*PART

The following keywords are used in this section:

PART_*{OPTION1}_{OPTION2}_{OPTION3}_{OPTION4}_{OPTION5}_{OPTION6}

*PART_ADAPTIVE_FAILURE

*PART_ANNEAL

PART_COMPOSITE_*{OPTION}

*PART_DUPLICATE

*PART_MODES

*PART_MOVE

*PART_SENSOR

*PART_STACKED_ELEMENTS

***PART**_*{OPTION1}_{OPTION2}_{OPTION3}_{OPTION4}_{OPTION5}_{OPTION6}*

For *OPTION1*, the available options are

<BLANK>

INERTIA

REPOSITION

For *OPTION2*, the available options are

<BLANK>

CONTACT

For *OPTION3*, the available options are

<BLANK>

PRINT

For *OPTION4*, the available options are

<BLANK>

ATTACHMENT_NODES

For *OPTION5*, the available options are

<BLANK>

AVERAGED

For *OPTION6*, the available options are

<BLANK>

FIELD

Options 1, 2, 3, 4, 5, and 6 may be specified in any order on the *PART card.

Purpose: Define parts, that is, combine material information, section properties, hourglass type, thermal properties, and a flag for part adaptivity.

The INERTIA option allows the inertial properties and initial conditions to be defined rather than calculated from the finite element mesh. This applies to rigid bodies (see *MAT_RIGID) only.

The REPOSITION option applies to deformable materials and is used to reposition deformable materials attached to rigid dummy components whose motion is controlled by either CAL3D or MADYMO. At the beginning of the calculation, each component controlled by CAL3D/MADYMO is automatically repositioned to be consistent with the CAL3D/MADYMO input. However, deformable materials attached to these components will not be repositioned unless this option is used.

The CONTACT option allows part-based contact parameters to be used with contact types a3, 4, a5, b5, a10, 13, a13, 15 and 26, that is

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE,

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR,

*CONTACT_SINGLE_SURFACE,

*CONTACT_AUTOMATIC_NODES_TO_SURFACE,

*CONTACT_AUTOMATIC_BEAMS_TO_SURFACE,

*CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE,

*CONTACT_AUTOMATIC_SINGLE_SURFACE,

*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR,

*CONTACT_AIRBAG_SINGLE_SURFACE,

*CONTACT_ERODING_SINGLE_SURFACE,

*CONTACT_AUTOMATIC_GENERAL.

The default values for these contact parameters can be specified on the input for *CONTACT_....

The PRINT option allows user control over whether output data is written into the ASCII files matsum and rbdout. See *DATABASE_ASCII.

The AVERAGED option may be applied *only* to parts consisting of a single (non-branching) line of truss elements. The average strain and strain rate over the length of the truss elements in the *part* are calculated, and the resulting average axial force is applied to all the elements in the part. Truss elements in an averaged part form one long continuous "macro-element." The time step size for an AVERAGED part is based on the total length of the assembled trusses rather than on the shortest truss.

Effectively, the truss elements of an AVERAGED part behave as a string under uniform tension. In an AVERAGED part, no internal forces act to keep the nodes separated, and other force contributions from the surrounding system *must* play that role. Therefore, the nodes connected to the truss elements should be attached to other structural members. This model is prototypically used for modeling cables in mechanical actuators. The AVERAGED option can be activated for all material types available for truss elements.

The FIELD option allows you to define spatially varying properties for the current part. The field card determines a unique correlation between the current part's properties and the corresponding fields of data. Field data are specified in the physical space on sets of points; see *DEFINE_FIELD and *DEFINE_POINT_CLOUD. The FIELD option currently allows you to define spatially varying baseline orientation vectors for non-isotropic materials. It is currently supported for isogeometric shell and solid elements defined with *IGA keywords and for solid elements with ELFORM = 1.

Card Summary:

Card Sets. Repeat as many sets of data cards as desired (Cards 1 through 10, depending on the keyword options). This input ends at the next keyword ("*") card.

Card 1. This card is required.

HEADING	

Card 2. This card is required.

PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID

Card 3. Include this card if using the INERTIA option.

XC	YC	ZC	TM	IRCS	NODEID		

Card 4. Include this card if using the INERTIA option.

IXX	IXY	IXZ	IYY	IYZ	IZZ		

Card 5. Include this card if using the INERTIA option.

VTX	VTY	VTZ	VRX	VRY	VRZ		

Card 6. Include this card if using the INERTIA option. It is optional unless IRCS = 1.

XL	YL	ZL	XLIP	YLIP	ZLIP	CID	

Card 7. Include this card if using the REPOSITION option.

CMSN	MDEP	MOVOPT					

Card 8. Include this card if using the CONTACT option.

FS	FD	DC	VC	OPTT	SFT	SSF	CPARM8

Card 9. Include this card if using the PRINT option.

PRBF							

Card 10. Include this card if using the ATTACHMENT_NODES option.

ANSID							

Card 11. Include this card if using the FIELD option.

FIDBO							

Data Card Definitions:

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

VARIABLE	DESCRIPTION
HEADING	Heading for the part

Card 2	1	2	3	4	5	6	7	8
Variable	PID	SECID	MID	EOSID	HGID	GRAV	ADPOPT	TMID
Type	I/A	I/A	I/A	I/A	I/A	I	I	I/A
Default	none	none	none	0	0	0	0	0

VARIABLE	DESCRIPTION
PID	Part identification. A unique number or label must be specified.

VARIABLE	DESCRIPTION
SECID	Section identification defined in a *SECTION keyword. See Remark 7.
MID	Material identification defined in the *MAT section. See Remark 7.
EOSID	Equation of state identification defined in the *EOS section. Nonzero only for solid elements using an equation of state to compute pressure. See Remark 7.
HGID	Hourglass/bulk viscosity identification defined in the *HOURGLASS section. See Remark 7. EQ.0: Default values are used.
GRAV	Flag to turn on gravity initialization according to *LOAD_DENSITY_DEPTH. EQ.0: Part will be initialized only if included in the part set PSID in *LOAD_DENSITY_DEPTH. EQ.1: Part will be initialized irrespective of PSID in *LOAD_DENSITY_DEPTH.
ADPOPT	Indicate if this part is adapted. (See also *CONTROL_ADAPTIVE): LT.0: *r*-adaptive remeshing for 2D solids, |ADPOPT| gives the load curve ID that defines the element size as a function of time. EQ.0: Adaptive remeshing is inactive for this part ID. EQ.1: *h*-adaptivity for 3D shells and shell/solid/shell sandwich composites. EQ.2: *r*-adaptive remeshing for 2D solids, 3D tetrahedrons and 3D EFG. EQ.3: Axisymmetric *r*-adaptive remeshing for 3D solid (see Remark 6). EQ.9: Passive *h*-adaptivity for 3D shells. The elements in this part will not be split unless their neighboring elements in other parts need to be split more than one level.
TMID	Thermal material property identification defined in the *MAT_-THERMAL section. Thermal properties must be specified for all solid, shell, and thick shell parts for thermal or coupled thermal-structural analyses. Discrete elements are not considered in

VARIABLE	DESCRIPTION
	thermal analyses. See Remark 7.

Inertia Card 1. Additional card for the INERTIA option. See Remarks 2, 3, and 4.

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

VARIABLE	DESCRIPTION
XC	Global x-coordinate of center of mass. If nodal point NODEID is defined, XC, YC, and ZC are ignored, and the coordinates of NODEID are taken as the center of mass.
YC	Global y-coordinate of center of mass
ZC	Global z-coordinate of center of mass
TM	Translational mass
IRCS	Flag for inertia tensor reference coordinate system: EQ.0: Global inertia tensor EQ.1: Local inertia tensor given in a system defined by the orientation vectors
NODEID	Nodal point defining the center of mass of the rigid body. This node should be, but is not required to be, included as an extra node for the rigid body. If this node is free, its motion will not be updated to correspond with the rigid body after the calculation begins.

Inertia Card 2. Additional card for the INERTIA option.

Card 4	1	2	3	4	5	6	7	8
Variable	IXX	IXY	IXZ	IYY	IYZ	IZZ		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
IXX	I_{xx}, xx component of the inertia tensor (see Remark 4)
IXY	I_{xy}, xy component of the inertia tensor (see Remark 4)
IXZ	I_{xz}, xz component of the inertia tensor (see Remark 4)
IYY	I_{yy}, yy component of the inertia tensor (see Remark 4)
IYZ	I_{yz}, yz component of the inertia tensor (see Remark 4)
IZZ	I_{zz}, zz component of the inertia tensor (see Remark 4)

Inertia Card 3. Additional card for the INERTIA option.

Card 5	1	2	3	4	5	6	7	8
Variable	VTX	VTY	VTZ	VRX	VRY	VRZ		
Type	F	F	F	F	F	F		

VARIABLE	DESCRIPTION
VTX	Initial translational velocity of the rigid body in the global x-direction (see Remark 5)
VTY	Initial translational velocity of the rigid body in the global y-direction (see Remark 5)
VTZ	Initial translational velocity of the rigid body in the global z-direction (see Remark 5)
VRX	Initial rotational velocity of the rigid body about the global x-axis (see Remark 5)
VRY	Initial rotational velocity of the rigid body about the global y-axis (see Remark 5)
VRZ	Initial rotational velocity of the rigid body about the global z-axis (see Remark 5)

Inertial Coordinate System Card. Optional card required for IRCS = 1 with INERTIA option. Define two local vectors or a local coordinate system ID.

Card 6	1	2	3	4	5	6	7	8
Variable	XL	YL	ZL	XLIP	YLIP	ZLIP	CID	
Type	F	F	F	F	F	F	I	
Remark	1	1	1	1	1	1		

VARIABLE	DESCRIPTION
XL	x-coordinate of the local x-axis. The origin lies at (0,0,0).
YL	y-coordinate of the local x-axis
ZL	z-coordinate of the local x-axis
XLIP	x-coordinate of vector in the local xy-plane
YLIP	y-coordinate of vector in the local xy -plane
ZLIP	z-coordinate of vector in the local xy-plane
CID	Local coordinate system ID, see *DEFINE_COORDINATE_… With this option, leave fields 1 - 6 blank.

Reposition Card. Include this card for the REPOSITION option.

Card 7	1	2	3	4	5	6	7	8
Variable	CMSN	MDEP	MOVOPT					
Type	I	I	I					

VARIABLE	DESCRIPTION
CMSN	CAL3D segment number/MADYMO system number. See the numbering in the corresponding program.
MDEP	MADYMO ellipse/plane number: GT.0: Ellipse number

VARIABLE	DESCRIPTION
	EQ.0: Default LT.0: Absolute value is plane number.
MOVOPT	Flag to deactivate moving for merged rigid bodies; see *CONSTRAINED_RIGID_BODIES. This option allows a merged rigid body to be fixed in space while the nodes and elements of the generated CAL3D / MADYMO parts are repositioned: EQ.0: The merged rigid body is repositioned. EQ.1: The merged rigid body is not repositioned.

Contact Card. An additional card is required for the CONTACT option.

Card 8	1	2	3	4	5	6	7	8
Variable	FS	FD	DC	VC	OPTT	SFT	SSF	CPARM8
Type	F	F	F	F	F	F	F	F

NOTE: If FS, FD, DC, and VC are specified, they will not be used unless FS is set to a negative value (-1.0) in the *CONTACT input. These frictional coefficients apply only to contact types:

SINGLE_SURFACE,
AIRBAG_SINGLE_SURFACE,
AUTOMATIC_GENERAL,
AUTOMATIC_SINGLE_SURFACE,
AUTOMATIC_SINGLE_SURFACE_MORTAR,
AUTOMATIC_NODES_TO_...,
AUTOMATIC_SURFACE_...,
AUTOMATIC_SURFACE_..._MORTAR,
AUTOMATIC_ONE_WAY_...,
ERODING_SINGLE_SURFACE

Default values are input via *CONTROL_CONTACT input.

VARIABLE	DESCRIPTION
FS	Static coefficient of friction. The frictional coefficient is assumed to

VARIABLE	DESCRIPTION
	be dependent on the relative velocity, v_{rel}, of the surfaces in contact, $$\mu_c = \mathrm{FD} + (\mathrm{FS} - \mathrm{FD})e^{-\mathrm{DC}\times\lvert v_{\mathrm{rel}}\rvert}.$$ For Mortar contact $\mu_c = \mathrm{FS}$, meaning dynamic effects are ignored.
FD	Dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity, v_{rel}, of the surfaces in contact $$\mu_c = \mathrm{FD} + (\mathrm{FS} - \mathrm{FD})e^{-\mathrm{DC}\times\lvert v_{\mathrm{rel}}\rvert}.$$ For Mortar contact $\mu_c = \mathrm{FS}$, meaning dynamic effects are ignored.
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity, v_{rel}, of the surfaces in contact $$\mu_c = \mathrm{FD} + (\ \mathrm{FS} - \mathrm{FD})e^{-\mathrm{DC}\times\lvert v_{\mathrm{rel}}\rvert}.$$ For Mortar contact, $\mu_c = \mathrm{FS}$ (dynamical effects are ignored).
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed by $$F_{\mathrm{lim}} = \mathrm{VC} \times A_{\mathrm{cont}},$$ where A_{cont} is the area of the segment contacted by the node. The suggested value for VC is to use the yield stress in shear; that is, $\mathrm{VC} = \sigma_0/\sqrt{3}$, where σ_0 is the yield stress of the contacted material.
OPTT	Optional contact thickness. For SOFT = 2, it applies to solids, shells, and beams. For SOFT = 0 and 1 and Mortar contacts, it only applies to shells and beams. However, for the MPP version only, OPTT does affect the contact behavior of solid elements for SOFT = 0 and 1 but *not* by changing the contact thickness. In the case of MPP with SOFT = 0 and 1 for solids, OPTT overrides the thickness of the solid elements used for the calculation of the contact penetration release (see Table 11-2) but does *not* affect the contact thickness. This is *not* available in SMP. OPTT does *not* affect the contact thickness when applied to the parts on the tracked side (SURFA) of an AUTOMATIC_NODES_-TO_SURFACE contact. However, it affects the contact thickness of this type of contact's reference side (SURFB).
SFT	Optional thickness scale factor in automatic contact (scales the true

VARIABLE	DESCRIPTION
	thickness). This option applies to contact with shell and beam elements. The true thickness is the element thickness.
SSF	Scale factor on default tracked surface penalty stiffness for this part ID whenever it appears in the contact definition. If zero, SSF is taken as unity.
CPARM8	Flag to exclude beam-to-beam contact from the same PID for *CONTACT_AUTOMATIC_GENERAL. This flag applies only to MPP. The global default may be set using CPARM8 on MPP1 of *CONTACT_…_MPP. EQ.0: Flag is not set (default). EQ.1: Flag is set. EQ.2: Flag is set. CPARM8 = 2 has the additional effect of permitting contact treatment of spot weld (type 9) beams in AUTOMATIC_GENERAL contacts; spot weld beams are otherwise disregarded entirely by AUTOMATIC_GENERAL contacts.

Print Card. An additional card is required for the PRINT option. This option applies to rigid bodies and provides a way to turn off ASCII output in files rbdout and matsum.

Card 9	1	2	3	4	5	6	7	8
Variable	PRBF							
Type	I							

VARIABLE	DESCRIPTION
PRBF	Print flag for rbdout and matsum files. EQ.0: Default is taken from *CONTROL_OUTPUT. EQ.1: Write data into the rbdout file only. EQ.2: Write data into the matsum file only. EQ.3: Do not write data into rbdout and matsum.

Attachment Nodes Card. An additional card is required for the ATTACHMENT_-NODES option. See Remark 8.

Card 10	1	2	3	4	5	6	7	8
Variable	ANSID							
Type	I							

VARIABLE	DESCRIPTION
ANSID	Attachment node set ID. See Remark 8. This option should be used very cautiously and applies only to rigid bodies. The attachment point nodes are updated each cycle, whereas other nodes in the rigid body are updated only in the output databases. All loads seen by the rigid body must be applied through this nodal subset or directly to the center of gravity of the rigid body. This set must include all interacting nodes if the rigid body is in contact. EQ.0: All nodal updates are skipped for this rigid body. The null option can be used if the rigid body is fixed in space or if the rigid body does not interact with other parts. For example, the rigid body is only used for some visual purpose.

Field Card. An additional Card is required for the FIELD option.

Card 11	1	2	3	4	5	6	7	8
Variable	FIDBO							
Type	I							

VARIABLE	DESCRIPTION
FIDBO	Field ID for baseline orientation vectors for the material referenced by MID. See Remark 9.

Remarks:

1. **Local inertia tensor coordinate system.** The local Cartesian coordinate system is defined as described in *DEFINE_COORDINATE_VECTOR. The local z-axis vector is the vector cross product of the x-axis and the in-plane vector. The

local y-axis vector is finally computed as the vector cross product of the z-axis vector and the x-axis vector. The local coordinate system defined by CID has the advantage that the local system can be defined by nodes in the rigid body, making repositioning the rigid body in a pre-processor much easier since the local system moves with the nodal points.

2. **INERTIA keyword option and shared rigid/deformable nodes.** When specifying mass properties for a rigid body using the INERTIA keyword option, the mass contributions of deformable bodies to nodes shared by the rigid body should be considered part of the rigid body.

3. **INERTIA keyword option lacks default values.** If the INERTIA keyword option is used, all mass and inertia properties of the body *must* be specified. *There are no default values.*

4. **Inertia tensor characteristics.** The inertia terms are always with respect to the center of mass of the rigid body. The reference coordinate system defines the orientation of the axes, not the origin. Note that the off-diagonal terms of the inertia tensor are opposite in sign from the products of inertia.

5. **Initial velocity card for rigid bodies.** The *INITIAL_VELOCITY card may overwrite the initial velocity of the rigid body.

6. **Axisymmetric remeshing.** Axisymmetric remeshing is for 3D orbital forming. The adaptive part using this option needs to meet the following requirements for both geometry and discretization:

   a) The geometry is (quasi-) symmetric with respect to the local z-axis, which in turn must be parallel to the global z-axis. See CID in *CONTROL_-REMESHING.

   b) A set of 2D cross-sections with a uniform angular interval around the z-axis is discretized by mixed triangular and quadrilateral elements in a similar pattern.

   c) A set of circular lines around the z-axis passes through the nodes of the cross-sections and forms the orbital pentahedrons and hexahedrons.

7. **Allowed ID values.** The variables SECID, MID, EOSID, HGID, and TMID in *PART, and in *SECTION, *MAT, *EOS, *HOURGLASS, and *MAT_THERMAL, respectively, may be input as a 10-character alphanumeric variable (20-characters if the long format is used), such as "HS Steel," or as a 10-digit integer (20-digit integer if the long format is used) that may not exceed the limit of a 4-byte integer (2,147,483,647) if a single precision executable is used. If any of the aforementioned variables are non-numeric, the execution line should include "plabel=y" for proper (but, unfortunately, slower) input processing.

8. **Attachment nodes option.** All nodes are treated as attachment nodes if this option is not used. Attachment nodes apply to rigid bodies only. The motion of these nodes, which must belong to the rigid body, is updated each cycle. Other nodes in the rigid body are updated only for output purposes. Include all nodes in the attachment node set that interact with the structure through joints, contact, merged nodes, applied nodal point loads, and applied pressure. Include all nodes in the attachment node set if their displacements, accelerations, and velocities are to be written into an ASCII output file. Body force loads are applied to the center of gravity of the rigid body.

9. **Spatially-varying baseline orientation vectors.** This FIELD option is active only for material models that allow the specification of baseline orientation vectors with parameter AOPT in the keyword *MAT. For ELFORM = 1 solid elements, this option is supported for materials 22, 23, 26, 33, 40, 54, 55, 58, 59, 91, 92, 103, 104, 122, 126, 133, 142, 143, 157, 199, 213, 215, 219, 221, 233, 245, 261, 262, 263, 264, 278, and 291. If vectors are specified with this FIELD option, then the baseline orientation vectors associated with the parameter AOPT are disregarded for the current part.

   For shells, three values per point are required, so NV should be set to 3 in *DEFINE_FIELD. These values correspond to the components of the baseline orientation vector, which are automatically interpolated at the integration points. The interpolated vector is projected to the reference surface of the shell. The first material axis coincides with the projected vector unless an additional rotation, with respect to the element's normal vector, is specified. The rotation angle between the projected baseline orientation vector and the first material axis can be defined with the material orientation angle cards on *SECTION_IGA_SHELL by setting ICOMP equal to 1.

   For solids, six values per point are required, so NV should be set to 6 in *DEFINE_FIELD. The first three per-point values define the first baseline orientation vector $\mathbf{a}$, while the last three per-point values define the vector $\mathbf{d}$. The components of these vectors are automatically interpolated at the integration points. The interpolated vectors $\mathbf{a}_I$ and $\mathbf{d}_I$ are used to compute the material axes: $\mathbf{a}_{\mathrm{mat}} = \mathbf{a}_I$, $\mathbf{c}_{\mathrm{mat}} = \mathbf{a}_{\mathrm{mat}} \times \mathbf{d}_I$ and $\mathbf{b}_{\mathrm{mat}} = \mathbf{c}_{\mathrm{mat}} \times \mathbf{a}_{\mathrm{mat}}$, where the symbol $\times$ denotes the cross product between two vectors.

*PART_ADAPTIVE_FAILURE

Purpose: This option applies to two-dimensional adaptivity and allows a part that is singly connected to be split into two parts.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	T	TERM					
Type	I	F	I					

VARIABLE	DESCRIPTION
PID	Part ID
T	Thickness. When the thickness of the part reaches this minimum value, the part is split into two parts. *The value for T should be on the order of the element thickness of a typical element.*
TERM	Control adaptivity after the part separates: EQ.0: continue to adapt part (default) EQ.1: remove only this part from the adaptivity. Other parts will continue to adapt as normal. If there are no remaining parts to be adapted, adaptivity is disabled EQ.2: adaptivity is disabled for all parts.

*PART_ANNEAL

Available options include:

<BLANK>

SET

Purpose: To initialize the stress states at integration points within a specified part to zero at a given time during the calculation. This option is valid for parts that use constitutive models where the stress is incrementally updated. This option applies to the Hughes-Liu beam elements, the integrated shell elements, thick shell elements, and solid elements. In addition to the stress tensor components, the effective plastic strain is also set to zero.

Part Cards. Include as many parts cards as desired. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
PID/PSID	Part ID or part set ID if the SET option is active.
TIME	Time when the stress states are reinitialized.

PART_COMPOSITE_*{OPTION1}_{OPTION2}_{OPTION3}

For *OPTION1* the available options are:

<BLANK>

TSHELL

IGA_SHELL

For *OPTION2* the available option is:

<BLANK>

LONG

For *OPTION3* the available option is:

CONTACT

Purpose: The following input provides a simplified method of defining a composite material model for shell elements and thick shell (TSHELL) elements that eliminates the need for user defined integration rules and part IDs for each composite layer. When *PART_COMPOSITE is used, a section definition, *SECTION_SHELL or *SECTION_-TSHELL, and integration rule definition, *INTEGRATION_SHELL, are unnecessary. Using *PART_COMPOSITE, the user specifies layer thicknesses, and integration points reside at the mid-points of layers (trapezoidal rule), if not otherwise defined through IRPL on irregular optional Card 2 (see Remark 6). So, while *PART_COMPOSITE is more straightforward than *INTEGRATION_SHELL, it does not offer the same generality as *INTEGRATION_SHELL in terms of defining an arbitrary integration rule.

The material ID, thickness, material angle and thermal material ID for each through-thickness layer of a composite shell or thick shell are provided in the input for this command. The total number of layers is determined by the number of entries on these cards.

Unless the *ELEMENT_SHELL_THICKNESS card is set, the thickness is assumed to be constant on each shell element. The thickness, then, is given by summing the THICK*i* values from Card 5a/b below over the layers *i*. When the *ELEMENT_SHELL_THICKNESS card is included, the THICK*i* values are scaled to fit the nodal thickness values assigned using the *ELEMENT_SHELL_THICKNESS keyword. For thick shells, the total thickness is obtained from the positions of the nodes on the top and bottom surfaces. In this case, the THICK*i* are also scaled to conform to the geometry defined by the element's nodes.

For a more general method of defining composite shells and thick shells, see *ELEMENT_SHELL_COMPOSITE and *ELEMENT_TSHELL_COMPOSITE. These commands permit unique layer stack-ups for each element without requiring a unique part ID for each element.

With *PART_COMPOSITE, each Layer Properties Card provides two layers with 4 constants each. With the LONG keyword option, one Layer Properties Card specifies a single layer. The LONG option can also be used with options TSHELL and IGA_SHELL.

To maintain a direct association of through-thickness integration point numbers with physical plies in the case where plies span over more than one part ID, see Remark 5.

The CONTACT option allows part-based contact parameters to be used with the automatic contact types a3, 4, a5, a10, 13, a13, 15 and 26, which are listed under the *PART definition above.

The IGA_SHELL option allows *PART_COMPOSITE to be used with IGA shells. The part ID specified in *IGA_SHELL can reference the part ID in *PART_COMPOSITE.

Card Summary:

Card 1. This card is required.

HEADING	

Card 2. The keyword reader will interpret the card following Card 1 as optional Card 2 if the first column of the card is occupied by the string "OPTCARD." Otherwise, it is interpreted as Card 3a, 3b, or 3c.

OPTC	IRPL						

Card 3a. This card is included for all keyword options other than TSHELL or IGA_-SHELL.

PID	ELFORM	SHRF	NLOC	MAREA	HGID	ADPOPT	THSHEL

Card 3b. This card is included if the TSHELL option is used.

PID	ELFORM	SHRF			HGID		TSHEAR

Card 3c. This card is included if the IGA_SHELL option is used.

PID	ELFORM	SHRF	NLOC		IRL		

Card 4. This card is included if the CONTACT option is used.

FS	FD	DC	VC	OPTT	SFT	SSF	

Card 5a. This card is included if the LONG option is not used. This card provides the layer data. Include as many cards as necessary. This input ends at the next keyword ("*") card.

MID1	THICK1	B1	TMID1	MID2	THICK2	B2	TMID2

Card 5b. This card is included if the LONG option is used. This card provides the layer data. Include as many cards as necessary. This input ends at the next keyword ("*") card.

MID1	THICK1	B1	TMID1	PLYID1	SHRFAC1		

Data Card Definitions:

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

VARIABLE	DESCRIPTION
HEADING	Heading for the part

Irregular Optional Card. The keyword reader will interpret the card following Card 1 as optional Card 2 if the first column of the card is occupied by the string "OPTCARD." Otherwise, it is interpreted as Card 3 (a, b or c); see below.

Card 2	1	2	3	4	5	6	7	8
Variable	OPTC	IRPL						
Type	A	I						

VARIABLE	DESCRIPTION
OPTC	Include string "OPTCARD" to activate this card.
IRPL	Through thickness integration rule per layer (see Remark 6) EQ.103: 3-point Simpson

Thin Shell Card. The following card is required for thin shell composites that are not IGA. Omit this card if the TSHELL option or the IGA_SHELL option is used.

Card 3a	1	2	3	4	5	6	7	8
Variable	PID	ELFORM	SHRF	NLOC	MAREA	HGID	ADPOPT	THSHEL
Type	I	I	F	F	F	I	I	I
Default	none	none	1.0	0.0	0.0	0	0	0

VARIABLE	DESCRIPTION
PID	Part ID
ELFORM	Element formulation options for thin shells:

EQ.1: Hughes-Liu,

EQ.2: Belytschko-Tsay,

EQ.3: BCIZ triangular shell,

EQ.4: C^0 triangular shell,

EQ.6: S/R Hughes-Liu,

EQ.7: S/R co-rotational Hughes-Liu,

EQ.8: Belytschko-Leviathan shell,

EQ.9: Fully integrated Belytschko-Tsay membrane,

EQ.10: Belytschko-Wong-Chiang,

EQ.11: Fast (co-rotational) Hughes-Liu,

EQ.16: Fully integrated shell element (very fast),

EQ.-16: Fully integrated shell element modified for higher accuracy,

EQ.17: Fully integrated DKT, triangular shell element,

EQ.18: Fully integrated linear DK quadrilateral/triangular shell,

EQ.20: Fully integrated linear assumed strain C0 shell,

EQ.21: Fully integrated linear assumed strain C0 shell (5 DOF),

EQ.23: 8-node quadratic quadrilateral shell (see IRQUAD in *CONTROL_SHELL),

VARIABLE	DESCRIPTION
	EQ.24: 6-node quadratic triangular shell, EQ.25: Belytschko-Tsay shell with thickness stretch, EQ.26: Fully integrated shell with thickness stretch, EQ.27: C0 triangular shell with thickness stretch, EQ.30: Fast fully integrated element with 2 in-plane integration points based on ELFORM 16 EQ.41: Mesh-free (EFG) shell local approach (more suitable for crashworthiness analysis), EQ.42: Mesh-free (EFG) shell global approach (more suitable for metal forming analysis), EQ.101: User defined shell, EQ.102: User defined shell, EQ.103: User defined shell, EQ.104: User defined shell, EQ.105: User defined shell.
SHRF	Shear correction factor which scales the transverse shear stress.
NLOC	Location of reference surface, available for thin shells only. If nonzero, the offset distance from the plane of the nodal points to the reference surface of the shell in the direction of the shell normal vector is a value: $\text{offset} = -0.50 \times \text{NLOC} \times (\text{average shell thickness})$. This offset is not considered in the contact subroutines unless CNTCO is set to 1 or 3 in *CONTROL_SHELL. Alternatively, the offset can be specified by using the OFFSET option in the *ELEMENT_SHELL input section. EQ.1.0: Top surface, EQ.0.0: Mid-surface (default), EQ.-1.0: Bottom surface.
MAREA	Non-structural mass per unit area. This is additional mass which comes from materials such as carpeting. This mass is not directly included in the time step calculation.
HGID	Hourglass/bulk viscosity identification defined in the *HOURGLASS Section:

VARIABLE	DESCRIPTION
	EQ.0: Default values are used.
ADPOPT	Indicate if this part is adapted or not. Also see, *CONTROL_-ADAPTIVITY: EQ.0: No adaptivity, EQ.1: *h*-adaptive for 3-D thin shells.
THSHEL	Thermal shell formulation EQ.0: Default is governed by THSHEL on *CONTROL_SHELL EQ.1: Thick thermal shell EQ.2: Thin thermal shell

Thick Shell Card. This is an additional card for the TSHELL option.

Card 3b	1	2	3	4	5	6	7	8
Variable	PID	ELFORM	SHRF			HGID		TSHEAR
Type	I	I	F			I		I
Default	none	none	1.0			0		0

VARIABLE	DESCRIPTION
PID	Part ID
ELFORM	Element formulation options for thick shells: EQ.1: One point reduced integration, EQ.2: Selective reduced 2 × 2 in plane integration, EQ.3: Assumed strain 2 × 2 in plane integration, EQ.5: Assumed strain reduced integration with brick materials EQ.6: Assumed strain reduced integration with shell materials EQ.7: Assumed strain 2 × 2 in plane integration
SHRF	Shear correction factor which scales the transverse shear stress.

VARIABLE	DESCRIPTION
HGID	Hourglass/bulk viscosity identification defined in the *HOURGLASS Section: EQ.0: Default values are used.
TSHEAR	Flag for transverse shear stress distribution (see Remarks 3 and 4): EQ.0: Parabolic, EQ.1: Constant through thickness.

IGA Shell Card. The following card is required for the IGA_SHELL option.

Card 3c	1	2	3	4	5	6	7	8
Variable	PID	ELFORM	SHRF	NLOC		IRL		
Type	I	I	F	F		I		
Default	none	0	1.0	0.0		0		

VARIABLE	DESCRIPTION
PID	Part ID
ELFORM	Element formulation options for IGA shells: EQ.0: Reissner-Mindlin with fibers at the control points EQ.1: Kirchhoff-Love with fibers at the control points EQ.2: Kirchhoff-Love with fibers at the integration points EQ.3: Reissner-Mindlin with fibers at the integration points EQ.5: Shell with thickness stretch based on the ELFORM = 0 EQ.6: Shell with thickness stretch based on ELFORM = 3
SHRF	Shear correction factor which scales the transverse shear stress.
NLOC	Location of reference surface; see the definition of NLOC in *SECTION_IGA_SHELL for more details.
IRL	Lamina integration rule: EQ.0: Reduced Gauss-Legendre

VARIABLE	DESCRIPTION
	EQ.1: Gauss-Legendre EQ.2: Patchwise reduced Gauss-Legendre (for biquadratic NURBS only)

Contact Card. Additional card is required for the CONTACT option.

Card 4	1	2	3	4	5	6	7	8
Variable	FS	FD	DC	VC	OPTT	SFT	SSF	
Type	F	F	F	F	F	F	F	

> **NOTE**: If FS, FD, DC, and VC are specified they will not be used unless FS is set to a negative value (-1.0) in the *CONTACT section. These frictional coefficients apply only to contact types:
>
> SINGLE_SURFACE,
> AUTOMATIC_GENERAL,
> AUTOMATIC_SINGLE_SURFACE,
> AUTOMATIC_NODES_TO_...,
> AUTOMATIC_SURFACE_...,
> AUTOMATIC_ONE_WAY_...,
> ERODING_SINGLE_SURFACE
>
> Default values are input via *CONTROL_CONTACT input.

VARIABLE	DESCRIPTION
FS	Static coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact as $$\mu_c = \mathrm{FD} + (\mathrm{FS} - \mathrm{FD})e^{-\mathrm{DC}\times\lvert v_{\mathrm{rel}}\rvert}.$$
FD	Dynamic coefficient of friction. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact as $$\mu_c = \mathrm{FD} + (\mathrm{FS} - \mathrm{FD})e^{-\mathrm{DC}\times\lvert v_{\mathrm{rel}}\rvert}.$$

VARIABLE	DESCRIPTION
DC	Exponential decay coefficient. The functional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact as $$\mu_c = \mathrm{FD} + (\mathrm{FS} - \mathrm{FD})e^{-\mathrm{DC}\times\lvert v_{\mathrm{rel}}\rvert}.$$
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed $F_{\mathrm{lim}} = \mathrm{VC} \times A_{\mathrm{cont}}$. A_{cont} being the area of the segment contacted by the node in contact. The suggested value for VC is to use the yield stress in shear $\mathrm{VC} = \sigma_o/\sqrt{3}$ where σ_0 is the yield stress of the contacted material.
OPTT	Optional contact thickness. This applies to shells only.
SFT	Optional thickness scale factor for PART ID in automatic contact (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
SSF	Scale factor on default tracked surface penalty stiffness for this part ID whenever it appears in the contact definition. If zero, SSF is taken as unity.

Layer Data Cards without Long Option. The material ID, thickness, and material angle for each through-thickness layer of a composite shell are provided below (up to two layers per card). The layer data should be given sequentially starting with the bottommost layer. The total number of layers is determined by the number of entries on these cards. Include as many cards as necessary. The next keyword ("*") card terminates this input.

Card 5a	1	2	3	4	5	6	7	8
Variable	MID1	THICK1	B1	TMID1	MID2	THICK2	B2	TMID2
Type	I	F	F	I	I	F	F	I

Layer Data Cards for Long Option. The material ID, thickness, and material angle for each through-thickness layer of a composite shell are provided below (one layer per card). The layer data should be given sequentially starting with the bottommost layer. The total number of layers is determined by the number of entries on these cards. Include as many cards as necessary. The next keyword ("*") card terminates this input.

Card 5b	1	2	3	4	5	6	7	8
Variable	MID1	THICK1	B1	TMID1	PLYID1	SHRFAC1		
Type	I	F	F	I	I	F		

VARIABLE	DESCRIPTION
MID*i*	Material ID of layer *i*, see *MAT_… Section.
THICK*i*	Thickness of layer *i*.
B*i*	Material angle of layer *i*. This material angle applies only to material types 21, 22, 23, 33, 33_96, 34, 36, 40, 41-50, 54, 55, 58, 59, 103, 103_P, 104, 108, 116, 122, 133, 135, 135_PLC, 136, 157, 158, 190, 219, 226, 233, 234, 235, 242, 243, 261, 262, 278, and 293.
TMID*i*	Thermal material ID of layer *i*
PLYID*i*	Ply ID of layer *i* (for post-processing purposes)
SHRFAC*i*	Transverse shear stress scale factor

Remarks:

1. **Orthotropic Materials.** In cases where there is more than one orthotropic material model referenced by *PART_COMPOSITE, the orthotropic material orientation parameters (AOPT, BETA, and associated vectors) from the material model of the first orthotropic integration point apply to all the orthotropic integration points. AOPT, BETA, etc. input for materials of subsequent integration points are ignored. B*i*, not to be confused with BETA, is applied into account for each integration point.

2. **SHRF Field and Zero Traction Condition.** Thick shell formulations 1, 2, and 3, and all shell formulations, except for BCIZ and DK elements, are based on first order shear deformation theory that yields constant transverse shear strains which violates the condition of zero traction on the top and bottom surfaces of the shell. For these elements, setting SHRF = 0.83333 will compensate for this

error and result in the correct transverse shear deformation, so long as all layers have the same transverse stiffness. SHRF is not used by thick shell forms 3, 5, or 7 except for materials 33, 36, 133, 135, and 243.

3. **Thick Shell 5 or 6 and Shear Stress.** When thick shell formulation 5 and 6 are used, LS-DYNA will use either a parabolic transverse shear stress distribution when TSHEAR = 0 or a constant shear stress distribution when TSHEAR = 1. The parabolic option is recommended when elements are used in a single layer to model a plate or beam. The constant option may be better when elements are stacked so there are two or more elements through the thickness.

4. **Laminated Shear Stress Theory to Minimize Discontinuities.** For composites that have a transverse shear stiffness that varies by layer, laminated shell theory, activated by LAMSHT on *CONTROL_SHELL, will correct the transverse shear stress to minimize stress discontinuities between layers and at the bottom and top surfaces by imposing a parabolic transverse shear stress. SHRF should be set to the default value of 1.0 when the shear stress distribution is parabolic. If thick shells are stacked so that there is more than one element through the thickness of a plate or beam model, setting TSHEAR = 1 will cause a constant shear stress distribution which may be more accurate than parabolic. The TSHEAR parameter is available for all thick shell forms when laminated shell theory is active. Alternatively, a scale factor can be defined for transverse shear stress in each layer of shell or thick shell composites using the SHRFAC parameter. The inputted SHRFAC values are normalized so that overall shear stiffness is unaffected by the distribution. Therefore, only the ratio of parameter values is significant.

5. **Assignment of Zero Thickness to Layers.** The ability to assign zero thickness layers in the stacking sequence allows the number of layers to remain constant even as the number of physical plies varies from part to part and eases post-processing since a particular layer corresponds to a physical ply. Such a capability is important when one or more of the physical plies are not continuous across a composite structure. To represent a missing ply in *PART_COMPOSITE, set THICK*i* to 0.0 for the corresponding layer and additionally, set either MID = -1 or, if the LONG option is used, set PLYID to any nonzero value.

   When the number of physical plies varies from element to element in a part, one can assign zero thickness to integration points in exactly the same manner as described above but on an element-by-element basis using *ELEMENT_SHELL_COMPOSITE(_LONG) or *ELEMENT_TSHELL_COMPOSITE.

   When post-processing the results using LS-PrePost version 4.5, read both the keyword deck and d3plot database into the code and then select *Option →N/A gray fringe*. Then, when viewing fringe plots for a particular integration point

(*FriComp* $\rightarrow$ *IPt* $\rightarrow$ *intpt#*), the element will be grayed out if the selected integration point is missing (or has zero thickness) in that element.

6. **Through Thickness Integration Rule per Layer.** If not otherwise stated, each layer will be numerically integrated with a one-point trapezoidal rule which should be sufficient for most composite layups and which will be the recommended setting. However, by adding an irregular optional card 1a, the user may define other through thickness integration rules that should be applied for the numerical integration of an individual ply. So far, a 3-point Simpson integration per layer is supported, where the integration points are located at the bottom, the middle and the top of each individual layer. If IRPL is set to an unknown integration rule, the standard default one-point integration will be used.

*PART_DUPLICATE_{OPTION}

The available *OPTION* is:

<BLANK>

NULL_OVERLAY

NULL_OVERLAY is used to generate null shells for contact. See Remark 5.

Purpose: To provide a method of duplicating parts or part sets without the need to use the *INCLUDE_TRANSFORM option.

Duplication Cards. This format is used when the keyword option is left <BLANK>. Include as many of these cards as desired. This input ends at the next keyword ("*") card.

Card 1a	1	2	3	4	5	6	7	8
Variable	PTYPE	TYPEID	IDPOFF	IDEOFF	IDNOFF	TRANID	BOXID	ZMIN
Type	A	I	I	I	I	I	I	F
Default	none	none	0	0	0	0	0	0.0

Null Duplication Cards. This format is used when the keyword option is set to NULL_-OVERLAY. Include as many of these cards as desired. This input ends at the next keyword ("*") card.

Card 1b	1	2	3	4	5	6	7	8
Variable	PTYPE	TYPEID	IDPOFF	IDEOFF	DENSITY	E	PR	
Type	A	I	I	I	F	F	F	
Default	none	none	0	0	0	0	0	

VARIABLE	DESCRIPTION
PTYPE	Type of entity to duplicate: EQ.PART: Duplicate a single part EQ.PSET: Duplicate a part set

VARIABLE	DESCRIPTION
TYPEID	ID of part or part set to be duplicated
IDPOFF	ID offset of newly created parts
IDEOFF	ID offset of newly created elements
IDNOFF	ID offset of newly created nodes
TRANID	ID of *DEFINE_TRANSFORMATION to transform the existing nodes in a part or part set; see Remark 4.
BOXID	Optional *DEFINE_BOX ID used to define the boundary of the transformed configuration; see Remark 6.
ZMIN	Minimum value of the Z-coordinate of the transformed part or part set if set to a value other than zero. This field applies to all transformations except for SCALE.
DENSITY	Density
E	Young's modulus
PR	Poisson's ratio

Remarks:

1. **Parts with Common Nodes.** All parts sharing common nodes must be grouped in a *PART_SET and duplicated in a single *PART_DUPLICATE command so that the newly duplicated parts still share common nodes.

2. **Allowed Elements.** The following elements which need a *PART to complete their definition can be duplicated by using this command: *ELEMENT_SOLID, *ELEMENT_DISCRETE, *ELEMENT_SHELL, *ELEMENT_TSHELL, *ELEMET_BEAM, and *ELEMENT_SEATBELT.

3. **Constraints.** This command only duplicates definition of nodes, elements and parts, not the associated constraints. For example, TC and RC defined in *NODE will not be passed to the newly created nodes.

4. **Transformation.** When IDNOFF = IDPOFF = IDEOFF = 0, the existing part, or part set, will be transformed as indicated by TRANID; no new node or elements will be created.

5. **Null Elements.** The NULL_OVERLAY option may be used to generate 3 and 4-node null shell elements from the 6- and 8-node quadratic elements for use in contact. No additional nodes are generated.

6. **Boundary Box.** The XMIN, XMAX, YMIN, YMAX, ZMIN, and ZMAX of *DEFINE_BOX define the boundary of the transformed configuration. If any node falls out of the box boundary after transformation, TRANID, is applied, an additional rigid body translation will be applied automatically to the whole part(s) to assure no box boundary violation.

Example:

The following partial keyword example will rotate a part set 45° about the global Y-axis, passing through the global origin. The minimum coordinate of any nodes in the part set after the transformation will be 431.0 mm.

```
*PART_DUPLICATE
$    PTYPE    TYPEID    IDPOFF    IDEOFF    IDNOFF    TRANID     BOXID      ZMIN
      PSET       300                                     989                431.
*DEFINE_TRANSFORMATION
$ TRANID
989
$OPTION, A1, A2, A3, A4, A5, A6, A7
ROTATE,0.0,1.0,0.0,,,,45.0
```

Revision Information:

The variable ZMIN is available starting in Dev 138833.

*PART_MODES

Purpose: Treat a part defined with *MAT_RIGID as a linearized flexible body (LFB) whereby deformations are calculated from mode shapes. Unlike superelements (*ELEMENT_DIRECT_MATRIX_INPUT), linearized flexible bodies are accurate for systems undergoing large displacements and large rotations.

The modes shapes obtained experimentally or in a finite element analysis, e.g., NASTRAN .pch file or an LS-DYNA d3eigv or d3mode file, enable modeling the deformations. These files may contain a combination of normal modes, constraint modes, and attachment modes. For stress recovery in linearized flexible bodies, we recommend using linear element formulations. The implementation assumes a lump mass matrix. See also *CONTROL_RIGID.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	NMFB	FORM	ANSID	FORMAT	KMFLAG	NUPDF	SIGREC
Type	I	I	I	I	I	I	I	

Card 2	1	2	3	4	5	6	7	8
Variable	FILENAME							
Type	C							
Default	none							

Kept Mode Cards. Additional card if KMFLAG = 1. Use as many cards as necessary to specify the NMFB kept modes. After defining NMFB modes, no further input is expected.

Card 3	1	2	3	4	5	6	7	8
Variable	MODE1	MODE2	MODE3	MODE4	MODE5	MODE6	MODE7	MODE8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

Optional Modal Damping Cards. This input ends at the next keyword ("*") card.

Card 4	1	2	3	4	5	6	7	8
Variable	MSTART	MSTOP	DAMPF					
Type	I	I	F					
Default	none	none	none					

VARIABLE	DESCRIPTION
PID	Part identification. This part must be a rigid body. See Remark 4.
NMFB	Number of kept modes in linearized flexible body. The number of modes in the file, FILENAME, must equal or exceed NMFB. If KMFLAG = 0, the first NMFB modes in the file are used.
FORM	Linearized flexible body formulation. See Remark 6 below. EQ.0: Exact EQ.1: Fast EQ.2: General formulation (default) EQ.3: General formulation without rigid body mode orthogonalization
ANSID	Attachment node set ID (optional).
FORMAT	Input format of modal information:

VARIABLE	DESCRIPTION
	EQ.0: NASTRAN .pch file.
	EQ.1: (not supported)
	EQ.2: NASTRAN .pch file (LS-DYNA binary version). The binary version of this file is automatically created if a NASTRAN .pch file is read. The name of the binary file is the name of the NASTRAN .pch file but with .bin appended. The binary file is smaller and can be read much faster.
	EQ.3: LS-DYNA d3eigv binary eigenvalue database (see *CONTROL_IMPLICIT_EIGENVALUE).
	EQ.4: LS-DYNA d3mode binary constraint/attachment mode database (see *CONTROL_IMPLICIT_MODE).
KMFLAG	Kept mode flag. It selects the method for identifying modes to keep. This flag is not supported for FORMAT = 4 (d3mode).
	EQ.0: The first NMFB modes in the file, FILENAME, are used.
	EQ.1: Define NMFB kept modes with additional input.
NUPDF	Nodal update flag. If active, an attachment node set, ANSID, must be defined.
	EQ.0: All nodes of the rigid part are updated each cycle.
	EQ.1: Only attachment nodes are fully updated. All nodes in the body are output based on the rigid body motion without the addition of the modal displacements. For maximum benefit an attachment node set can also be defined with the PART_ATTACHMENT_NODES option. The same attachment node set ID should be used here.
SIGREC	Stress recovery flag:
	EQ.0: Do not recover stress.
	EQ.1: Recover stress.
	EQ.2: Recover stress and then set the recovery stress as initial stress when switching to a deformable body using *DEFORMABLE_TO_RIGID_AUTOMATIC. (shell formulations 16, 18, 20, and 21 and solid formulation 2).
	EQ.3: Recover stress based on shell formulation 21, and then set the recovery stress as initial stress for shell formulation 16 when switching to a deformable body using *DEFORMABLE_TO_RIGID_AUTOMATIC (shell formulation 16

VARIABLE	DESCRIPTION
	only).
FILENAME	The path and name of a file which contains the modes for this rigid body.
MODE*n*	Keep normal mode, MODE*n*.
MSTART	First mode for damping, ($1 \leq$ MSTART $\leq$ NMFB).
MSTOP	Last mode for damping, MSTOP, ($1 \leq$ MSTOP $\leq$ NMFB). All modes between MSTART and MSTOP inclusive are subject to the same modal damping coefficient, DAMPF.
DAMPF	Modal damping coefficient, ζ. See Remark 5.

Remarks:

1. **Shared nodes and interconnections.** Currently, linearized flexible bodies cannot share nodes with other linearized flexible bodies or rigid bodies; however, interconnections to other linearized flexible bodies or to rigid bodies can use the penalty joint option. The linearized flexible bodies are not implemented with the Lagrange multiplier joint option (see LMF in *CONTROL_RIGID).

2. **Normal modes file format.** The format of the file which contains the normal modes follows the file formats of NASTRAN output for modal information.

3. **Mode set.** The mode set typically combines both normal modes and attachment modes. The eigenvalues for the attachment modes are computed from the stiffness and mass matrices.

4. **PID.** The part ID specified must be either a single rigid body or a lead rigid body (see *CONSTRAINED_RIGID_BODIES) which can be made up of many rigid parts.

5. **Modal damping.** The modal damping is defined by the modal damping coefficient ζ, where a value of 1.0 equals critical damping. For a one degree of freedom model system, the relationship between the damping and the damping coefficient is $c = 2\zeta\omega_n m$, where c is the damping, m is the mass, and ω_n is the natural frequency, $\sqrt{k/m}$.

6. **Linearized flexible body formulations.** There are four linearized flexible body formulations. The first is a formulation that contains all the terms of the linearized flexible body equations, and its cost grows approximately as the square of the number of modes. The second formulation ignores most of the second-order

terms appearing in the exact equations, and its cost grows linearly with the number of modes. If the angular velocities are small and if the deflections are small with respect to the geometry of the system, the cost savings of the second formulation may make it more attractive than the first method.

Please note that the first two formulations are only applicable when the modes are eigenmodes computed for the free-free problem, that is, the 6 rigid body modes are included. The third formulation, the default, is a more general formulation that allows more general mode shapes. The fourth formulation does not orthogonalize the modes with respect to the rigid body modes and may allow boundary conditions to be imposed more simply in some cases than the third formulation. This last formulation is selected regardless of user input if attachment modes are used.

*PART_MOVE

Purpose: Translate a part by an incremental displacement in either a local or global coordinate system or by a distance along a defined vector. This option currently applies to parts defined by either shell or solid elements. All nodal points of the given part ID are moved. Care must be observed since parts that share boundary nodes with the part being moved must also be moved to avoid severe mesh distortions – the variable IFSET can be used to handle the situation.

Part/Part Set Move Cards. Include as many of following cards as desired. This input ends at the next keyword ("*") cards.

Card 1	1	2	3	4	5	6	7	8	9	10
Variable	PID	XMOV		YMOV		ZMOV		CID	IFSET	
Type	I	F		F		F		I	I	
Default	none	0.0		0.0		0.0		0	0	

VARIABLE	DESCRIPTION
PID	Part or part set identification number.
XMOV	Move shell/solid part ID, PID, in the x-direction by the incremental distance, XMOV. Ignored if CID < 0.
YMOV	Move shell/solid part ID, PID, in the y-direction by the incremental distance, YMOV. Ignored if CID < 0.
ZMOV	Move shell/solid part ID, PID, in the z-direction by the incremental distance, ZMOV. CID.LT.0: ZMOV is the distance the part is translated along CID.
CID	Coordinate system ID to define incremental displacement or vector ID to define the direction for the translation. GT.0: Local coordinate system ID. All displacements, XMOV, YMOV, and ZMOV, are with respect to CID. EQ.0: Global coordinate system. All displacements, XMOV, YMOV, and ZMOV, are with respect to the global coordinate system. LT.0: |CID| is a vector ID for a vector that points in the direction

VARIABLE	DESCRIPTION
	of the part translation. XMOV and YMOV are ignored in this case.
IFSET	Indicate if part set ID is used in PID definition. EQ.0: Part ID is used. EQ.1: Part set ID is used.

Remarks:

1. **Multiple Parts.** IFSET addresses the movement of multiple parts that share common boundary nodes, such as tailor-welded blanks. This field allows for all parts in a part set to move simultaneously.

2. **Draw Beads.** Draw beads can be modeled as beam elements that move the same distance and direction as either the die or punch, depending on the draw types.

3. **Supported Keywords.** Keywords *SET_PART_COLLECT and *SET_PART_-ADD are supported.

Example:

The example partial deck given below automatically positions all tools in a toggle draw of a decklid inner, with a tailor welded blank consisting of PID 1 and PID5, as shown in Figure 37-1. The keyword *CONTROL_FORMING_AUTOPOSITION_PARAMETER_-SET calculates where to position various part sets as follows:

a) the tailor-welded blank part set ID 1 is to be positioned in the global *Z*-direction on top of the lower die cavity (part set ID 4);

b) the binder (part set ID 3) is to be positioned on top of the blank;

c) the upper punch (part set ID 2) is to be positioned on top of the blank.

The three positioning distances for the blank, upper binder and upper punch are calculated and stored in variables &blnkmv, &upbinmv, and &uppunmv, respectively. The keyword *PART_MOVE, with IFSET set to 1, is responsible for moving the three part sets, using the three corresponding positioning variables.

```
*PARAMETER
R   blnkmv       0.0
R  upbinmv       0.0
R  uppunmv       0.0
*SET_PART_LIST
```

```
1
1,5
*SET_PART_LIST
2
2
*SET_PART_LIST
3
3
*SET_PART_LIST
4
4
*CONTROL_FORMING_AUTOPOSITION_PARAMETER_SET
$   PID/SID        CID        DIR  MPID/MSID   Position    PREMOVE     THICK
PARORDER
         1                    3          4          1                    1.5
blnkmv
         3                    3          1          1                    1.5
upbinmv
         2                    3          1          1                    1.5
uppunmv
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+---
-8
*PART_MOVE
$     PID              XMOV              YMOV              ZMOV      CID    IFSET
        1               0.0               0.0           &blnkmv               1
        3               0.0               0.0          &upbinmv               1
        2               0.0               0.0          &uppunmv               1
```

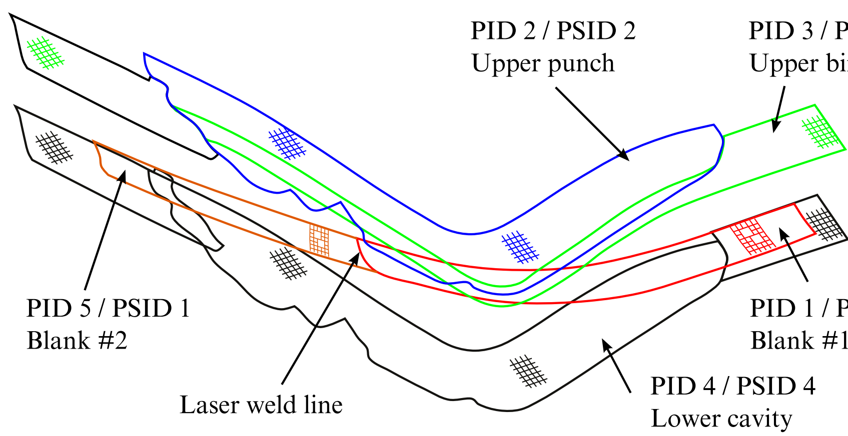

Figure 37-1. A tailor welded blank is positioned in a decklid (toggle draw).

*PART_SENSOR

Purpose: Activate and deactivate parts, based on sensor defined in *ELEMENT_SEAT-BELT_SENSOR. This option applies to discrete beam element only.

Sensor Part Coupling Cards. Include as many of the following cards as desired. This input ends at the next keyword ("*") card.

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

VARIABLE	DESCRIPTION
PID	Part ID which is controlled by sensor
SIDA	Sensor ID to activate or deactivate part
ACTIVE	Part activation flag: EQ.0: part is active from time zero until a signal is received by the part to deactivate. EQ.1: part is inactive from time zero and becomes active when a signal is received by the part to activate. The history variables for inactive parts are initialized at time zero.

*PART_STACKED_ELEMENTS

Purpose: This keyword provides a method of defining a stacked element model for shell-like structures. These types of plane load-bearing components possess a thickness which is small compared to their other (in-plane) dimensions. Their physical properties vary in the thickness direction according to distinct layers. Application examples include sandwich plate systems, composite laminates, plywood, and laminated glass.

With this keyword it is possible to discretize layered structures by an arbitrary sequence of shell and/or solid elements over the thickness. Whether a physical ply should be discretized by shell or solid elements depends on the individual thickness and other mechanical properties. Every single layer can consist of one shell element over the thickness or one or several solid elements over thickness.

The stacked element mesh can either be provided directly or it be automatically generated by LS-DYNA itself. For automatic generation extrusion methods are used to determine the new node locations that characterize the out-of-plane geometry. Each layer gets its own predefined properties such as individual thickness, material characteristic, and element type.

A more detailed description of this feature including a detailed description of the appropriate mesh generation procedure is given in Erhart [2015].

Card 1	1	2	3	4	5	6	7	8
Variable	HEADING							
Type	C							
Default	none							

Card 2	1	2	3	4	5	6	7	8
Variable	PIDREF	NUMLAY	ADPOPT	INPLCMP				
Type	I	I	I	I				
Default	none	none	0	0				

Layer Data Cards. The part ID, section ID, material ID, hourglass ID, thermal material ID, thickness, and number of through thickness solid elements for each layer *i* of a stacked element model are provided below. The layer data should be given sequentially starting with the bottommost layer. This card should be included NUMLAY times (one for each layer). The definitions in Card 3 replace the usual *PART cards.

Card 3	1	2	3	4	5	6	7	8
Variable	PID*i*	SID*i*	MID*i*	HGID*i*	TMID*i*	THK*i*	NSLD*i*	
Type	I	I	I	I	I	F	I	
Default	none	none	none	0	0	none	none	

VARIABLE	DESCRIPTION
HEADING	Heading for the part composition
PIDREF	Part ID of reference shell element mesh
NUMLAY	Number of layers
ADPOPT	Indicate if parts are adapted or not. (See also *CONTROL_ADAPTIVE): EQ.0: Inactive EQ.1: *h*-adaptive refinement
INPLCMP	Option for in-plane composed parts or to invoke a more robust search algorithm: EQ.0: Off EQ.1: On; see Remarks 6 and 7.
PID*i*	Part identification
SID*i*	Section identification for layer *i* defined in a *SECTION keyword
MID*i*	Material identification for layer *i* defined in a *MAT keyword
HGID*i*	Hourglass identification for layer *i* defined in a *HOURGLASS keyword

VARIABLE	DESCRIPTION
TMID*i*	Thermal material identification for layer *i* defined in a *MAT_-THERMAL keyword
THK*i*	Thickness of layer *i*
NSLD*i*	Number of through-thickness solid elements for layer *i*

Remarks:

1. **Provided versus automatically generated meshes.** In general, there are two different options for this keyword:

   a) The user provides a finished mesh comprising stacked shell and/or solid elements and then combines the corresponding part IDs using this keyword. This mode does not require a reference mesh in the PIDREF field nor does it require that either the layer thickness (THK*i* fields) and the the number of through-thickness solids elements (NSLD*i* fields) be specfied.

   b) The user may provide a shell reference mesh (PIDREF) together with the layup sequence. The stacked element mesh is automatically generated during the initialization phase of LS-DYNA. In that second case, layer thickness (THK*i*) and number of through-thickness solids (NSLD*i*) must be defined.

2. **Shell solid overlap.** In the mesh generation case, two consecutive layers (solid-solid or solid-shell) are firmly connected, meaning that they share nodes in the most obvious way possible (except they are both shell element layers, see Remark 3 for that case). This condition leads to the necessity that shell and solid elements partly overlap if they follow each other in the stacking sequence. This deficiency can be corrected afterwards by subsequent relocation of the shell mid-surfaces via NLOC on *SECTION_SHELL (only works for first or last layer in this stacked element approach) or appropriate adjustment of the material stiffness for the solid elements.

3. **Stacked shells.** Starting with the release of LS-DYNA version R10, it is possible to define shell element layers directly on top of each other (i.e. without solid elements in between). A potential connection/interaction of such layers has to be declared separately by additional contact definitions (standard, tied, or tie-break) otherwise they are free to penetrate each other.

4. **Chained calculations.** This keyword (*PART_STACKED_ELEMENTS) can also be used for modeling multi-stage processes. The *INTERFACE_SPRING-

BACK_LSDYNA card can be used (where PSID should contain all the parts PID*i*, not PIDREF) to save a final state including deformed geometry, stresses, and strains to a dynain file. In a subsequent calculation the *INCLUDE keyword can be used with that dynain to apply the layup sequence without regenerating the mesh. LS-DYNA automatically detects if a reference shell element mesh is present or not.

5. **Example.** The following input gives an example of specifying a three-layer shell-solid-shell structure:

```
*PART_STACKED_ELEMENTS
$ title
sandwich
$   pidref    numlay
        11         3
$#     pid       sid       mid      hgid      tmid       thk      nsld
       100       200         1         1         0      0.25         0
       101       201         2         0         0      0.60         3
       102       200         1         1         0      0.15         0
*SECTION_SHELL
$      sid    elform      shrf       nip     propt   qr/irid
       200         2     0.833       5.0       1.0       0.0
$       t1        t2        t3        t4
      0.25      0.25      0.25      0.25
*SECTION_SOLID
$      sid    elform
       201        -1
```

Shell elements (SID = 200) on the outer layers with part identifiers 100 and 102 discretize the sandwich structure. The interior of the "sandwich" consists of three solid elements (SID = 201, NSLD = 3) with part identifier 101. In this case, the reference shell mesh belongs to part 11 (PIDREF). The value of the field THK defines the thickness of each layer. It overwrites the thickness values from *SECTION_SHELL. Related materials (MID, TMID) and hourglass types (HGID) receive the usual treatment, so we do not show them here. .

6. **In-Plane composed parts.** In the case of a reference mesh made up of several parts with matching nodes at their boundaries (such as tailor welded blanks), define one *PART_STACKED_ELEMENTS keyword for each reference part. Each of these *PART_STACKED_ELEMENTS keywords must have INPLCMP set to 1. This feature requires merging the coincident nodes of the reference parts in a pre-processor before running the simulation. Additionally, the input deck must include *NODE_MERGE_SET for these newly generated parts so that they stay together after LS-DYNA generates the new mesh.

7. **Robust search algorithm.** The option INPLCMP = 1 also invokes a more robust search algorithm that can be used if the regular method fails, even if parts are not in-plane composed. For large models, this algorithm leads to some additional computational effort, meaning more CPU time.

8. **Boundary conditions.** All node sets (*SET_NODE) containing nodes of the reference shell element mesh are automatically converted to node sets that include the corresponding new nodes of the generated mesh. Therefore, it is possible to transfer boundary conditions of the reference mesh to the stacked model when corresponding keywords *BOUNDARY_... or *LOAD_... refer to such node sets.