal forming operations where the accumulated strain and the thickness change is of interest.

The strain tensors are defined at the inner and outer integration points and are used for post-processing only. There is no interpolation with this option, and the strains are defined in the global Cartesian coordinate system.

Note: The in-plane integration point locations, defined with R and S in this keyword, may not match the integration point locations used during the computation. In such cases, LS-DYNA uses the information of the closest data point in this keyword for the analysis.

Card Sets. Define as many IGA_SHELL elements in this section as desired, *one set of cards per element*. The input is assumed to terminate when a new keyword ("*") card is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	NPLANE	ITHK	LARGE				
Type	I	I	I	I				
Default	none	none	0	0				

Strain Cards for LARGE = 0.

For the NPLANE integration points, define two pairs of cards below, one pair for the inner integration point and the other pair for the outer integration point, respectively.

Card 2	1	2	3	4	5	6	7	8
Variable	R	S	T					
Type	F	F	F					
Default	none	none	none					

Card 3	1	2	3	4	5	6	7	8
Variable	EPSXX	EPSYY	EPSZZ	EPSXY	EPSYZ	EPSZX	THKI	
Type	F	F	F	F	F	F	F	

Strain Cards for LARGE = 1.

For the NPLANE integration points, define two pairs of cards below, one pair for the inner integration point and the other pair for the outer integration point, respectively.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	R		S		T		EPSXX		EPSYY	
Type	F		F		F		F		F	

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	EPSZZ		EPSXY		EPSYZ		EPSZX		THKI	
Type	F		F		F		F		F	

VARIABLE	DESCRIPTION
EID	IGA shell element ID
NPLANE	Number of in-plane integration points being output
ITHK	Flag for initialization of thicknesses at in-plane IPs: EQ.0: Off EQ.1: On
LARGE	Large format flag: EQ.0: Off EQ.1: On. Each strain field is twice as long for higher precision.

VARIABLE	DESCRIPTION
R	Parametric *r*-coordinate of location of in-plane integration point (with respect to *IGA_2D_NURBS_XYZ patch definition)
S	Parametric s-coordinate of location of in-plane integration point (with respect to *IGA_2D_NURBS_XYZ patch definition)
T	Parametric coordinate of through thickness integration point between -1 and 1 inclusive.
EPS*ij*	Define the *ij* strain component. The strains are defined in the *global* cartesian system.
THKI	Thickness value at in-plane integration point

*INITIAL_STRAIN_SHELL_{OPTION}

The available options include:

<BLANK>

SET

Purpose: Initialize strain tensor for a shell element. This option is primarily for multi-stage metal forming operations where the accumulated strain is of interest.

These strain tensors are defined at the inner and outer integration points and are used for post-processing only. There is no interpolation with this option and the strains are defined in the global Cartesian coordinate system. The *DATABASE_EXTENT_BINARY flag STRFLG must be set to unity for this option to work. When the keyword option is blank, the strains at all the integration points can be defined by providing nonzero NPLANE and NTHICK and setting INTOUT flag of *DATABASE_EXTENT_BINARY to either "STRAIN" or "ALL."

Card Sets. Define as many shell elements in this section as desired, *one set of cards per element*. The input is assumed to terminate when a new keyword is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	NPLANE	NTHICK	LARGE				ILOCAL
Type	I	I	I	I				I
Default	none	none	none	0				0

Ordering of Integration Points

When NPLANE and NTHICK are defined, include NPLANE × NTHICK cards below. For each through thickness point define NPLANE points. NPLANE should be either 1 or 4 corresponding to either 1 or 4 Gauss integration points. If four integration points are specified, they should be ordered such that their in-plane parametric coordinates are at:

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\quad \left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\quad \left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\quad \left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right)$$

respectively.

Strain Cards for LARGE = 0.

When NPLANE and NTHICK are not defined or when the SET option is used, define two cards below, one for the inner integration point and the other for the outer integration point, respectively. Otherwise define NPLANE × NTHICK cards, one card for each integration point.

Card 2	1	2	3	4	5	6	7	8
Variable	EPSxx	EPSyy	EPSzz	EPSxy	EPSyz	EPSzx	T	
Type	F	F	F	F	F	F	F	
Default	none	none	none	0.			0.	

Strain Cards for LARGE = 1.

When NPLANE and NTHICK are not defined or when the SET option is used, define two pairs of cards below, one pair for the inner integration point and the other pair for the outer integration point, respectively. Otherwise define NPLANE × NTHICK pairs of cards; one pair for each integration point.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	EPSxx		EPSyy		EPSzz		EPSxy		EPSyz	
Type	F		F		F		F		F	

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	EPSzx		T							
Type	F		F							

VARIABLE	DESCRIPTION
EID	Element ID or shell element set ID when the SET option is used.

VARIABLE	DESCRIPTION
NPLANE	Number of in-plane integration points being output (not read when the SET option is used).
NTHICK	Number of integration points through the thickness (not read when the SET option is used).
LARGE	Large format flag: EQ.0: off, EQ.1: on. Each strain field is twice as long for higher precision.
ILOCAL	Flag for coordinate system of strain components: EQ.0: global, EQ.1: local (not supported).
EPS*ij*	Define the *ij* strain component. The strains are defined in the GLOBAL Cartesian system.
T	Parametric coordinate of through thickness integration point between -1 and 1 inclusive.

*INITIAL_STRAIN_SHELL_NURBS_PATCH

Purpose: Initialize strain tensor and thicknesses at in-plane integration points for isogeometric shell elements. This option is primarily for multi-stage metal forming operations where the accumulated strain and the thickness change is of interest.

The strain tensors are defined at the inner and outer integration points and are used for post-processing only. There is no interpolation with this option and the strains are defined in the global Cartesian coordinate system.

Card Sets. Define as many NURBS shell elements in this section as desired, *one set of cards per element*. The input is assumed to terminate when a new keyword is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	NPLANE	ITHK	LARGE				
Type	I	I	I	I				
Default	none	none	0	0				

Strain Cards for LARGE = 0.

For the NPLANE integration points, define two pairs of cards below, one pair for the inner integration point and the other pair for the outer integration point, respectively.

Card 2	1	2	3	4	5	6	7	8
Variable	R	S	T					
Type	F	F	F					
Default	none	none	none					

Card 3	1	2	3	4	5	6	7	8
Variable	EPSXX	EPSYY	EPSZZ	EPSXY	EPSYZ	EPSZX	THKI	
Type	F	F	F	F	F	F	F	

Strain Cards for LARGE = 1.

For the NPLANE integration points, define two pairs of cards below, one pair for the inner integration point and the other pair for the outer integration point, respectively.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	R		S		T		EPSXX		EPSYY	
Type	F		F		F		F		F	

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	EPSZZ		EPSXY		EPSYZ		EPSZX		THKI	
Type	F		F		F		F		F	

VARIABLE	DESCRIPTION
EID	NURBS element ID
NPLANE	Number of in-plane integration points being output
ITHK	Flag for initialization of thicknesses at in-plane IPs: EQ.0: off EQ.1: on
LARGE	Large format flag: EQ.0: off EQ.1: on. Each strain field is twice as long for higher precision.

VARIABLE	DESCRIPTION
R	Parametric *r*-coordinate of location of in-plane integration point (with respect to NURBS patch definition)
S	Parametric s-coordinate of location of in-plane integration point (with respect to NURBS patch definition)
T	Parametric coordinate of through thickness integration point between -1 and 1 inclusive.
EPS*ij*	Define the *ij* strain component. The strains are defined in the GLOBAL Cartesian system.
THKI	Thickness value at in-plane integration point

*INITIAL_STRAIN_SOLID_{OPTION}

The available options include:

<BLANK>

SET

Purpose: Initialize strain tensor at element center. This keyword can be used for multi-stage metal forming operations where the accumulated strain is of interest.

These strain tensors are defined at the element center and are used for post-processing only. The strains are defined in the global Cartesian coordinate system. The ones digit of STRFLG in *DATABASE_EXTENT_BINARY is automatically set to unity when *INITIAL_STRAIN_SOLID_{OPTION} is included in the input deck. This capability is not available for cohesive elements since it is based on displacements, not strains.

Card Sets. Define as many solid elements in this section as desired: *one pair of cards per element*. The input is assumed to terminate when a new keyword ("*") card is detected.

Element ID Cards.

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

Strain Cards.

Card 2	1	2	3	4	5	6	7	8
Variable	EPSxx	EPSyy	EPSzz	EPSxy	EPSyz	EPSzx		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
EID	Element ID or solid element set ID when the SET keyword option is used.

VARIABLE	DESCRIPTION
EPS*ij*	Define the *ij*th strain component. The strains are defined in the global cartesian system.

*INITIAL_STRAIN_SOLID_NURBS_PATCH

Purpose: Initialize strain at integration points for isogeometric solid elements.

These strain tensors are used for post-processing only. There is no interpolation with this option and the strains are defined in the global Cartesian coordinate system.

Card Sets. Define as many NURBs solid elements in this section as desired, *one set of cards per element*. The input is assumed to terminate when a new keyword ("*") card is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	NIP	LARGE					
Type	I	I	I					
Default	none	none	0					

Strain Cards for LARGE = 0.

For each NIP integration points, define a pair of cards below.

Card 2	1	2	3	4	5	6	7	8
Variable	R	S	T					
Type	F	F	F					
Default	none	none	none					

Card 3	1	2	3	4	5	6	7	8
Variable	EPSXX	EPSYY	EPSZZ	EPSXY	EPSYZ	EPSZX		
Type	F	F	F	F	F	F		

Strain Cards for LARGE = 1.

For each NIP integration points, define a pair of cards below.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	R		S		T		EPSXX		EPSYY	
Type	F		F		F		F		F	

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	EPSZZ		EPSXY		EPSYZ		EPSZX			
Type	F		F		F		F			

VARIABLE	DESCRIPTION
EID	NURBS element ID
NIP	Number of integration points being output
LARGE	Large format flag: EQ.0: off EQ.1: on. Each strain field is twice as long for higher precision.
R	Parametric *r*-coordinate of location of in-plane integration point (with respect to NURBS patch definition)
S	Parametric *s*-coordinate of location of in-plane integration point (with respect to NURBS patch definition)
T	Parametric *t*-coordinate of location of in-plane integration point (with respect to NURBS patch definition)
EPS*ij*	Define the *ij* strain component. The strains are defined in the GLOBAL Cartesian system.

*INITIAL_STRAIN_TSHELL_{OPTION}

The available options include:

<BLANK>

SET

Purpose: Initialize the strain tensors for thick shell elements.

Strain tensors are defined at the inner and outer integration points and are used for post-processing only. Strain tensors are defined in the global Cartesian coordinate system. The STRFLG flag on *DATABASE_EXTENT_BINARY must be set to unity for this keyword to work. Initialize as many elements as needed.

Card Sets. For each element, include a set of Cards 1, 2, and 3, where Card 2 is for the inner layer and Card 3 is for the outer layer. The input is assumed to terminate when a new keyword (*) card is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID/SID							
Type	I							
Default	none							

Strain Cards. Card 2 is the strain at the inner layer. Card 3 is the strain at the outer layer.

Cards 2, 3	1	2	3	4	5	6	7	8
Variable	EPSXX	EPSYY	EPSZZ	EPSXY	EPSYZ	EPSZX		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
EID/SID	Element ID or thick shell element set ID when the SET option is used, see *SET_TSHELL.

VARIABLE	DESCRIPTION
EPS*ij*	Define the ij^{th} strain component. The strains are defined in the global cartesian system.

*INITIAL_STRESS_BEAM

Purpose: Initialize stresses, plastic strains, and history variables for Hughes-Liu beam elements and truss beam elements, or the axial force, moment resultants, and history variables for Belytschko-Schwer beam elements and discrete beams.

Card Summary:

Card Sets. Define as many beams in this section as desired. Each set consists of one Card 1 and several additional cards depending on variables NPTS, LARGE, NHISV, and NAXES. The input terminates when a new keyword ("*") card is detected.

Card 1. This card is required.

EID	RULE	NPTS	LOCAL	LARGE	NHISV	NAXES	

Card 2a. This card is included for Belytschko-Schwer beams if LARGE = 0.

F11	T11	M12	M13	M22	M23	PARM	

Card 2b.1. This card is included for Belytschko-Schwer beams if LARGE = 1.

F11	T11	M12	M13	M22

Card 2b.2. This card is included for Belytschko-Schwer beams if LARGE = 1. Include additional cards of this format to include all of the history variables.

M23	PARM	HISV1	HISV2	HISV3

Card 2c. Include this card NPTS times for Hughes-Liu or truss beams if LARGE = 0.

SIG11	SIG22	SIG33	SIG12	SIG23	SIG31	EPS	

Card 2d.1. Include NPTS sets of this card and Card 2d.2 for Hughes-Liu or truss beams if LARGE = 1.

SIG11	SIG22	SIG33	SIG12	SIG23

Card 2d.2. For each of the NPTS sets include additional cards of this format to include all of the history variables.

SIG31	EPS	HISV1	HISV2	HISV3

Card 3.1. This card is included if NAXES = 12.

AX1	AX2	AX3	AX4	AX5

Card 3.2. This card is included if NAXES = 12.

AX6	AX7	AX8	AX9	AX10

Card 3.3. This card is included if NAXES = 12.

AX11	AX12			

Card 1	1	2	3	4	5	6	7	8
Variable	EID	RULE	NPTS	LOCAL	LARGE	NHISV	NAXES	
Type	I	I	I	I	I	I	I	
Default	none	2	none	0	0	0	0	

VARIABLE	DESCRIPTION
EID	Element ID
RULE	Integration rule type number: EQ.1: 1 × 1 Gauss quadrature, EQ.2: 2 × 2 Gauss quadrature (default beam), EQ.3: 3 × 3 Gauss quadrature, EQ.4: 3 × 3 Lobatto quadrature, EQ.5: 4 × 4 Gauss quadrature.
NPTS	Number of integration points. For the Belytschko-Schwer resultant beam element, NPTS = 1.
LOCAL	Coordinate system for stresses: EQ.0: Stress components are defined in the global coordinate system. EQ.1: Stress components are defined in the local beam system. In the local system components SIG22, SIG33, and SIG23 are set to 0.0.
LARGE	Format size: EQ.0: Off,

VARIABLE	DESCRIPTION
	EQ.1: On. Each field is twice as long for higher precision.
NHISV	Number of additional history variables. Only available for LARGE = 1.
NAXES	Number of variables giving the beam's local axes (0 or 12). See Remark 2.

Belytschko-Schwer Card for LARGE = 0. Additional card for the Belytschko-Schwer beam.

Card 2a	1	2	3	4	5	6	7	8
Variable	F11	T11	M12	M13	M22	M23	PARM	
Type	F	F	F	F	F	F	F	

Belytschko-Schwer Cards for LARGE = 1. Additional cards for the Belytschko-Schwer beam. Include as many cards as necessary to collect NHISV (see Card 1) history variables.

Card 2b.1	1	2	3	4	5	6	7	8	9	10
Variable	F11		T11		M12		M13		M22	
Type	F		F		F		F		F	

Card 2b.2	1	2	3	4	5	6	7	8	9	10
Variable	M23		PARM		HISV1		HISV2		HISV3	
Type	F		F		F		F		F	

Hughes-Liu Cards for LARGE = 0. Additional cards for the Hughes-Liu or truss beam. Include NPTS additional cards, one per integration point. See Remark 1.

Card 2c	1	2	3	4	5	6	7	8
Variable	SIG11	SIG22	SIG33	SIG12	SIG23	SIG31	EPS	
Type	F	F	F	F	F	F	F	

Hughes-Liu Cards for LARGE = 1. Additional cards for the Hughes-Liu or truss beam. Include NPTS additional card sets, one per integration point. Include as many cards in one card set as necessary to collect NHISV (see Card 1) history variables. See Remark 1.

Card 2d.1	1	2	3	4	5	6	7	8	9	10
Variable	SIG11		SIG22		SIG33		SIG12		SIG23	
Type	F		F		F		F		F	

Card 2d.2	1	2	3	4	5	6	7	8	9	10
Variable	SIG31		EPS		HISV1		HISV2		HISV3	
Type	F		F		F		F		F	

VARIABLE	DESCRIPTION
F11	Axial force resultant along local beam axis 1
T11	Torsional moment resultant about local beam axis 1
M12	Moment resultant at node 1 about local beam axis 2
M13	Moment resultant at node 1 about local beam axis 3
M22	Moment resultant at node 2 about local beam axis 2
M23	Moment resultant at node 2 about local beam axis 3
PARM	Generally not used.
SIG*ij*	Define the *ij* stress component

VARIABLE	DESCRIPTION
EPS	Effective plastic strain
HISV*n*	Define the *n*th history variable

Optional Local Axes Cards for NAXES = 12. Additional cards for definition of local axes values. These 12 values are internally used by LS-DYNA for the mapping between local beam element system and global coordinate system. They are automatically written to the dynain file if *INTERFACE_SPRINGBACK_LSDYNA or *CONTROL_STAGED_CONSTRUCTION is used.

Card 3.1	1	2	3	4	5	6	7	8	9	10
Variable	AX1		AX2		AX3		AX4		AX5	
Type	F		F		F		F		F	

Card 3.2	1	2	3	4	5	6	7	8	9	10
Variable	AX6		AX7		AX8		AX9		AX10	
Type	F		F		F		F		F	

Card 3.3	1	2	3	4	5	6	7	8	9	10
Variable	AX11		AX12							
Type	F		F							

VARIABLE	DESCRIPTION
AX*n*	The *n*th local axes value. See Remark 2.

Remarks:

1. **Axial stress for truss beams.** Note that only SIG11 is nonzero for truss elements since they only carry axial forces.

2. **NAXES.** NAXES = 12 indicates providing AX1, …, AX12 for determining the local axes of the beam. These values are useful in cases where an initial analysis is performed, the beam has rotated during the analysis, and a subsequent analysis is to be performed. LS-DYNA uses them to properly orient the beam in the subsequent analysis. *The user is not expected to enter these values.* LS-DYNA automatically writes this keyword with these values to a dynain file when indicated in the input deck (such as with *INTERFACE_SPRINGBACK_LSDYNA). These AX1, …, AX12 correspond to the $\hat{Y}$ terms seen in Equation 7.12 of the R14 Theory manual.

*INITIAL_STRESS_DEPTH_{OPTION}

Available options include:

<BLANK>

SET

Purpose: Initialize solid element stresses where stress is a function of depth. This feature is intended only for material models whose stresses are updated incrementally and whose stress state does not depend on internal history variables.

Card 1	1	2	3	4	5	6	7	8
Variable	PID/PSID	RO_G	ZDATUM	KFACT	LC	LCH	LCK0	
Type	I	F	F	F	I	I	I	
Default	none	none	none	0.0	opt	opt	opt	

VARIABLE	DESCRIPTION
PID/PSID	Part ID or Part Set ID for the SET option
RO_G	Stress per unit elevation above datum, which is usually ρ_g = density × gravity.
ZDATUM	z-coordinate of datum
KFACT	x and y-stresses = KFACT × z-stress
LC	Optional curve of stress as a function of the z-coordinate (ZDATUM is ignored with this option)
LCH	Optional curve of horizontal stress as function of the z-coordinate (KFACT is ignored with this option)
LCK0	Optional curve of K0 (ratio of horizontal stress to vertical stress) as a function z-coordinate. KFACT and LCH are ignored with this option. The x-axis of the curve is the z-coordinate, the y-axis is K0.

Remarks:

1. **Model Description.** With this keyword stress is calculated according to,

$$\sigma_z = \mathrm{RO_G} \times (\mathrm{Z_{element}} - \mathrm{ZDATUM}) ,$$

where $\mathrm{Z_{element}}$ is the z-coordinate of the centroid of the element. It is assumed that the z-axis points vertically upwards. For a 2D problem (axisymmetric or plane strain), replace z in this documentation with y. To generate compressive stresses, the datum should be above the highest element, that is, the datum should have a more positive z-coordinate. For instance, this is at the surface of the soil in geotechnics simulations. If the curve, LC, is defined, it overrides RO_-G and ZDATUM. Note that the points in the curve should be ordered with most negative z-coordinate first.

2. **Pore Water.** If pore water is present, the stresses input here are effective (meaning soil skeleton stresses only, not including the pore pressure). The pore water pressures will automatically be initialized to hydrostatic, or by *INITIAL_-PWP_DEPTH or *BOUNDARY_PWP_TABLE if those cards are present. Effective vertical stress at a given depth is usually equal to the buoyant weight of the soil between that depth and the surface, where buoyant weight is calculated from the density of the saturated soil minus the density of water. The density of saturated soil is the RO given on the *MAT card, and the density of water is the PF_RHO given on *BOUNDARY_PORE_FLUID or *CONTROL_PORE_FLUID. Note also that the density of saturated soil is somewhat higher than the density of dry soil, and the value of RO on the *MAT card should reflect this.

3. **Stress at Integration Points.** For fully integrated elements, the stress calculated for the element centroid is applied at all the integration points. There is no stress gradation up the height of the element. This is to ensure equilibrium with gravity loading, which is applied at the nodes and therefore results in stress that is uniformly distributed up the height of the element.

*INITIAL_STRESS_DES

Purpose: Initialize stresses and coordination number for DES elements.

Element Cards. Define as many DES elements in this section as desired. The new keyword ("*") terminates this input.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	COOR
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
EID	DES particle ID
SIG*ij*	Define the ij^{th} stress component. Stresses are defined in the global cartesian system.
COOR	Define the coordination number of the DES element

*INITIAL_STRESS_IGA_SHELL

Purpose: Initialize stresses, history variables, and the effective plastic strain for isogeometric shell elements, defined via *IGA_SHELL. Materials that do not use an incremental formulation for the stress update may not be initializable with this card.

Note: The in-plane integration point locations, defined with R and S in this keyword, might not match the integration point locations used during the computation. In such cases, LS-DYNA uses the information of the closest data point in this keyword for the analysis. When the number of through-thickness integration points differs between this keyword and the actual analysis, LS-DYNA interpolates accordingly.

Card Sets per Element. Define as many IGA_SHELL elements in this section as desired. The input is assumed to terminate when a new keyword ("*") card is detected.

Element Card.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	NPLANE	NTHICK	NHISV	LARGE			
Type	I	I	I	I	I			
Default	none	none	none	0	0			

Solid Mechanics Data Card for LARGE = 0.

The following set of cards: "Stress Cards" and "History Variable Cards" should be included NPLANE × NTHICK times (one set for each integration point).

Stress Card 1. Additional Card for LARGE = 0.

Card 2	1	2	3	4	5	6	7	8
Variable	R	S	T					
Type	F	F	F					

Stress Card 2. Additional Card for LARGE = 0.

Card 3	1	2	3	4	5	6	7	8
Variable	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPSP	
Type	F	F	F	F	F	F	F	

History Variable Cards. Additional Cards for LARGE = 0. Include as many cards as necessary to collect NHISV (see Card 1) history variables.

Card 4	1	2	3	4	5	6	7	8
Variable	HISV1	HISV2	HISV3	HISV4	HISV5	HISV6	HISV7	HISV8
Type	F	F	F	F	F	F	F	F

Solid Mechanics Data Card for LARGE = 1.

The following set of cards: "Stress Cards" and "History Variable Cards" should be included NPLANE × NTHICK times (one set for each integration point).

Stress Card 1. Additional Card for LARGE = 1.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	R		S		T		SIGXX		SIGYY	
Type	F		F		F		F		F	

Stress Card 2. Additional Card for LARGE = 1.

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	SIGZZ		SIGXY		SIGYZ		SIGZX		EPS	
Type	F		F		F		F		F	

History Variable Cards. Additional Cards for LARGE = 1. Include as many cards as necessary to collect NHISV (see Card 1) history variables.

Card 4	1	2	3	4	5	6	7	8	9	10
Variable	HISV1		HISV2		HISV3		HISV4		HISV5	
Type	F		F		F		F		F	

VARIABLE	DESCRIPTION
EID	IGA shell element ID
NPLANE	Number of in-plane integration points being output
NTHICK	Number of integration points through the thickness
NHISV	Number of additional history variables.
LARGE	Format size. See cards above. EQ.0: Off EQ.1: On
R	Parametric *r*-coordinate of location of in-plane integration point (with respect to *IGA_2D_NURBS_XYZ-patch definition)
S	Parametric *s*-coordinate of location of in-plane integration point (with respect to *IGA_2D_NURBS_XYZ-patch definition)
T	Parametric coordinate of through thickness integration point between -1 and 1 inclusive.
SIG*ij*	Define the *ij* stress component. The stresses are defined in the *global* cartesian system.
EPS	Effective plastic strain
HISV*n*	Define the n^{th} history variable

*INITIAL_STRESS_SECTION

Purpose: Initialize (pre-load) the stress normal to a plane intersecting the model using a curve (*DEFINE_CURVE). The initialization is applied to a set of solid elements that are specified using the *DATABASE_CROSS_SECTION card. This option is compatible only with those material models that are incrementally updated (such as elastic, viscoelastic, and elastoplastic). Except as noted in Remark 3, rubbers, foams, and materials that are combined with equations-of-state cannot be initialized with this card.

Card 1	1	2	3	4	5	6	7	8
Variable	ISSID	CSID	LCID	PSID	VID	IZSHEAR	ISTIFF	
Type	I	I	I	I	I	I	I	
Default	none	none	none	none	none	0	0	

VARIABLE	DESCRIPTION
ISSID	ID for this card.
CSID	Cross-section ID. See *DATABASE_CROSS_SECTION.
LCID	Load curve ID defining preload stress as a function of time. When the load curve ends or goes to zero, the initialization is assumed to be completed. See Remark 2.
PSID	Part set ID. Stress is initialized on only those parts included in *both* PSID from this card and the PSID field from the associated *DATABASE_CROSS_SECTION card.
VID	Vector ID (see *DEFINE_VECTOR) specifies the direction normal to the cross section (CSID). VID *must* be set when the "SET" variant of *DATABASE_CROSS_SECTION (*DATABASE_CROSS_SECTION_SET) is used. If the cross section is defined using the PLANE option, the normal used in the definition of the plane is used when VID is left undefined.
IZSHEAR	Shear stress flag: EQ.0: Shear stresses are prescribed as zero during the time the curve is acting to prescribe normal stress.

VARIABLE	DESCRIPTION
	EQ.1: Shear stresses are allowed to develop during the time the curve is acting to prescribe normal stress. For implicit the section can also take bending and is identical to 2; see Remark 5.
	EQ.2: Shear and bending stresses are allowed to develop during the time the curve is acting to prescribe normal stress; see Remark 5.
ISTIFF	Artificial stiffness. Simulates additional linearly elastic "ghost" elements in the cross section. These elements prevent mesh distortion by stiffening up the structure.
	GT.0: Load curve ID defining stiffness fraction as a function of time. The stiffness of the ghost elements is the load curve value times the stiffness of the material in the part. Since the ghost element stress counteracts the preload stress the fraction should be low (1% or less). The total section stress is the preload stress minus the ghost element stress.
	LT.0: |ISTIFF| is the load curve ID for the stiffness fraction as a function of time. The preload stress is here automatically adjusted ($\pm$10% of original prestress values) such that the total section stress corresponds to the curve in LCID.

Remarks:

1. **Dynamic Relaxation Issues.** To achieve convergence during explicit dynamic relaxation, applying damping options is very important. If contact is active, contact damping is recommended with a value between 10 - 20 percent. Additional damping applied using *DAMPING_PART_STIFFNESS also speeds up convergence where a coefficient of 0.10 is effective. If damping is not used, convergence may not be possible.

2. **Best Practices for Quasi-Static Load Ramp-up.** When defining the load curve, LCID, a ramp starting at the origin should be used to increase the stress to the desired value. The time duration of the ramp should produce a quasi-static response. When the end of the load curve is reached, or when the value of the load decreases from its maximum value, the initialization stops. If the load curve begins at the desired stress value, that is, no ramp, convergence will take much longer, since the impulsive like load created by the initial stress can excite nearly every frequency in the structural system where stress is initialized.

3. **Supported Materials.** This option currently applies only to materials that are incrementally updated. Hyperelastic materials and materials that require an equation-of-state are not currently supported. However, materials 57, 73, and 83 can be initialized with this approach.

4. **Supported Elements.** Solid elements types 1, 2, 3, 4, 9, 10, 13, 15, 16, 17, and 18 are supported. ALE elements are not supported. Element forms 16 and 17 with mid-side nodes may only work well if the area of the cross-section where the preload stress is applied does not vary. In other words, element forms 16 or 17 will not work to preload a tapered part.

5. **IZSHEAR for Solid Elements.** By default, IZSHEAR = 0, that is, *only* the stress normal to the section is prescribed while *all* other stress components are set to zero. Therefore, the stress tensor components with respect to a local system aligned with the section would have $\sigma_{xx} = \sigma$ (the prescribed value) and $\sigma_{yy} = \sigma_{zz} = \sigma_{xy} = \sigma_{xz} = \sigma_{yz} = 0$, which holds for each integration point of each element in the section. This initialization in turn means that these elements, thus the entire section, are infinitely weak in bending and shear which might have a potential side effect of unexpected and nonsensical deformations for certain loading and surrounding geometry conditions.

   Turning on IZSHEAR = 1 will for *explicit analysis* essentially relax the zero constraint on the shear stresses, resulting in a stress tensor where σ_{xy}, σ_{xz} and σ_{yz} are updated according to the laws of the material model, thus adding some shear stiffness to the section to avoid spurious deformations.

   For *implicit analysis* (and currently only for low order element formulations -1, -2, 1, 2, 10, 13, 15 and 115 with sections defined using *DATABASE_CROSS_SECTION_PLANE or *DATABASE_CROSS_SECTION_SET), the following approach is taken for IZSHEAR = 1: Each collection of elements in a section (for example, of a bolt) is identified and seen as an entity for which only the *mean normal stress* of the section is prescribed, in contrast to imposing constraints on each integration point. For example, if the goal is to prestress 10 bolts modeled with solid elements, LS-DYNA will, based on element connectivity, identify the 10 unique sections (10 clusters of elements) corresponding to each bolt, and subsequently prescribe the mean stress in each of these sections independently. The advantage of this approach is that the section can globally resist bending and shear, and thus preserve the structural integrity of the bolt more adequately.

   The IZSHEAR = 1 approach for implicit is also available in explicit analysis through IZSHEAR = 2. Note that if this approach is used with *DATABASE_CROSS_SECTION_SET, the vector option must be used to define the normal direction. Using *DATABASE_CROSS_SECTION_SET leads to a better distribution of the stress than *DATABASE_CROSS_SECTION_PLANE by, for instance, imposing the stress for a sufficient length of the shank. Distributing

the stress with this method prevents elements from collapsing, thereby preserving structural integrity.

IZSHEAR = 2 can also be used when the initial stress (meaning stress at time zero) in the section is nonzero. For instance, bolts that are to be preloaded may have already gone through a sequence of load steps from prior simulations, resulting in a nonzero initial stress state. The load curve for the stress should still start at the origin and increase to the desired stress value. LS-DYNA will preserve stress continuity at the start of the simulation by internally scaling the curve appropriately.

*INITIAL_STRESS_SHELL_{OPTION}

Available options include:

<BLANK>

SET

Purpose: Initialize stresses, history variables, and the effective plastic strain for shell elements. Materials that do not use an incremental formulation for the stress update may not be initializable with this card.

Card Sets per Element. Define as many shell elements or shell element sets in this section as desired. The input is assumed to terminate when a new keyword ("*") card is detected.

Element Card.

Card 1	1	2	3	4	5	6	7	8
Variable	EID/SID	NPLANE	NTHICK	NHISV	NTENSR	LARGE	NTHINT	NTHHSV
Type	I	I	I	I	I	I	I	I
Default	none	none	none	0	0	0	0	0

Ordering of Integration Points.

For each through thickness point define NPLANE points. NPLANE should be either 1 or 4 corresponding to either 1 or 4 Gauss integration points. If four integration points are specified, they should be ordered such that their in-plane parametric coordinates are at:

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\quad \left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\quad \left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\quad \left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),$$

respectively. It is not necessary for the location of the through thickness integration points to match those used in the elements which are initialized. The data will be interpolated by LS DYNA.

Solid Mechanics Data Card for LARGE = 0.

The following set of cards, "Stress Card" through "Tensor Cards," should be included NPLANE × NTHICK times (one set for each integration point).

Stress Card. Additional Card for LARGE = 0.

Card 2	1	2	3	4	5	6	7	8
Variable	T	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPS
Type	F	F	F	F	F	F	F	F

History Variable Cards. Additional Cards for LARGE = 0. Include as many cards as necessary to collect NHISV (see Card 1) history variables.

Card 3	1	2	3	4	5	6	7	8
Variable	HISV1	HISV2	HISV3	HISV4	HISV5	HISV6	HISV7	HISV8
Type	F	F	F	F	F	F	F	F

Tensor Cards. Additional card for LARGE = 0. Include as many cards as necessary to collect NTENSR (see Card 1) entries. Tensor cards contain only 6 entries per card.

Card 4	1	2	3	4	5	6	7	8
Variable	TENXX	TENYY	TENZZ	TENXY	TENYZ	TENZX		
Type	F	F	F	F	F	F		

Solid Mechanics Data Card for LARGE = 1.

The following set of cards, "Stress Card 1" through "Tensor Cards," should be included NPLANE × NTHICK times (one set for each integration point).

Stress Card 1. Additional Card for LARGE = 1.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	T		SIGXX		SIGYY		SIGZZ		SIGXY	
Type	F		F		F		F		F	

Stress Card 2. Additional Card for LARGE = 1.

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	SIGYZ		SIGZX		EPS					
Type	F		F		F					

History Variable Cards. Additional Cards for LARGE = 1. Include as many cards as necessary to collect NHISV (see Card 1) history variables.

Card 4	1	2	3	4	5	6	7	8	9	10
Variable	HISV1		HISV2		HISV3		HISV4		HISV5	
Type	F		F		F		F		F	

Tensor Cards. Include as many pairs of Cards 5 and 6 as necessary to collect NTENSR entries. Note that Cards 5 and 6 must appear as pairs, and that Card 6 may include at most one value, as indicated below.

Card 5	1	2	3	4	5	6	7	8	9	10
Variable	TENXX		TENYY		TENZZ		TENXY		TENYZ	
Type	F		F		F		F		F	

Card 6	1	2	3	4	5	6	7	8	9	10
Variable	TENZX									
Type	F									

Thermal Data Cards for LARGE = 1.

For each element, thermal data cards come after the *entire* set of mechanical data cards. For each of the NTHINT thermal integration points, include the following set of cards.

Thermal Time History Cards. Additional cards for LARGE = 1. Include as many cards as needed to collect all the of NTHHSV time history variables per thermal integration point.

Card 7	1	2	3	4	5
Variable	THHSV1	THHSV2	THHSV3	THHSV4	THHSV5
Type	F	F	F	F	F

VARIABLE	DESCRIPTION
EID/SID	Element ID or shell set ID, see *SET_SHELL_…
NPLANE	Number of in plane integration points being output.
NTHICK	Number of integration points through the thickness.
NHISV	Number of additional history variables.
NTENSR	Number of components of tensor data taken from the element history variables stored.
LARGE	Format size. See cards above. EQ.0: off EQ.1: on
T	Parametric coordinate of through thickness integration point between -1 and 1 inclusive.
SIG*ij*	Define the *ij* stress component. The stresses are defined in the GLOBAL cartesian system.
EPS	Effective plastic strain.
HISV*n*	Define the n^{th} history variable.
TEN*ij*	Define the ij^{th} component of the tensor taken from the history variables. The tensor is defined in the GLOBAL Cartesian system. Define enough lines to provide a total of NTENSOR components, stored six components per line. This applies to material 190 only.
NTHINT	Number of thermal integration points.
NTHHSV	Number of thermal history variables per thermal integration point.

VARIABLE	DESCRIPTION
THHSV*n*	n^{th} history variable at the thermal integration point.

*INITIAL_STRESS_SHELL_NURBS_PATCH

Purpose: Initialize stresses, history variables, and the effective plastic strain for isogeometric shell elements. Materials that do not use an incremental formulation for the stress update may not be initializable with this card.

Card Sets per Element. Define as many NURBS shell elements in this section as desired. The input is assumed to terminate when a new keyword ("*") card is detected.

Element Card.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	NPLANE	NTHICK	NHISV	LARGE			
Type	I	I	I	I	I			
Default	none	none	none	0	0			

Solid Mechanics Data Card for LARGE = 0.

The following set of cards: "Stress Cards" and "History Variable Cards" should be included NPLANE × NTHICK times (one set for each integration point).

Stress Card1. Additional Card for LARGE = 0.

Card 2	1	2	3	4	5	6	7	8
Variable	R	S	T					
Type	F	F	F					

Stress Card2. Additional Card for LARGE = 0.

Card 3	1	2	3	4	5	6	7	8
Variable	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPSP	
Type	F	F	F	F	F	F	F	

History Variable Cards. Additional Cards for LARGE = 0. Include as many cards as necessary to collect NHISV (see Card 1) history variables.

Card 4	1	2	3	4	5	6	7	8
Variable	HISV1	HISV2	HISV3	HISV4	HISV5	HISV6	HISV7	HISV8
Type	F	F	F	F	F	F	F	F

Solid Mechanics Data Card for LARGE = 1.

The following set of cards: "Stress Cards" and "History Variable Cards" should be included NPLANE × NTHICK times (one set for each integration point).

Stress Card 1. Additional Card for LARGE = 1.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	R		S		T		SIGXX		SIGYY	
Type	F		F		F		F		F	

Stress Card 2. Additional Card for LARGE = 1.

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	SIGZZ		SIGXY		SIGYZ		SIGZX		EPS	
Type	F		F		F		F		F	

History Variable Cards. Additional Cards for LARGE = 1. Include as many cards as necessary to collect NHISV (see Card 1) history variables.

Card 4	1	2	3	4	5	6	7	8	9	10
Variable	HISV1		HISV2		HISV3		HISV4		HISV5	
Type	F		F		F		F		F	

VARIABLE	DESCRIPTION
EID	NURBS Element ID
NPLANE	Number of in plane integration points being output.
NTHICK	Number of integration points through the thickness.
NHISV	Number of additional history variables.
LARGE	Format size. See cards above. EQ.0: off EQ.1: on
R	Parametric *r*-coordinate of location of in-plane integration point (with respect to NURBS-patch definition)
S	Parametric *s*-coordinate of location of in-plane integration point (with respect to NURBS-patch definition)
T	Parametric coordinate of through thickness integration point between -1 and 1 inclusive.
SIG*ij*	Define the *ij* stress component. The stresses are defined in the GLOBAL cartesian system.
EPS	Effective plastic strain.
HISV*n*	Define the n^{th} history variable.

*INITIAL_STRESS_SOLID_{OPTION}

Available options include:

<BLANK>

SET

Purpose: Initialize stresses and plastic strains for solid elements.

Material models 2, 5, 7, 21, 23, 27, 31, 38, 57, 73, 77, 83,132, 179, 181, 183, and 189 support a reference configuration (see *INITIAL_FOAM_REFERENCE_GEOMETRY and the variable REF in the material description), and if that feature is invoked, it will take precedence over *INITIAL_STRESS_SOLID. Furthermore, if FMATRX = 2 in *CONTROL_SOLID, *INITIAL_STRESS_SOLID is ignored for the aforementioned material models.

For *MAT_014 and any material that requires an equation-of-state (*EOS), the specified initial stresses are adjusted to be in accordance with the initial pressure calculated from the equation of state.

Card Sets per Element or Element Set. For this keyword, each data card set consists of an element or element set card and all of its corresponding data cards, both thermal and mechanical. For LARGE = 1, this can involve several (even tens of) cards per set. Include cards for as many solid elements or solid element sets as desired. The input is assumed to terminate when a new keyword ("*") card is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID/SID	NINT	NHISV	LARGE	IVEFLG	IALEGP	NTHINT	NTHHSV
Type	I	I	I	I	I	I	I	I
Default	none	none	0	0	0	0	0	0

Ordering of Integration Points.

NINT may be 1, 8, or 14 for hexahedral solid elements, depending on the element formulation. If eight Gauss integration points are specified, they should be ordered such that their parametric coordinates are located at:

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),$$

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),$$

respectively. If eight points are defined for 1 point LS-DYNA solid elements, the average value will be taken.

NINT may be 1, 4, or 5 for tetrahedral solid elements, depending on the element formulation and NIPTETS in *CONTROL_SOLID. NINT may be 1 or 2 for pentahedral solid elements, depending on the element formulation.

Solid Mechanics Data Card for LARGE = 0.

Stress Card. Additional Card for LARGE = 0. This card should be included NINT times (one for each integration point).

Card 2	1	2	3	4	5	6	7	8
Variable	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPS	
Type	F	F	F	F	F	F	F	

Mechanical Data Cards for LARGE = 1.

The following set of cards "Stress Card 1" through "Additional History Cards." Should be included NINT times (one set for each integration point).

Stress Card 1. Additional cards for LARGE = 1.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	SIGXX		SIGYY		SIGZZ		SIGXY		SIGYZ	
Type	F		F		F		F		F	

Stress Card 2. Additional cards for LARGE = 1.

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	SIGZX		EPS		HISV1		HISV2		HISV3	
Type	F		F		F		F		F	

Additional History Cards. Additional cards for LARGE = 1. If NHISV > 3 define as many additional cards as necessary. NOTE: the value of IVEFLG (see Card 1) can affect the number of history variables on these cards.

Card 4	1	2	3	4	5	6	7	8	9	10
Variable	HISV4		HISV5		HISV6		HISV7		HISV8	
Type	F		F		F		F		F	

Thermal Data Cards for LARGE = 1.

For each element, thermal data cards come after the *entire* set of mechanical data cards. For each of the NTHINT thermal integration points, include the following set of cards.

Thermal Time History Cards. Additional cards for LARGE = 1. Include as many cards as needed to capture all the of NTHHSV time history variables per thermal integration point.

Card 5	1	2	3	4	5	6	7	8	9	10
Variable	THHSV1		THHSV2		THHSV3		THHSV4		THHSV5	
Type	F		F		F		F		F	

VARIABLE	DESCRIPTION
EID/SID	Element ID or solid set ID; see *SET_SOLID_...
NINT	Number of integration points (should correspond to the solid element formulation).
NHISV	Number of additional history variables, which is typically equal to the number of history variables stored at the integration point + IVEFLG.
LARGE	Format size. If zero, NHISV must also be set to zero (this is the format used by LS-DYNA versions 970 and earlier), and, if set to 1, a larger format is used and NHISV is used.

VARIABLE	DESCRIPTION
IVEFLG	Initial Volume/energy flag (**only used in large format**): EQ.0: Last history variable is used as normal. EQ.1: Last history variable is used as the initial volume of the element. One additional history variable is required if IVEFLG = 1. EQ.2: Last two history variables are used to define the initial volume and the internal energy per unit initial volume. Two additional history variables must be allocated; see NHISV above, if IVEFLG = 2. If the initial volume is set to zero, the actual element volume is used.
IALEGP	The ALE multi-material group (AMMG) ID; only if the element is of ALE multi-material formulation (ELFORM = 11). In this case, each AMMG has its own sets of stress and history variables, so we must specify to which AMMG the stress data are assigned. For mixed elements, multiple cards are needed to complete the stress initialization in this element as each AMMG needs to have its own set of stress data. EQ.0: Assuming the element is fully filled by the AMMG that the element part belongs to. Please refer to *ALE_MULTI-MATERIAL_GROUP card. EQ.n: Assigning the stress to nth AMMG in that element.
SIG*ij*	Define the ij^{th} stress component. Stresses are defined in the GLOBAL Cartesian system.
EPS	Effective plastic strain
HISV*i*	Define NHISV history variables.
NTHINT	Number of thermal integration points
NTHHSV	Number of thermal history variables per thermal integration point
THHSV*n*	n^{th} thermal time history variable

Remarks:

1. **Cohesive Elements.** The elastic material model for cohesive elements is a total Lagrangian formulation, and the initial stress will therefore be ignored for it.

2. **Tensorial History Variables.** If the history variable field contains tensorial quantities, such as back stresses or similar tensors (such as for *MAT_003), then *INCLUDE_TRANSFORM with *DEFINE_TRANSFORMATION including a rotation (options MIRROR, ROTATE, POS6P, POS6N) should not be used. Only the actual stress tensor is transformed correctly at the moment.

*INITIAL_STRESS_SOLID_NURBS_PATCH

Purpose: Initialize stresses, effective plastic strain, and history variables for isogeometric solid elements. This command is not applicable to hyperelastic materials or any material model based on a Total Lagrangian formulation.

Card Sets. Define as many NURBSsolid elements in this section as desired, *one set of cards per element*. The input is assumed to terminate when a new keyword ("*") card is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	NINT	NHISV	LARGE				
Type	I	I	I	I				
Default	none	none	0	0				

Solid Mechanics Data Card for LARGE = 0.

The following set of cards, "Stress Cards" and "History Variable Cards," should be included NINT times (one set for each integration point).

Stress Card1. Additional card for LARGE = 0.

Card 2	1	2	3	4	5	6	7	8
Variable	R	S	T					
Type	F	F	F					

Stress Card2. Additional card for LARGE = 0.

Card 2	1	2	3	4	5	6	7	8
Variable	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPS	
Type	F	F	F	F	F	F	F	

History Variable Cards. Additional card for LARGE = 0. Include as many cards as necessary to collect NHISV (see Card 1) history variables.

Card 3	1	2	3	4	5	6	7	8
Variable	HISV1	HISV2	HISV3	HISV4	HISV5	HISV6	HISV7	HISV8
Type	F	F	F	F	F	F	F	F

Solid Mechanics Data Card for LARGE = 1.

The following set of cards, "Stress Cards" and "History Variable Cards," should be included NINT times (one set for each integration point).

Stress Card 1. Additional card for LARGE = 1.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	R		S		T		SIGXX		SIGYY	
Type	F		F		F		F		F	

Stress Card 2. Additional card for LARGE = 1.

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	SIGZZ		SIGXY		SIGYZ		SIGZX		EPS	
Type	F		F		F		F		F	

History Variable Cards. Additional card for LARGE = 1. Include as many cards as necessary to collect NHISV (see Card 1) history variables.

Card 4	1	2	3	4	5	6	7	8	9	10
Variable	HISV1		HISV2		HISV3		HISV4		HISV5	
Type	F		F		F		F		F	

VARIABLE	DESCRIPTION
EID	NURBS Element ID
NINT	Number of in integration points being output.
NHISV	Number of additional history variables.
LARGE	Format size: EQ.0: off EQ.1: on
R	Parametric *r*-coordinate of location of in-plane integration point (with respect to NURBS patch definition)
S	Parametric *s*-coordinate of location of in-plane integration point (with respect to NURBS patch definition)
T	Parametric *t*-coordinate of location of in-plane integration point (with respect to NURBS patch definition)
SIG*ij*	Define the *ij* stress component. The stresses are defined in the GLOBAL cartesian system.
EPS	Effective plastic strain.
HISV*n*	Define the n^{th} history variable.

*INITIAL_STRESS_SPH

Purpose: Initialize stresses and plastic strains for SPH elements. This command is not applicable to hyperelastic materials or any material model based on a Total Lagrangian formulation. For *MAT_005, *MAT_014, and any material that requires an equation-of-state (*EOS), the initialized stresses are deviatoric stresses, not total stresses.

Element Cards. Define as many SPH elements in this section as desired. The input is assumed to terminate when a new keyword is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPS
Type	I	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
EID	SPH particle ID
SIG*ij*	Define the ij^{th} stress component. Stresses are defined in the GLOBAL Cartesian system.
EPS	Effective plastic strain.

***INITIAL_STRESS_TSHELL**_*{OPTION}*

Available options include:

<BLANK>

SET

Purpose: Initialize stresses and plastic strains for thick shell elements.

Card Sets per Element. Define as many thick shell elements in this section as desired. The input is assumed to terminate when a new keyword is detected.

Card 1	1	2	3	4	5	6	7	8
Variable	EID/SID	NPLANE	NTHICK	NHISV	LARGE			
Type	I	I	I	I	I			
Default	none	none	none	0	0			

Ordering of Integration Points.

For each through thickness point define NPLANE points. NPLANE should be either 1 or 4 corresponding to either 1 or 4 Gauss integration points. If four integration points are specified, they should be ordered such that their in plane parametric coordinates are at:

$$\left(-\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\quad \left(\frac{\sqrt{3}}{3},-\frac{\sqrt{3}}{3}\right),\quad \left(\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right),\quad \left(-\frac{\sqrt{3}}{3},\frac{\sqrt{3}}{3}\right)$$

respectively. It is not necessary for the location of the through thickness integration points to match those used in the elements which are initialized. The data will be interpolated by LS DYNA.

Data Card for LARGE = 0.

The following set of cards "Stress Card" and "History Cards." Should be included NPLANE × NTHICK times (one set for each integration point).

Stress Card. Additional card for LARGE = 0.

Card 2	1	2	3	4	5	6	7	8
Variable	T	SIGXX	SIGYY	SIGZZ	SIGXY	SIGYZ	SIGZX	EPS
Type	F	F	F	F	F	F	F	F

History Cards. Additional Card for LARGE = 0. Include as many History Cards as needed to define all NHIST history variables.

Optional	1	2	3	4	5	6	7	8
Variable	HISV1	HSIV2	HSIV3	HSIV4	HSIV5	HSIV6	HSIV7	HSIV8
Type	F	F	F	F	F	F	F	F

Data Card for LARGE = 1.

The following set of cards "Stress Cards" and "History Cards." Should be included NPLANE × NTHICK times (one set for each integration point).

Stress Card 1. Additional card for LARGE = 1.

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	T		SIGXX		SIGYY		SIGZZ		SIGXY	
Type	F		F		F		F		F	

Stress Card 2. Additional card for LARGE = 1.

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	SIGYZ		SIGZX		EPS					
Type	F		F		F					

History Cards. Additional Card for LARGE = 1. Include as many History Cards as needed to define all NHIST history variables.

Optional	1	2	3	4	5	6	7	8	9	10
Variable	HISV1		HISV2		HISV3		HISV4		HISV5	
Type	F		F		F		F		F	

VARIABLE	DESCRIPTION
EID/SID	Thick shell element ID or for the SET keyword option thick shell set ID (see *SET_TSHELL_…)
NPLANE	Number of in plane integration points.
NTHICK	Number of integration points through the thickness.
T	Parametric coordinate of through thickness integration point between -1 and 1 inclusive.
NHISV	Number of additional history variables.
LARGE	Format size. See keywords above. EQ.0: off EQ.1: on
SIG*ij*	Define the *ij* stress component. The stresses are defined in the GLOBAL Cartesian system.
EPS	Effective plastic strain

INITIAL_TEMPERATURE_*OPTION

Available options include:

NODE

SET

Purpose: Define initial nodal point temperatures using nodal set IDs or node numbers. These initial temperatures are used in a thermal only analysis or a coupled thermal/structural analysis. See also *CONTROL_THERMAL_SOLVER, *CONTROL_THERMAL_TIMESTEP, and CONTROL_THERMAL_NONLINEAR.

For thermal loading in a structural only analysis, see *LOAD_THERMAL_*OPTION*.

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

Card 1	1	2	3	4	5	6	7	8
Variable	NSID/NID	TEMP	LOC					
Type	I	I	I					
Default	↓	0.	0					
Remarks	1							

VARIABLE	DESCRIPTION
NSID/NID	Nodal set ID or nodal point ID, see also *SET_NODES: EQ.0: all nodes are included (set option only).
TEMP	Temperature at node or node set.
LOC	For a thick thermal shell, the temperature will be applied to the surface identified by LOC. See parameter, THSHEL, on the *CONTROL_SHELL keyword. EQ.-1: lower surface of thermal shell element EQ.0: middle surface of thermal shell element EQ.1: upper surface of thermal shell element

Remarks:

1. **SPH Particles.** This keyword can be used to define initial nodal point temperatures for SPH particles by using nodal set IDs or node numbers from SPH particles.

*INITIAL_VAPOR_PART

Purpose: Initialization of a part as a vapor material for *PART using *MAT_NULL and *EOS_PHASE_CHANGE. All elements in the part are initialized as a vapor material.

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

VARIABLE	DESCRIPTION
PID	Part ID of the part to be initialized as a vapor material

*INITIAL_VEHICLE_KINEMATICS

Purpose: Define initial kinematical information for a vehicle. In its initial orientation, the vehicle's yaw, pitch, and roll axes must be aligned with the global axes. Successive simple rotations are taken about these body fixed axes.

Card 1	1	2	3	4	5	6	7	8
Variable	GRAV	PSID	XO	YO	ZO	XF	YF	ZF
Type	I	I	F	F	F	F	F	F
Default	none	none	0.	0.	0.	0.	0.	0.

Card 2	1	2	3	4	5	6	7	8
Variable	VX	VY	VZ	AAXIS	BAXIS	CAXIS	IVADD	
Type	F	F	F	I	I	I	I	
Default	0.	0.	0.	0	0	0	2	

Card 3	1	2	3	4	5	6	7	8
Variable	AANG	BANG	CANG	WA	WB	WC		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
GRAV	Gravity direction code (see Remark 1): EQ.0: Gravity not considered EQ.1: Global $+x$ direction EQ.-1: Global $-x$ direction

VARIABLE	DESCRIPTION
	EQ.2: Global $+y$ direction EQ.-2: Global $-y$ direction EQ.3: Global $+z$ direction EQ.-3: Global $-z$ direction Note: This must be the same for all vehicles present in the model.
PSID	Part set ID
XO	x-coordinate of initial position of mass center
YO	y-coordinate of initial position of mass center
ZO	z-coordinate of initial position of mass center
XF	x-coordinate of final position of mass center
YF	y-coordinate of final position of mass center
ZF	z-coordinate of final position of mass center
VX	Global x-component of mass center velocity
VY	global y-component of mass center velocity
VZ	global z-component of mass center velocity
AAXIS	First rotation axis code. EQ.1: Initially aligned with global x-axis EQ.2: Initially aligned with global y-axis EQ.3: Initially aligned with global z-axis
BAXIS	Second rotation axis code
CAXIS	Third rotation axis code
IVADD	Flag for velocity overwrite / add: EQ.1: Add to pre-defined velocities. EQ.2: Overwrite pre-defined velocities (default).
AANG	Rotation angle about the first rotation axis (degrees)
BANG	Rotation angle about the second rotation axis (degrees)

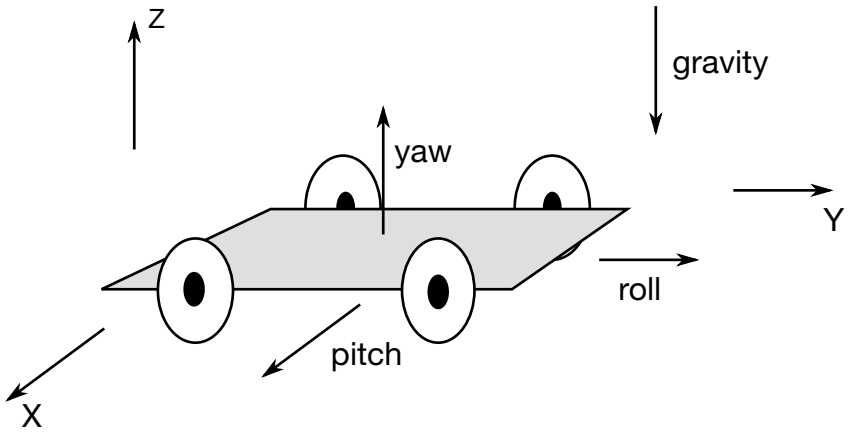

Figure 28-9. The vehicle pictured is to be oriented with a successive rotation sequence about the yaw, pitch, and roll axes, respectively. Accordingly, AAXIS = 3, BAXIS = 1, and CAXIS = 2. The direction of gravity is given by GRAV = -3.

VARIABLE	DESCRIPTION
CANG	Rotation angle about the third rotation axis (degrees)
WA	Angular velocity component for the *x* body-fixed axis (radian/second)
WB	Angular velocity component for the *y* body-fixed axis (radian/second)
WC	Angular velocity component for the *z* body-fixed axis (radian/second)

Remarks:

1. **GRAV.** The GRAV field is used to preserve the spatial location in the direction specified. To illustrate this feature, suppose a part set that represents a vehicle is initially oriented such that the wheels are on the ground (the ground plane) at *Z* = 100 and gravity is in the -*Z*-direction. To perform a roof drop simulation, the part set of the vehicle must be re-oriented, but the ground plane must be maintained. To do this, GRAV must be set to -3 which preserves *Z* = 100 as the lowest *Z*-coordinate, thereby maintaining the ground plane.

*INITIAL_VELOCITY

Purpose: Define initial nodal point velocities using nodal set ID's. This may also be used for sets in which some nodes have other velocities. See NSIDEX below.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	NSIDEX	BOXID	IRIGID	ICID			
Type	I	I	I	I	I			
Default	↓	{∅}	0	0	0			
Remark	1							

Card 2	1	2	3	4	5	6	7	8
Variable	VX	VY	VZ	VXR	VYR	VZR		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

Exempted Node Card. Additional card for NSIDEX > 0.

Card 3	1	2	3	4	5	6	7	8
Variable	VXE	VYE	VZE	VXRE	VYRE	VZRE		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODES, containing nodes for initial velocity. If NSID = 0, the initial velocity is applied to all nodes.

VARIABLE	DESCRIPTION
NSIDEX	Nodal set ID, see *SET_NODES, containing nodes that are exempted from the imposed velocities and may have other initial velocities.
BOXID	All nodes in box which belong to NSID are initialized. Nodes outside the box are not initialized. Exempted nodes are initialized to velocities defined by VXE, VYE, and VZE below, regardless of their location relative to the box.
IRIGID	Option to overwrite rigid body velocities defined on *PART_INERTIA or *CONSTRAINED_NODAL_RIGID_BODY_INERTIA cards. GE.1: part set ID, containing ID of parts to overwrite. Center of gravity of part must lie within box BOXID. If BOXID is not defined, then all parts defined in the set are overwritten. EQ.-1: Overwrite velocities for all rigid bodies defined with *PART_INERTIA or *CONSTRAINED_NODAL_-RIGID_BODY_INERTIA that have a center of gravity within box BOXID. If BOXID is not defined, then all are overwritten. EQ.-2: Overwrite velocities for all rigid bodies defined with *PART_INERTIA or *CONSTRAINED_NODAL_-RIGID_BODY_INERTIA.
ICID	Local coordinate system ID. The initial velocity is specified in the local coordinate system if ICID is greater than zero. Furthermore, if ICID is greater than zero, *INCLUDE_TRANSFORM does not rotate the initial velocity values specified by VX, VY, …, VZRE.
VX	Initial translational velocity in x-direction
VY	Initial translational velocity in y-direction
VZ	Initial translational velocity in z-direction
VXR	Initial rotational velocity about the x-axis
VYR	Initial rotational velocity about the y-axis
VZR	Initial rotational velocity about the z-axis
VXE	Initial velocity in x-direction of exempted nodes

VARIABLE	DESCRIPTION
VYE	Initial velocity in y-direction of exempted nodes
VZE	Initial velocity in z-direction of exempted nodes
VXRE	Initial rotational velocity in x-direction of exempted nodes
VYRE	Initial rotational velocity in y-direction of exempted nodes
VZRE	Initial rotational velocity in z-direction of exempted nodes

Remarks:

1. **Deck Restrictions.** This generation input must not be used with *INITIAL_-VELOCITY_GENERATION keyword.

2. **Multiple Nodal Velocity Initializations.** If a node velocity is initialized on more than one input card set, then the last set input will determine its velocity. However, if the nodal velocity is also specified on a *INITIAL_VELOCITY_NODE card, then the velocity specification on this card will be used.

3. **Overwriting Rigid Body Velocities.** Unless the option IRIGID is specified, initial velocities for rigid bodies given by *PART_INERTIA will overwrite generated initial velocities. The IRIGID option will cause the rigid body velocities specified on the *PART_INERTIA input to be overwritten. To directly specify the motion of a rigid body without using the keyword, *PART_INERTIA, which also requires the definition of the mass properties, use the keyword option, *INITIAL_VELOCITY_RIGID_BODY.

4. **Rigid Body Motion Consistency.** Nodes which belong to rigid bodies must have motion consistent with the translational and rotational velocity of the center of gravity (c.g.) of the rigid body. During initialization the rigid body translational and rotational rigid body momentum's are computed based on the prescribed nodal velocity field. From this rigid body momentum, the translational and rotational velocities of the nodal points are computed and reset to the new values. These new values may or may not be the same as the values prescribed for the nodes that make up the rigid body. Sometimes this occurs in single precision due to numerical round-off. If a problem like this occurs, specify the velocity using the keyword *INITIAL_VELOCITY_RIGID_BODY.

5. **Mid-Side Nodes.** Mid-side nodes generated by *ELEMENT_SOLID_TET4TO10 will not be initialized since the node numbers are not known a priori to the user. Instead use *INITIAL_VELOCITY_GENERATION if you intend to initialize the velocities of the mid-side nodes.

*INITIAL_VELOCITY_GENERATION

Purpose: Define initial velocities for rotating and/or translating bodies. These velocities are invoked at time = 0 unless a start time or sensor switch is specified with *INITIAL_VELOCITY_GENERATION_START_TIME (see Remark 7). If a dynamic relaxation phase is invoked, these velocities do not apply during that phase.

Card 1	1	2	3	4	5	6	7	8
Variable	ID	STYP	OMEGA	VX	VY	VZ	IVATN	ICID
Type	I	I	F	F	F	F	I	I
Default	{all}	↓	0.	0.	0.	0.	0	global

Card 2	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	NX	NY	NZ	PHASE	IRIGID
Type	F	F	F	F	F	F	I	I
Default	0.	0.	0.	0.	0.	0.	0	0

VARIABLE	DESCRIPTION
ID	Part ID, part set ID, or node set ID. If zero, STYP is ignored, and all velocities are set. WARNING for if IVATN = 0: If a part ID of a rigid body is specified, only the nodes that belong to elements of the rigid body are initialized. Nodes added with *CONSTRAINED_EXTRA_NODES are not initialized. Set IVATN = 1 to initialize the velocities of constrained nodes and parts.
STYP	Set type (see Remark 5): EQ.1: Part set ID (see *SET_PART) EQ.2: Part ID (see *PART) EQ.3: Node set ID (see *SET_NODE)
OMEGA	Angular velocity about the rotational axis. See Remark 6.

VARIABLE	DESCRIPTION
VX	Initial translational velocity in x-direction (see ICID below)
VY	Initial translational velocity in y-direction (see ICID below)
VZ	Initial translational velocity in z-direction (see ICID below)
IVATN	Flag for setting the initial velocities of constrained nodes and parts: EQ.0: Constrained parts are ignored. EQ.1: Constrained parts and nodes constrained to parts will be assigned initial velocities like the part to which they are constrained.
ICID	Local coordinate system ID. If ICID = 0, the specified translational velocities (VX,VY,VZ) and the direction cosines of the rotation axis (NX,NY,NZ) are in the global system; otherwise, they are in the local system specified by ICID. Therefore, if ICID is defined, *INCLUDE_TRANSFORM does not transform (VX,VY,VZ) and (NX,NY,NZ).
XC	Global x-coordinate on rotational axis
YC	Global y-coordinate on rotational axis
ZC	Global z-coordinate on rotational axis
NX	x-direction cosine. If set to -999, NY and NZ are interpreted as the 1st and 2nd nodes defining the rotational axis, in which case the coordinates of node NY are used as XC, YC, ZC. If ICID is defined, the direction cosine, (NX,NY,NZ), is projected along coordinate system ICID to yield the direction cosines of the rotation axis only if NX ≠ -999.
NY	y-direction cosine or the 1st node of the rotational axis when NX = -999.
NZ	z-direction cosine or the 2nd node of the rotational axis when NX = -999.
PHASE	Flag determining basis for initialization of velocity (see Remarks 6 and 7): EQ.0: Initial velocities are applied at $t = 0$ of the regular transient phase of the analysis and are based on the undeformed geometry. Rigid bodies whose velocities are initialized

VARIABLE	DESCRIPTION
	using this keyword should always use PHASE = 0. EQ.1: Initial velocities of deformable bodies are based on geometry that includes deformation incurred prior to the application of the initial velocities. That deformation could be due to a dynamic relaxation phase or due to a nonzero start time specified with *INITIAL_VELOCITY_GENERATION_START_TIME.
IRIGID	Controls hierarchy of initial velocities set with *INITIAL_VELOCITY_GENERATION versus those set with *PART_INERTIA / *CONSTRAINED_NODAL_RIGID_BODY_INERTIA when the commands conflict. EQ.0: *PART_INERTIA / *CONSTRAINED_NODAL_RIGID_BODY_INERTIA controls initial velocities. EQ.1: *INITIAL_VELOCITY_GENERATION controls initial velocities. This option does not apply if STYP = 3.

Remarks:

1. **Exclusions.** This generation input must not be used with *INITIAL_VELOCITY or *INITIAL_VELOCITY_NODE options.

2. **Order dependence.** The velocities are initialized in the order the *INITIAL_VELOCITY_GENERATION input is defined. Later input using the *INITIAL_VELOCITY_GENERATION keyword may overwrite the velocities previously set.

3. **Consistency for rigid body nodes.** Nodes that belong to rigid bodies must have motion consistent with the translational and rotational velocity of the rigid body. During initialization, the translational and rotational rigid body momentums are computed based on the prescribed nodal velocities. From this rigid body motion, the velocities of the nodal points are computed and reset to the new values. These new values may or may not be the same as the values prescribed for the node.

4. **SPH.** Prior to R15, SPH elements can only be initialized with node sets (STYP = 3). Starting with R15, SPH elements can additionally be initialized with parts and part sets.

5. **Constrained nodal rigid bodies.** Part IDs of *CONSTRAINED_NODAL_RIGID_BODYs that do not include the INERTIA option are not recognized by

the code in the case of STYP = 1 or 2. Use STYP = 3 (a node set ID) when initializing the velocity of such nodal rigid bodies.

6. **Rotating bodies.** When velocities for a rotating body (nonzero OMEGA) are initialized at $t = 0$ following a dynamic relaxation phase in which body forces (*LOAD_BODY) serve to preload the spinning body, velocities of deformable parts/nodes of the spinning body should be included in an *INITIAL_VELOCITY_GENERATION command with PHASE = 1. Furthermore, any rigid parts/nodes of the spinning body should be initialized using a second *INITIAL_VELOCITY_GENERATION command with PHASE = 0.

7. **Nonzero start time for initial velocity.** *INITIAL_VELOCITY_GENERATION_START_TIME can cause a delayed start time for the velocity. A delayed start time requires that the input deck contain an *INITIAL_VELOCITY_GENERATION command with PHASE = 1. Similar to the situation described in Remark 6, if there are any rigid body nodes whose velocity is being initialized, those nodes must be included in a second *INITIAL_VELOCITY_GENERATION command with PHASE = 0. Note that all velocity set with *INITIAL_VELOCITY_GENERATION begins at the start time if the input deck contains at least one *INITIAL_VELOCITY_GENERATION keyword with PHASE = 1. If the deck *only* includes one *INITIAL_VELOCITY_GENERATION with PHASE = 0, the start time is ignored, and the velocity is initialized at $t = 0$.

   The following example illustrates the initialization of translational velocity in the y-direction to 8.8 for node set 55 at $t = 13$. Node set 55 includes both rigid body nodes and deformable body nodes. Note that the same all-inclusive node set may be used for both *INITIAL_VELOCITY_GENERATION commands.

```
$ *INITIAL_VELOCITY_GENERATION
$#      id      styp     omega        vx        vy        vz     ivatn      icid
        55         3                           8.8
$#      xc        yc        zc        nx        ny        nz     phase    irigid
                                                                     0
*INITIAL_VELOCITY_GENERATION
$#      id      styp     omega        vx        vy        vz     ivatn      icid
        55         3                           8.8
$#      xc        yc        zc        nx        ny        nz     phase    irigid
                                                                     1
*INITIAL_VELOCITY_GENERATION_START_TIME
$#   stime
       13.
```

*INITIAL_VELOCITY_GENERATION_START_TIME

Purpose: Define a time to initialize velocities after time zero or provide a *SENSOR_SWITCH switch ID for a switch that causes the initialization of the velocities upon meeting the switch condition. Time zero starts after dynamic relaxation if used for initialization.

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

VARIABLE	DESCRIPTION
STIME	Start time or switch ID: GT.0.0: Start time at which LS-DYNA initializes the velocities. LT.0.0: |STIME| is a *SENSOR_SWITCH switch ID. When the switch condition is met, LS-DYNA initializes the velocities.

Remarks:

1. **One start time.** Only one *INITIAL_VELOCITY_GENERATION_START_TIME can be specified. Multiple start times are not allowed.

2. **Using start time.** All *INITIAL_VELOCITY_GENERATION commands adhere to the start time or switch condition provided the requirement is met that at least one of those commands has PHASE set to 1.

3. **Velocities of other nodes.** When *INITIAL_VELOCITY_GENERATION_START_TIME is active, nodes that are not part of the initial velocity generation definitions will be re-initialized with velocities as they were at $t = 0$.

*INITIAL_VELOCITY_NODE

Purpose: Define initial nodal point velocities for a node.

Card	1	2	3	4	5	6	7	8
Variable	NID	VX	VY	VZ	VXR	VYR	VZR	ICID
Type	I	F	F	F	F	F	F	I
Default	none	0.	0.	0.	0.	0.	0.	0

VARIABLE	DESCRIPTION
NID	Node ID
VX	Initial translational velocity in x-direction
VY	Initial translational velocity in y-direction
VZ	Initial translational velocity in z-direction
VXR	Initial rotational velocity about the x-axis
VYR	Initial rotational velocity about the y-axis
VZR	Initial rotational velocity about the z-axis
ICID	Local coordinate system ID. The specified velocities are in the local system if ICID is greater than zero. Furthermore, if ICID is greater than zero, *INCLUDE_TRANSFORM does not rotate the initial velocity values specificied by VX, VY, …, VZR.

See Remarks on *INITIAL_VELOCITY card.

*INITIAL_VELOCITY_RIGID_BODY

Purpose: Define the initial translational and rotational velocities for a rigid body or a nodal rigid body. Depending on the value of CMO (see *CONSTRAINED_NODAL_-RIGID_BODY and *MAT_RIGID), the velocities are either at the center of gravity or at an arbitrary point in space. This input overrides all other velocity input for the rigid body and the nodes which define the rigid body.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	VX	VY	VZ	VXR	VYR	VZR	ICID
Type	I	F	F	F	F	F	F	I
Default	none	0.	0.	0.	0.	0.	0.	0

VARIABLE	DESCRIPTION
PID	Part ID of the rigid body or the nodal rigid body.
VX	Initial translational velocity in global x-direction.
VY	Initial translational velocity in global y-direction.
VZ	Initial translational velocity in global z-direction.
VXR	Initial rotational velocity about the global x-axis.
VYR	Initial rotational velocity about the global y-axis.
VZR	Initial rotational velocity about the global z-axis.
ICID	Local coordinate system ID. The specified velocities are in the local system if ICID is greater than zero. Furthermore, if ICID is greater than zero, *INCLUDE_TRANSFORM does not transform the initial velocity values specified by VX, VY, …, VZR.

See Remarks 3 and 4 of the *INITIAL_VELOCITY input description.

***INITIAL_VOID**_*OPTION*

Available options include:

PART

SET

Purpose: Define initial voided part set IDs or part numbers. This command can be used only when ELFORM = 12 in *SECTION_SOLID. Void materials cannot be created during the calculation. Fluid elements which are evacuated, such as by a projectile moving through the fluid, during the calculation are approximated as fluid elements with very low densities. The constitutive properties of fluid materials used as voids must be identical to those of the materials which will fill the voided elements during the calculation. Mixing of two fluids with different properties is not permitted with this option.

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

VARIABLE	DESCRIPTION
PSID/PID	Part set ID or part ID, see also *SET_PART:

Remarks:

This void option and multiple materials per element, see *ALE_MULTI-MATERIAL_-GROUP are incompatible and cannot be used together in the same run.

*INITIAL_VOLUME_FRACTION_{OPTION}

Available option:

<BLANK>

LSDA

NALEGP

Purpose: Define initial volume fractions of different materials in multi-material ALE / S-ALE elements. Without the NALEGP or LSDA option, the keyword allows up to 7 ALE multi-material groups. The NALEGP option adds in an additional card immediately after the keyword to let users input the number of ALE multi-material groups to be read in for each element. The LSDA option is for S-ALE only. It allows you to include the lsda file produced from a previous S-ALE simulation that contained either *INITIAL_VOLUME_FRACTION_GEOMETRY or *ALE_STRUCTURED_MESH_VOLUME_FILLING. The default name of the lsda file is salevfrc.lsda. This file is used to pre-fill and trim the mesh which can save simulation time.

Card Summary:

Card 1a. This card is included if there is no keyword option. Repeat this card as many times as desired (one card for each element volume fraction to be defined). This input ends at the next keyword ("*") card.

EID	VF1	VF2	VF3	VF4	VF5	VF6	VF7

Card 1b. This card is included if the keyword option is NALEGP. This card set (Card 1b, 1b.1, and 1b.2) is repeated as many times as desired (one set for each element volume fraction to be defined).

NALEGP							

Card 1b.1. This card is included if the option is NALEGP.

EID	VF1	VF2	VF3	VF4	VF5	VF6	VF7

Card 1b.2. This card is included if the option is NALEGP and NALEGP > 7. Repeat this card in each set enough times to define NALEGP volume fractions.

VF*i*	VF(*i*+1)	VF(*i*+2)	VF(*i*+3)	VF(*i*+4)	VF(*i*+5)	VF(*i*+6)	VF(*i*+7)

Card 1c. Include this card if the LSDA keyword option is used.

FILENAME

Data Card Definitions:

Volume Fraction Card. This card is included if no keyword option is used. Include as many of this card as desired. This input ends at the next keyword ("*") card.

Card 1a	1	2	3	4	5	6	7	8
Variable	EID	VF1	VF2	VF3	VF4	VF5	VF6	VF7
Type	I	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
EID	Element ID.
VF1	Volume fraction of multi-material group 1, AMMGID = 1.
VF2	Volume fraction of multi-material group 2. Only needed in simulations with at least 3 material groups. Otherwise VF2 = 1 − VF1.
VF3	Volume fraction of multi-material group 3, AMMGID = 3.
VF4	Volume fraction of multi-material group 4, AMMGID = 4.
VF5	Volume fraction of multi-material group 5, AMMGID = 5.
VF6	Volume fraction of multi-material group 6, AMMGID = 6.
VF7	Volume fraction of multi-material group 7, AMMGID = 7.

NALEGP Card. Include this card for the NALEGP keyword option. This card set (Card 1b, 1b.1, and 1b.2) is repeated as many times as desired (one set for each element volume fraction to be defined).

Card 1b	1	
Variable	NALEGP	
Type	I	

VARIABLE	DESCRIPTION
NALEGP	Number of volume fractions.

NALEGP Volume Fractions Card. Include this card for NALEGP keyword option. This card set (Card 1b, 1b.1, and 1b.2) is repeated as many times as desired (one set for each element volume fraction to be defined).

Card 1b.1	1	2	3	4	5	6	7	8
Variable	EID	VF1	VF2	VF3	VF4	VF5	VF6	VF7
Type	I	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	0.0

NALEGP Extra Volume Fractions Card. This card is included if the option is NALEGP and NALEGP > 7. Repeat this card in each set enough times to define NALEGP volume fractions.

Card 1b.2	1	2	3	4	5	6	7	8
Variable	VF*i*	VF(*i*+1)	VF(*i*+2)	VF(*i*+3)	VF(*i*+4)	VF(*i*+5)	VF(*i*+6)	VF(*i*+7)
Type	I	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
EID	Element ID.
VF1	Volume fraction of multi-material group 1, AMMGID = 1.
VF2	Volume fraction of multi-material group 2. Only needed in simulations with at least 3 material groups. Otherwise VF2 = 1 − VF1.
VF3	Volume fraction of multi-material group 3, AMMGID = 3.
VF4	Volume fraction of multi-material group 4, AMMGID = 4.
VF5	Volume fraction of multi-material group 5, AMMGID = 5.

VARIABLE	DESCRIPTION
VF6	Volume fraction of multi-material group 6, AMMGID = 6.
VF7	Volume fraction of multi-material group 7, AMMGID = 7.
VF(N)	Volume fraction of multi-material group N, AMMGID = N. Define NALEGP volume fractions.

LSDA File Card. This card is included for keyword option LSDA.

Card 1c	1	2	3	4	5	6	7	8
Variable	FILENAME							
Type	A							

VARIABLE	DESCRIPTION
FILENAME	Name of the lsda file produced in a previous S-ALE run that contained either *INITIAL_VOLUME_FRACTION_GEOMETRY or *ALE_STRUCTURED_MESH_VOLUME_FILLING. The default name of the lsda file is salevfrc.lsda.

*INITIAL_VOLUME_FRACTION_GEOMETRY

Purpose: This is a volume-filling command for defining the volume fractions of various ALE multi-material groups (AMMG) that initially occupy several spatial regions in an ALE mesh. This applies only to ELFORMs 11 and 12 in *SECTION_SOLID and ALE-FORM 11 in *SECTION_ALE2D. For ELFORM 12, AMMGID 2 is void. See Remark 2.

Card Summary:

Card 1. This card is required.

FMSID	FMIDTYP	BAMMG	NTRACE				

Card 2. For each container include this card (Container Card) and one geometry card (Card 3*α*). Include as many pairs as desired. This input ends at the next keyword ("*") card.

CNTTYP	FILLOPT	FAMMG	VX	VY	VZ		

Card 3a. Include this card if CNTTYP = 1.

SID	STYPE		XOFFST				

Card 3b. Include this card if CNTTYP = 2.

SGSID		XOFFST					

Card 3c. Include this card if CNTTYP = 3.

X0	Y0	Z0	XCOS	YCOS	ZCOS		

Card 3d. Include this card if CNTTYP = 4.

X0	Y0	Z0	X1	Y1	Z1	R1	R2

Card 3e. Include this card if CNTTYP = 5.

X0	Y0	Z0	X1	Y1	Z1	LCSID	

Card 3f. Include this card if CNTTYP = 6.

X0	Y0	Z0	R0				

Card 3g. Include this card if CNTTYP = 7.

IDFUNC							

Data Card Definitions:

Background ALE Mesh Card. Defines the background ALE mesh set & an AMMGID that initially fills it.

Card 1	1	2	3	4	5	6	7	8
Variable	FMSID	FMIDTYP	BAMMG	NTRACE				
Type	I	I	I	I				
Default	none	0	none	3				

VARIABLE	DESCRIPTION
FMSID	Background ALE (fluid) mesh SID to be initialized or filled with various AMMGs. This set ID refers to one or more ALE parts.
FMIDTYP	ALE mesh set ID type: EQ.0: FMSID is an ALE part set ID (PSID). EQ.1: FMSID is an ALE part ID (PID).
BAMMG	The background fluid group ID or ALE Multi-Material group ID (AMMGID) that initially fills the entire ALE mesh region defined by FMSID. For S-ALE, AMMG name (AMMGNM) could be also used in place of AMMGID. See Remark 8.
NTRACE	Number of sampling points for volume filling detection. Typically, NTRACE ranges from 3 to maybe 10 (or more). The higher it is, the finer the ALE element is divided so that small gaps between 2 Lagrangian shells may be filled in. See Remark 4.

Container Card. Defines the container type and the AMMGID that fills the region defined by the container type.

Card 2	1	2	3	4	5	6	7	8
Variable	CNTTYP	FILLOPT	FAMMG	VX	VY	VZ		
Type	I	I	I	F	F	F		
Default	none	0	none	0	0	0		

VARIABLE	DESCRIPTION
CNTTYP	A "container" defines a Lagrangian surface boundary of a spatial region, inside (or outside) of which, an AMMG would fill up. CNTTYP defines the container geometry type of this surface boundary (or shell structure). EQ.1: The container geometry is defined by a part ID (PID) or a part set ID (PSID), where the parts (see *PART or *SET_PART) should be defined by shell elements in 3D (beam elements in 2D). If the parts are meshed with solids in 3D (shells in 2D), their boundaries define the container geometry (see Remark 7). EQ.2: The container geometry is defined by a segment set (SGSID). EQ.3: The container geometry is defined by a plane: a point and a normal vector. EQ.4: The container geometry is defined by a conical surface: 2 end points and 2 corresponding radii (in 2D see Remark 6). EQ.5: The container geometry is defined by a cuboid or rectangular box: 2 opposing end points, minimum to maximum coordinates. EQ.6: The container geometry is defined by a sphere: 1 center point, and a radius. EQ.7: The container geometry is defined with a user-defined function implemented using *DEFINE_FUNCTION. The arguments of the function should be the coordinates of a point (x,y,z). The function should return 1.0 if the point is inside the geometry.

VARIABLE	DESCRIPTION
FILLOPT	A flag to indicate which side of the container surface the AMMG is supposed to fill. CNTTYP = 1, 2, and 3, the "head" side of a container surface/segment is defined as the side pointed to by the heads of the normal vectors of the segments ("tail" side refers to opposite direction to "head"). See Remark 5. Note that for CNTTYP = 1 and 2, the fluid interface can be offset from the container walls with XOFFST. XOFFST does not apply to the other container geometries. EQ.0: The "head" side of the geometry defined above will be filled with fluid (default). For CNTTYP = 4, 5, 6, and 7, the inside of the container is filled. EQ.1: The "tail" side of the geometry defined above will be filled with fluid. For CNTTYP = 4, 5, 6, and 7, the outside of the container is filled.
FAMMG	This defines the fluid group ID or ALE Multi-Material group ID (AMMGID) which will fill up the interior (or exterior) of the space defined by the "container". *The order of AMMGIDs is determined by the order in which they are listed under *ALE_MULTI-MATERIAL_GROUP card.* For example, the first data card under the *ALE_MULTI-MATERIAL_GROUP keyword defines the multi-material group with ID (AMMGID) 1, the second data card defined AMMGID = 2, and so on. In case of S-ALE, AMMG name (AMMGNM) could be also used in place of AMMGID. See Remark 8. LT.0: |FAMMG| is a *SET_MULTI-MATERIAL_GROUP_LIST ID listing pairs of group IDs. For each pair, the 2nd group replaces the first one in the "container".
VX	Initial velocity in the global *x*-direction for this AMMGID.
VY	Initial velocity in the global *y*-direction for this AMMGID.
VZ	Initial velocity in the global *z*-direction for this AMMGID.

Part/Part Set Container Card. Additional card for CNTTYP = 1.

Card 3a	1	2	3	4	5	6	7	8
Variable	SID	STYPE	NORMDIR	XOFFST				
Type	I	I	I	F				
Default	none	0	0	0.0				
Remark			obsolete					

VARIABLE	DESCRIPTION
SID	A set ID pointing to a part ID (PID) or part set ID (PSID) of the Lagrangian shell element structure defining the "container" geometry to be filled (see *PART or *SET_PART).
SSTYPE	Set ID type: EQ.0: Container SID is a Lagrangian part set ID (PSID). EQ.1: Container SID is a Lagrangian part ID (PID).
NORMDIR	Obsolete (see Remark 5).
XOFFST	|XOFFST| is the absolute length for offsetting the fluid interface from the nominal fluid interface LS-DYNA would otherwise define by default. The sign of XOFFST determines which direction the interface is offset. It is based on the normal vectors of the segments associated with the container. XOFFST.GT.0: Interface is offset along the positive direction of the segments of the container. XOFFST.LT.0: Interface is offset in the negative direction of the normal vectors of the segments of the container. This is applicable to cases in which high pressure fluid is contained within a container. The offset allows LS-DYNA time to prevent leakage. In general, this may be set to roughly 5-10% of the ALE element width. It may be important only for when ILEAK is turned ON to give the code time to "catch" the leakage (see *CONSTRAINED_LAGRANGE_IN_SOLID). If ILEAK is not ON, this may not be necessary.

Segment Set Container Card. Additional card for CNTTYP = 2.

Card 3b	1	2	3	4	5	6	7	8
Variable	SGSID	NORMDIR	XOFFST					
Type	I	I	F					
Default	none	0	0.0					
Remark		obsolete						

VARIABLE	DESCRIPTION
SGSID	Segment Set ID defining the "container", see *SET_SEGMENT.
NORMDIR	Obsolete (see Remark 5).
XOFFST	|XOFFST| is the absolute length for offsetting the fluid interface from the nominal fluid interface LS-DYNA would otherwise define by default. The sign of XOFFST determines which direction the interface is offset. It is based on the normal vectors of the segments associated with the container. XOFFST.GT.0: Interface is offset along the positive direction of the segments of the container. XOFFST.LT.0: Interface is offset in the negative direction of the normal vectors of the segments of the container. This is applicable to cases in which high pressure fluid is contained within a container. The offset allows LS-DYNA time to prevent leakage. In general, this may be set to roughly 5-10% of the ALE element width. It may be important only for when ILEAK is turned ON to give the code time to "catch" the leakage (see *CONSTRAINED_LAGRANGE_IN_SOLID). If ILEAK is not ON, this may not be necessary.

Plane Card. Additional card for CNTTYP = 3.

Card 3c	1	2	3	4	5	6	7	8
Variable	X0	Y0	Z0	XCOS	YCOS	ZCOS		
Type	F	F	F	F	F	F		
Default	none	none	none	none	none	none		

VARIABLE	DESCRIPTION
X0, Y0, Z0	x, y and z coordinate of a spatial point on the plane.
XCOS, YCOS, ZCOS	x, y and z direction cosines of the plane normal vector. The filling will occur on the side pointed to by the plane normal vector (or "head" side).

Cylinder/Cone Container Card. Additional Card for CNTTYP = 4 (see Remark 6 for 2D).

Card 3d	1	2	3	4	5	6	7	8
Variable	X0	Y0	Z0	X1	Y1	Z1	R1	R2
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
X0, Y0, Z0	x, y and z coordinate of the center of the 1st base of the cone.
X1, Y1, Z1	x, y and z coordinate of the center of the 2nd base of the cone.
R1	Radius of the 1st base of the cone
R2	Radius of the 2nd base of the cone

Rectangular Box Container Card. Additional Card for CNTTYP = 5.

Card 3e	1	2	3	4	5	6	7	8
Variable	X0	Y0	Z0	X1	Y1	Z1	LCSID	
Type	F	F	F	F	F	F	I	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
X0, Y0, Z0	Minimum *x*, *y* and *z* coordinates of the box.
X1, Y1, Z1	Maximum *x*, y and *z* coordinates of the box.
LCSID	Local coordinate system ID, if defined, the box is aligned with the local coordinate system instead of global coordinate system. Please see *DEFINE_COORDINATE_*OPTION*for details.

Sphere Container Card. Additional card for CNTTYP = 6.

Card 3f	1	2	3	4	5	6	7	8
Variable	X0	Y0	Z0	R0				
Type	F	F	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
X0, Y0, Z0	*x*, *y* and *z* coordinate of the center of the sphere.
R0	Radius of the sphere

User Defined Container Card. Additional card for CNTTYP = 7.

Card 3g	1	2	3	4	5	6	7	8
Variable	IDFUNC							
Type	I							
Default	none							

VARIABLE	DESCRIPTION
IDFUNC	Function ID (*DEFINE_FUNCTION) for the function of the coordinates (x,y,z) that gives the container geometry. If the point (x,y,z) is in the container, the function should return 1.0.

Remarks:

1. **Structure of Data Cards.** After Card 1 which defines the basic mesh filled by a certain fluid group (AMMGID), each "filling action" will require 2 additional lines of input: Cards 2 and 3α, where α depends on the CNTTYP value. At the minimum there will be 3 cards required for this command (1, 2, and 3α) for 1 "filling action".

   There can be one or more "filling actions" prescribed for each instance of this command. The "filling actions" take place in the prescribed order and the effects are cumulative. Later "filling actions" will over-write the previous ones. Therefore, any complex filling logic will require some planning. For example, the following card sequence with 2 "filling actions" is allowable:

```
*INITIAL_VOLIME_FRACTION_GEOMETRY
[Card 1]
[Card 2, CNTTYP = 1]
[Card 3a]
[Card 2, CNTTYP = 3]
[Card 3c]
```

   This sequence of cards prescribes a background ALE mesh with 2 "filling actions" to be executed. The 1st is a filling of a CNTTYP = 1 and the 2nd of CNTTYP = 3.

2. **Group IDs for ELFORM 12.** If ELFORM = 12, the single-material-and-void element formulation, is used in *SECTION_SOLID, then the non-void material defaults to AMMG = 1 and the void to AMMG = 2. These multi-material groups

are implied even though no *ALE_MULTI-MATERIAL_GROUP card is required.

3. **Using Shells to Divide Space.** A simple ALE background mesh (for example, a cuboid mesh) can be constructed enveloping some Lagrangian shell structure (or container). The ALE region inside this Lagrangian shell container may be filled with one multi-material group (AMMG1), and the outside region with another (AMMG2). This approach simplifies the mesh generation requirements for ALE material parts with complex geometries.

4. **NTRACE.** The default number of sampling points is NTRACE = 3 in which case the total number is

$$(2 \times \text{NTRACE} + 1)^3 = 7^3$$

This means an ALE element is subdivided into $7 \times 7 \times 7$ regions. Each is to be filled in with the appropriate AMMG. An example of this application would be the filling of initial gas between multiple layers of Lagrangian airbag shell elements sharing the same ALE element.

5. **Interior/Exterior Fill Setting.** To set which side of a container is to be filled: (1) define the shell (or segment) container with inward normal vectors; then (2) set the FILLOPT field on "Card 2" to 0, corresponding to the head of the normal, for the interior, and to 1, corresponding to the tail of the normal, for the exterior.

6. **Two Dimensional Geometry.** If the ALE model is 2D (*SECTION_ALE2D instead of *SECTION_SOLID), CNTTYP = 4 defines a quadrilateral. In this case the fields which, in the 3D case define a cone, are interpreted as the corner coordinates of a clockwise defined (inward normal) quadrilateral having the vertices: (X1, Y1), (X2, Y2), (X3, Y3), and (X4, Y4). The CNTTYP = 4 input fields X0, Y0, Z0, X1, Y1, Z1, R1, and R2 becomes X1, Y1, X2, Y2, X3, Y3, X4, and Y4 respectively. CNTTYP = 6 should be used to fill a circle.

7. **3D Solid (2D Shell) Mesh for CNTTYP = 1.** If a part, *P*, defines the geometry container for CNTTYP = 1, is meshed with solids in 3D or shells in 2D, and is in the ALE group FAMMG (*P* should be in *ALE_MULTI-MATERIAL_GROUP and it should have the ALE formulation 11 in the *SECTION keyword), the ALE elements superimposed with *P* will be filled with FAMMG. The mesh of *P* will become a dummy rigid part.

8. **AMMG NAME for S-ALE.** For the general ALE solver, you define each AMMG with *ALE_MULTI-MATERIAL_GROUP. In this case, each AMMG can only be referred to by their AMMGID. The AMMGID for each AMMG is based on the order of appearance of the AMMG in the input deck. For the S-ALE solver, you can define the AMMG using *ALE_STRUCTURED_MULTI-MATERIAL_-GROUP instead of *ALE_MULTI-MATERIAL_GROUP. With *ALE_STRUCTURED_MULTI-MATERIAL_GROUP, you give each AMMG a name with the

field AMMGNM. Each AMMG defined with that keyword can then be referred with either its name or its AMMGID (which is again based on order of appearance). We recommend using the name as it leads to fewer errors. For instance, if you add or delete AMMGs, then the AMMGIDs may change. Then, you must find all those references and change them accordingly. With the name, you do not need to modify the input deck for unchanged AMMGs.

9. **Files Output for Subsequent Analyses.** This keyword causes file alevfrc.inc to be output for both ALE and S-ALE simulations at the end of the run. For S-ALE, it also outputs file salevfrc.lsda which is an lsda version of alefrc.inc. alefrc.inc can be included with *INCLUDE for subsequent analyses while salevfrc.lsda can be included with *INITIAL_VOLUME_FRACTION_LSDA. These files pre-fill the volume fractions and store the trimmed mesh which can save time for subsequent analyses.

Example:

Consider a simple ALE model with ALE parts H1-H5 (5 AMMGs possible) and 1 Lagrangian shell (container) part S6. Only parts H1 and S6 initially have their meshes defined. We will perform 4 "filling actions". The volume filling results after each step will be shown below to clarify the concept used. The input for the volume filling looks like this.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
$ H1 = AMMG 1 = fluid 1 initially occupying whole ALE mesh= background mesh
$ H5 = AMMG 5 = fluid 5 fills below a plane = filling action 1 = CNTTYP=3
$ H2 = AMMG 2 = fluid 2 fills outside S6    = filling action 2 = CNTTYP=1
$ H3 = AMMG 3 = fluid 3 fills inside a cone = filling action 3 = CNTTYP=4
$ H4 = AMMG 4 = fluid 4 fills inside a box  = filling action 4 = CNTTYP=5
$ S6 =          Lagrangian shell container
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*ALE_MULTI-MATERIAL_GROUP
         1         1
         2         1
         3         1
         4         1
         5         1
*INITIAL_VOLUME_FRACTION_GEOMETRY
$ The 1st card fills the whole pid H1 with AMMG 1=background ALE mesh
$    FMSID   FMIDTYP     BAMMG     <=== card 1: background fluid
         1         1         1
$ filling action 1 = AMMG 5 fill all elms below a plane
$ CNTTYPE   FILLOPT FILAMMGID     <=== card a  : container: CNTTYPE=3=plane
         3         0         5
$   X0, Y0,   Z0,    NX,  NY, NZ   <=== card b-3: details on container =plane
  25.0,20.0, 0.0,   0.0,-1.0,0.0
$ filling action 2: AMMG 2 fills OUTSIDE (FILLOPT=1) shell S6 (inward normals);
$ CNTTYPE   FILLOPT     FAMMG  <== card a  : container #1; FILLOPT=1=fill tail
         1         1         2
$    SETID   SETTYPE   NORMDIR  <== card b-1: details on container #1
         6         1         0
$ filling action 3 = AMMG 3 fill all elms inside a CONICAL region
$ CNTTYPE   FILLOPT     FAMMG  CNTTYP = 4 = Container = conical region
         4         0         3
$       X1        Y1        Z1        X2        Y2        Z2        R1        R2
      25.0      75.0       0.0      25.0      75.0       1.0       8.0       8.0
```

```
$ filling action 4 = AMMG 4 fill all elms inside a BOX region
$ CNTTYPE   FILLOPT  FFLUIDID                 : CNTTYP=5 = "BOX"
         5         0         4
$      XMIN       YMIN       ZMIN       XMAX       YMAX       ZMAX
      65.0       35.0        0.0       85.0       65.0        1.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

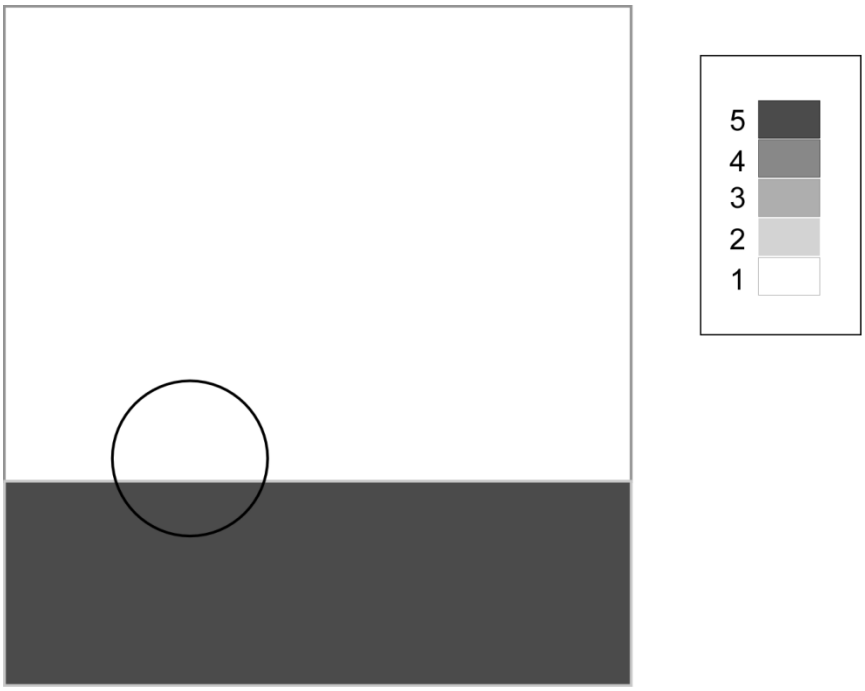

Figure 28-10. Before the 1st "filling action" the whole ALE mesh of part H1 is filled with AMMG 1 (white). After the 1st "filling action", AMMG 5 fills below the specified plane.

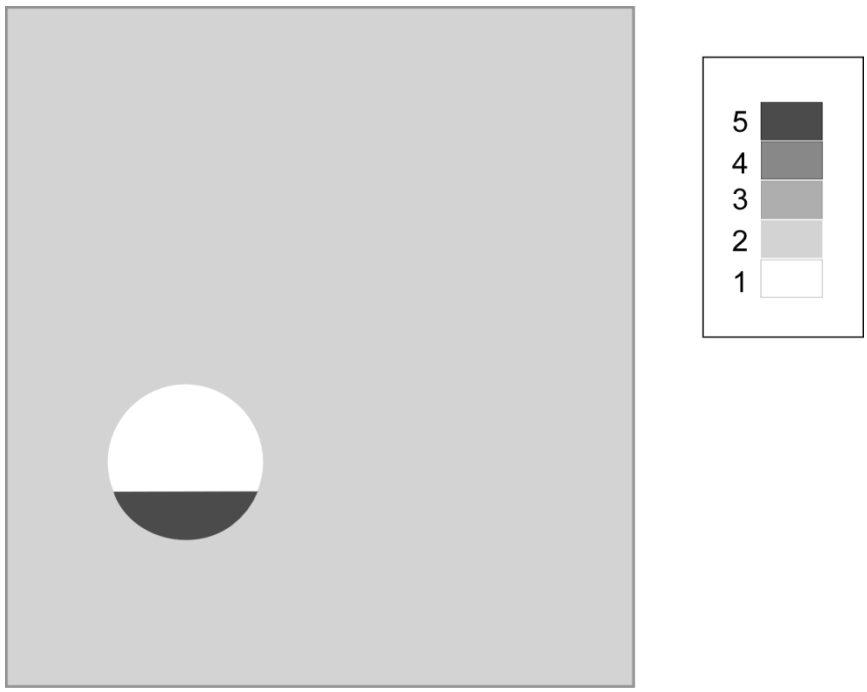

Figure 28-11. After the 1st and 2nd "filling actions", outside the shell (S6) is filled with AMMG 2.

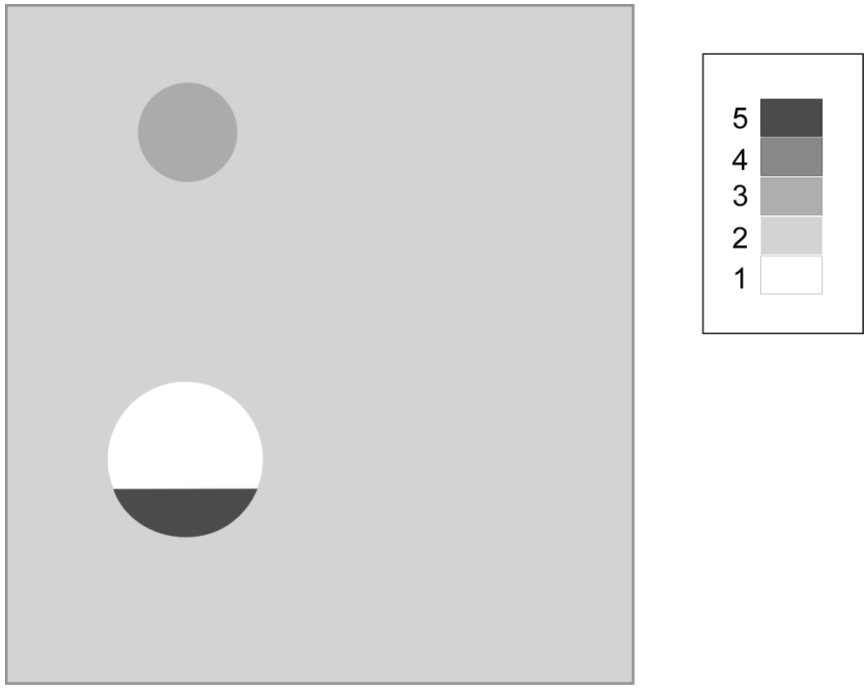

Figure 28-12. After the 1st, 2nd and 3rd "filling actions", it fills in the analytical sphere with AMMG 3.

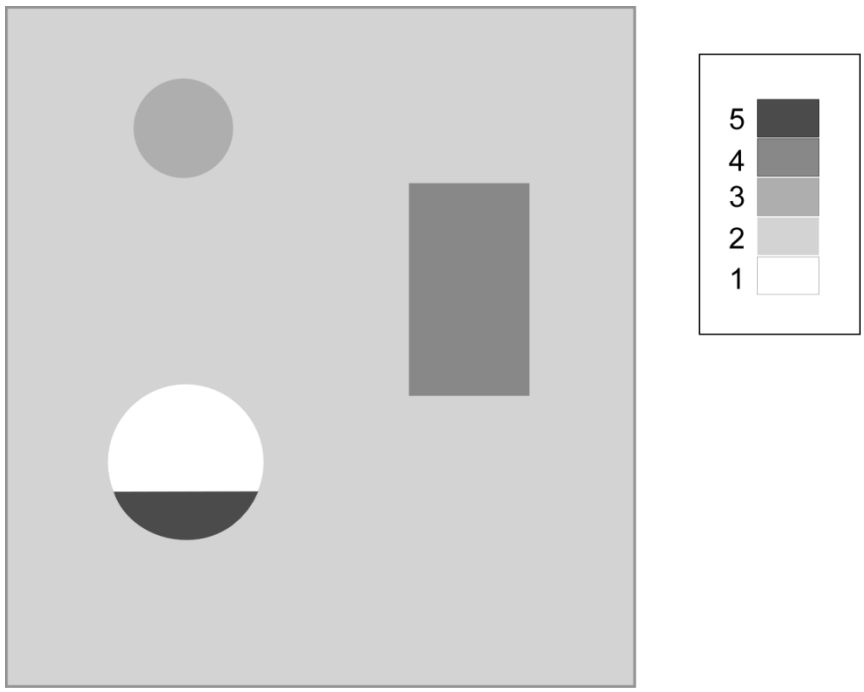

Figure 28-13. After the 1st, 2nd, 3rd and 4th "filling actions", the analytical box is filled with AMMG 4.