*CONSTRAINED

The keyword *CONSTRAINED provides a way of constraining degrees of freedom to move together in some way. The keyword cards in this section are defined in alphabetical order:

*CONSTRAINED_ADAPTIVITY

*CONSTRAINED_BEAM_IN_SOLID

*CONSTRAINED_BUTT_WELD

*CONSTRAINED_COORDINATE

*CONSTRAINED_EULER_IN_EULER

CONSTRAINED_EXTRA_NODES_*OPTION

CONSTRAINED_GENERALIZED_WELD_*WELDTYPE

*CONSTRAINED_GLOBAL

*CONSTRAINED_IMMERSED_IN_SPG

*CONSTRAINED_INTERPOLATION

*CONSTRAINED_INTERPOLATION_SPOTWELD

CONSTRAINED_JOINT_*TYPE

CONSTRAINED_JOINT_COOR_*TYPE

CONSTRAINED_JOINT_STIFFNESS_*OPTION

*CONSTRAINED_JOINT_USER_FORCE

*CONSTRAINED_LAGRANGE_IN_SOLID

*CONSTRAINED_LINEAR_GLOBAL

*CONSTRAINED_LINEAR_LOCAL

*CONSTRAINED_LOCAL

*CONSTRAINED_MULTIPLE_GLOBAL

*CONSTRAINED_NODAL_RIGID_BODY

*CONSTRAINED_NODE_INTERPOLATION

*CONSTRAINED_NODE_SET

*CONSTRAINED_NODE_TO_NURBS_PATCH

*CONSTRAINED_POINTS

*CONSTRAINED_RIGID_BODIES

*CONSTRAINED_RIGID_BODY_INSERT

*CONSTRAINED_RIGID_BODY_STOPPERS

*CONSTRAINED_RIVET

*CONSTRAINED_SHELL_IN_SOLID

*CONSTRAINED_SHELL_TO_SOLID

*CONSTRAINED_SOIL_PILE

*CONSTRAINED_SOLID_IN_SOLID

*CONSTRAINED_SPLINE

*CONSTRAINED_SPOTWELD_*{OPTION}*_*{OPTION}*

*CONSTRAINED_SPR2

*CONSTRAINED_TIEBREAK

*CONSTRAINED_TIED_NODES_FAILURE

*CONSTRAINED_ADAPTIVITY

Purpose: Constrains a node created during *h*-adaptivity of three-dimensional shells to the midpoint along an edge of an element. This keyword is automatically created by LS-DYNA during an *h*-adaptive simulation involving three-dimensional shells. See the theory manual for details about this type of adaptivity.

Card 1	1	2	3	4	5	6	7	8
Variable	DNID	NID1	NID2					
Type	I	I	I					
Default	none	none	none					

VARIABLE	DESCRIPTION
DNID	Dependent node. This is the node constrained at the midpoint of an edge of an element.
NID1	Node at one end of an element edge
NID2	Node at the other end of that same element edge

Remarks:

Not all nodes created during *h*-adaptivity are constrained to an element edge. It only occurs when a node on the edge of an element created during an adaptive step is *not* on the boundary of the mesh and is *not* a node belonging to an element on the other side of the edge. If the element on the other side of the edge is adapted, then this constraint may be deleted.

Figure 10-1 illustrates which elements become dependent after adapting a single element mesh. The left element is adapted to create the elements on the right. In this figure, the red dots are dependent nodes while the blue dots are the edge nodes constraining the dependent nodes. For instance, node 2 is a dependent node constrained at the midpoint of the edge between nodes 1 and 3 and node 5 is a dependent node constrained at the midpoint of the edge between nodes 4 and 6.

Suppose that the element to the left of node 2 is adapted once. Then node 2 is no longer a dependent node and node 3 is no longer a constraining node as shown in Figure 10-2. These nodes are purple dots to illustrate that they are no longer part of a *CON-

STRAINED_ADAPTIVITY definition. However, node 1 continues to constrain the node to its right and now constrains the node to its left (node 7) with node 8.

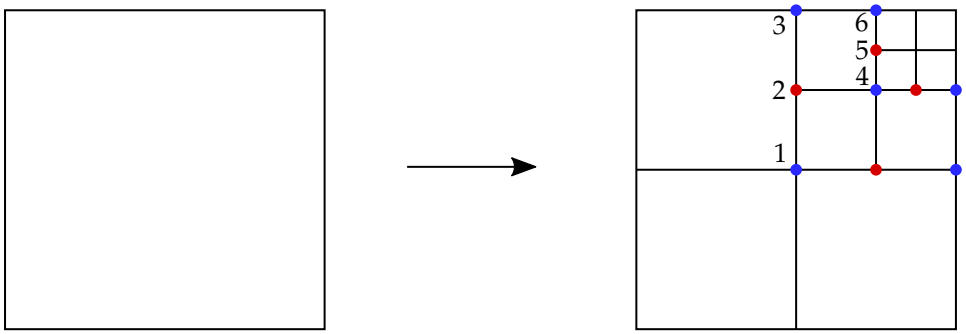


Figure 10-1. Illustration of which nodes are constrained after adaptive steps. The red nodes are dependent while the blue nodes constrain the dependent node that shares the edge.

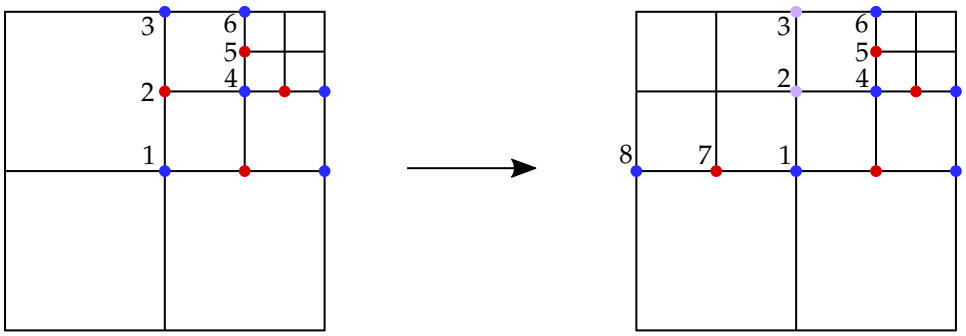


Figure 10-2. Illustration of which nodes are constrained after the element to the left of Node 2 is adapted. The purple nodes indicate which ones are no longer part of constraints.

*CONSTRAINED_BEAM_IN_SOLID_{OPTION1}_{OPTION2}

This keyword could take the following two forms:

*CONSTRAINED_BEAM_IN_SOLID

*CONSTRAINED_BEAM_IN_SOLID_PENALTY

To define an ID and heading, the following options are available:

ID

TITLE

Purpose: This keyword provides either constraint-based or penalty-based coupling between beams embedded in solid or thick shell elements. For shells embedded in solid or thick shell elements, see *CONSTRAINED_SHELL_IN_SOLID.

*CONSTRAINED_BEAM_IN_SOLID invokes constraint-based coupling. It constrains beam structures to move with Lagrangian solids/tshells. Both acceleration and velocity are constrained. This feature is intended to sidestep certain limitations in the CTYPE = 2 implementation of *CONSTRAINED_LAGRANGE_IN_SOLID. Notable features of this keyword include:

1. **CDIR = 1 Feature.** With the CDIR = 1 option, coupling occurs only in the normal directions. This option allows for releasing the constraints along the beam axial direction and provides a means to model the debonding process.

2. **Axial Coupling Force.** Debonding processes can be modeled if a shear force-slip relationship is given and CDIR is set to 1 to release the axial constraints. There are three ways to specify the bond force-slip relationship:

   a) A function defined with *DEFINE_FUNCTION. To activate this, set the AXFOR flag to a negative integer which refers to the *DEFINE_FUNCTION ID.

   b) A user-defined subroutine. You provide a subroutine giving the axial shear force based on the slip between rebar nodes and concrete solid elements. To enable this, you set AXFOR to a number greater than 1000, and the user must modify the subroutine `rebar_bondslip_get_force()` in the user-defined subroutine Fortran files to add in one or more debonding laws, each tagged with a "lawid." An AXFOR value greater than 1000 will call the user subroutine and pass AXFOR in as "lawid."

   c) A general built-in force-slip relationship. Starting with R14.0, AXFOR = 999 invokes a general bond force-slip relationship based on Bulletin 65 of the fib Model Code for Concrete Structures 2010.

3. **NCOUP Feature.** NCOUP is used to invoke coupling at both beam nodes and coupling points between the two beam nodes of each beam element. This implementation resolves the errors in energy balance observed when using the alternative implementation, *CONSTRAINED_LAGRANGE_IN_SOLID, CTYPE = 2.

4. **Tetrahedral and Pentahedral Solid Elements.** These element shapes are supported in this keyword and are no longer treated as degenerated hexahedra as in the *CONSTRAINED_LAGRANGE_IN_SOLID CTYPE = 2 implementation.

5. **Constraints on Nodes.** The *CONSTRAINED_LAGRANGE_IN_SOLID CTYPE = 2 implementation does not constrain beam nodes embedded in elements whose nodes have prescribed motion or other constraints.

6. **R-Adaptivity Support.** Severely deformed solid elements could terminate the simulation prematurely. One way to address this problem is to use LS-DYNA's 3D tetrahedral *r*-adaptivity method to remesh and restart. This keyword supports *r*-adaptivity and can be used to model manufacturing processes with severe deformations, such as compression molding with embedded fibers.

7. **Implicit Support.** Both SMP and MPP are supported.

8. **Optimized Sorting.** The sorting subroutine is optimized for larger problems to achieve better performance with less memory usage.

9. **Thermal Support.** In case the implicit thermal solver is invoked to perform either a thermal analysis (SOLN = 1 in *CONTROL_SOLUTION) or a coupled structural thermal analysis (SOLN = 2 in *CONTROL_SOLUTION), the temperature field is also constrained between beam and solid nodes.

*CONSTRAINED_BEAM_IN_SOLID_PENALTY invokes penalty-based coupling. A penalty spring is attached between coupling points on the beam and in the solid/tshell element. Penalty spring stiffness is calculated based on the geometric mean of the beam and the solid's bulk moduli. The magnitude of this coupling force can be controlled through field PSSF (penalty spring stiffness scale factor). This penalty coupling conserves kinetic energy much better in transient problems like blast loading. Please note that the CDIR and AXFOR fields only work with constraint-based coupling and are not available with penalty-based coupling.

Card Summary:

Card ID. This card is included if the TITLE or ID keyword option is used.

COUPID	TITLE

Card 1. This card is required.

BSID	SSID	BSTYP	SSTYP			NCOUP	CDIR

Card 2. This card is required.

START	END		AXFOR		PSSF		XINT

Card 3. This card is included if AXFOR = 999.

BONDC	BAREA	FCM	S1	S2	CLEAR	ALPHA	

Data Card Definitions:

Title Card. Additional card for TITLE and ID keyword options.

Card ID	1	2	3	4	5	6	7	8
Variable	COUPID	TITLE						
Type	I	A70						

VARIABLE	DESCRIPTION
COUPID	Coupling (card) ID number (I10). If not defined, LS-DYNA will assign an internal coupling ID based on the order of appearance in the input deck.
TITLE	A description of this coupling definition

Card 1	1	2	3	4	5	6	7	8
Variable	BSID	SSID	BSTYP	SSTYP			NCOUP	CDIR
Type	I	I	I	I			I	I
Default	none	none	0	0			0	0

VARIABLE	DESCRIPTION
BSID	Part or part set ID of the Lagrangian beam structure (see *PART, *SET_PART).
SSID	Part or part set ID of the Lagrangian solid elements or thick shell elements (see *PART or *SET_PART).
BSTYP	Set type of BSID: EQ.0: Part set ID (PSID) EQ.1: Part ID (PID)
SSTYP	Set type of SSID: EQ.0: Part set ID (PSID) EQ.1: Part ID (PID)
NCOUP	Number of coupling points generated in one beam element. If set to 0, coupling only happens at beam nodes. Otherwise, coupling is done at both the beam nodes and those automatically generated coupling points.
CDIR	Coupling direction. Only available in constraint form. EQ.0: Constraint applied along all directions. EQ.1: Constraint only applied along normal directions; along the beam axial direction, there is no constraint.

Card 2	1	2	3	4	5	6	7	8
Variable	START	END		AXFOR		PSSF		XINT
Type	F	F		I		F		F
Default	0	10^{20}		0		0.1		10^{16}

VARIABLE	DESCRIPTION
START	Start time to activate the coupling: LT.0.0: Start time is set to |START|. When negative, start time is followed during the dynamic relaxation phase of the

VARIABLE	DESCRIPTION
	calculation. After the completion of dynamic relaxation, coupling is activated regardless of the value of END. EQ.0.0: Start time is inactive, meaning coupling is always active GT.0.0: If END = -9999, START is interpreted as the curve or table ID defining multiple pairs of start-time and end-time. Otherwise, if END > 0, the start time applies during and after dynamic relaxation.
END	End time to deactivate the coupling: LT.0.0: If END = -9999, START is interpreted as the curve or table ID defining multiple pairs of start-time and end-time. Otherwise, negative END indicates that coupling is inactive during dynamic relaxation. After dynamic relaxation, the start and end times are followed and set to |START| and |END|, respectively. EQ.0.0: END defaults to 10^{20}. GT.0.0: END sets the time at which the coupling is deactivated.
AXFOR	Function that calculates the coupling force in the beam axial direction. Only available in constraint form with CDIR = 1. GE.0.and.LT.999: Off LT.0: |AXFOR| is a function ID; see *DEFINE_FUNCTION. See Example 1. EQ.999: General bond stress-slip relationship, which follows Bulletin 65 of fib Model Code for Concrete Structures 2010. GT.1000: Debonding law ID, "lawid," in the user-defined subroutine `rebar_bond-slip_get_force()`. See Example 2.
PSSF	Penalty spring stiffness scale factor. Only available in penalty form.
XINT	Interval distance. This field is designed to deal with beam elements with wide length variations. Coupling points are generated at an interval of length equal to XINT. Hence the number of coupling points in a beam element is no longer a fixed number (NCOUP) but instead variable, depending on the length of the beam element.

VARIABLE	DESCRIPTION
	This field can be used together with NCOUP. In that case, we will take the larger number of coupling points from these two options in each element.

General Bond Stress-Slip Relationship Card. Additional card if AXFOR = 999.

Card 3	1	2	3	4	5	6	7	8
Variable	BONDC	BAREA	FCM	S1	S2	CLEAR	ALPHA	
Type	I	F	F	F	F	F	F	
Default	none	none	none	none	none	none	none	

VARIABLE	DESCRIPTION
BONDC	Bond condition flag. BONDC consists of a two-digit integer, BONDC = [BA]: $$\text{BONDC} = B \times 10 + A$$ The 1's digit controls the transverse reinforcement: A.EQ.0: Unconfined A.EQ.1: Stirrups The 10's digit controls the bond condition: B.EQ.0: Good bond condition B.EQ.1: All other bond condition
BAREA	Beam area
FCM	Concrete compression strength
S1	Slip at maximum stress (pull-out)
S2	Slip at the end of stress plateau (pull-out)
CLEAR	Clearance distance between ribs
ALPHA	α, see Bulletin 65 of fib Model Code for Concrete Structures 2010

Examples:

1. **Function for Modeling Debonding.** The example below shows how to define and use a function to prescribe the debonding process. User-defined functions are supported. The function computes the debonding force and has two internally calculated arguments: slip and leng. Slip is the relative axial displacement between the beam node (or coupling point) and the material in which the beam is embedded. Leng is the tributary length of the beam node or coupling point. Implicit calculations require a third argument which is output by the function: stiff. Stiff is the debonding spring stiffness. The asterisk in front of stiff (*stiff) is required to indicate that it is called by reference, meaning that its value is returned after the function is evaluated. Please note that this asterisk cannot be placed in the first column of the function body because the LS-DYNA keyword reader assumes asterisks start new keywords.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*CONSTRAINED_BEAM_IN_SOLID
$#    bsid      ssid     bstyp     sstyp                         ncoup      cdir
         2         1         1         1                             2         1
$#   start       end               axfor
     0.000     0.000                 -10
*DEFINE_FUNCTION
        10
float force(float slip,float leng, float *stiff)
{
  float force,pi,d,area,shear,pf;
  pi=3.1415926;
  d=0.175;
  area=pi*d*leng;
  pf=1.0;
  if (slip<0.25) {
    shear=slip*pf;
  } else {
    shear=0.25*pf;
  }
  force=shear*area;
  *stiff=pf*area;
  return force;
}
```

2. **User Subroutine for Modeling Debonding.** The example below shows how to define a user subroutine and use it to prescribe the debonding process.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*CONSTRAINED_BEAM_IN_SOLID
$#    bsid      ssid     bstyp     sstyp                         ncoup      cdir
         2         1         1         1                             2         1
$#   start       end               axfor
     0.000     0.000                1001
*CONSTRAINED_BEAM_IN_SOLID
$#    bsid      ssid     bstyp     sstyp                         ncoup      cdir
         3         1         1         1                             2         1
$#   start       end               axfor
     0.000     0.000                1002
*USER_LOADING
$    parm1     parm2     parm3     parm4     parm5     parm6     parm7     parm8
       1.0       6.0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
```

The user debonding law subroutine:

```
      subroutine rebar_bondslip_get_force(slip,dl,force,hsv,
     .  userparm,lawid)
      real hsv
      dimension hsv(12),cm(8),userparm(*)
c
c in this subroutine user defines debonding properties and
c call his debonding subroutine to get force
      cm(1)=userparm(1)
      cm(2)=userparm(2)
      cm(3)=2.4*(cm(2)/5.0)**0.75
      cm(8)=0.
c
      pi=3.1415926
      d=0.175
      area=pi*0.25*d*d*dl
      pf=1.0
c
      if (lawid.eq.1001) then
        if (slip.lt.0.25) then
          shear=slip*pf
        else
          shear=0.25*pf
        endif
        force=sign(1.0,slip)*shear*area
      elseif (lawid.eq.1002) then
        if (slip.lt.0.125) then
          shear=slip*pf
        else
          shear=0.125*pf
        endif
      endif
      return
      end
```

*CONSTRAINED_BUTT_WELD

Purpose: Define a line of coincident nodes that represent a structural butt weld between two parts defined by shell elements. Failure is based on nodal plastic strain for ductile failure and stress resultants for brittle failure. This input is much simpler than the alternative approach for defining butt welds, *CONSTRAINED_GENERALIZED_WELD_BUTT. With this keyword, LS-DYNA automatically determines the local coordinate system, the effective length, and thickness for each pair of butt welded nodes. For *CONSTRAINED_GENERALIZED_WELD_BUTT, these quantities must be defined in the input.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID1	NSID2	EPPF	SIGF	BETA			
Type	I	I	F	F	F			
Default	none	none	0.	10^{16}	1.0			
Remarks		1, 2	3, 4	3	3			

VARIABLE	DESCRIPTION
NSID1	Node set ID for one side of the butt weld, see *SET_NODE_*OPTION*.
NSID2	Node set ID for the other side of the butt weld
EPPF	Plastic strain at failure
SIGF	σ_f, stress at failure for brittle failure
BETA	β, failure parameter for brittle failure

Remarks:

1. **Butt Weld Geometry and Node Sets.** Nodes in NSID1 and NSID2 must be given in the order they appear as one moves along the edge of the surface. An equal number of coincident nodes must be defined in each set. In a line weld the first and last node in a string of nodes can be repeated in the two sets. If the first and last pair of nodal points are identical, LS-DYNA assume the geometry

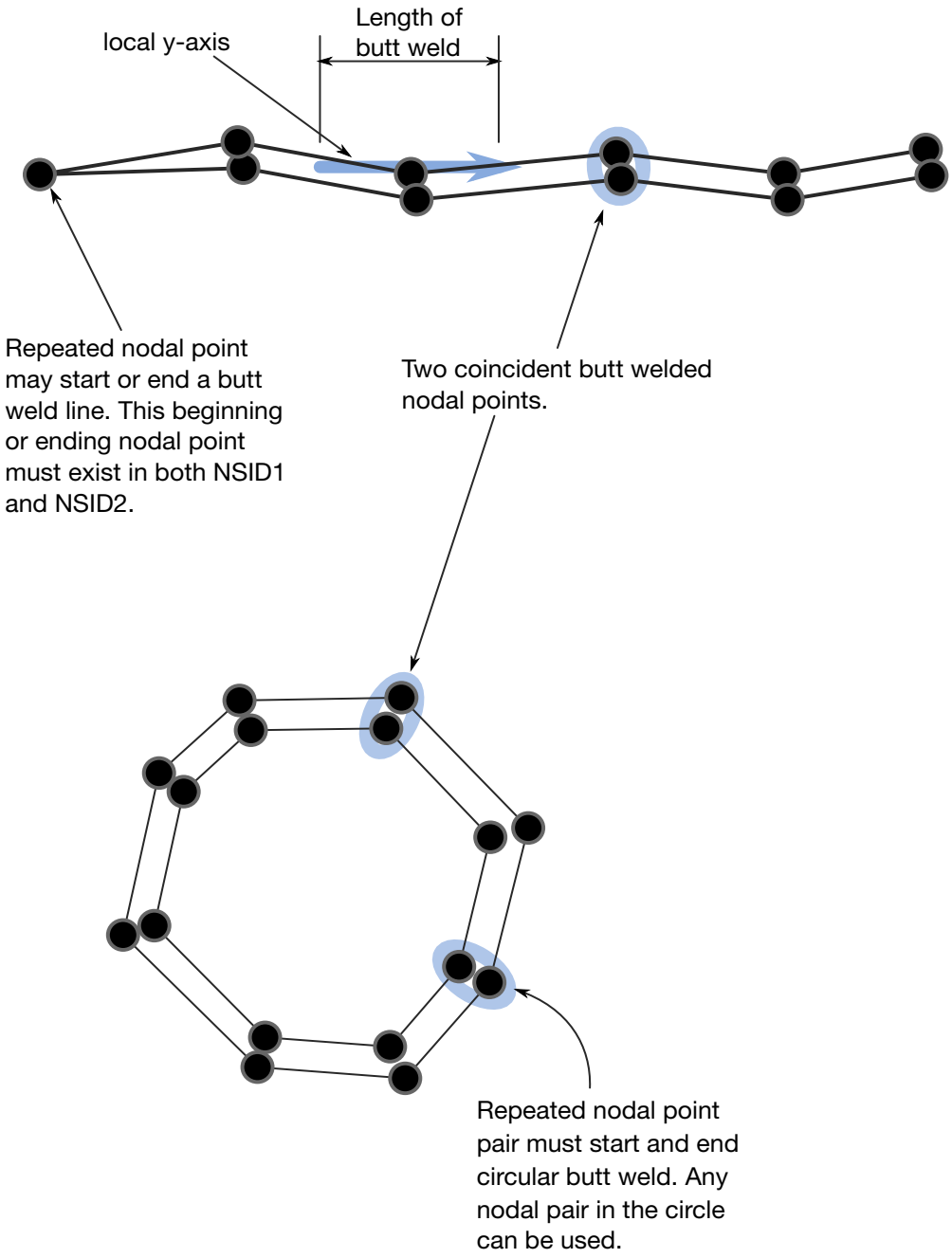


Figure 10-3. Definition of butt welds are shown above. The butt weld can be represented by a line of nodal points or by a closed loop

of the butt weld is circular or a closed loop. See Figure 10-3, where the line butt weld and closed loop weld are illustrated.

2. **Restrictions.** Butt welds may not cross. For complicated welds, this keyword can be combined with the input in *CONSTRAINED_GENERALIZED_WELD_-BUTT to handle the case where crossing occurs. Nodes in a butt weld must not be members of rigid bodies.

3. **Failure Model.** If the average volume-weighted effective plastic strain in the shell elements adjacent to a node exceeds the specified plastic strain at failure, the node is released. Brittle failure of the butt welds occurs when:

$$\beta\sqrt{\sigma_n^2 + 3(\tau_n^2 + \tau_t^2)} \geq \sigma_f$$

where,

σ_n = normal stress (local x)
τ_n = shear stress in direction of weld (local y)
τ_t = shear stress normal to weld (local z)
σ_f = failure stress
β = failure parameter

The component σ_n is nonzero for tensile values only. The nodes on each side of the butt weld must coincide.

The local coordinates at each set of coincident nodes are determined as follows. The normal vector (local x) is found by summing the unit normal vectors of all the shell elements on the *NSID2 side* sharing the butt welded node. The direction of the normal vector at the node is chosen so that the local x-direction points towards the elements on the NSID1 side in order to identify tensile versus compressive stresses. The local y-axis is taken as the vector in the direction of a line connecting the mid-points of the line segments lying on either side of the node in NSID2 (see Figure 10-3). The local z-axis at a node in NSID2 is normal to the plane of the butt weld at the node determined by looking at the *NSID2 side*.

The thickness and length of the butt weld are needed to compute the stress values. The thickness is based on the average thickness of the shell elements that share the butt welded nodal pair, and the chosen length of the butt weld is shown in Figure 10-3.

4. **Failure along a Line.** Butt welds may be used to simulate the effect of failure along a predetermined line, such as a seam or structural joint. When the failure criterion is reached at a nodal pair, the nodes begin to separate. As this effect propagates, the weld will appear to "unzip," thus simulating failure of the connection.

*CONSTRAINED_COORDINATE_{OPTION}

To define constraints based on position coordinates the following options are available:

<BLANK>

LOCAL

Purpose: Define constraints on position coordinates for springback simulations. With the frequent application of an adaptive mesh in stamping simulations, nodes needed for springback constraints are often unavailable until the last process simulation before springback is complete, making defining springback constraints at the beginning of the simulation process setup impossible. However, if the nodes are available, their positions may not be exactly at the desired locations required for springback constraints. With this keyword, it is possible to define constraints in both scenarios.

Card 1	1	2	3	4	5	6	7	8
Variable	ID	PID	IDIR	X1	Y1	Z1	CID	
Type	I	I	I	F	F	F	I	
Default	none	none	none	0.0	0.0	0.0	0	

VARIABLE	DESCRIPTION
ID	Identification number of the constraint
PID	Part or part set to be constrained: GT.0: Part ID LT.0: |PID| is a part set ID.
IDIR	Constraint flag. IDIR > 0 works for implicit (not explicit dynamic) simulations only with no active adaptive refinement (adaptive blank is acceptable); the magnitude of IDIR defines the degrees of freedom below to be constrained at (X1,Y1,Z1). IDIR < 0 works for both implicit and explicit simulations with active adaptive refinement; the magnitude of IDIR determines the degrees of freedom below to be constrained at the node nearest to (X1,Y1,Z1). |IDIR|.EQ.1: x translational degree of freedom |IDIR|.EQ.2: y translational degree of freedom

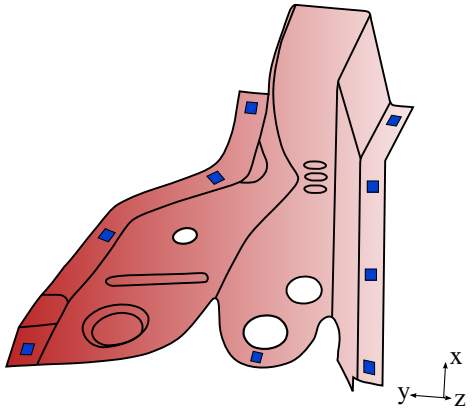


Figure 10-4. Constrained locations of a trim panel (NUMISHEET 2005 cross member).

VARIABLE	DESCRIPTION
	|IDIR|.EQ.3: z translational degree of freedom |IDIR|.EQ.11: x rotational degree of freedom |IDIR|.EQ.12: y rotational degree of freedom |IDIR|.EQ.13: z rotational degree of freedom |IDIR|.EQ.14: x and y rotational degrees of freedom |IDIR|.EQ.15: y and z rotational degrees of freedom |IDIR|.EQ.16: z and x rotational degrees of freedom |IDIR|.EQ.17: x, y and z rotational degrees of freedom
X1, Y1, Z1	X, Y, Z coordinates of the location being constrained
CID	Local coordinate system ID

Remarks:

1. **Constraint IDs.** The identification number of a constraint must be unique; the IDs must be unique even for two constraints with the same (X, Y, Z) position that constrains different degrees of freedom. When the LOCAL option is invoked, a local coordinate system ID (see *DEFINE_COORDINATE_*{OPTION}*) should be provided in the CID field.

2. **Moving parts.** The constraints defined by this keyword take effect after the deformable part is moved by *PART_MOVE. This is necessary, for example, in the case of a formed panel springback supported by a checking fixture, where the panel is automatically positioned by *CONTROL_FORMING_AUTOPOSITION_PARAMETER.

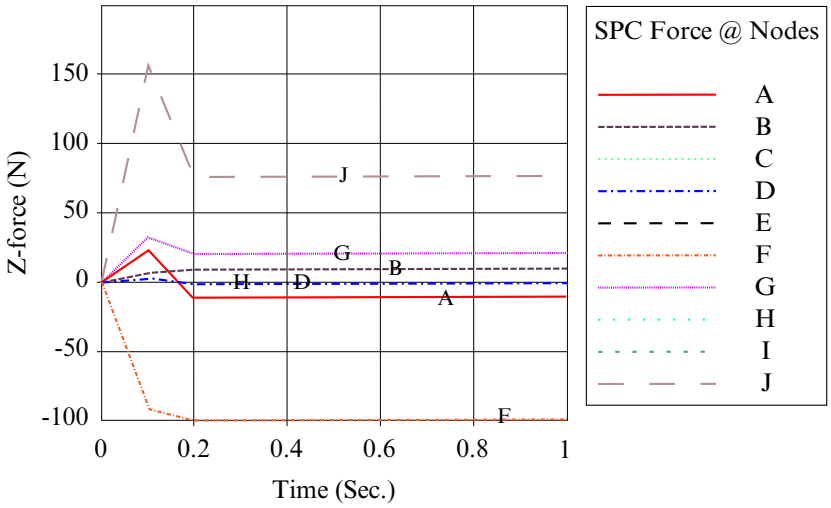


Figure 10-5. SPC Z-forces at 10 nodes

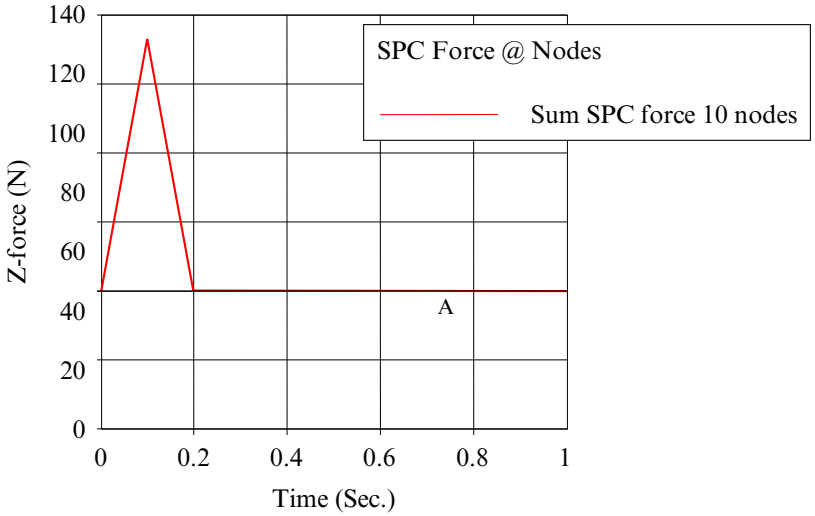


Figure 10-6. SPC Z-force summation of the 10 nodes

3. **SPC forces.** It is possible to output SPC forces on the coordinates constrained. For each position coordinate set, an extra node will be generated and SPC forces are calculated and output to the SPCFORC file. The frequency of the output is specified with the keyword *DATABASE_SPCFORC. For example, Figure 10-4 shows the Z-constrained locations on the trimmed panel (half with symmetric conditions at the smaller end) of the NUMISHEET 2005 cross member. SPC forces in the Z-direction of these 10 locations were recovered after a multi-step static implicit springback with this over-constrained boundary condition (see Figure 10-5). Figure 10-6 shows the summation of these Z-forces. The summation approaches zero as the residual stresses are balanced out by the springback shape absent of gravity.

Example:

A partial keyword input deck is shown below. A part with PID 18 is constrained in 6 locations in a local coordinate system with ID 9. Constrained DOFs are indicated by IDIR.

```
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*CONSTRAINED_COORDINATE
$       ID      IDPT      IDIR         x         y         z       CID
         1        18         2  -555.128      86.6   1072.29         9
         2        18         3  -555.128      86.6   1072.29         9
         3        18         3  -580.334    -62.15   1068.32         9
         4        18         1   568.881   81.2945   1033.72         9
         5        18         2   568.881   81.2945   1033.72         9
         6        18         3   568.881   81.2945   1033.74         9
*DEFINE_COORDINATE_SYSTEM
$      CID        X0        Y0        Z0        XL        YL        ZL
         9       0.0       0.0       0.0       0.0      10.0       0.0
$       XP        YP        ZP
      10.0      10.0       0.0
```

*CONSTRAINED_EULER_IN_EULER

Purpose: Define the coupling interaction between Eulerian materials in two overlapping, geometrically similar, multi-material Eulerian mesh sets. The command allows a frictionless "contact" between two or more different Eulerian materials.

Card 1	1	2	3	4	5	6	7	8
Variable	PSID1	PSID2	PFAC					
Type	I	I	F					
Default	0	0	0.1					

VARIABLE	DESCRIPTION
PSID1	Part set ID of the 1st ALE or Eulerian set of mesh(es)
PSID2	Part set ID of the 2nd ALE or Eulerian set of mesh(es)
PFAC	A penalty factor for the coupling interaction between the two PSIDs

Remarks:

1. **Model Description.** The 2 meshes must be of Eulerian formulation (the meshes are fixed in space, not moving). Consider 2 overlapping Eulerian meshes. Each Eulerian mesh contains 2 physical materials, say a vacuum and a metal. This keyword provides a frictionless "contact" or interaction between the 2 metals since each resides in a different Eulerian mesh system. Due to its restrictive nature, this option is currently only an experimental feature.

2. **Contact Pressure.** Contact pressure is built up in two overlapping Eulerian elements if their combined material fill fraction exceeds 1.0 (penalty formulation).

3. **Materials.** This feature needs to be combined with *MAT_VACUUM (element formulation 11).

Example:

Consider an ALE/Eulerian multi-material model (ELFORM = 11) consisting of:

PID 1 = *MAT_NULL (material 1)

PID 2 = *MAT_VACUUM ⇒ PID 1 is merged at its boundary to PID 2.

PID 3 = *MAT_NULL (material 3)

PID 4 = *MAT_VACUUM ⇒ PID 3 is merged at its boundary to PID 4.

The mesh set containing PID 1 & 2 intersects or overlaps with the mesh set containing PID 3 & 4. PID 1 is given an initial velocity in the positive *x*-direction. This will cause material 1 to contact material 3 (note that materials 2 & 4 are void). The interaction between materials 1 & 3 is possible by defining this coupling command. In this case material 1 can flow within the mesh region of PID 1 & 2 only, and material 3 can flow within the mesh region of PID 3 & 4 only.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
*ALE_MULTI-MATERIAL_GROUP
$       SID   SIDYTPE
          1         1
          2         1
          3         1
          4         1
*CONSTRAINED_EULER_IN_EULER
$     PSID1     PSID2     PENAL
         11        12       0.1
*SET_PART_LIST
         11
          1         2
*SET_PART_LIST
         12
          3         4
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
```

***CONSTRAINED_EXTRA_NODES**_*OPTION*

Available options include:

NODE

SET

Purpose: Define extra nodes for rigid body.

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

VARIABLE	DESCRIPTION
PID	Part ID of rigid body to which the nodes will be added, see *PART.
NID / NSID	Node (keyword option: NODE) or node set ID (keyword option: SET), see *SET_NODE, of added nodes.
IFLAG	This flag is meaningful if and only if the inertia properties of the Part ID are defined in PART_INERTIA. If set to unity, the center-of-gravity, the translational mass, and the inertia matrix of the PID will be updated to reflect the merged nodal masses of the node or node set. If IFLAG is defaulted to zero, the merged nodes will not affect the properties defined in PART_INERTIA since it is assumed the properties already account for merged nodes.

Remarks:

Extra nodes for rigid bodies may be placed anywhere, even outside the body, and they are assumed to be part of the rigid body. They have many uses including:

1. The definition of draw beads in metal forming applications by listing nodes along the draw bead.

2. Placing nodes where joints will be attached between rigid bodies.

3. Defining a node where point loads are to be applied or where springs may be attached.

4. Defining a lumped mass at a particular location.

The coordinates of the extra nodes are updated according to the rigid body motion.

Examples:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_EXTRA_NODES_NODE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Rigidly attach nodes 285 and 4576 to part 14. (Part 14 MUST be a rigid body.)
$
*CONSTRAINED_EXTRA_NODES_NODE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid       nid
        14       285
        14      4576
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$


$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_EXTRA_NODES_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Rigidly attach all nodes in set 4 to part 17. (Part 17 MUST be a rigid body.)
$
$  In this example, four nodes from a deformable body are attached
$  to rigid body 17 as a means of joining the two parts.
$
*CONSTRAINED_EXTRA_NODES_SET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      pid      nsid
        17         4
$
$
*SET_NODE_LIST
$      sid
         4
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       665       778       896       827
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

CONSTRAINED_GENERALIZED_WELD_*WELDTYPE_{OPTION}

CONSTRAINED_GENERALIZED_WELD_*WELDTYPE is a family of keywords all sharing a common set of data cards and option flags. The available weld variants are:

SPOT

FILLET

BUTT

CROSS_FILLET

COMBINED

To define an ID for the weld use the option:

ID

Purpose: Define spot, fillet, butt, and other types of welds. Coincident nodes are permitted if the local coordinate ID is defined. For the spot weld a local coordinate ID is not required if the nodes are offset. Failures can include both the plastic and brittle failures. These can be used either independently or together. Failure occurs when either criteria is met. The welds may undergo large rotations since the equations of rigid body mechanics are used to update their motion. Weld constraints between solid element nodes are not supported.

Card Summary:

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

WID							

Card 1. This card is required.

NSID	CID	FILTER	WINDOW	NPR	NPRT		

Card 2a. This card is included if and only if the weld type is SPOT.

TFAIL	EPSF	SN	SS	N	M		

Card 2b. This card is included if and only if the weld type is FILLET.

TFAIL	EPSF	SIGF	BETA	L	W	A	ALPHA

Card 2c. This card is included if and only if the weld type is BUTT.

TFAIL	EPSF	SIGY	BETA	L	D		

Card 2d.1. This card is included if and only if the weld type is CROSS_FILLET.

TFAIL	EPSF	SIGY	BETA	L	W	A	ALPHA

Card 2d.2. This card is included if and only if the weld type is CROSS_FILLET. Include NPR of this card.

NODEA	NODEB	NCID					

Card 2e.1. This card is included if and only if the weld type is COMBINED. Read in NPR pairs of Cards 2e.1 and 2e.2 for a total of 2 × NPR cards.

TFAIL	EPSF	SIGY	BETA	L	W	A	ALPHA

Card 2e.2. This card is included if and only if the weld type is COMBINED. Read in NPR pairs of Cards 2e.1 and 2e.2 for a total of 2 × NPR cards.

NODEA	NODEB	NCID	WTYP				

Data Card Definitions:

ID Card. Additional card for ID keyword option.

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

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	CID	FILTER	WINDOW	NPR	NPRT		
Type	I	I	I	I	I	I		
Default	none	0	0	0	↓	0		

VARIABLE	DESCRIPTION
WID	Optional weld ID
NSID	Nodal set ID, see *SET_NODE
CID	Coordinate system ID for output of spot weld data to SWFORC in local system; see *DEFINE_COORDINATE_*OPTION*. CID is not required for spot welds if the nodes are not coincident.
FILTER	Number of force vectors saved for filtering. This option can eliminate spurious failures due to numerical force spikes; however, memory requirements are significant since 6 force components are stored with each vector. LE.1: No filtering GE.2: Simple average of force components divided by FILTER or the maximum number of force vectors that are stored for the time window option below.
WINDOW	Time window for filtering. This option requires the specification of the maximum number of steps (defined by FILTER) which can occur within the filtering time window. A time weighted average is used. If the time step decreases too far, then the filtering time window will be ignored and the simple average is used. EQ.0: Time window is not used.
NPR	Number of individual nodal pairs in the cross fillet or combined general weld.
NPRT	Print option in file rbdout. EQ.0: Default from the control card, *CONTROL_OUTPUT, is used; see variable name IPRTF. EQ.1: Data is printed. EQ.2: Data is not printed.

Spot Weld Card. Additional Card required for the SPOT keyword option.

Card 2a	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SN	SS	N	M		
Type	F	F	F	F	F	F		
Default	10^{16}	10^{16}	10^{16}	10^{16}	2.0	2.0		

VARIABLE	DESCRIPTION
TFAIL	Failure time for constraint set, t_f
EPSF	Effective plastic strain at failure, ϵ^p_{fail}, defines ductile failure.
SN	S_n, normal force at failure; only for the brittle failure of spot welds
SS	S_s, shear force at failure; only for the brittle failure of spot welds
N	n, exponent for normal force; only for the brittle failure of spot welds
M	m, exponent for shear force; only for the brittle failure of spot welds

Remarks:

Spot weld failure due to plastic straining occurs when the effective nodal plastic strain exceeds the input value, $\varepsilon^p_{\text{fail}}$. This option can model the tearing out of a spot weld from the sheet metal since the plasticity is in the material that surrounds the spot weld, not the spot weld itself. A least squares algorithm is used to generate the nodal values of plastic strains at the nodes from the element integration point values. The plastic strain is integrated through the element and the average value is projected to the nodes using a least square fit. This option should only be used for the material models related to metallic plasticity and can result in slightly increased run times.

Brittle failure of the spot welds occurs when:

$$\left[\frac{\max(f_n, 0)}{S_n}\right]^n + \left[\frac{|f_s|}{S_s}\right]^m \geq 1 \ ,$$

where f_n and f_s are the normal and shear interface force. Component f_n contributes for tensile values only. When the failure time, t_f, is reached the nodal rigid body becomes inactive, and the constrained nodes may move freely. In Figure 10-7 the ordering of the

nodes is shown for the 2 node and 3 node spot welds. This order is with respect to the local coordinate system where the local z-axis determines the tensile direction. The nodes in the spot weld may coincide. The failure of the 3 node spot weld may occur gradually with first one node failing and later the second node may fail. For n noded spot welds the failure is progressive starting with the outer nodes (1 and n) and then moving inward to nodes 2 and $n-1$. Progressive failure is necessary to preclude failures that would create new rigid bodies.

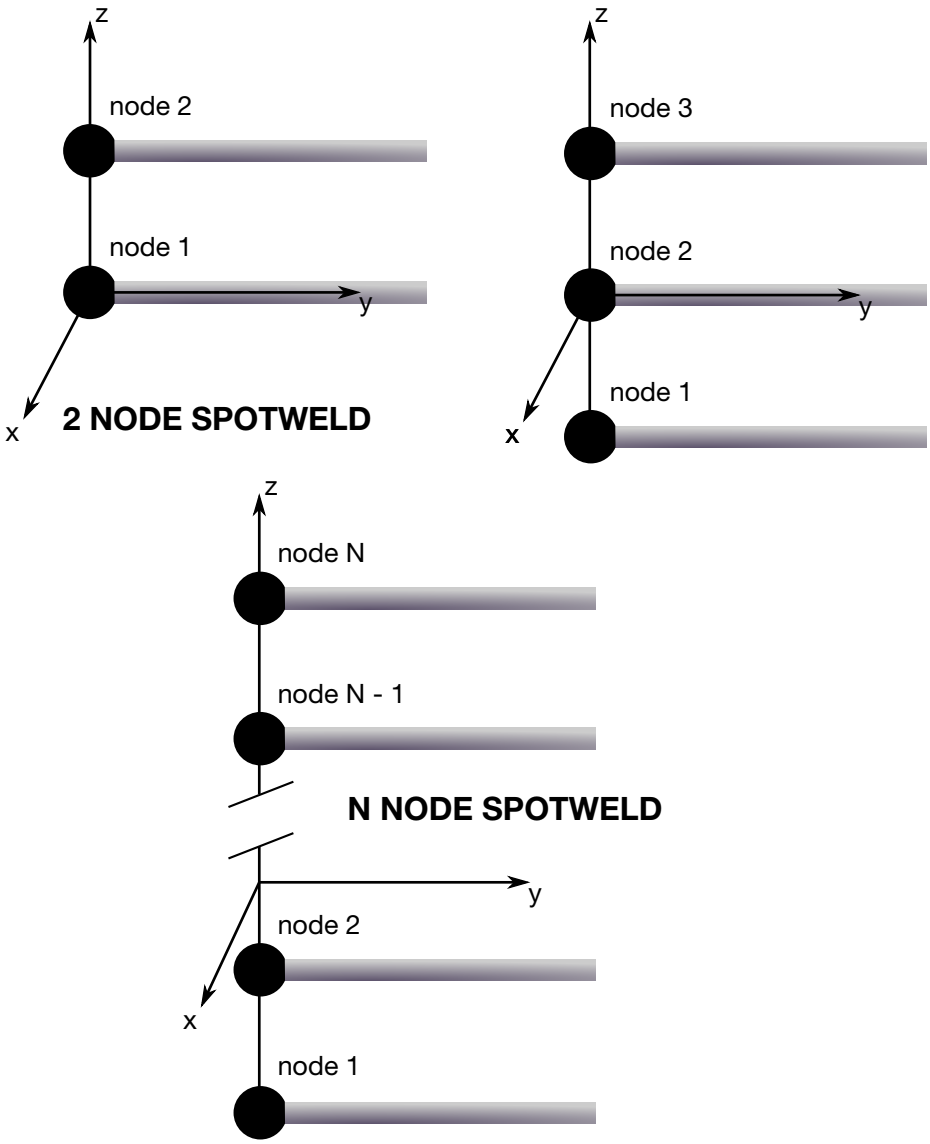


Figure 10-7. Nodal ordering and orientation of the local coordinate system is important for determining spotweld failure.

Fillet Weld Card. Additional Card required for the FILLET keyword option.

Card 2b	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGF	BETA	L	W	A	ALPHA
Type	F	F	F	F	F	F	F	F
Default	10^{16}	10^{16}	10^{16}	1.0	none	none	none	none

VARIABLE	DESCRIPTION
TFAIL	Failure time for constraint set, t_f
EPSF	Effective plastic strain at failure, ϵ^p_{fail}, defines ductile failure.
SIGF	σ_f, stress at failure for brittle failure
BETA	β, failure parameter for brittle failure
L	L, length of fillet weld (see Figure 10-8)
W	w, separation of parallel fillet welds (see Figure 10-8)
A	a, fillet weld throat dimension (see Figure 10-8)
ALPHA	α, weld angle (see Figure 10-8) in degrees

Remarks:

Ductile fillet weld failure, due to plastic straining, is treated identically to spot weld failure. Brittle failure occurs when the following weld stress condition is met on the narrowest fillet weld cross section (across the throat):

$$\beta\sqrt{\sigma_n^2 + 3(\tau_n^2 + \tau_t^2)} \geq \sigma_f \quad ,$$

where

σ_n = normal stress
τ_n = shear stress in local zx-plane
τ_t = shear stress in local-y direction
σ_f = failure stress
β = failure parameter

The component σ_n is nonzero for tensile values only. When the failure time, t_f , is reached the nodal rigid body becomes inactive and the constrained nodes may move freely.

Figure 10-8 shows the ordering of the nodes for the 2 node and 3 node fillet welds. This order is with respect to the local coordinate system where the local *z*-axis determines the tensile direction. The initial orientation of the local coordinate system is defined by CID. If CID = 0, then the global coordinate system is used. The local coordinate system is updated according to the rotation of the rigid body representing the weld. The failure of the 3 node fillet weld may occur gradually with first one node failing and later the second node may fail.

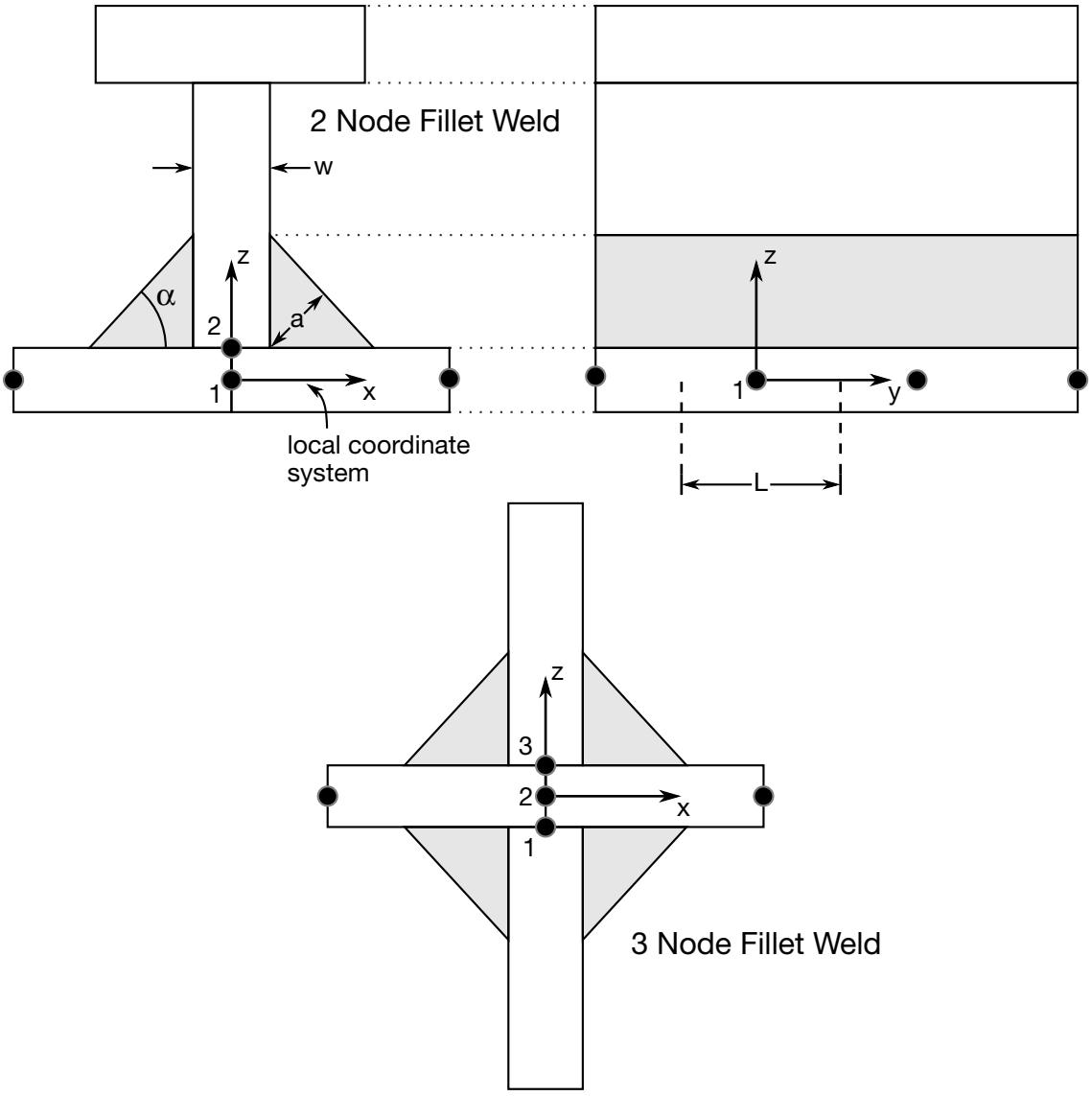


Figure 10-8. Nodal ordering and orientation of the local coordinate system is shown for fillet weld failure. The angle is defined in degrees.

Butt Weld Card. Additional Card required for the BUTT keyword option.

Card 2c	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	BETA	L	D		
Type	F	F	F	F	F	F		
Default	10^{16}	10^{16}	10^{16}	1.0	none	none		

VARIABLE	DESCRIPTION
TFAIL	Failure time for constraint set, t_f
EPSF	Effective plastic strain at failure, ϵ^p_{fail}, defines ductile failure.
SIGY	σ_f, stress at failure for brittle failure
BETA	β, failure parameter for brittle failure
L	L, length of butt weld (see Figure 10-9)
D	d, thickness of butt weld (see Figure 10-9)

Remarks:

Ductile butt weld failure, due to plastic straining, is treated identically to spot weld failure. Brittle failure of the butt welds occurs when:

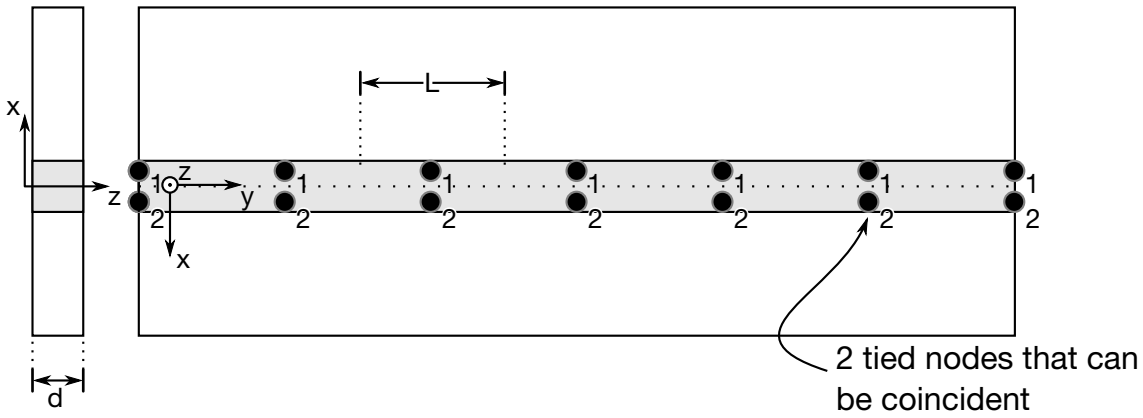


Figure 10-9. Orientation of the local coordinate system and nodal ordering is shown for butt weld failure.

$$\beta\sqrt{\sigma_n^2 + 3(\tau_n^2 + \tau_t^2)} \geq \sigma_f \ ,$$

where

σ_n = normal stress
τ_n = shear stress in direction of weld (local y)
τ_t = shear stress normal to weld (local z)
σ_f = failure stress
β = failure parameter

The component σ_n is nonzero for tensile values only. When the failure time, t_f, is reached the nodal rigid body becomes inactive and the constrained nodes may move freely. The nodes in the butt weld may coincide.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_GENERALIZED_WELD_BUTT
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Weld two plates that butt up against each other at three nodal pair
$  locations.  The nodal pairs are 32-33, 34-35 and 36-37.
$
$  This requires 3 separate *CONSTRAINED_GENERALIZED_WELD_BUTT definitions,
$  one for each nodal pair.  Each weld is to have a length (L) = 10,
$  thickness (D) = 2, and a transverse length (Lt) = 1.
$
$  Failure is defined two ways:
$    Ductile failure if effective plastic strain exceeds 0.3
$    Brittle failure if the stress failure criteria exceeds 0.25
$       - scale the brittle failure criteria by beta = 0.9.
$    Note: beta > 1 weakens weld          beta < 1 strengthens weld
$
*CONSTRAINED_GENERALIZED_WELD_BUTT
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid       cid
        21
$    tfail      epsf      sigy      beta         L         D        Lt
               0.3     0.250       0.9      10.0       2.0       1.0
$
$
*CONSTRAINED_GENERALIZED_WELD_BUTT
$     nsid       cid
        23
$    tfail      epsf      sigy      beta         L         D        Lt
               0.3     0.250       0.9      10.0       2.0       1.0
$
$
*CONSTRAINED_GENERALIZED_WELD_BUTT
$     nsid       cid
        25
$    tfail      epsf      sigy      beta         L         D        Lt
               0.3     0.250       0.9      10.0       2.0       1.0
$
$
*SET_NODE_LIST
```

```
$       sid
         21
$      nid1       nid2
         32         33
*SET_NODE_LIST
         23
         34         35
*SET_NODE_LIST
         25
         36         37
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

Cross Fillet Weld Card. Additional Card for the CROSS_FILLET keyword option.

Card 2d.1	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	BETA	L	W	A	ALPHA
Type	F	F	F	F	F	F	F	F
Default	10^{16}	10^{16}	10^{16}	1.0	none	none	none	none

Nodal Pair Cards. Read NPR additional cards for the CROSS_FILLET keyword option.

Card 2d.2	1	2	3	4	5	6	7	8
Variable	NODEA	NODEB	NCID					
Type	I	I	I					

VARIABLE	DESCRIPTION
TFAIL	Failure time for constraint set, t_f
EPSF	Effective plastic strain at failure, ϵ^p_{fail}, defines ductile failure
SIGY	σ_f, stress at failure for brittle failure
BETA	β, failure parameter for brittle failure
L	L, length of fillet weld (see Figure 10-8)
W	w, separation of parallel fillet welds (see Figure 10-8)
A	a, throat dimension of fillet weld (see Figure 10-8)
ALPHA	α, weld angle (see Figure 10-8) in degrees
NODEA	Node ID, A, in weld pair. See Figure 10-10.
NODEB	Node ID, B, in weld pair
NCID	Local coordinate system ID

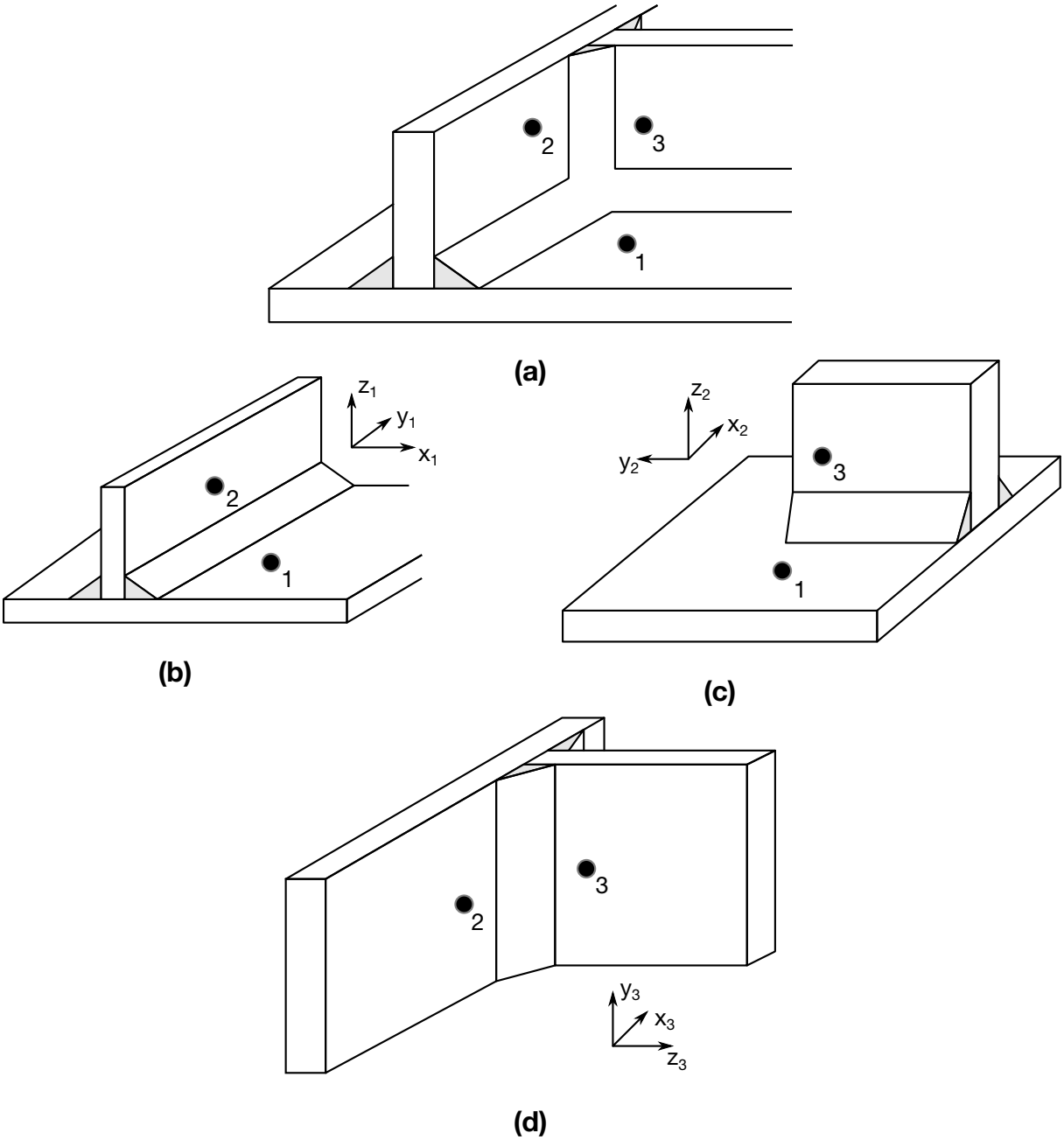


Figure 10-10. A simple cross fillet weld illustrates the required input. Here NPR = 3 with nodal pairs (A = 2, B = 1), (A = 3, B = 1), and (A = 3, B = 2). The local coordinate axes are shown. These axes are fixed in the rigid body and are referenced to the local rigid body coordinate system which tracks the rigid body rotation.

Combined Weld Cards:
Additional cards for the COMBINED keyword option. Read in NPR pairs of Cards 2e.1 and 2e.2 for a total of 2 × NPR cards.

Card 2e.1	1	2	3	4	5	6	7	8
Variable	TFAIL	EPSF	SIGY	BETA	L	W	A	ALPHA
Type	F	F	F	F	F	F	F	F
Default	10^{16}	10^{16}	10^{16}	1.0	none	none	none	none

Card 2e.2	1	2	3	4	5	6	7	8
Variable	NODEA	NODEB	NCID	WTYP				
Type	I	I	I	I				

VARIABLE	DESCRIPTION
TFAIL	Failure time, t_f, for constraint set
EPSF	Effective plastic strain at failure, ϵ^p_{fail}, defines ductile failure.
SIGY	σ_f, stress at failure for brittle failure.
BETA	β, failure parameter for brittle failure.
L	L, length of fillet/butt weld (see Figure 10-8 and 10-9).
W	w, width of flange (see Figure 10-8).
A	a, width of fillet weld (see Figure 10-8).
ALPHA	α, weld angle (see Figure 10-8) in degrees.
NODEA	Node ID, A, in weld pair
NODEB	Node ID, B, in weld pair
NCID	Local coordinate system ID

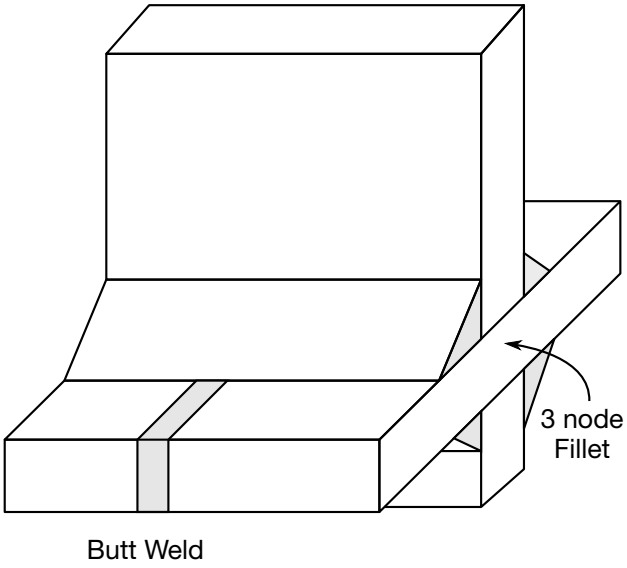


Figure 10-11. A combined weld is a mixture of fillet and butt welds.

VARIABLE	DESCRIPTION
WTYPE	Weld pair type. See Figure 10-11. EQ.0: Fillet weld EQ.1: Butt weld

*CONSTRAINED_GLOBAL

Purpose: Define a global boundary constraint plane. This card implements the trivial constraint equation; namely, constrained degrees of freedom are held fixed in time.

Card 1	1	2	3	4	5	6	7	8
Variable	TC	RC	DIR	X	Y	Z	TOL	
Type	I	I	I	F	F	F	F	
Default	0	0	none	0.0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
TC	Translational Constraint: EQ.0: no translational constraints EQ.1: constrained x translation EQ.2: constrained y translation EQ.3: constrained z translation EQ.4: constrained x and y translations EQ.5: constrained y and z translations EQ.6: constrained x and z translations EQ.7: constrained x, y, and z translations
RC	Rotational Constraint: EQ.0: no rotational constraints EQ.1: constrained x-rotation EQ.2: constrained y-rotation EQ.3: constrained z-rotation EQ.4: constrained x and y rotations EQ.5: constrained y and z rotations EQ.6: constrained z and x rotations EQ.7: constrained x, y, and z rotations

VARIABLE	DESCRIPTION
DIR	Direction of normal for constraint plane: EQ.1: global x EQ.2: global y EQ.3: global z
X	Global x-coordinate of a point on the constraint plane
Y	Global y-coordinate of a point on the constraint plane
Z	Global z-coordinate of a point on the constraint plane
TOL	User-defined tolerance in length units. If non-zero, the internal mesh-size dependent tolerance gets replaced by this value.

Remarks:

Nodes within a mesh-size-dependent tolerance are constrained on a global plane. This option is recommended for use with *r*-method adaptive remeshing where nodal constraints are lost during the remeshing phase. See *CONSTRAINED_LOCAL for specifying constraints to nodes lying on a local plane.

*CONSTRAINED_IMMERSED_IN_SPG

Purpose: Define coupling of beams and SPG solid elements through immerse method.

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

FEM Beams Card. Include this card as many times as needed. Input ends at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	IPID1	IPID2	IPID3	IPID4	IPID5	IPID6	IPID7	IPID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
SPGPID	Part ID of SPG solids into which where FEM beams are immersed
IPID*i*	Part IDs of FEM beams

*CONSTRAINED_INTERPOLATION_{OPTION}

Available options include:

<BLANK>

LOCAL

Purpose: Define an interpolation constraint. With this constraint type, the motion of a single dependent node is interpolated from the motion of a set of independent nodes.

This option is useful for the redistribution of a load applied to the dependent node by the surrounding independent nodes. This load may be a translational force or a rotational moment. This keyword is typically used to model shell-brick and beam-brick interfaces.

The mass and rotary inertia of the dependent nodal point is also redistributed. This constraint is applied in the global coordinate system unless the option LOCAL is active. *One *CONSTRAINED_INTERPOLATION card is required for each constraint definition.* The input list of independent nodes is terminated when the next keyword ("*") card is found.

In explicit calculations, the independent and dependent nodes cannot be dependent nodes in other constraints such as nodal rigid bodies; however, implicit calculations are not bound by this limitation. See Remark 2.

Card 1	1	2	3	4	5	6	7	8
Variable	ICID	DNID	DDOF	CIDD	ITYP	IDNSW	FGM	
Type	I	I	I	I	I	I	I	
Default	none	none	123456	global	0	1	0	

Independent Node Card Sets:

If LOCAL option is not set, for each independent node include the following card; if the LOCAL keyword option is set, include the following *pair* of cards. The next keyword ("*") card terminates this input.

Card 2	1	2	3	4	5	6	7	8
Variable	INID	IDOF	TWGHTX	TWGHTY	TWGHTZ	RWGHTX	RWGHTY	RWGHTZ
Type	I	I	F	F	F	F	F	F
Default	0	123456	1.0	TWGHTX	TWGHTX	TWGHTX	TWGHTX	TWGHTX

Local Coordinate Card. Additional card for the LOCAL keyword option to be paired with Card 2.

Card 3	1	2	3	4	5	6	7	8
Variable	CIDI							
Type	I							
Default	global							

VARIABLE	DESCRIPTION
ICID	Interpolation constraint ID.
DNID	Dependent node ID. This node should not be a member of a rigid body, or elsewhere constrained in the input.
DDOF	Dependent degrees-of-freedom. The list of dependent degrees-of-freedom consists of a number with up to six digits, with each digit representing a degree of freedom. For example, the value 1356 indicates that degrees of freedom 1, 3, 5, and 6 are controlled by the constraint. The default is 123456. DDOF are in the global coordinate system regardless of whether the LOCAL option is used or not. Digit degree of freedom ID's: EQ.1: x

VARIABLE	DESCRIPTION
	EQ.2: y
	EQ.3: z
	EQ.4: Rotation about x axis
	EQ.5: Rotation about y axis
	EQ.6: Rotation about z axis
CIDD	Local coordinate system ID if LOCAL option is active. If blank, the global coordinate system is assumed.
ITYP	Specifies the meaning of INID.
	EQ.0: INID is a node ID
	EQ.1: INID is a node set ID
INDSW	Switch for controlling the explicit solution when an independent (or dependent) node is deleted.
	EQ.0: Default to option 1.
	EQ.1: Terminate the explicit analysis when an independent node or the dependent node is deleted.
	EQ.2: Continue the explicit analysis with the constraints unchanged.
FGM	Flag for special treatment of this constraint for implicit problems only:
	EQ.0: Use standard constraint processing for implicit.
	EQ.1: Use special processing for this constraint for implicit only; see Remarks.
INID	Independent node ID or node set ID
IDOF	Independent degrees-of-freedom using the same form as for the dependent degrees-of-freedom, DDOF, above.
TWGHTX	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the x-translational component. It is normally sufficient to define only TWGHTX even if its degree-of-freedom is inactive since the other factors are set equal to this input value as the default. There is no requirement on the values that are chosen as the weighting factors, that is, that they sum to unity. The default value for the weighting factor is unity.

VARIABLE	DESCRIPTION
TWGHTY	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the y-translational component.
TWGHTZ	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the z-translational component.
RWGHTX	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the x-rotational component.
RWGHTY	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the y-rotational component.
RWGHTZ	Weighting factor for node INID with active degrees-of-freedom IDOF. This weight scales the z-rotational component.
CIDI	Local coordinate system ID if LOCAL option is active. If blank the global coordinate system is assumed.

Remarks:

1. **Dependent node not attached to the finite element model.** This constraint is sometimes used with a dependent node that is not attached to the finite element model. If point loads or discrete masses are put on this dependent node, then this constraint transfers the point load or the mass to the set of independent nodes. In another case, a dependent node can track a particular point in the geometry through this constraint as the model deforms and rotates, such as a point on the axis of rotation of a turbine. For these cases, where the dependent node is not attached to the finite element model but is an artificial node used to distribute forces and/or masses or to track a geometric position, we recommend setting FGM to 1.

2. **Limitations in enforcing constraints with the explicit solver.** As mentioned previously, dependent and independent nodes in this constraint cannot be dependent nodes for other constraints when using the explicit solver. LS-DYNA sequentially enforces constraints in explicit. Thus, there is no guarantee that LS-DYNA will evaluate the constraints in the desired order. For instance, suppose we have the following two constraints:

$$\text{constraint 1:} \quad x_1 = x_2$$
$$\text{constraint 2:} \quad x_2 = x_3$$

Then, at time step $n+1$, $x_1^{n+1} = x_2^n$ and $x_2^{n+1} = x_3^{n+1}$ because of the order of evaluation. Thus, we get a result that doesn't match the desired result of $x_1^{n+1} = x_2^{n+1} = x_3^{n+1}$.

Note that between different types of constraints (such as a rigid body constraint and a *CONSTRAINED_INTERPOLATION), the last evaluated constraint overwrites the previous results. Thus, one constraint may have no influence on the results and be effectively ignored.

Example 1:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_INTERPOLATION  (Beam to solid coupling)
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Tie a beam element to a solid element.
$
$  The node of the beam to be tied does not share a common node with the solids.
$  If the beam node is shared, for example, then set ddof=456.
$
*CONSTRAINED_INTERPOLATION
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     icid      dnid      ddof
         1        45    123456
$     inid      idof    twghtx    twghty    twghtz    rwghtx    rwghty    rwghtz
        22       123
        44       123
        43       123
$
*.........
$
```

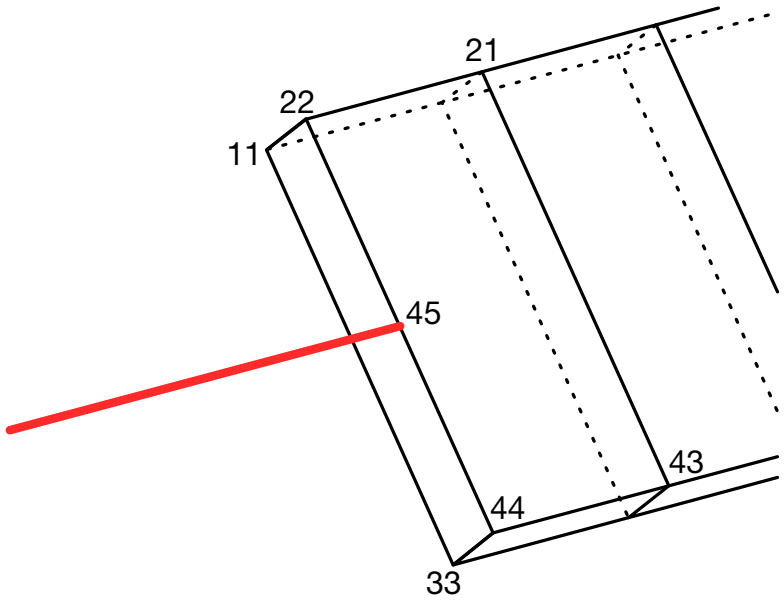


Figure 10-12. Illustration of Example 1.

Example 2:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_INTERPOLATION  (Load redistribution)
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Moment about normal axis of node 100 is converted to an equivalent load by
$  applying x-force resultants to the nodes lying along the right boundary
$
*DEFINE_CURVE
1,0,0.,0.,0.,0.,0
0.,0.
.1,10000.
*LOAD_NODE_POINT
100,6,1,1.0
$
*CONSTRAINED_INTERPOLATION
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     icid      dnid      ddof
         1       100         5
$     inid      idof    twghtx    twghty    twghtz    rwghtx    rwghty    rwghtz
        96       1
        97       1
        98       1
        99       1
       177       1
       178       1
       179       1
       180       1
$
*.........
$
```

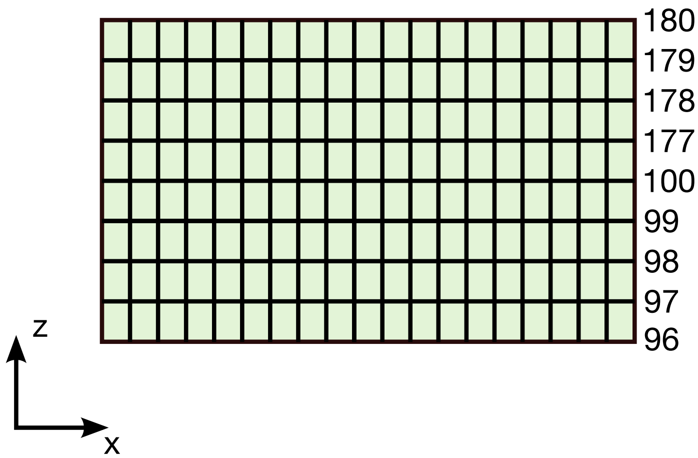


Figure 10-13. Illustration of Example 2.

*CONSTRAINED_INTERPOLATION_SPOTWELD_{OPTION}

(prior notation *CONSTRAINED_SPR3 still works)

Available options include:

<BLANK>

TITLE

Purpose: Define a spot weld with failure. This feature uses a plasticity-damage model that reduces the force and moment resultants to zero as the spot weld fails. A single node specifies the location of the spot weld at the center of two connected sheets. A radius approximately equal to the spot weld's radius gives the domain of influence. LS-DYNA performs a normal projection from the two sheets to the spot weld node and locates all nodes within the domain of influence. The numerical implementation of this feature is similar to the SPR2 model (*CONSTRAINED_SPR2).

Starting with LS-DYNA R13.1, this approach can connect not only shell element parts but also solid elements. However, the new capability is still experimental and should be treated with care. For instance, it is advantageous if the solid element parts still have some kind of obvious planar structure.

Card Summary:

Card Title. Include this card if the TITLE keyword option is used.

TITLE

Card 1. This card is required.

PID1	PID2	NSID	THICK	R	STIFF	ALPHA1	MODEL

Card 2. This card is required.

RN	RS	BETA1	LCF	LCUPF	LCUPR	DENS	INTP

Card 3. Include this card when MODEL = 2, 12, or 22.

UPFN	UPFS	ALPHA2	BETA2	UPRN	UPRS	ALPHA3	BETA3

Card 4. Include this card when MODEL = 2, 12, or 22.

MRN	MRS						

Card 5. This card is read only when MODEL = 1, 3, 11, 21, 31, or 41. It is optional.

STIFF2	STIFF3	STIFF4	LCDEXP	GAMMA	SROPT	PIDVB	

Card 6. This card is read only when MODEL = 1, 3, 11, 21, 31, or 41. It is optional.

SCARN	SCARS	DAMP	SPROFF				

Card 7. This card is read only when MODEL = 1, 11, 21, 31, or 41. It is optional.

FFN	FFB	FFS	EXFC	STIFP	MFSFC	DEFC	NPFC

Data Card Definitions:

Title Card. Additional card for the TITLE keyword option.

Card Title	1	2	3	4	5	6	7	8
Variable	TITLE							
Type	A80							

VARIABLE	DESCRIPTION
TITLE	Name or description of the SPR3 defined in this keyword. This name will be used as part title for the visualization beams, such as in the d3plot database. If undefined, that part title is "SPR3_-NSID_...".

Card 1	1	2	3	4	5	6	7	8
Variable	PID1	PID2	NSID	THICK	R	STIFF	ALPHA1	MODEL
Type	I	I	I	F	F	F	F	F
Default	none	none	none	none	none	none	none	1.0

VARIABLE	DESCRIPTION
PID1	Part ID or part set ID of the first sheet

VARIABLE	DESCRIPTION
	GT.0: Part ID LT.0: |PID1| is part set ID (for in-plane composed sheets, such as Tailored Blanks).
PID2	Part ID or part set ID of the second sheet. PID2 can be identical to PID1 if the spot weld location nodes from NSID lie in between the shell elements that should be self-connected. GT.0: Part ID LT.0: |PID2| is part set ID (for in-plane composed sheets, such as Tailored Blanks).
NSID	Node set ID of spot weld location nodes
THICK	Total thickness of both sheets
R	Spot weld radius
STIFF	Elastic stiffness. Function ID if MODEL > 10 (see Remark 2). GT.0: Constant value LT.0: |STIFF| refers to separate material data in *MAT_CONSTRAINED_SPR3 (*MAT_265).
ALPHA1	Scaling factor α_1. Function ID if MODEL > 10 (see Remark 2).
MODEL	Material behavior and damage model (see Remarks 3 and 4). EQ.1: SPR3 (default) EQ.2: SPR4 EQ.3: SPR3 with strongly objective formulation EQ.11: Same as 1 with selected material parameters as functions EQ.12: Same as 2 with selected material parameters as functions EQ.21: Same as 11 with slight modification EQ.22: Same as 12 with slight modification EQ.31: Same as 11 but with 12 more material parameters as functions EQ.41: Same as 31 with slight modification

Card 2	1	2	3	4	5	6	7	8
Variable	RN	RS	BETA1	LCF	LCUPF	LCUPR	DENS	INTP
Type	F	F	F	I	I	I	F	F
Default	none	none	none	none	↓	↓	none	0

VARIABLE	DESCRIPTION
RN	Tensile strength factor, R_n. GT.0.0: Constant value unless MODEL > 10. Function ID if MODEL > 10 (see Remark 2). LT.0.0: Load curve with ID |RN| giving R_n as a function of peel ratio (see Remark 5)
RS	Shear strength factor, R_s. Function ID if MODEL > 10 (see Remark 2).
BETA1	Exponent for plastic potential β_1. Function ID if MODEL > 10 (see Remark 2).
LCF	Load curve or table ID. Load curve ID describing force as a function of plastic displacement, that is, $F^0(\bar{u}^{\text{pl}})$. Table ID describing force as a function of mode mixity (table values) and plastic displacement (curves), that is, $F^0(\bar{u}^{\text{pl}}, \kappa)$.
LCUPF	Load curve ID describing plastic initiation displacement as a function of mode mixity, that is, $\bar{u}_0^{\text{pl}}(\kappa)$. Only for MODEL = 1, 11, or 21. For MODEL = 1, LCUPF can also be a table ID giving plastic initiation displacement as a function of peel ratio (table values) and mode mixity (curves). See Remark 5.
LCUPR	Load curve ID describing plastic rupture displacement as a function of mode mixity, that is, $\bar{u}_f^{\text{pl}}(\kappa)$. Only for MODEL = 1, 11, or 21. For MODEL = 1, LCUPF can also be a table ID giving plastic initiation displacement as a function of peel ratio (table values) and mode mixity (curves). See Remark 5.
DENS	Spot weld density (necessary for time step calculation).
INTP	Flag for interpolation. INTP is interpreted digit-wise, namely as,

VARIABLE	DESCRIPTION
	$\text{INTP} = [LK] = K + 10 \times L$ The first digit, K, controls the interpolation method: K.EQ.0: Linear (default), K.EQ.1: Uniform (not recommended for asymmetrical arrangement of the upper and lower nodes), K.EQ.2: Inverse distance weighting, K.EQ.3: Quadratic. The second digit, L, controls if the nodal area is considered when computing the weight: L.EQ.0: Nodal area is not considered (default), L.EQ.1: Nodal area is considered.

Additional Card for MODEL = 2, 12, or 22.

Card 3	1	2	3	4	5	6	7	8
Variable	UPFN	UPFS	ALPHA2	BETA2	UPRN	UPRS	ALPHA3	BETA3
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
UPFN	Plastic initiation displacement in the normal direction, $\bar{u}^{\text{pl},n}_{0,\text{ref}}$.
UPFS	Plastic initiation displacement in the shear direction, $\bar{u}^{\text{pl},s}_{0,\text{ref}}$.
ALPHA2	Plastic initiation displacement scaling factor, α_2.
BETA2	Exponent for plastic initiation displacement, β_2.
UPRN	Plastic rupture displacement in the normal direction, $\bar{u}^{\text{pl},n}_{f,\text{ref}}$.
UPRS	Plastic rupture displacement in shear direction, $\bar{u}^{\text{pl},s}_{f,\text{ref}}$.
ALPHA3	Plastic rupture displacement scaling factor, α_3.

VARIABLE	DESCRIPTION
BETA3	Exponent for plastic rupture displacement, β_3.

Additional Card for MODEL = 2, 12, or 22.

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

VARIABLE	DESCRIPTION
MRN	Proportionality factor for dependency, m_{R_n}.
MRS	Proportionality factor for dependency, m_{R_s}.

Optional Card for MODEL = 1, 3, 11, 21, 31, or 41.

Card 5	1	2	3	4	5	6	7	8
Variable	STIFF2	STIFF3	STIFF4	LCDEXP	GAMMA	SROPT	PIDVB	
Type	F	F	F	F	F	F	F	
Default	STIFF	STIFF	STIFF	0.0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
STIFF2	Elastic shear stiffness. Function ID if MODEL > 30 (see Remark 2). GT.0: Constant value or function ID. LT.0: |STIFF2| is a load curve ID for the curve giving shear stiffness as a function of load angle (ranging from 0 for pure shear to $\pi/2$ for pure tension).
STIFF3	Elastic bending stiffness. Function ID if MODEL > 30 (see Remark 2).

VARIABLE	DESCRIPTION
STIFF4	Elastic torsional stiffness. Function ID if MODEL > 30 (see Remark 2).
LCDEXP	Load curve for damage exponent as a function of mode mixity
GAMMA	Scaling factor, γ_1. Function ID if MODEL > 30 (see Remark 2).
SROPT	Shear rotation option that defines local kinematics system: EQ.0: Pure shear does not create a normal component (default). EQ.1: Pure shear creates a normal component.
PIDVB	Part ID for beams used to represent SPR3 in post-processing. EQ.0: Part ID automatically set (default). GT.0: PIDVB defines the part ID. LT.0: |PIDVB| defines the part ID, but an alternative type of beam representation (deformation) is invoked. Also, beams are deleted after failure.

Optional Card for MODEL = 1, 3, 11, 21, 31, 41.

Card 6	1	2	3	4	5	6	7	8
Variable	SCARN	SCARS	DAMP	SPROFF				
Type	F	F	F	F				
Default	1.0	1.0	0.0	0.0				

VARIABLE	DESCRIPTION
SCARN	Scale factor for tensile strength factor RN. This option also scales the displacements in LCF, LCUPF, and LCUPR so that the shape of the force-displacement curve stays similar. LT.0.0: |SCARN| refers to a load curve ID, giving the scale factor as a function of the state of the neighboring SPR3.See Remark 6.
SCARS	Scale factor for tensile strength factor RS. This option also scales the displacements in LCF, LCUPF, and LCUPR so that the shape of

VARIABLE	DESCRIPTION
	the force-displacement curve stays similar. LT.0.0: |SCARS| refers to a load curve ID, giving the scale factor as a function of the state of the neighboring SPR3. See Remark 6.
DAMP	Viscous coefficient to model stiffness-proportional damping effects. We recommend setting DAMP to a value between 0.05 and 0.15.
SPROFF	Specifies the relative location of the spot weld between PID1 and PID2. The value must be between -1.0 and 1.0. A value of -1.0 places the spot weld on PID1, while a value of 1.0 places it on PID2. By default (SPROFF = 0.0), the spot weld is located at the geometric center between PID1 and PID2

Optional Card for MODEL = 1, 11, 21, 31, 41. See Remark 3.

Card 7	1	2	3	4	5	6	7	8
Variable	FFN	FFB	FFS	EXFC	STIFP	MFSFC	DEFC	NPFC
Type	F	F	F	F	F	F	F	F
Default	optional	optional	optional	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
FFN	Resultant normal force at failure. FFN is a function ID if MODEL > 30 (see Remark 2).
FFB	Resultant bending force at failure. FFB is a function ID if MODEL > 30 (see Remark 2).
FFS	Resultant shear force at failure. FFS is a function ID if MODEL > 30 (see Remark 2).
EXFC	Failure function exponent. EXFC is a function ID if MODEL > 30 (see Remark 2).
STIFP	Plastic stiffness. If greater than zero, this replaces LCF by a simple linear hardening law:

VARIABLE	DESCRIPTION
	$F^0(\bar{u}^{pl}) = 1.0 + \text{STIFP} \times \bar{u}^{pl}$. STIFP is a function ID if MODEL > 30 (see Remark 2).
MFSFC	Scaling factor for torsion term in resultant shear force. MFSC is a function ID if MODEL > 30 (see Remark 2).
DEFC	Fading energy for damage. DEFC is a function ID if MODEL > 30 (see Remark 2).
NPFC	Plastic displacement offset for damage initiation. NPFC is a function ID if MODEL > 30 (see Remark 2).

Remarks:

1. **Drilling rotation constraint method.** With this keyword, we recommend using the drilling rotation constraint method for the connected components in explicit analysis. To do this, specify DRPSID of *CONTROL_SHELL to include all shell parts involved in INTERPOLATION_SPOTWELD connections.

2. **Function inputs.** If MODEL is chosen to be greater than 10 (but less than 30), five variables must be defined as function IDs: STIFF, ALPHA1, RN, RS, and BETA1. These functions incorporate the following input values: thicknesses of both weld partners (t1, t2) and maximum engineering yield stresses, also called necking points, (sm1, sm2). For ALPHA1 = 100 such a function could look like,

```
*DEFINE_FUNCTION
       100
func(t1,t2,sm1,sm2)=sm1/sm2
```

(This function is only a demonstration; it does not make any physical sense.) For MODEL = 11 or 12, the first sheet (PID1) is the first weld partner represented by t1 and sm1. For MODEL = 21 or 22, the thinner part is the first weld partner. Since material parameters must be identified from both weld partners during initialization, this feature is only available for a subset of material models, namely, materials 3, 24, 36, 81, 120, 123, 124, 133, 187, 224, 251, and 324.

If MODEL is chosen to be greater than 30, seventeen variables must be defined as function IDs: STIFF, ALPHA1, RN, RS, BETA1, STIFF2, STIFF3, STIFF4, GAMMA, FFN, FFB, FFS, EXFC, STIFP, MFSFC, DEFC, and NPFC. These functions incorporate the following input values: thicknesses of both weld partners (t1, t2); maximum engineering yield stresses, also called necking points, (sm1, sm2); initial yield stresses at plastic strain of 0.002 (sy1, sy2); resultant

velocity (v); and normal/bending/shear term in failure function (fn, fb, fs). For ALPHA1 = 100 such a function could look like:

```
*DEFINE_FUNCTION
       100
func(t1,t2,sm1,sm2,sy1,sy2,v,fn,fb,fs)=sm1/sm2
```

For MODEL = 31, the first sheet (PID1) is the first weld partner represented by t1, sm1, and sy1. For MODEL = 41, the thinner part is the first weld partner.

3. **Model = 1, 11, 21, 31, or 41 ("SPR3").** This numerical model is similar to the self-piercing rivet model SPR2 (see *CONSTRAINED_SPR2) but with some differences to make it more suitable for spot welds. The first difference is the symmetric behavior of the spot weld connection; that is, there is no distinction between an upper sheet and a lower sheet. This symmetry is done by averaging the normals of both parts and always distributing the balance moments equally to both sides.

The second difference is that three (or four if STIFF4 > 0) quantities describe the kinematics instead of two. These quantities are the normal relative displacement, δ_n; the tangential relative displacement, δ_t; and the relative rotation, ω_b (and the relative twist, ω_t). These quantities are all with respect to the plane-of-maximum opening. We can define a relative displacement vector as:

$$\mathbf{u} = (\delta_n, \delta_t, \omega_b) \ .$$

The third difference is the underlying material model. With the described kinematic quantities, LS-DYNA first computes an elastic effective force vector:

$$\tilde{\mathbf{f}} = (f_n, f_t, m_b) = \text{STIFF} \times \mathbf{u} = \text{STIFF} \times (\delta_n, \delta_t, \omega_b) \ .$$

Or, if individual stiffnesses are defined on optional Card 5, LS-DYNA treats each component separately like this (also allows torsional stiffness to be added):

$$f_n = \text{STIFF} \times \delta_n$$

$$f_t = \text{STIFF2} \times \delta_t$$

$$m_b = \text{STIFF3} \times \omega_b$$

$$m_t = \text{STIFF4} \times \omega_t$$

From that, the algorithm computes two resultant forces for the normal and tangential directions (shear), respectively, using

$$F_n = \langle f_n \rangle + \alpha_1 m_b, \qquad F_s = f_t + \gamma_1 |m_t| \ .$$

A yield function is defined for plastic behavior

$$\phi(\tilde{\mathbf{f}}, \bar{u}^{\text{pl}}) = P(\tilde{\mathbf{f}}) - F^0(\bar{u}^{\text{pl}}) \leq 0$$

with the potential

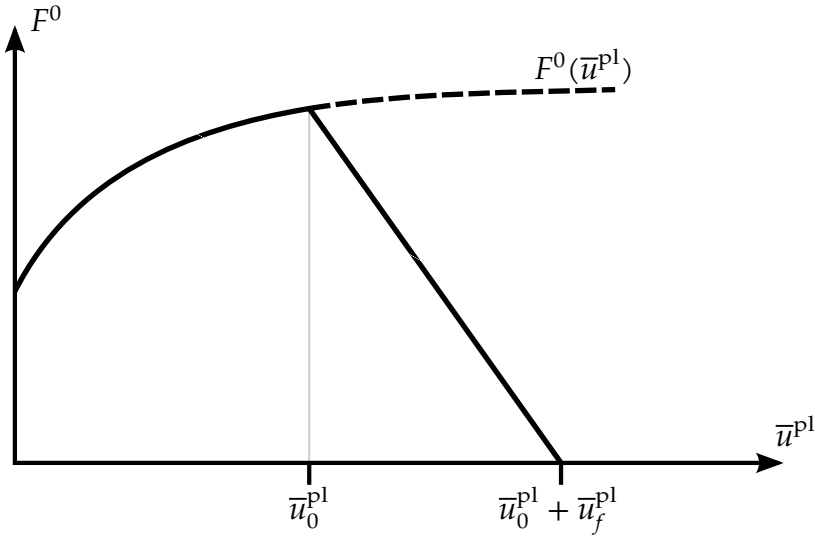


Figure 10-14. Force-displacement curve: plasticity and linear damage

$$P(\tilde{\mathbf{f}}) = \left[\left(\frac{F_n}{R_n} \right)^{\beta_1} + \left(\frac{F_s}{R_s} \right)^{\beta_1} \right]^{1/\beta_1} ,$$

and isotropic hardening described by load curve LCF (see Figure 10-14):

$$F^0 = F^0\left(\bar{u}^{\mathrm{pl}}\right) .$$

Therefore, the relative plastic displacement, $\bar{u}^{\mathrm{pl}}$, is calculated using the following relation:

$$P(\tilde{\mathbf{f}})\ \bar{u}^{\mathrm{pl}} = \sum_i F_i\ u_i^{\mathrm{pl}}, \qquad i = \{n, s\} .$$

In addition, if values on Card 7 are not defined, a linear softening evolution is incorporated, where damage is defined as:

$$d = \frac{\bar{u}^{\mathrm{pl}} - \bar{u}_0^{\mathrm{pl}}(\kappa)}{\bar{u}_f^{\mathrm{pl}}(\kappa)}, \qquad 0 < d < 1$$

with mode mixity

$$\kappa = \frac{2}{\pi} \arctan\left(\frac{F_n}{F_s} \right), \qquad 0 < \kappa < 1 .$$

Alternatively, nonlinear damage evolution can be invoked by using LCDEXP, where the damage exponent m is defined as a function of the mode mixity:

$$d = \frac{1 - \exp\left(-m\dfrac{\bar{u}^{\mathrm{pl}} - \bar{u}_0^{\mathrm{pl}}(\kappa)}{\bar{u}_f^{\mathrm{pl}}(\kappa)}\right)}{1 - \exp(-m)}, \qquad 0 < d < 1 \ .$$

Finally, the nominal force is computed as:

$$\mathbf{f} = (1 - d)\tilde{\mathbf{f}} \ .$$

If the parameters on Card 7 are defined, meaning at least FFN, FFB, and FFS are nonzero, a force-based failure criterion with an energy-based damage evolution is used. The failure criterion is given as:

$$f_c = \left(\frac{\langle f_n \rangle}{\mathrm{FFN}}\right)^{\mathrm{EXFC}} + \left(\frac{m_b}{\mathrm{FFB}}\right)^{\mathrm{EXFC}} + \left(\frac{f_t + \mathrm{MFSFC} \bullet m_t}{\mathrm{FFS}}\right)^{\mathrm{EXFC}} - 1 \ .$$

The damage is defined as:

$$d = \frac{(f_n \delta_n + f_t \delta_t)}{\mathrm{DEFC}}.$$

The implicit solver automatically uses a strongly objective formulation for the spot weld. To activate the strongly objective formulation in explicit, set MODEL to 3. For the explicit solver, this formulation tends to improve stability and accuracy when the elastic constants are high or when the model undergoes large rigid body rotation. Not all options available within MODEL = 1 are compatible with this formulation. For further details, regarding MODEL = 3, contact Ansys or your local distributor.

4. **MODEL = 2, 12, or 22 ("SPR4").** In this approach, the relative displacement vector is defined as in Model = 1:

$$\mathbf{u} = (\delta_n, \delta_t) \ .$$

The elastic effective force vector is computed using the elastic stiffness STIFF

$$\tilde{\mathbf{f}} = (f_n, f_t) = \mathrm{STIFF} \times \mathbf{u} = \mathrm{STIFF} \times (\delta_n, \delta_t) \ .$$

A yield function is defined for plastic behavior as

$$\phi(\tilde{\mathbf{f}}, \bar{u}^{\mathrm{pl}}) = P(\tilde{\mathbf{f}}) - F^0(\bar{u}^{\mathrm{pl}}) \leq 0$$

with relative plastic displacement $\bar{u}^{\mathrm{pl}}$ and potential

$$P(\tilde{\mathbf{f}}) = \left[\left(\frac{f_n}{\tilde{R}_n}\right)^{\beta_1} + \left(\frac{f_t}{\tilde{R}_s}\right)^{\beta}\right]^{1/\beta_1}$$

in which $\tilde{R}_n$ and $\tilde{R}_s$ represent the load capacity in the normal and tangential directions. They are calculated using the values of R_n and R_s as follows with the influence of relative rotation angle, ω_b, scaled by α_1:

$$\tilde{R}_s = R_s$$
$$\tilde{R}_n = R_n(1 - \alpha_1 \omega_b)$$

In addition, a linear softening evolution is incorporated, where damage is defined as:

$$d = \frac{\bar{u}^{\mathrm{pl}} - \bar{u}_0^{\mathrm{pl}}}{\bar{u}_f^{\mathrm{pl}}}, \qquad 0 < d < 1 \ .$$

The values of $\bar{u}_0^{\mathrm{pl}}$ and $\bar{u}_f^{\mathrm{pl}}$ are calculated by solving the following equations

$$\left\{ \left[\frac{\bar{u}_0^{\mathrm{pl},n}}{\bar{u}_{0,\mathrm{ref}}^{\mathrm{pl},n}(1 - \alpha_2 \omega_b)} \right]^{\beta_2} + \left(\frac{\bar{u}_0^{\mathrm{pl},s}}{\bar{u}_{0,\mathrm{ref}}^{\mathrm{pl},s}} \right)^{\beta_2} \right\}^{1/\beta_2} - 1 = 0, \qquad \begin{aligned} \bar{u}_0^{\mathrm{pl},n} &= \sin(\varphi)\, \bar{u}_0^{\mathrm{pl}} \\ \bar{u}_0^{\mathrm{pl},s} &= \cos(\varphi)\, \bar{u}_0^{\mathrm{pl}} \end{aligned}$$

$$\left\{ \left[\frac{\bar{u}_f^{\mathrm{pl},n}}{\bar{u}_{f,\mathrm{ref}}^{\mathrm{pl},n}(1 - \alpha_3 \omega_b)} \right]^{\beta_3} + \left(\frac{\bar{u}_f^{\mathrm{pl},s}}{\bar{u}_{f,\mathrm{ref}}^{\mathrm{pl},s}} \right)^{\beta_3} \right\}^{1/\beta_3} - 1 = 0, \qquad \begin{aligned} \bar{u}_f^{\mathrm{pl},n} &= \sin(\varphi)\, \bar{u}_f^{\mathrm{pl}} \\ \bar{u}_f^{\mathrm{pl},s} &= \cos(\varphi)\, \bar{u}_f^{\mathrm{pl}} \end{aligned}$$

in which the load angle is

$$\varphi = \arctan\left(\frac{f_n}{f_s} \right) \ .$$

For rate-dependent behavior, a plastic deformation rate $\dot{\bar{u}}^{\mathrm{pl}}$ is defined by

$$\dot{\bar{u}}^{\mathrm{pl}} = \frac{\Delta \bar{u}^{\mathrm{pl}}}{\Delta t}$$

where $\Delta \bar{u}^{\mathrm{pl}}$ is the plastic increment in the current time step and Δt is the time step size. If MRN and MRS are defined, $\tilde{R}_n$ and $\tilde{R}_s$ are calculated as

$$\tilde{R}_n\left(\dot{\bar{u}}^{\mathrm{pl}}\right) = \left(R_n + m_{R_n} \dot{\bar{u}}^{\mathrm{pl}}\right)(1 - \alpha_1\, \omega_b)$$
$$\tilde{R}_s\left(\dot{\bar{u}}^{\mathrm{pl}}\right) = R_s + m_{R_s} \dot{\bar{u}}^{\mathrm{pl}}$$

A detailed description of the SPR4 approach (MODEL = 2) is given in Bier and Sommer [2013], where this model is called "SPR3_IWM".

5. **Peel ratio.** For MODEL = 1, you can optionally define R_n (RN), $\bar{u}_0^{\mathrm{pl}}$ (LCUPF), and $\bar{u}_f^{\mathrm{pl}}$ (LCUPR) to be dependent on the peel ratio. The peel ratio is the ratio of the bending moment to the resultant axial force. This value is kept constant when plastic yield is reached.

6. **State of the neighboring SPR3.** The scale factors defined with SCARN and SCARS can depend on the state of the neighboring SPR3. SCARN and SCARS less than 0 enable this feature. The state measures how close the neighboring SPR3 is to failing. It is a value between 0 and 3. A value between 0 and 1 means the elastic regime, 1 and 2 means the plastic regime, and 2 and 3 means the

damage (softening) regime. Thus, as the state approaches 3, the neighboring SPR3 approaches failure. Having the scale factor depend upon the state applies when the material neighboring the SPR3 weakens as the SPR3 fails.

7. **History variables output.** If NEIPB = 7 is defined on *DATABASE_EXTENT_BINARY, then the following history variables are written to the d3plot database for SPR3 beams:

History Variable	Description
1	Load state (0,…, 1 = elastic, 1,…, 2 = plastic, 2,…, 3 = damage) $$\text{SPR state} = \begin{cases} \dfrac{P(\tilde{\mathbf{f}})}{F^0(\bar{u}^{\text{pl}})} & \text{if } \phi < 0 \\ 1 + \dfrac{\bar{u}^{\text{pl}}}{\bar{u}_0^{\text{pl}}} & \text{if } d = 0 \\ 2 + d & \text{if } d > 0 \end{cases}$$
2	Resultant normal force, F_n
3	Resultant shear force, F_s
4	Normal relative displacement
5	Tangential relative displacement
6	Relative rotation (bending)
7	Relative twist (torsion), only if STIFF4 > 0

***CONSTRAINED_JOINT**_*TYPE_{OPTION}_{OPTION}_{OPTION}*

Purpose: Define a joint between two rigid bodies.

CONSTRAINED_JOINT_*TYPE is a family of keywords all sharing a common set of data cards and option flags. The available joint variants are (one is mandatory):

*CONSTRAINED_JOINT_SPHERICAL

*CONSTRAINED_JOINT_REVOLUTE

*CONSTRAINED_JOINT_CYLINDRICAL

*CONSTRAINED_JOINT_PLANAR

*CONSTRAINED_JOINT_UNIVERSAL

*CONSTRAINED_JOINT_TRANSLATIONAL

*CONSTRAINED_JOINT_LOCKING

*CONSTRAINED_JOINT_TRANSLATIONAL_MOTOR

*CONSTRAINED_JOINT_ROTATIONAL_MOTOR

*CONSTRAINED_JOINT_GEARS

*CONSTRAINED_JOINT_RACK_AND_PINION

*CONSTRAINED_JOINT_CONSTANT_VELOCITY

*CONSTRAINED_JOINT_PULLEY

*CONSTRAINED_JOINT_SCREW

If the force output data is to be transformed into a local coordinate, use the option:

LOCAL

to define a joint ID and heading the following option is available:

ID

and to define failure for penalty-based joints (LMF = 0 in *CONTROL_RIGID) use:

FAILURE

The ordering of the bracketed options is arbitrary.

For the first seven joint types above excepting the Universal joint, the nodal points within the nodal pairs (1, 2), (3, 4), and (5, 6) (see Figures 10-16 through 10-22) should coincide in the initial configuration, and the nodal pairs should be as far apart as possible to obtain the best behavior. For the Universal Joint the nodes within the nodal pair (3, 4) do not coincide, but the lines drawn between nodes (1, 3) and (2, 4) must be perpendicular.

For the Gear joint the nodes within the nodal pair (1, 2) must not coincide, but for implicit the nodes within the pair (5, 6) must coincide and be located at the contact point of the gear teeth for correct results.

For cylindrical joints, by setting node 3 to zero, it is possible to use a cylindrical joint to join a node that is not on a rigid body (node 1) to a rigid body (nodes 2 and 4).

Card Summary:

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

JID	HEADING

Card 1. This card is required.

N1	N2	N3	N4	N5	N6	RPS	DAMP

Card 2. This card is required for joint types: MOTOR, GEARS, RACK_AND_PINION, PULLEY, and SCREW.

PARM	LCID	TYPE	R1	H_ANGLE			

Card 3. This card is included if the LOCAL keyword option is used.

RAID	LST						

Card 4. This card is included if the FAILURE keyword option is used.

CID	TFAIL	COUPL					

Card 5. This card is included if the FAILURE keyword option is used.

NXX	NYY	NZZ	MXX	MYY	MZZ		

Data Card Definitions:

ID Card. Additional card for ID keyword option. The heading is picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

Card ID	1	2	3	4	5	6	7	8
Variable	JID	HEADING						
Type	I	A70						

VARIABLE	DESCRIPTION
JID	Joint ID. This must be a unique number.
HEADING	Joint descriptor. It is suggested that unique descriptions be used.

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	N3	N4	N5	N6	RPS	DAMP
Type	I	I	I	I	I	I	F	F
Default	0	0	0	0	0	0	1.0	1.0

VARIABLE	DESCRIPTION
N1	Node 1, in rigid body A. Define for all joint types.
N2	Node 2, in rigid body B. Define for all joint types.
N3	Node 3, in rigid body A. Define for all joint types except SPHERICAL.
N4	Node 4, in rigid body B. Define for all joint types except SPHERICAL.
N5	Node 5, in rigid body A. Define for joint types TRANSLATIONAL, LOCKING, ROTATIONAL_MOTOR, CONSTANT_VELOCITY, GEARS, RACK_AND_PINION, PULLEY, and SCREW. For implicit GEARS, N5 should coincide with N6 and be located at the contact point of the gear teeth.

VARIABLE	DESCRIPTION
N6	Node 6, in rigid body B. Define for joint types TRANSLATIONAL, LOCKING, ROTATIONAL_MOTOR, CONSTANT_VELOCITY, GEARS, RACK_AND_PINION, PULLEY, and SCREW. For implicit GEARS, N6 should coincide with N5 and be located at the contact point of the gear teeth.
RPS	Relative penalty stiffness (default = 1.0): GT.0.0: Constant value, LT.0.0: Time dependent value given by load curve ID = -RPS (only for SPHERICAL, REVOLUTE, and CYLINDRICAL).
DAMP	Damping scale factor on default damping value. (Revolute and Spherical Joints): EQ.0.0: Default is set to 1.0, GT.0.0.AND.LE.0.01: No damping is used.

Rotational Properties Card. Additional card for joint types MOTOR, GEARS, RACK_AND_PINION, PULLEY, and SCREW.

Card 2	1	2	3	4	5	6	7	8
Variable	PARM	LCID	TYPE	R1	H_ANGLE			
Type	F	I	I	F	F			
Default	none	0	none	↓	0.0			

VARIABLE	DESCRIPTION
PARM	Parameter, which is a function of joint type: Gears: Define R_2/R_1 Rack and Pinion: Define h Pulley: Define R_2/R_1 Screw: Define $\dot{x}/\omega$ Motors: Leave blank
LCID	Define load curve ID for MOTOR joints.

VARIABLE	DESCRIPTION
TYPE	Define integer flag for MOTOR joints as follows: EQ.0: Translational/rotational velocity EQ.1: Translational/rotational acceleration EQ.2: Translational/rotational displacement
R1	Radius, R_1, for the gear and pulley joint type. If undefined, nodal points 5 and 6 are assumed to be on the outer radius. The values of R1 and R2 affect the outputted reaction forces. The forces are calculated from the moments by dividing them by the radii.
H_ANGLE	Helix angle in degrees. This is only necessary for the gear joint if the gears do not mesh tangentially, such as worm gears. See Remark 2 below for a definition. See Remark 4 for a discussion of which joint formulation to use.

Local Card. Additional card required for LOCAL keyword option.

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

VARIABLE	DESCRIPTION
RAID	Rigid body or accelerometer ID. The force resultants are output in the local system of the rigid body or accelerometer.
LST	Flag for local system type: EQ.0: Rigid body EQ.1: Accelerometer

Failure Card 1. Additional card for FAILURE keyword option.

Card 4	1	2	3	4	5	6	7	8
Variable	CID	TFAIL	COUPL					
Type	I	F	F					
Default	global	0.	0.					

Failure Card 2. Additional card for FAILURE keyword option.

Card 5	1	2	3	4	5	6	7	8
Variable	NXX	NYY	NZZ	MXX	MYY	MZZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
CID	Coordinate ID for resultants in the failure criteria. EQ.0: Global coordinate system
TFAIL	Time for joint failure. EQ.0.0: Joint never fails.
COUPL	Coupling between the force and moment failure criteria. LE.0.0: The failure criteria are identical to that of spotwelds. GT.0.0: The force and moment results are considered independently. See the Remark 1 below.
NXX	Axial force resultant N_{xx_F} at failure. EQ.0.0: failure due to this component is not considered.
NYY	Force resultant N_{yy_F} at failure. EQ.0.0: failure due to this component is not considered.

VARIABLE	DESCRIPTION
NZZ	Force resultant N_{zz_F} at failure. EQ.0.0: failure due to this component is not considered.
MXX	Torsional moment resultant M_{xx_F} at failure. EQ.0.0: failure due to this component is not considered.
MYY	Moment resultant M_{yy_F} at failure. EQ.0.0: failure due to this component is not considered.
MZZ	Moment resultant M_{zz_F} at failure. EQ.0.0: failure due to this component is not considered.

Remarks:

1. **Failure Criteria.** The moments for the revolute, cylindrical, planar, translational, and locking joints are calculated at the midpoint of nodes N1 and N3. The moments for the spherical, universal, constant velocity, gear, pulley, and rack and pinion joints are calculated at node N1. When COUPL is less than or equal to zero, the failure criterion is

$$\left(\frac{N_{xx}}{N_{xx_F}}\right)^2+\left(\frac{N_{yy}}{N_{yy_F}}\right)^2+\left(\frac{N_{zz}}{N_{zz_F}}\right)^2+\left(\frac{M_{xx}}{M_{xx_F}}\right)^2+\left(\frac{M_{yy}}{M_{yy_F}}\right)^2+\left(\frac{M_{zz}}{M_{zz_F}}\right)^2-1=0\ .$$

   Otherwise, it consists of both

$$\left(\frac{N_{xx}}{N_{xx_F}}\right)^2+\left(\frac{N_{yy}}{N_{yy_F}}\right)^2+\left(\frac{N_{zz}}{N_{zz_F}}\right)^2-1=0\ ,$$

   and

$$\left(\frac{M_{xx}}{M_{xx_F}}\right)^2+\left(\frac{M_{yy}}{M_{yy_F}}\right)^2+\left(\frac{M_{zz}}{M_{zz_F}}\right)^2-1=0\ .$$

2. **Helix Angle for Gear Joints.** For a gear joint, the relative direction and magnitude of rotation between the two gears is determined by the *helix angle*. Let $\mathbf{e}_1$ be the unit vector directed from node 2 to 4, which corresponds to the second gear's rotation axis. See Figure 10-25. Let $\mathbf{e}_2$ be defined as the positively oriented tangent vector to motion of the teeth when spun about the $\mathbf{e}_1$ axis (the gear's axis). See Figure 10-15. The helix angle α characterizes the deviation of the teeth axis from the gear axis. In particular, α is defined as the angle between the direction of teeth, called $\mathbf{e}_3$, and the axis of the gear $\mathbf{e}_1$,

$$\mathbf{e}_3=\cos\alpha\mathbf{e}_1+\sin\alpha\mathbf{e}_2\ .$$

The gears are assumed to be setup so that the teeth initially fit having matching $\mathbf{e}_3$ directions. A nonzero helix angle is typically used to model worm gears.

3. **Penalty Formulation.** When the penalty formulation is used (see LMF on *CONTROL_RIGID), at each time step, the relative penalty stiffness is multiplied by a function dependent on the step size to give the maximum stiffness that will not destroy the stability of the solution. Instabilities can result in the explicit time integration scheme if the penalty stiffness is too large. If instabilities occur, the recommended way to eliminate these problems is to decrease the time step or reduce the scale factor on the penalties.

4. **Joint Formulation for Gear Joints with a Helix Angle.** If a gear joint requires a helix angle (such as a worm joint), we recommend using the Lagrange multiplier formulation (LMF = 1 on *CONTROL_RIGID) instead of the default penalty formulation (LMF = 0). The penalty formulation does not account for the helix angle and will give incorrect results.

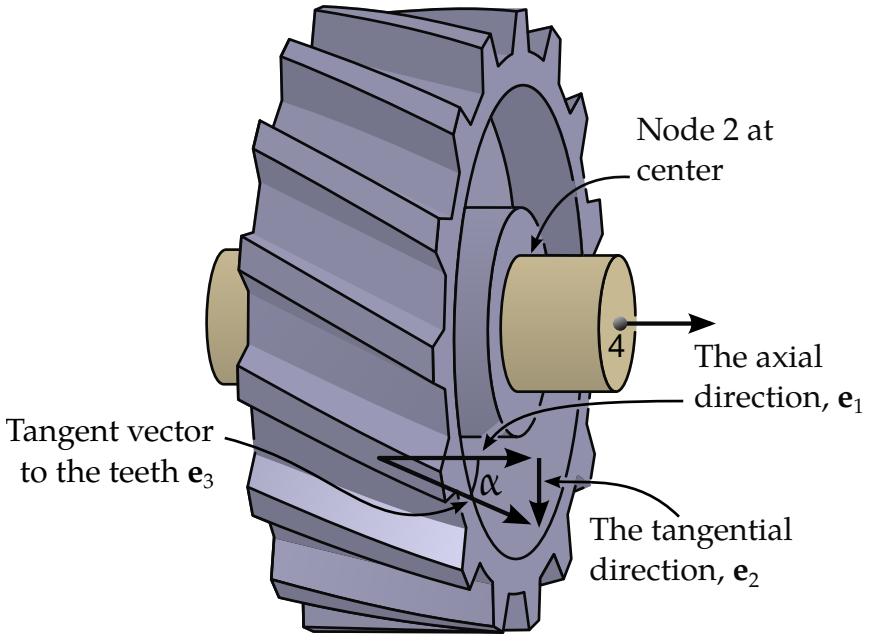


Figure 10-15. Helix angle α definition, gear #2 viewed from the extension of node n_2 to node n_6.

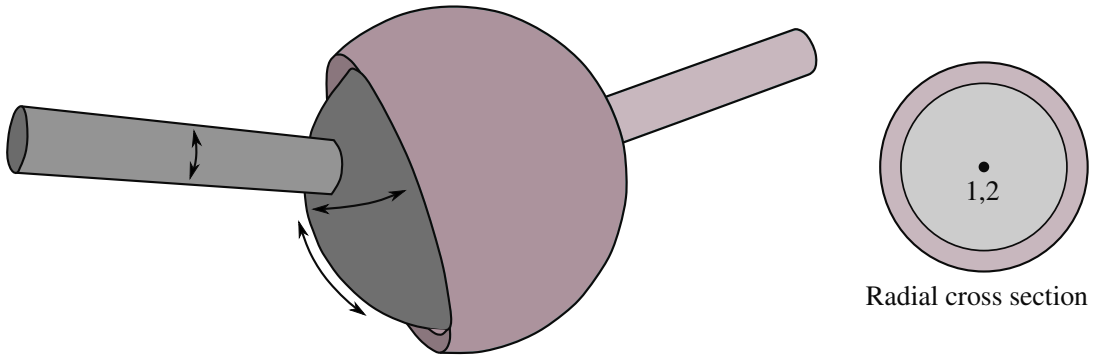


Figure 10-16. *Spherical joint.* The relative motion of the rigid bodies is constrained so that nodes which are initially coincident remain coincident. In the above figure the socket's node is not interior to the socket—LS-DYNA does not require that a rigid body's nodes be interior to the body.

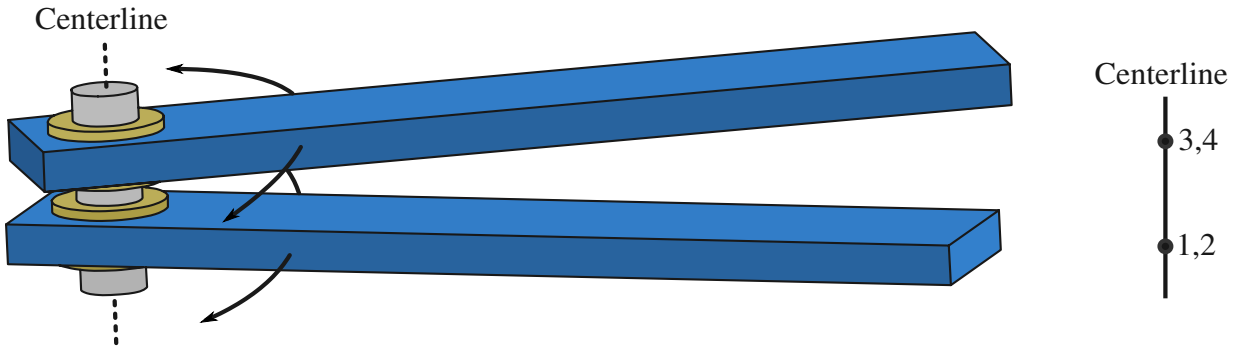


Figure 10-17. *Revolute Joint.* Nodes 1 and 2 are coincident; nodes 3 and 4 are coincident. Nodes 1 and 3 belong to rigid body A; nodes 2 and 4 belong to rigid body B. The relative motion of the two rigid bodies is restricted to rotations about the axis formed by the two pairs of coincident nodes. This axis is labeled the "centerline."

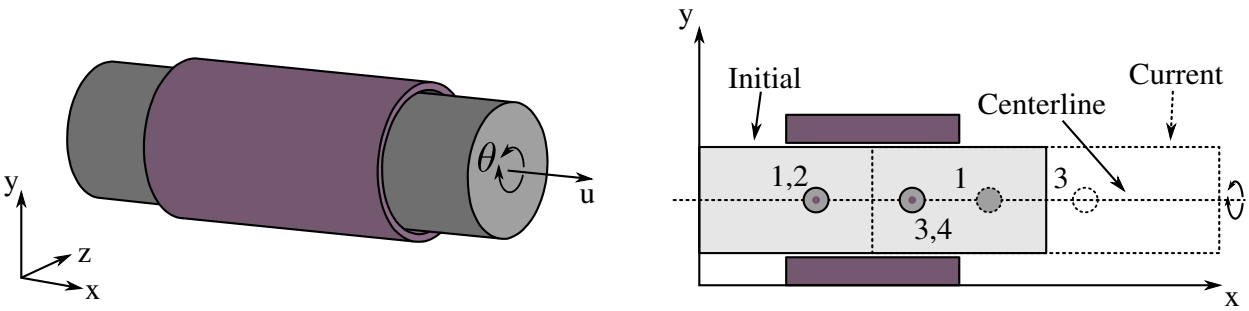


Figure 10-18. *Cylindrical Joint*. This joint is derived from the rotational joint by relaxing the constraints along the centerline. This joint admits relative rotation and translation along the centerline.

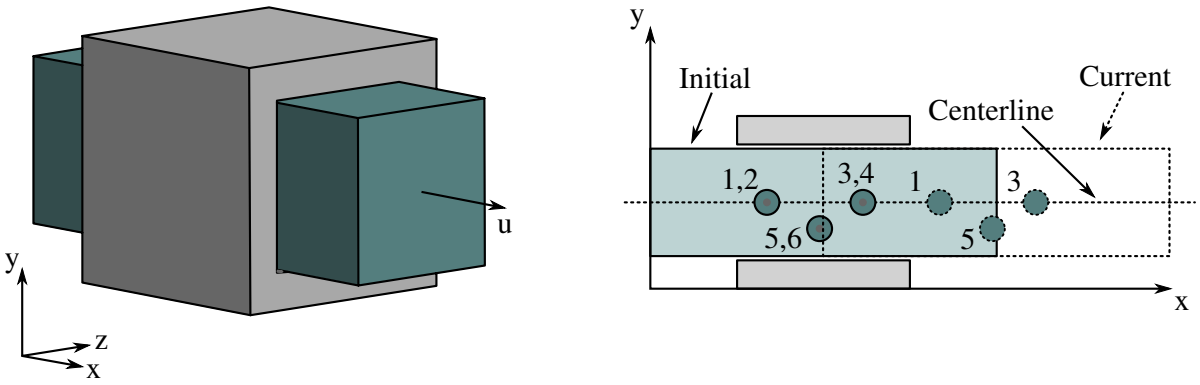


Figure 10-19. *Translational joint*. This is a cylindrical joint with a third pair of off-centerline nodes which restrict rotation. Aside from translation along the centerline, the two rigid bodies are stuck together.

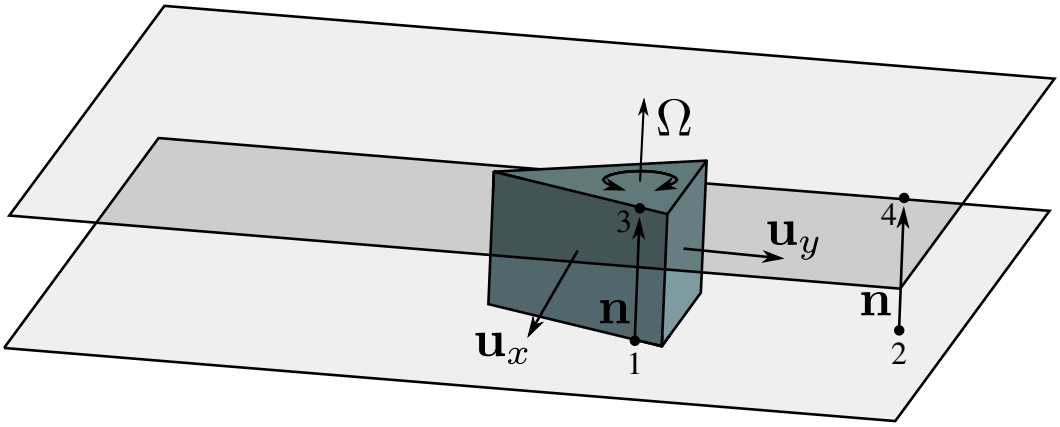


Figure 10-20. *Planar joint*. This joint is derived from the rotational joint by relaxing the constraints normal to the centerline. Relative displacements along the direction of the centerline are excluded.

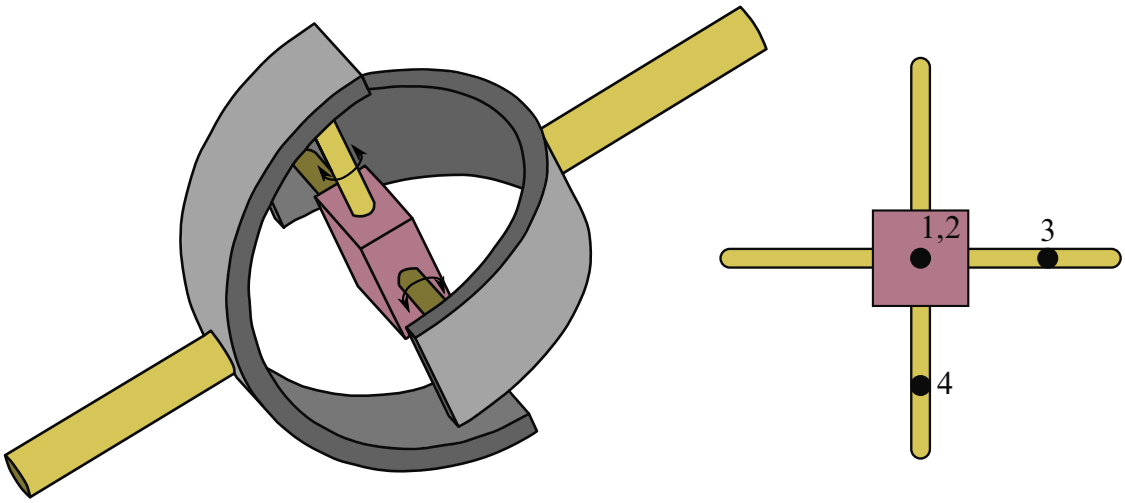


Figure 10-21. *Universal Joint*. Nodes 1 and 2 are initially coincident. The segments formed by nodal pairs (1, 3) and (2, 4) must be orthogonal; they serve as axes about which the two bodies may undergo relative rotation. The universal joint excludes all other relative motion and the axes remain orthogonal at all time.

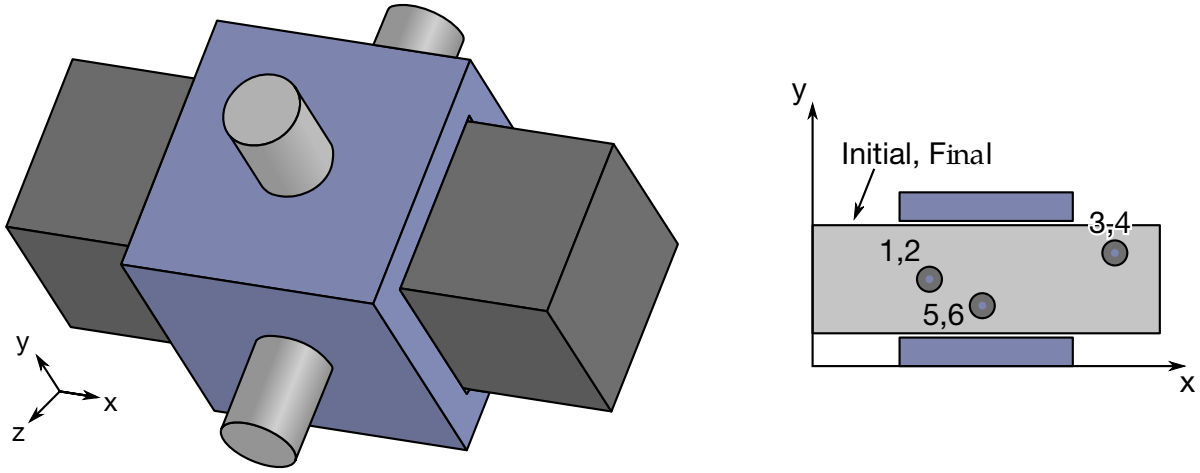


Figure 10-22. *Locking Joint*. A locking joint couples two rigid bodies in all six degrees-of-freedom. The forces and moments required to form this coupling are written to the jntforc file (*DATABASE_JNTFORC). As stated in the Remarks, forces and moments in jntforc are calculated halfway between N1 and N3. Nodal pairs (1, 2), (3, 4) and (5, 6) must be coincident. The three spatial points corresponding to three nodal pairs must be neither collocated nor collinear.

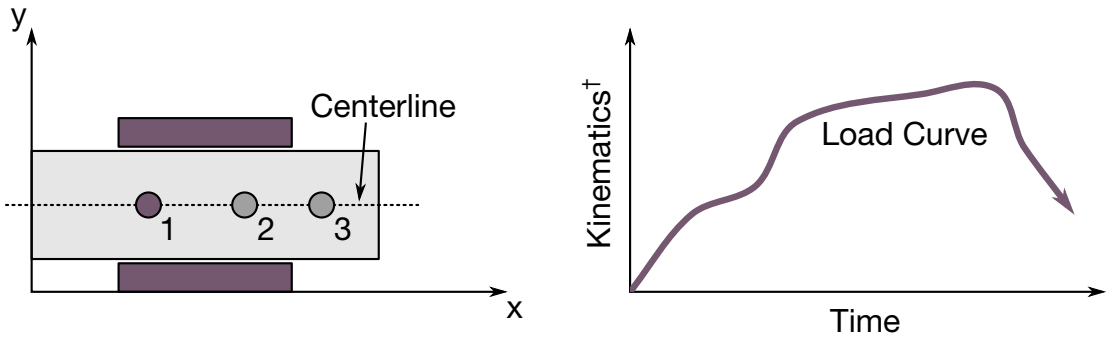


Figure 10-23. *Translational motor joint.* This joint is usually used in combination with a translational or a cylindrical joint. Node 1 and node 2 belong to the first rigid body and the second rigid body, resp. Furthermore, nodes 1 and 2 must be *coincident*. Node 3 may belong to either rigid body. Curve LCID defines the velocity/acceleration/displacement of N1 minus that of N2, where velocity/acceleration/displacement is measured in the direction of the vector N2-to-N3. Node 4 is not used and can be left blank. The value of the load curve may specify any of several kinematic measures; see TYPE.

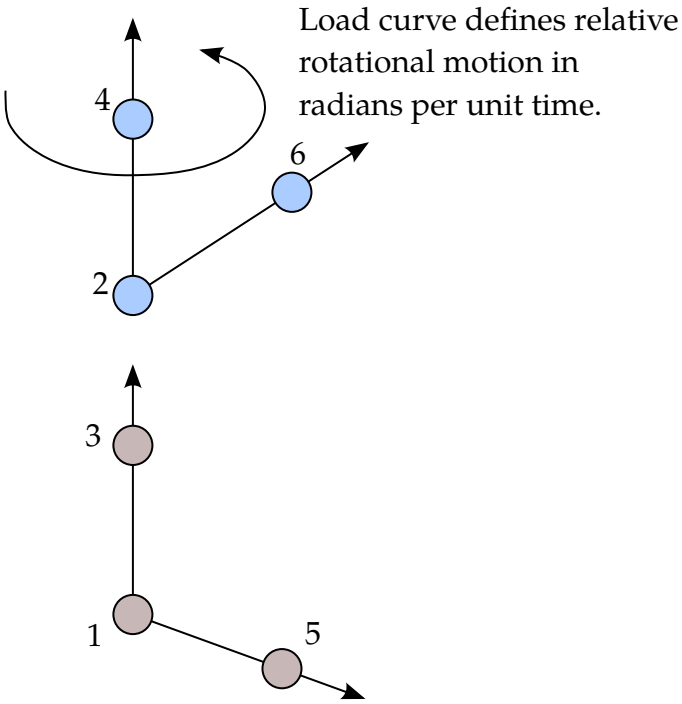


Figure 10-24. *Rotational motor joint.* This joint can be used in combination with other joints such as the revolute or cylindrical joints.

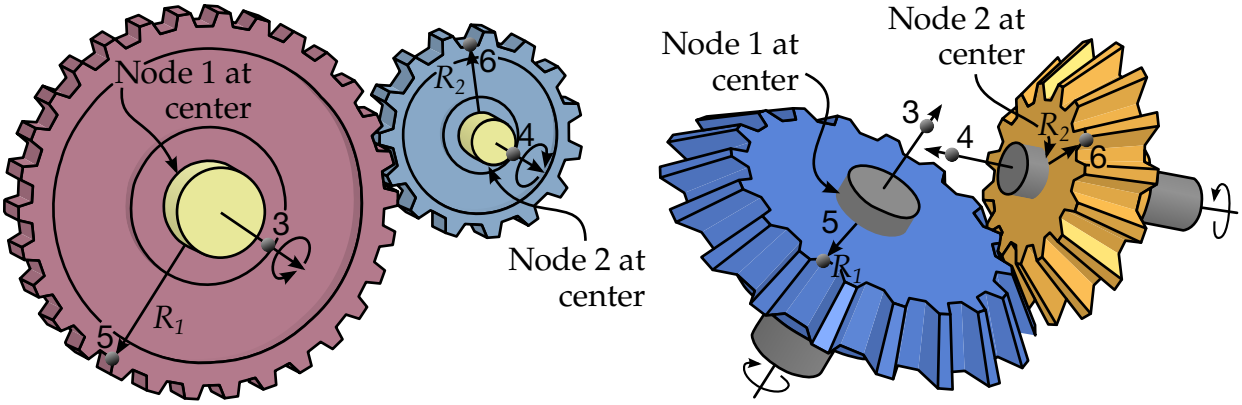


Figure 10-25. *Gear joints.* Nodal pairs (1, 3) and (2, 4) define axes that are orthogonal to the gears. Nodal pairs (1, 5) and (2, 6) define vectors in the plane of the gears. The ratio R_2/R_1 is specified but need not necessarily correspond to the geometry, if for instance the gear consists of spiral grooves. Note that the gear joint in itself does not maintain the contact point but this requires additional treatment, such as accompanying it with other joints. For implicit, nodes 5 and 6 must initially coincide for correct result in general.

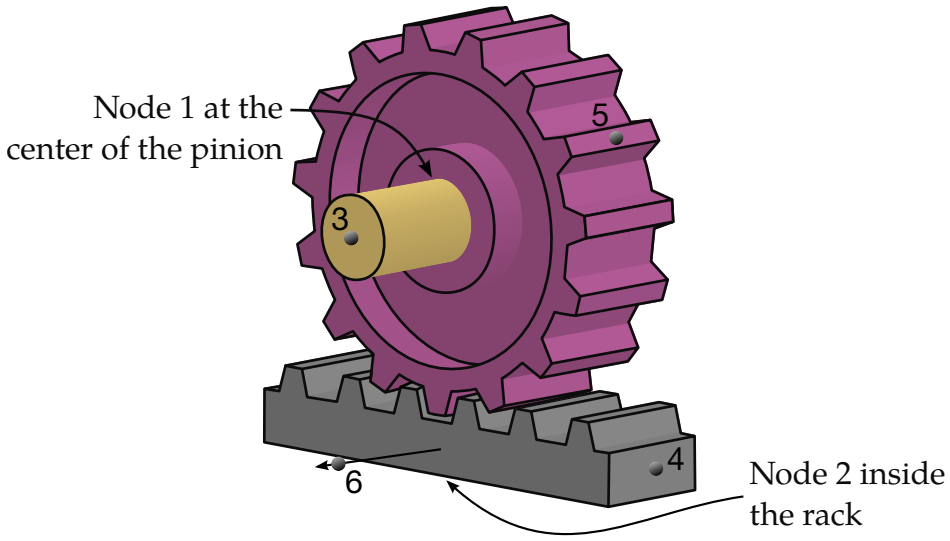


Figure 10-26. *Rack and pinion joint.* Nodal pair (1, 3) defines the axis of rotation of the first body (the pinion). Nodal pair (1, 5) is a vector in the plane of the pinion and is orthogonal to nodal pair (1, 3). Nodal pair (2, 4) defines the direction of travel for the second body (the rack). Nodal pair (2, 6) is parallel to the axis of the pinion and is thus parallel to nodal pair (1, 3). The value h is specified. The velocity of the rack is $h\omega_{\text{pinion}}$ in its direction of travel, where ω_{pinion} is positive according to the right-hand rule about the axis of the pinion.

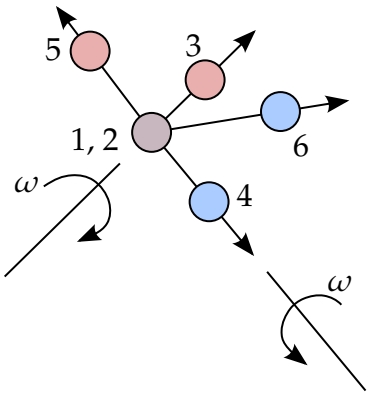


Figure 10-27. *Constant velocity joint.* Nodal pairs (1, 3) and (2, 4) define axes for the constant angular velocity. Nodal pairs (1, 5) and (2, 6) are orthogonal to nodal pairs (1, 3) and (2, 4), respectively. Here nodal points 1 and 2 must be coincident.

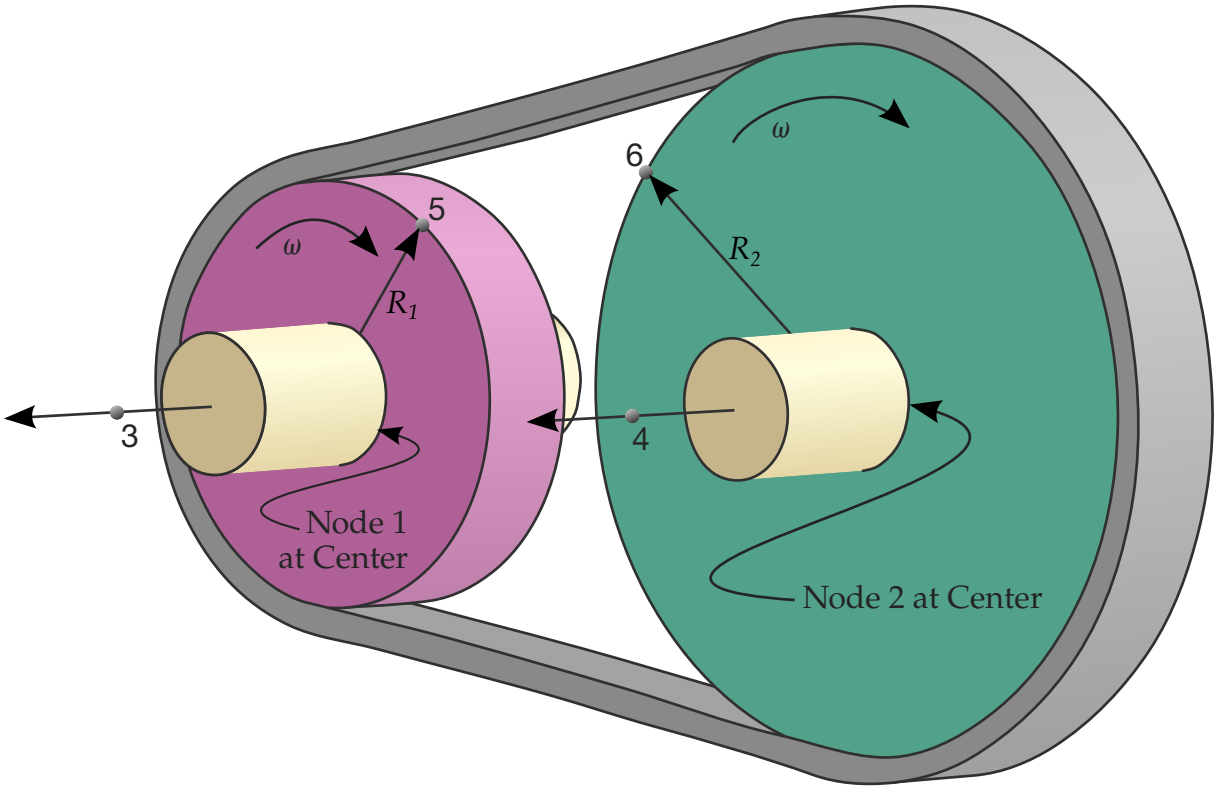


Figure 10-28. *Pulley joint.* Nodal pairs (1, 3) and (2, 4) define axes that are orthogonal to the pulleys. Nodal pairs (1, 5) and (2, 6) define vectors in the plane of the pulleys. The ratio R_2/R_1 is specified.

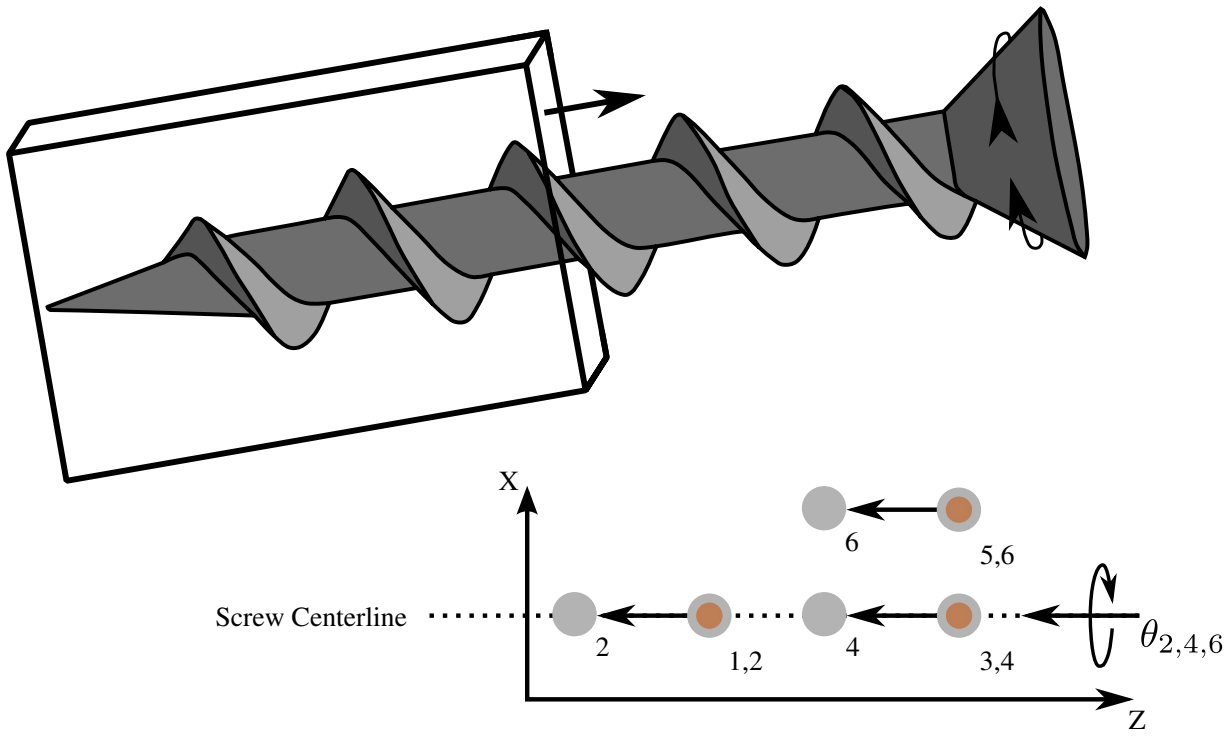


Figure 10-29. *Screw joint.* The second body (nodes 2, 4, and 6) translates in response to the spin of the first body (nodes 1, 3, and 5). Nodal pairs (1, 3) and (2, 4) lie along the same axis and nodal pairs (1, 5) and (2, 6) are orthogonal to the axis. PARM on Card 2 sets the helix ratio, $\dot{x}/\omega$. Note that, for a positive spin, the second body moves in the direction from node 1 to node 5.

Example 1:

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_JOINT_PLANAR
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a planar joint between two rigid bodies.
$    - Nodes 91 and 94 are on rigid body 1.
$    - Nodes 21 and 150 are on rigid body 2.
$    - Nodes 91 and 21 must be coincident.
$        * These nodes define the origin of the joint plane.
$    - Nodes 94 and 150 must be coincident.
$        * To accomplish this, massless node 150 is artificially created at
$           the same coordinates as node 94 and then added to rigid body 2.
$        * These nodes define the normal of the joint plane (e.g., the
$           vector from node 91 to 94 defines the planes' normal).
$
*CONSTRAINED_JOINT_PLANAR
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$       n1        n2        n3        n4        n5        n6       rps
        91        21        94       150                     0.000E+00
$
$
*NODE
$    nid               x               y               z      tc      rc
     150            0.00            3.00            0.00       0       0
$
*CONSTRAINED_EXTRA_NODES_SET
$      pid      nsid
         2         6
$*SET_NODE_LIST
$      sid
         6
$     nid1
       150
$
$$$  request output for joint force data
$
*DATABASE_JNTFORC
$  dt/cycl      lcdt
    0.0001
$
```

Example 2:

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_JOINT_REVOLUTE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Create a revolute joint between two rigid bodies.  The rigid bodies must
$  share a common edge to define the joint along.  This edge, however, must
$  not have the nodes merged together.  Rigid bodies A and B will rotate
$  relative to each other along the axis defined by the common edge.
$
$  Nodes 1 and 2 are on rigid body A and coincide with nodes 9 and 10
$  on rigid body B, respectively.  (This defines the axis of rotation.)
$
```

```
$  The relative penalty stiffness on the revolute joint is to be 1.0,
$
*CONSTRAINED_JOINT_REVOLUTE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$       n1        n2        n3        n4        n5        n6       rps      damp
         1         9         2        10                            1.0
$
$  Note:  A joint stiffness is not mandatory for this joint to work.
$          However, to see how a joint stiffness can be defined for this
$          particular joint, see the corresponding example listed in:
$             *CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

***CONSTRAINED_JOINT_COOR**_*TYPE_{OPTION}_{OPTION}_{OPTION}*

CONSTRAINED_JOINT_COOR_*TYPE is a family of keywords all sharing a common set of data cards and option flags. The available joint variants are (one is mandatory):

*CONSTRAINED_JOINT_COOR_SPHERICAL

*CONSTRAINED_JOINT_COOR_REVOLUTE

*CONSTRAINED_JOINT_COOR_CYLINDRICAL

*CONSTRAINED_JOINT_COOR_PLANAR

*CONSTRAINED_JOINT_COOR_UNIVERSAL

*CONSTRAINED_JOINT_COOR_TRANSLATIONAL

*CONSTRAINED_JOINT_COOR_LOCKING

*CONSTRAINED_JOINT_COOR_TRANSLATIONAL_MOTOR

*CONSTRAINED_JOINT_COOR_ROTATIONAL_MOTOR

*CONSTRAINED_JOINT_COOR_GEARS

*CONSTRAINED_JOINT_COOR_RACK_AND_PINION

*CONSTRAINED_JOINT_COOR_CONSTANT_VELOCITY

*CONSTRAINED_JOINT_COOR_PULLEY

*CONSTRAINED_JOINT_COOR_SCREW

If the force output data is to be transformed into a local coordinate, use the option:

LOCAL

to define a joint ID and heading the following option is available:

ID

and to define failure for penalty-based joints (LMF = 0 in *CONTROL_RIGID) use:

FAILURE

The ordering of the bracketed options is arbitrary.

Purpose: Define a joint between two rigid bodies; see Figures 10-16 through 10-29 of the *CONSTRAINED_JOINT section. The connection coordinates are given instead of the

nodal point IDs required in the previous section, *CONSTRAINED_JOINT_{*OPTION*}. Nodes are automatically generated for each coordinate and are constrained to the rigid body. Where coincident nodes are expected in the initial configuration, only one connection coordinate is needed since the connection coordinate for the second node, if given, is ignored. The created nodal IDs are chosen to exceed the maximum user ID. The coordinates of the joint nodes are specified on Cards 2 - 7. The input which follows Card 7 is identical to that in the previous section.

In the first seven joint types above excepting the Universal joint, the coordinate points within the nodal pairs (1, 2), (3, 4), and (5, 6) (see Figures 10-16 through 10-22) should coincide in the initial configuration, and the nodal pairs should be as far apart as possible to obtain the best behavior. For the Universal Joint the nodes within the coordinate pair (3, 4) do not coincide, but the lines drawn between nodes (1, 3) and (2, 4) must be perpendicular.

For the Gear joint the nodes within the coordinate pair (1, 2) must not coincide.

For cylindrical joints, by setting node 3 to zero, it is possible to use a cylindrical joint to join a node that is not on a rigid body (node 1) to a rigid body (nodes 2 and 4).

Card Summary:

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

JID	HEADING

Card 1. This card is required.

RBID_A	RBID_B	RPS	DAMP	TMASS	RMASS		

Card 2. This card is required.

X1	Y1	Z1					

Card 3. This card is required.

X2	Y2	Z2					

Card 4. This card is required.

X3	Y3	Z3					

Card 5. This card is required.

X4	Y4	Z4					

Card 6. This card is required.

X5	Y5	Z5					

Card 7. This card is required.

X6	Y6	Z6					

Card 8. This card is required for joint types: MOTOR, GEARS, RACK_AND_PINION, PULLEY, and SCREW.

PARM	LCID	TYPE	R1				

Card 9. This card is included if and only if the LOCAL keyword option is used.

RAID	LST						

Card 10. This card is included if and only if the FAILURE keyword option is used.

CID	TFAIL	COUPL					

Card 11. This card is included if and only if the FAILURE keyword option is used.

NXX	NYY	NZZ	MXX	MYY	MZZ		

Data Card Definitions:

ID Card. Additional card for the ID keyword option. The heading is picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

Card ID	1	2	3	4	5	6	7	8
Variable	JID	HEADING						
Type	I	A70						

VARIABLE	DESCRIPTION
JID	Joint ID. This must be a unique number.
HEADING	Joint descriptor. It is suggested that unique descriptions be used.

Card 1	1	2	3	4	5	6	7	8
Variable	RBID_A	RBID_B	RPS	DAMP	TMASS	RMASS		
Type	I	I	F	F	F	F		

VARIABLE	DESCRIPTION
RBID_A	Part ID of rigid body A. See Remark 2.
RBID_B	Part ID of rigid body B. See Remark 2.
RPS	Relative penalty stiffness (default = 1.0). See Remark 1.
DAMP	Damping scale factor on default damping value. (Revolute and Spherical Joints): EQ.0.0: default is set to 1.0, GT.0.0 and LE.0.01: no damping is used.
TMASS	Lumped translational mass. The mass is equally split between the first points defined for rigid bodies A and B. See Remark 3.
RMASS	Lumped rotational inertia. The inertia is equally split between the first points defined for rigid bodies A and B. See Remark 3.

Card 2	1	2	3	4	5	6	7	8
Variable	X1	Y1	Z1					
Type	F	F	F					

Card 3	1	2	3	4	5	6	7	8
Variable	X2	Y2	Z2					
Type	F	F	F					

Card 4	1	2	3	4	5	6	7	8
Variable	X3	Y3	Z3					
Type	F	F	F					

Card 5	1	2	3	4	5	6	7	8
Variable	X4	Y4	Z4					
Type	F	F	F					

Card 6	1	2	3	4	5	6	7	8
Variable	X5	Y5	Z5					
Type	F	F	F					

Card 7	1	2	3	4	5	6	7	8
Variable	X6	Y6	Z6					
Type	F	F	F					

VARIABLE	DESCRIPTION
X1, Y1, Z1	Coordinate of point 1, in rigid body A. Define for all joint types.
X2, Y2, Z2	Coordinate of point 2, in rigid body B. If points 1 and 2 are coincident in the specified joint type, the coordinate for point 1 is used.
X3, Y3, Z3	Coordinate of point 3, in rigid body A. Define for all joint types.
X4, Y4, Z4	Coordinate of point 4, in rigid body B. If points 3 and 4 are coincident in the specified joint type, the coordinate for point 3 is used.
X5, Y5, Z5	Coordinate of point 5, in rigid body A. Define for all joint types.

VARIABLE	DESCRIPTION
X6, Y6, Z6	Coordinate of point 6, in rigid body B. If points 5 and 6 are coincident in the specified joint type, the coordinate for point 5 is used.

Rotational Properties Card. Additional card for joint types MOTOR, GEARS, RACK_AND_PINION, PULLEY, and SCREW.

Card 8	1	2	3	4	5	6	7	8
Variable	PARM	LCID	TYPE	R1				
Type	F	I	I	F				
Default	none	0	0	↓				

VARIABLE	DESCRIPTION
PARM	Parameter, which is a function of joint type: Gears: define R_2/R_1 Rack and Pinion: define h Pulley: define R_2/R_1 Screw: define $\dot{x}/\omega$ Motors: leave blank
LCID	Define load curve ID for MOTOR joints.
TYPE	Define integer flag for MOTOR joints as follows: EQ.0: translational/rotational velocity EQ.1: translational/rotational acceleration EQ.2: translational/rotational displacement
R1	Radius, R_1, for the gear and pulley joint type. If left undefined, nodal points 5 and 6 are assumed to be on the outer radius. R1 is the moment arm that goes into calculating the joint reaction forces. The ratio R_2/R_1 gives the transmitted moments, but not the forces. The force is moment divided by distance R1.

Local Card. Additional card for LOCAL keyword option.

Card 9	1	2	3	4	5	6	7	8
Variable	RAID	LST						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
RAID	Rigid body or accelerometer ID. The force resultants are output in the local system of the rigid body or accelerometer.
LST	Flag for local system type: EQ.0: rigid body EQ.1: accelerometer

Failure Card 1. Additional card for the FAILURE keyword option.

Card 10	1	2	3	4	5	6	7	8
Variable	CID	TFAIL	COUPL					
Type	I	F	F					
Default	global	0.0	0.0					

Failure Card 2. Additional card for the FAILURE keyword option.

Card 11	1	2	3	4	5	6	7	8
Variable	NXX	NYY	NZZ	MXX	MYY	MZZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0.		

VARIABLE	DESCRIPTION
CID	Coordinate ID for resultants in the failure criteria. EQ.0: global coordinate system
TFAIL	Time for joint failure. EQ.0.0: joint never fails.
COUPL	Coupling between the force and moment failure criteria: LE.0.0: failure criteria are identical to that of spotwelds. GT.0.0: the force and moment results are considered independently. See the remarks in *CONSTRAINED_JOINT_{OPTION}.
NXX	Axial force resultant N_{XX_F} at failure. EQ.0.0: failure due to this component is not considered.
NYY	Force resultant N_{YY_F} at failure. EQ.0.0: failure due to this component is not considered.
NZZ	Force resultant N_{ZZ_F} at failure. EQ.0.0: failure due to this component is not considered.
MXX	Torsional moment resultant M_{XX_F} at failure. EQ.0.0: failure due to this component is not considered.
MYY	Moment resultant M_{YY_F} at failure. EQ.0.0: failure due to this component is not considered.

VARIABLE	DESCRIPTION
MZZ	Moment resultant M_{ZZ_F} at failure. EQ.0.0: failure due to this component is not considered.

Remarks:

1. **Penalty Method.** When the penalty method is used (see *CONTROL_RIGID), at each time step, the relative penalty stiffness is multiplied by a function dependent on the step size to give the maximum stiffness that will not destroy the stability of the solution. LS-DYNA's explicit time integrator can become unstable when the penalty stiffness is too large. If instabilities occur, the recommended way to eliminate these problems is to decrease the time step or reduce the scale factor on the penalties.

2. **Joint Nodes.** The coordinates specified in this formulation are used to generate up to six nodes and assign them to the appropriate rigid body for updating. RBID_A updates coordinates 1, 3, and 5, while RBID_B updates coordinates 2, 4, and 6. There is no method for LS-DYNA to detect when the wrong bodies are assigned, and therefore the user has to make sure that the ordering of RBID_A and RBID_B is correct. For some joints, such as the spherical joint, the coordinate points are the same and the ordering does not matter. However, for others, such as the rack and pinion, the ordering is critical.

3. **Mass and Inertia.** Unless the rigid bodies are massless, or nearly massless, there is usually no need to add mass or inertia to the rigid bodies, and TMASS and RMASS may be safely set to zero. Specifying values that are larger than necessary may perturb the motion of the bodies and result in significant errors. Since the masses and inertias are included in the mass calculations in the output, the values should be set to zero if the masses and inertia tensors are to be used outside of LS-DYNA.

***CONSTRAINED_JOINT_STIFFNESS**_OPTION_{OPTION}

Available options include:

FLEXION-TORSION

GENERALIZED

TRANSLATIONAL

CYLINDRICAL

If desired a description of the joint stiffness can be provided with the option:

TITLE

which is written into the d3hsp and jntforc files.

Purpose: Define optional rotational, translational or cylindrical joint stiffness for joints defined by *CONSTRAINED_JOINT_*OPTION*. These definitions apply to all joints even though degrees of freedom that are considered in the joint stiffness capability may be constrained out in some joint types. The energy that is dissipated with the joint stiffness option is written for each joint in joint force file with the default name, jntforc. In the global energy balance this energy is included with the energy of the discrete elements, that is, the springs and dampers.

Card Summary:

Card Title. This card is included if and only if the TITLE keyword option is used.

TITLE

Card 1. This card is required.

JSID	PIDA	PIDB	CIDA	CIDB	JID	RPS	

Card 2a.1. This card is included if the FLEXION-TORSION keyword option is used.

LCIDAL	LCIDG	LCIDBT	DLCIDAL	DLCIDG	DLCIDBT		

Card 2a.2. This card is included if the FLEXION-TORSION keyword option is used.

ESAL	FMAL	ESBT	FMBT				

Card 2a.3. This card is included if the FLEXION-TORSION keyword option is used.

SAAL	NSABT	PSABT					

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

LCIDPH	LCIDT	LCIDPS	DLCIDPH	DLCIDT	DLCIDPS		

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

ESPH	FMPH	EST	FMT	ESPS	FMPS		

Card 2b.3. This card is included if the GENERALIZED keyword option is used.

NSAPH	PSAPH	NSAT	PSAT	NSAPS	PSAPS		

Card 2c.1. This card is included if the TRANSLATIONAL keyword is used.

LCIDX	LCIDY	LCIDZ	DLCIDX	DLCIDY	DLCIDZ		

Card 2c.2. This card is included if the TRANSLATIONAL keyword is used.

ESX	FFX	ESY	FFY	ESZ	FFZ		

Card 2c.3. This card is included if the TRANSLATIONAL keyword is used.

NSDX	PSDX	NSDY	PSDY	NSDZ	PSDZ	FS	FD

Card 2d.1. This card is included if the CYLINDRICAL keyword is used.

LCIDR		LCIDZ	DLCIDR	DLCIDP	DLCIDZ	LCIDT	DLCIDT

Card 2d.2. This card is included if the CYLINDRICAL keyword is used.

ESR	FFR			ESZ	FFZ	RAD1	RAD2

Card 2d.3. This card is included if the CYLINDRICAL keyword is used.

	PSDR			NSDZ	PSDZ	FS	FD

Data Card Definitions:

Title Card. Additional card for the TITLE keyword option.

Card Title	1	2	3	4	5	6	7	8
Variable	TITLE							
Type	A80							

Card 1	1	2	3	4	5	6	7	8
Variable	JSID	PIDA	PIDB	CIDA	CIDB	JID	RPS	
Type	I	I	I	I	I	I	F	
Default	none	none	none	none	CIDA	0	1.0	

VARIABLE	DESCRIPTION
TITLE	Description of joint stiffness for output files jntforc and d3hsp
JSID	Joint stiffness ID
PIDA	Part ID for rigid body A; see *PART.
PIDB	Part ID for rigid body B; see *PART.
CIDA	Coordinate ID for rigid body A; see *DEFINE_COORDINATE_OPTION. For the translational and cylindrical stiffnesses, the local coordinate system must be defined by nodal points, *DEFINE_COORDINATE_NODES, since the first nodal point in each coordinate system is used to track the motion.
CIDB	Coordinate ID for rigid body B. If zero, the coordinate ID for rigid body A is used; see *DEFINE_COORDINATE_OPTION. For the translational and cylindrical stiffnesses, the local coordinate system must be defined by nodal points, *DEFINE_COORDINATE_NODES, since the first nodal point in each coordinate system is used to track the motion.
JID	Joint ID for the joint reaction forces. If zero, tables can not be used in place of load curves for defining the frictional moments.

VARIABLE	DESCRIPTION
RPS	Relative penalty stiffness used for joint friction calculation. It is the same parameter as RPS in *CONSTRAINED_JOINT_*TYPE*. It only applies for keyword options TRANSLATIONAL and CYLINDRICAL. FS and FD must be defined in either Card 2c. 3 or Card 2d. 3. It can be used to calculate the joint force, so we can define it here instead of in *CONSTRAINED_JOINT_*TYPE*.

Flexion-Torsion Joint Stiffness Cards:

Card 2 for FLEXION-TORSION option.

Card 2a.1	1	2	3	4	5	6	7	8
Variable	LCIDAL	LCIDG	LCIDBT	DLCIDAL	DLCIDG	DLCIDBT		
Type	I	I	I	I	I	I		
Default	0.0	1.0	0.0	0	1.0	0		

VARIABLE	DESCRIPTION
LCIDAL	Load curve ID for α-moment as a function of rotation in radians. See Figure 10-30 where it should be noted that $0 \le \alpha \le \pi$. See *DEFINE_CURVE. EQ.0: The applied moment is set to 0.0.
LCIDG	Load curve ID for γ as a function of a scale factor which scales the bending moment due to the α rotation. This load curve should be defined in the interval $-\pi \le \gamma \le \pi$. See *DEFINE_CURVE. EQ.0: The scale factor defaults to 1.0.
LCIDBT	Load curve ID for β-torsion moment as a function of twist in radians. See *DEFINE_CURVE. EQ.0: The applied twist is set to 0.0.
DLCIDAL	Load curve ID for α-damping momentas a function of rate of rotation in radians per unit time. See *DEFINE_CURVE. EQ.0: damping is not considered.
DLCIDG	Load curve ID for γ-damping scale factor as a function of rate of rotation in radians per unit time. This scale factor scales the α-damping moment. See *DEFINE_CURVE. EQ.0: The scale factor defaults to 1.0.
DLCIDBT	Load curve ID for β-damping torque as a function of rate of twist. See *DEFINE_CURVE. EQ.0: Damping is not considered.

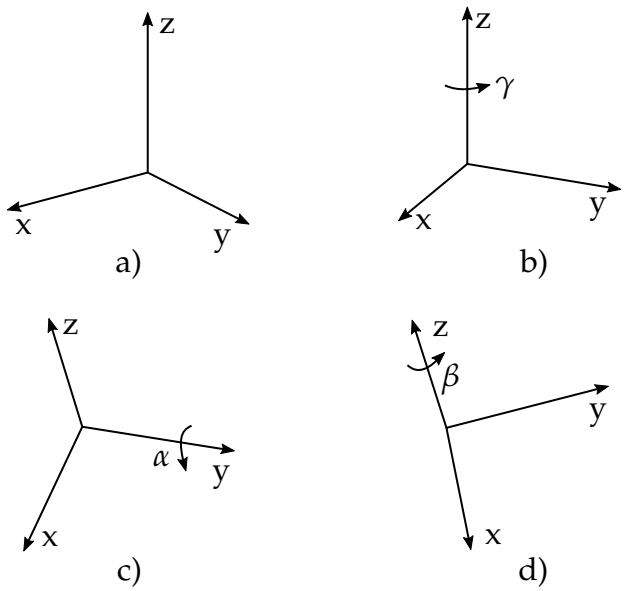


Figure 10-30. The angles γ, α, and β align rigid body one with rigid body two for the FLEXION-TORSION option, using z-y-z Euler angles (in the order shown by the figure labels) to calculate the moments. An illustrative version of this figure is shown in Figure 10-31.

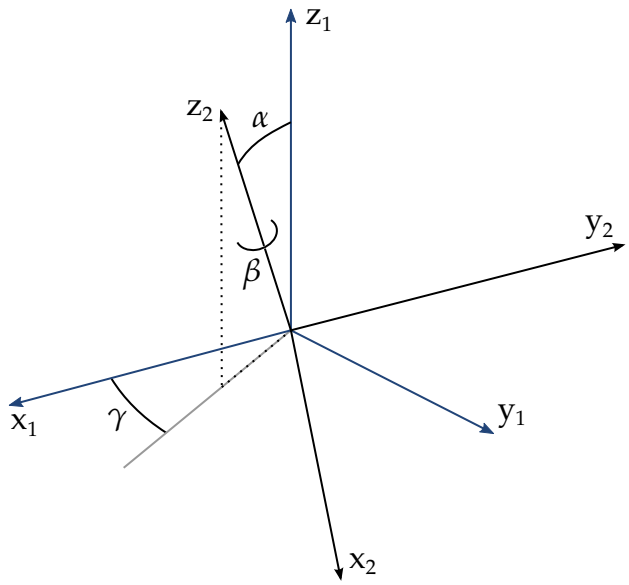


Figure 10-31. Flexion-torsion joint angles. If the initial positions of the local coordinate axes of the two rigid bodies connected by the joint do not coincide, the angles, α and γ, are initialized and torques will develop instantaneously based on the defined load curves. The angle β is also initialized but no torque will develop about the local axis on which β is measured. Rather, β will be measured relative to the computed offset.

Card 3 for FLEXION-TORSION option.

Card 2a.2	1	2	3	4	5	6	7	8
Variable	ESAL	FMAL	ESBT	FMBT				
Type	F	F/I	F	F/I				
Default	0.0	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
ESAL	Elastic stiffness per unit radian for friction and stop angles for α rotation. See Figure 10-32. EQ.0.0: Friction and stop angles are inactive for α rotation.
FMAL	Frictional moment limiting value for α rotation. This option may also be thought of as an elastic-plastic spring. See Figure 10-32. EQ.0.0: Friction is inactive for α rotation. LT.0: -FMAL is the load curve or table ID defining the yield moment as a function of α rotation. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on Card 1.
ESBT	Elastic stiffness per unit radian for friction and stop angles for β twist. EQ.0.0: Friction and stop angles are inactive for β twist.
FMBT	Frictional moment limiting value for β twist. This option may also be thought of as an elastic-plastic spring. EQ.0.0: The friction is inactive for β twist. LT.0: -FMBT is the load curve or table ID defining the yield moment versus β rotation. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on Card 1.

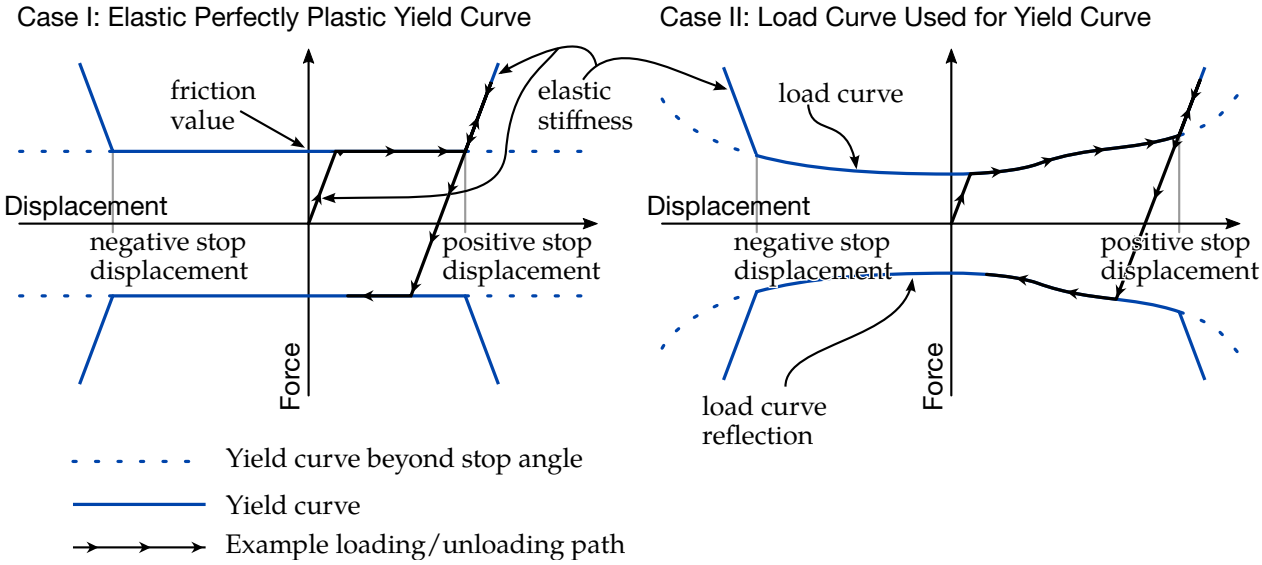


Figure 10-32. *Friction model.* The friction model is motivated by plasticity, and it is implemented for both rotational and translational joints. In the context of a rotational joint, the y-axis is to be interpreted as moment (rotational force) and the x-axis is to be interpreted as rotation. *Case I* (left) is activated by a positive friction value. *Case II* (right) is activated by a negative integer friction value, the absolute value of which specifies a load curve. *See the friction, elastic, and stop angle/displacement parameters from the input cards* (FM[var], ES[var], NSA[var], PSA[var]).

Card 4 for FLEXION-TORSION option.

Card 2a.3	1	2	3	4	5	6	7	8
Variable	SAAL	NSABT	PSABT					
Type	F	F	F					
Default	0.0	0.0	0.0					

VARIABLE	DESCRIPTION
SAAL	Stop angle in degrees for α rotation where $0.0 \leq \alpha \leq \pi$. Ignored if zero. See Figure 10-32.
NSABT	Stop angle in degrees for negative β rotation. Ignored if zero.
PSABT	Stop angle in degrees for positive β rotation. Ignored if zero.

Remarks:

This option simulates the flexion-torsion behavior of a joint in a slightly different fashion than with the generalized joint option.

After the stop angles are reached, the torques increase linearly to resist further angular motion using the stiffness values on Card 2a.2. If the stiffness value is too low or zero, the stop will be violated.

The moment resultants generated from the moment as a function of rotation curve, damping moment as a function of rate-of-rotation curve, and friction are evaluated independently and are added together.

Generalized Joint Stiffness Cards:

Card 2 for GENERALIZED stiffness option.

Card 2b.1	1	2	3	4	5	6	7	8
Variable	LCIDPH	LCIDT	LCIDPS	DLCIDPH	DLCIDT	DLCIDPS		
Type	I	I	I	I	I	I		
Default	0.0	0.0	0.0	0	0	0		

VARIABLE	DESCRIPTION
LCIDPH	Load curve ID for ϕ-moment as a function of rotation in radians. See Figure 10-33. See *DEFINE_CURVE. EQ.0: The applied moment is set to 0.0.
LCIDT	Load curve ID for θ-moment as a function of rotation in radians. See *DEFINE_CURVE. EQ.0: The applied moment is set to 0.0.
LCIDPS	Load curve ID for ψ-moment as a function of rotation in radians. See *DEFINE_CURVE. EQ.0: The applied moment is set to 0.0.
DLCIDPH	Load curve ID for ϕ-damping moment as a function of rate of rotation in radians per unit time. See *DEFINE_CURVE. EQ.0: Damping is not considered.
DLCIDT	Load curve ID for θ-damping moment as a function of rate of rotation in radians per unit time. See *DEFINE_CURVE. EQ.0: Damping is not considered.
DLCIDPS	Load curve ID for ψ-damping torque as a function of rate of rotation in radians per unit time. See *DEFINE_CURVE. EQ.0: Damping is not considered.

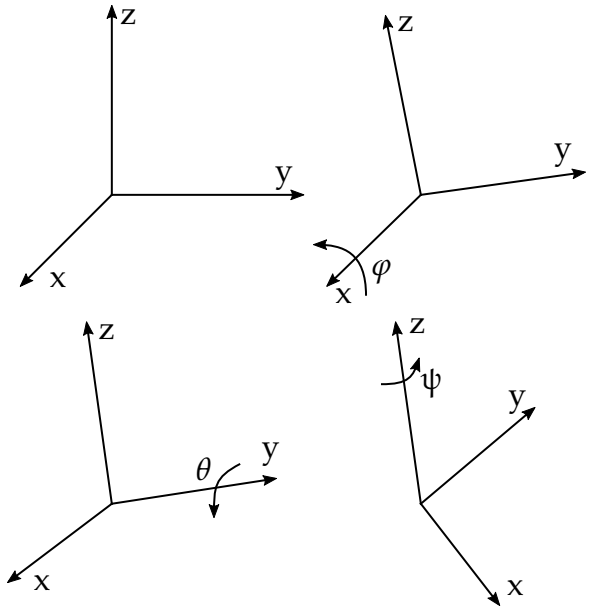


Figure 10-33. Definition of angles for the GENERALIZED joint stiffness.

Card 3 for GENERALIZED stiffness option.

Card 2b.2	1	2	3	4	5	6	7	8
Variable	ESPH	FMPH	EST	FMT	ESPS	FMPS		
Type	F	F/I	F	F/I	F	F/I		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
ESPH	Elastic stiffness per unit radian for friction and stop angles for ϕ rotation. EQ.0.0: Friction and stop angles are inactive for ϕ rotation.
FMPH	Frictional moment limiting value for ϕ rotation. This option may also be thought of as an elastic-plastic spring. See Figure 10-32. EQ.0.0: Friction is inactive for ϕ rotation. LT.0: -FMPH is the load curve or table ID defining the yield moment as a function of ϕ rotation. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on Card 1.

VARIABLE	DESCRIPTION
EST	Elastic stiffness per unit radian for friction and stop angles for θ rotation. See Figure 10-32. EQ.0.0: Friction and stop angles are inactive for θ rotation.
FMT	Frictional moment limiting value for θ rotation. This option may also be thought of as an elastic-plastic spring. EQ.0.0: Friction is inactive for θ rotation. LT.0: -FMT is the load curve or table ID defining the yield moment as a function of θ rotation. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on Card 1.
ESPS	Elastic stiffness per unit radian for friction and stop angles for ψ rotation. EQ.0.0: Friction and stop angles are inactive for ψ rotation.
FMPS	Frictional moment limiting value for ψ rotation. This option may also be thought of as an elastic-plastic spring. EQ.0.0: Friction is inactive for ψ rotation. LT.0: -FMPS is the load curve or table ID defining the yield moment as a function of ψ rotation. A table permits the moment to also be a function of the joint reaction force and requires the specification of JID on Card 1.

Card 4 for GENERALIZED stiffness option.

Card 2b.3	1	2	3	4	5	6	7	8
Variable	NSAPH	PSAPH	NSAT	PSAT	NSAPS	PSAPS		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
NSAPH	Stop angle in degrees for negative ϕ rotation. See Figure 10-32. Ignored if zero.

VARIABLE	DESCRIPTION
PSAPH	Stop angle in degrees for positive ϕ rotation. Ignored if zero.
NSAT	Stop angle in degrees for negative θ rotation. Ignored if zero.
PSAT	Stop angle in degrees for positive θ rotation. Ignored if zero.
NSAPS	Stop angle in degrees for negative ψ rotation. Ignored if zero.
PSAPS	Stop angle in degrees for positive ψ rotation. Ignored if zero.

Remarks:

After the stop angles are reached, the torques increase linearly to resist further angular motion using the stiffness values on Card 2b.2. Reasonable stiffness values must be chosen. If the stiffness values are too low or zero, the stop will be violated.

If the initial local coordinate axes do not coincide, the angles, ϕ, θ, and ψ, will be initialized and torques will develop instantaneously based on the defined moment as a function of rotation curves.

There are two methods available to calculate the rotation angles between the coordinate systems. For more information, see the JNTF parameter on *CONTROL_RIGID.

Example:

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a joint stiffness for the revolute joint described in
$    *CONSTRAINED_JOINT_REVOLUTE
$
$  Attributes of the joint stiffness:
$    - Used for defining a stop angle of 30 degrees rotation
$        (i.e., the joint allows a positive rotation of 30 degrees and
$           then imparts an elastic stiffness to prevent further rotation)
$    - Define between rigid body A (part 1) and rigid body B (part 2)
$    - Define a local coordinate system such that local x corresponds
$         to the joint's axis of revolution and the angle phi is the angle
$         of rotation about that axis.
$    - The elastic stiffness per unit radian for the stop angle is 100.
$    - Variables left blank are not used during the simulation.
$
*CONSTRAINED_JOINT_STIFFNESS_GENERALIZED
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     jsid      pida      pidb      cida      cidb
         1         1         2         5         5
```

```
$
$   lcidph     lcidt    lcidps   dlcidph    dlcidt   dlcidps

$
$     esph      fmps       est       fmt      esps      fmps
     100.0
$
$    nsaph     psaph      nsat      psat     nsaps     psaps
               30.0
$
$
*DEFINE_COORDINATE_NODES
$      cid        n1        n2        n3
         5         1         2         3
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

Translational Joint Stiffness Cards:

Card 2 for TRANSLATIONAL stiffness option.

Card 2c.1	1	2	3	4	5	6	7	8
Variable	LCIDX	LCIDY	LCIDZ	DLCIDX	DLCIDY	DLCIDZ		
Type	I	I	I	I	I	I		
Default	0.0	0.0	0.0	0	0	0		

VARIABLE	DESCRIPTION
LCIDX	Load curve ID for x-force as a function of x-distance between the origins of CIDA and CIDB based on the x-direction of CIDB. See *DEFINE_CURVE. EQ.0: The applied force is set to 0.0.
LCIDY	Load curve ID for y-force as a function of y-distance between the origins of CIDA and CIDB based on the y-direction of CIDB. See *DEFINE_CURVE. EQ.0: The applied force is set to 0.0.
LCIDZ	Load curve ID for z-force as a function of z-distance between the origins of CIDA and CIDB based on the z-direction of CIDB. See *DEFINE_CURVE. EQ.0: The applied force is set to 0.0.
DLCIDX	Load curve ID for x-damping force as a function of rate of x-translational displacement per unit time between the origins of CIDA and CIDB based on the x-direction of CIDB. See *DEFINE_CURVE. EQ.0: Damping is not considered.
DLCIDY	Load curve ID for y-damping force as a function of rate of y-translational displacement per unit time between the origins of CIDA and CIDB based on the y-direction of CIDB. See *DEFINE_CURVE. EQ.0: Damping is not considered.

VARIABLE	DESCRIPTION
DLCIDZ	Load curve ID for z-damping force as a function of rate of z-translational displacement per unit time between the origins of CIDA and CIDB based on the z-direction of CIDB. See *DEFINE_CURVE. EQ.0: Damping is not considered.

Card 3 TRANSLATIONAL stiffness option.

Card 2c.2	1	2	3	4	5	6	7	8
Variable	ESX	FFX	ESY	FFY	ESZ	FFZ		
Type	F	F/I	F	F/I	F	F/I		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
ESX	Elastic stiffness for friction and stop displacement for x-translation. See Figure 10-32. EQ.0.0: Friction and stop angles are inactive for x-translation.
FFX	Frictional force limiting value for x-translation. This option may also be thought of as an elastic-plastic spring. See Figure 10-32. EQ.0.0: Friction is inactive for x-translation. LT.0: -FFX is the load curve ID defining the yield force as a function x-translation.
ESY	Elastic stiffness for friction and stop displacement for y-translation. EQ.0.0: Friction and stop angles are inactive for y-translation.
FFY	Frictional force limiting value for y-translation. This option may also be thought of as an elastic-plastic spring. EQ.0.0: Friction is inactive for y-translation. LT.0: -FFY is the load curve ID defining the yield force as a function of y-translation.

VARIABLE	DESCRIPTION
ESZ	Elastic stiffness for friction and stop displacement for z-translation. EQ.0.0: Friction and stop angles are inactive for z-translation.
FFZ	Frictional force limiting value for z-translation. This option may also be thought of as an elastic-plastic spring. EQ.0.0: Friction is inactive for z-translation. LT.0: -FFZ is the load curve ID defining the yield force as a function of z-translation.

Card 4 for TRANSLATIONAL stiffness option.

Card 2c.3	1	2	3	4	5	6	7	8
Variable	NSDX	PSDX	NSDY	PSDY	NSDZ	PSDZ	FS	FD
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
NSDX	Stop displacement for negative x-translation. Ignored if zero. See Figure 10-32.
PSDX	Stop displacement for positive x-translation. Ignored if zero.
NSDY	Stop displacement for negative y-translation. Ignored if zero.
PSDY	Stop displacement for positive y-translation. Ignored if zero.
NSDZ	Stop displacement for negative z-translation. Ignored if zero.
PSDZ	Stop displacement for positive z-translation. Ignored if zero.
FS	Static friction coefficient
FD	Dynamic friction coefficient

Remarks:

After the stop displacements are reached, the force increases linearly to resist further translational motion using the stiffness values on Card 2c.2. Reasonable stiffness values must be chosen. If the stiffness values are too low or zero, the stop will be violated.

Cylindrical Joint Stiffness Cards:

Card 2 for CYLINDRICAL stiffness option.

Card 2d.1	1	2	3	4	5	6	7	8
Variable	LCIDR		LCIDZ	DLCIDR	DLCIDP	DLCIDZ	LCIDT	DLCIDT
Type	I		I	I	I	I	I	I
Default	0		0	0	0	0	0	0

VARIABLE	DESCRIPTION
LCIDR	Load curve ID for *r*-force as a function of *r*-distance between the origins of CIDA and CIDB. See *DEFINE_CURVE. EQ.0: The applied force is set to 0.0.
LCIDZ	Load curve ID for *z*-force as a function of *z*-distance between the origins of CIDA and CIDB. See *DEFINE_CURVE. EQ.0: The applied force is set to 0.0.
DLCIDR	Load curve or table ID for *r*-damping force as a function of rate of *r*-distance per unit time and optionally *r*-distance (if table) between the origins of CIDA and CIDB. See *DEFINE_CURVE or *DEFINE_TABLE. EQ.0: Damping is not considered.
DLCIDP	Load curve or table ID for *p*-damping force as a function of rate of *p*-distance per unit time and optionally *r*-distance (if table) between the origins of CIDA and CIDB. See *DEFINE_CURVE or *DEFINE_TABLE. EQ.0: Damping is not considered.
DLCIDZ	Load curve or table ID for *z*-damping force as a function of rate of *z*-distance per unit time and optionally *r*-distance (if table) between the origins of CIDA and CIDB. See *DEFINE_CURVE or *DEFINE_TABLE. EQ.0: Damping is not considered.

VARIABLE	DESCRIPTION
LCIDT	Load curve ID for θ-moment as a function of angle θ between the z-directions of CIDA and CIDB. See *DEFINE_CURVE. EQ.0: The applied moment is set to 0.0.
DLCIDT	Load curve ID for θ-moment as a function of rate of angle θ between the z-directions of CIDA and CIDB. See *DEFINE_CURVE. EQ.0: The applied moment is set to 0.0.

Card 3 CYLINDRICAL stiffness option.

Card 2d.2	1	2	3	4	5	6	7	8
Variable	ESR	FFR			ESZ	FFZ	RAD1	RAD2
Type	F	F/I			F	F/I	F	F
Default	0.0	0.0			0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
ESR	Elastic stiffness for friction and stop displacement for r-translation. See Figure 10-32. EQ.0.0: Friction and stop angles are inactive for r-translation.
FFR	Frictional force limiting value for r-translation. This option may also be thought of as an elastic-plastic spring. See Figure 10-32. EQ.0.0: Friction is inactive for r-translation. LT.0: -FFR is the load curve ID defining the yield force as a function r-translation.
ESZ	Elastic stiffness for friction and stop displacement for z-translation. EQ.0.0: Friction and stop angles are inactive for z-translation.
FFZ	Frictional force limiting value for z-translation. This option may also be thought of as an elastic-plastic spring. EQ.0.0: Friction is inactive for z-translation.

VARIABLE	DESCRIPTION
	LT.0: -FFZ is the load curve ID defining the yield force as a function of z-translation.
RAD1	Radius of pin, must be strictly positive.
RAD2	Radius of hole, must be strictly larger than RAD1 to model play in the connection.

Card 4 for CYLINDRICAL stiffness option.

Card 2d.3	1	2	3	4	5	6	7	8
Variable		PSDR			NSDZ	PSDZ	FS	FD
Type		F			F	F	F	F
Default		0.0			0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
PSDR	Stop displacement for r-translation. Ignored if zero.
NSDZ	Stop displacement for negative z-translation. Ignored if zero.
PSDZ	Stop displacement for positive z-translation. Ignored if zero.
FS	Static friction coefficient
FD	Dynamic friction coefficient

Remarks:

As shown in Figure 10-34, the cylindrical joint stiffness models a pin in a hole with a gap. The pin has a radius r_1 (RAD1) while the hole has a radius r_2 (RAD2), with $0 < r_1 < r_2$, so the gap is at most $g = r_2 - r_1$. This joint stiffness needs no associated joint, but all directions can be constrained by appropriate usage of this keyword.

Let $\mathbf{x}_i$ be the origin of coordinate system associated with body i, $i = 1, 2$, defined by *DEFINE_COORDINATE_NODES. Then the radial distance to be used in the curves/tables is defined as

$$r = \left|\left(\mathbf{I} - \mathbf{z}\mathbf{z}^T\right)(\mathbf{x}_2 - \mathbf{x}_1)\right| \ ,$$

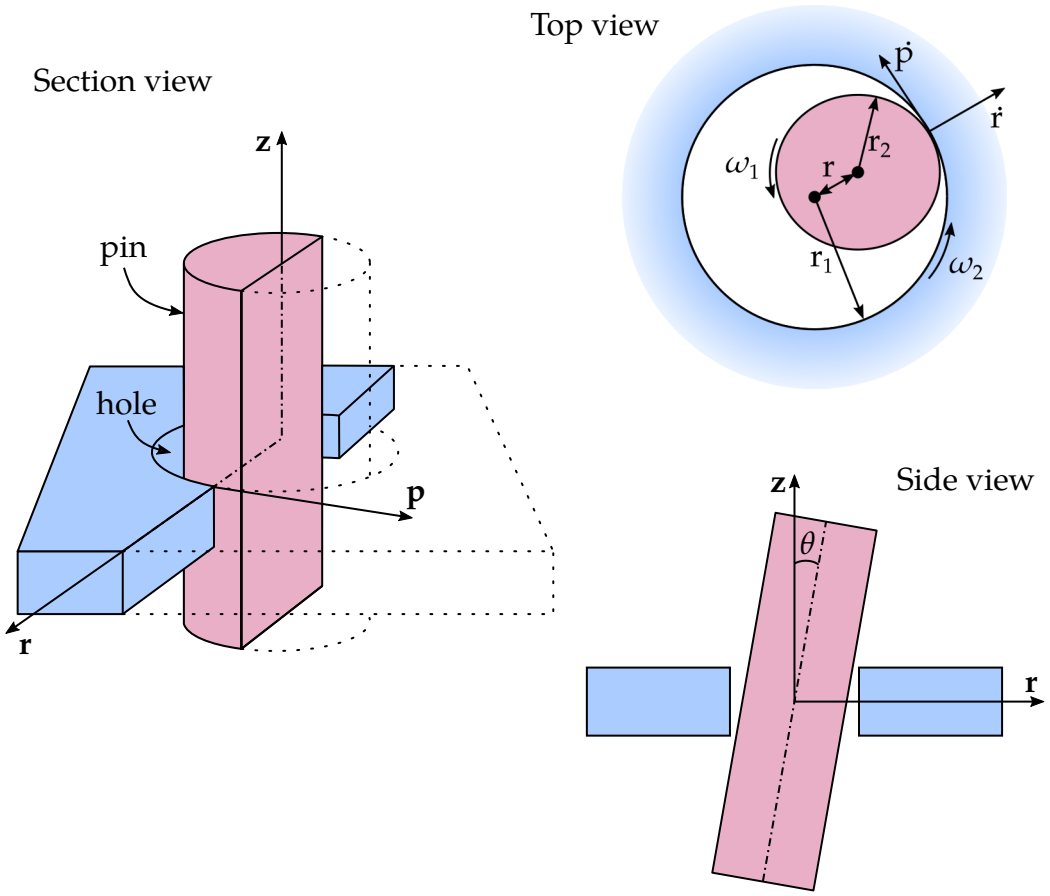


Figure 10-34. Cylindrical joint stiffness models a pin in a hole.

and the radial direction is defined as

$$\mathbf{r} = \frac{\left(\mathbf{I} - \mathbf{z}\mathbf{z}^T\right)(\mathbf{x}_2 - \mathbf{x}_1)}{r} \ .$$

See LCIDR, ESR, FFR and PSDR. Here $\mathbf{I}$ is the unit matrix, and $\mathbf{z}$ is the local z-direction of the coordinate system in body 2. We basically assign a cylindrical coordinate system $\{\mathbf{r}, \mathbf{p}, \mathbf{z}\}$ with the z-direction co-linear with the cartesian z-direction in body 2. The radial distance is thus the (non-directional) distance between body 1 and body 2 in the local xy-plane of body 2.

Likewise, we define the z-distance to be used in the curves/tables as

$$z = \left|\mathbf{z}^T(\mathbf{x}_2 - \mathbf{x}_1)\right| \ ,$$

which is the *directional* distance from body 1 to body 2 in the z-direction. See LCIDZ, ESZ, FFZ, NSDZ and PSDZ.

From this, the rate of r and z are obtained by taking the time derivative quantities, $\dot{r}$ and $\dot{z}$, that are also used in curves/tables for damping purposes; see DLCIDR and DLCIDZ. If these are defined by tables, the second argument is the radial distance, r, which can be used to not apply any damping when the gap is positive. For numerical purposes, and

for the sake of introducing a hysteresis, $\dot{r}$ is always positive, so DLCIDR only needs to be defined for positive $\dot{r}$.

Last, we define **p** as the cross production of **z** and **r**, that is,

$$\mathbf{p} = \mathbf{z} \times \mathbf{r}$$

This is the tangential direction along the circumference of the pin or the hole, depending on how you observe it. The tangential velocity is defined as

$$\dot{p} = \mathbf{p}^T(\dot{\mathbf{x}}_2 - (r_1 + r)\boldsymbol{\omega}_2 \times \mathbf{r} - \dot{\mathbf{x}}_1 + r_1\boldsymbol{\omega}_1 \times \mathbf{r}) \ ,$$

where $\boldsymbol{\omega}_1$ and $\boldsymbol{\omega}_2$ are the rotational velocities of bodies 1 and 2, respectively. The interpretation of this is the relative velocity of the contact point between the hole (body 2) and the pin (body 1) in direction **p** when the gap is zero. This can be used to define damping in the p-direction (see DLCIDP) as a type of simplified friction model. Also, here a table can be used with r as the second argument.

To maintain co-linearity of the z-directions of the pin and hole, the angle θ is defined as illustrated in the picture. This quantity, and its time derivative $\dot{\theta}$, can be used to enforce this co-linearity; see LCIDT and DLCIDT. For numerical purposes, and for the sake of introducing hysteresis, $\dot{\theta}$ is always positive so DLCIDT only needs to be defined for positive $\dot{\theta}$.

*CONSTRAINED_JOINT_USER_FORCE

Purpose: Define input data for a user subroutine to generate force resultants as a function of time and joint motion.

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

User Subroutine Constants Cards. Define up to 48 optional user constants (6 cards total) for the user subroutine. This input is terminated after 48 constants are defined or when the next keyword ("*") card is encountered.

Card 2	1	2	3	4	5	6	7	8
Variable	CONST1	CONST2	CONST3	CONST4	CONST5	CONST6	CONST7	CONST8
Type	F	F	F	F	F	I	I	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
FID	Joint user force ID.
JID	Joint ID for which this user force input applies.
NHISV	Number of history variables required for this definition. An array NHISV long is allocated and passed into the user subroutine. This array is updated in the user subroutine.
CONST*n*	A constant which is passed into the user subroutine.

*CONSTRAINED_LAGRANGE_IN_SOLID_{OPTION1}_{OPTION2}

Purpose: This command provides the coupling mechanism for modeling Fluid-Structure Interaction (FSI). The structure can be constructed from Lagrangian shell and/or solid entities. The multi-material fluids are modeled by ALE formulation.

Available options for *OPTION1* include:

<BLANK>

EDGE

This option may be used to allow the coupling between the edge of a shell part or part set and one or more ALE multi-material groups (AMMG). It accounts for the shell thickness in the coupling calculation. The edge thickness is the same as the shell thickness. This option only works when the Lagrangian set is defined as a part or a part set ID. It will not work for a segment set. One application of this option is a simulation of a Lagrangian blade (a shell part) cutting through some ALE material.

Available options for *OPTION2* include:

<BLANK>

TITLE

To define a coupling (card) ID number and title for each coupling card. If a title is not defined, LS-DYNA will automatically create an internal title for this coupling definition. The ID number can be used to delete coupling action in a restart input deck using the *DELETE_FSI card.

Card Summary:

Card ID. This card is included if and only if the TITLE keyword option is used.

COUPID	TITLE

Card 1. This card is required.

LSTRSID	ALESID	LSTRSTYP	ALESTYP	NQUAD	CTYPE	DIREC	MCOUP

Card 2. This card is required.

START	END	PFAC	FRIC	FRCMIN	NORM	NORMTYP	DAMP

Card 3. This card is required.

K	HMIN	HMAX	ILEAK	PLEAK	LCIDPOR	NVENT	IBLOCK

Card 4. This card is required for CTYPE = 11 and 12. Otherwise it is optional. If Card 5 is included, then this card must be included.

IBOXID	IPENCHK	INTFORC	IALESOF	LAGMUL	PFACMM	THKF	

Card 5. This card is required for CTYPE = 11 and 12. Otherwise it is optional. If this card is included, then Card 4 must be included.

A1	B1	A2	B2	A3	B3		POREINI

Card 6. NVENT of this card must be defined (one card for defining each vent hole). The last NVENT cards for *CONSTRAINED_LAGRANGE_IN_SOLID are assumed to be Card(s) 6; therefore, Cards 4 and 5 are not mandatory when Card(s) 6 are defined.

VENTSID	VENTYP	VTCOEF	POPPRES	COEFLC			

Data Card Definitions:

Title Card. Additional card for the TITLE keyword option.

Card ID	1	2	3	4	5	6	7	8
Variable	COUPID	TITLE						
Type	I	A70						

VARIABLE	DESCRIPTION
COUPID	Coupling (card) ID number. If not defined, LS-DYNA will assign an internal coupling ID based on the order of appearance in the input deck.
TITLE	A description of this coupling definition (A70).

Card 1	1	2	3	4	5	6	7	8
Variable	LSTRSID	ALESID	LSTRSTYP	ALESTYP	NQUAD	CTYPE	DIREC	MCOUP
Type	I	I	I	I	I	I	I	I
Default	none	none	0	0	0	2	1	0

VARIABLE	DESCRIPTION
LSTRSID	Set ID defining a part, part set, or segment set ID of the Lagrangian structure (see *PART, *SET_PART or *SET_SEGMENT). See Remark 1.
ALESID	Set ID defining a part or part set ID of the ALE solid elements (see *PART or *SET_PART). See Remark 1.
LSTRSTYP	LSTRSID set type (see Remark 1): EQ.0: Part set ID (PSID) EQ.1: Part ID (PID) EQ.2: Segment set ID (SSID)
ALESTYP	ALESID set type (see Remark 1): EQ.0: Part set ID (PSID) EQ.1: Part ID (PID)
NQUAD	Number of coupling points distributed over each coupled Lagrangian surface segment. See Remark 2. EQ.0: NQUAD will be set by default to 2, GT.0: An NQUAD × NQUAD coupling points distribution over each Lagrangian segment is defined, LT.0: NQUAD is reset to a positive value. Coupling at nodes is obsolete.
CTYPE	Fluid-Structure coupling method. CTYPEs 1 and 2 are not supported in MPP. EQ.1: Constrained acceleration EQ.2: Constrained acceleration and velocity (default, see Remark 3) EQ.3: Constrained acceleration and velocity, normal direction only EQ.4: Penalty coupling for shell and solid elements (without erosion) **NOTE:** For RIGID Lagrangian structure PARTS a penalty coupling method (CTYPE = 4) must be used.

VARIABLE	DESCRIPTION
	EQ.5: Penalty coupling allowing erosion in the Lagrangian entities EQ.6: Penalty coupling designed for airbag modeling which automatically controls the DIREC parameter internally. It is equivalent to setting {CTYPE = 4; DIREC = 1} for the unfolded region; and {CTYPE = 4; DIREC = 2} for the folded region. For both cases: {ILEAK = 2; FRCMIN = 0.3}. EQ.11: Coupling designed to couple Lagrangian porous shell to ALE material. When this option is used, THKF, the 7th column parameter of optional Card 4a and the first 2 parameters of optional Card 4b must be defined. See *LOAD_BODY_POROUS and Remark 14 below. EQ.12: Coupling designed to couple Lagrangian porous solid to ALE material. When this option is used, Ai & Bi parameters of optional Card 4b must be defined (Card 4a must be defined but can be blank). See *LOAD_BODY_POROUS and Remark 15 below.
DIREC	For CTYPE = 4 or 5 Coupling direction (see Remark 4): EQ.1: Normal direction, compression, and tension (default) EQ.2: Normal direction, compression only EQ.3: All directions For CTYPE = 12 Flag to activate an element coordinate system: EQ.0: The forces are applied in the global directions. EQ.1: The forces are applied in a local system attached to the Lagrangian solid. The system is consistent with AOPT = 1 in *LOAD_BODY_POROUS. (See Remark 15).
MCOUP	For CTYPE = 4, 5, 6, 11, or 12 Multi-material option (see Remark 5): EQ.0: Couple with all multi-material groups EQ.1: Couple with material with highest density

VARIABLE	DESCRIPTION
	LT.0: MCOUP must be an integer. -MCOUP refers to a set ID of an ALE multi-material group. See *SET_MULTI-MATERIAL_GROUP.

Card 2	1	2	3	4	5	6	7	8
Variable	START	END	PFAC	FRIC	FRCMIN	NORM	NORMTYP	DAMP
Type	F	F	F	F	F	I	I	F
Default	0.0	10^{10}	0.1	0.0	0.5	0	0	0.0

VARIABLE	DESCRIPTION
START	Start time for coupling.
END	End time for coupling. If less than zero, coupling will be turned off during dynamic relaxation. After dynamic relaxation phase is finished, the absolute value will be taken as end time.
PFAC	For CTYPE = 4,5 or 6 Penalty factor. PFAC is a scale factor for scaling the estimated stiffness of the interacting (coupling) system. It is used to compute the coupling forces to be distributed on the Lagrangian and ALE parts GT.0: Fraction of estimated critical stiffness. LT.0: PFAC must be an integer, and -PFAC is a load curve ID. The curve defines the coupling pressure on the *y*-axis as a function of the penetration along the *x*-axis. See Remark 6. See "How to correct leakage." For CTYPE = 11 or 12 Time step factor
FRIC	Coefficient of friction (used with DIREC = 1 and 2 only).
FRCMIN	Minimum volume fraction of a coupled ALE multi-material group (AMMG) or fluid in a multi-material ALE element to activate coupling. Default value is 0.5. Reducing FRCMIN (typically, between 0.1 and 0.3) would turn on coupling earlier to prevent leakage in high velocity impact cases.

VARIABLE	DESCRIPTION
NORM	A Lagrangian segment will couple to fluid on only one side of the segment. NORM determines which side. See Remark 7. EQ.0: Couple to fluid (AMMG) on head-side of Lagrangian segment normal vector. EQ.1: Couple to fluid (AMMG) on tail-side of Lagrangian segment normal vector.
NORMTYP	Penalty coupling spring (or force) direction (DIREC = 1 or 2): EQ.0: Normal vectors are interpolated from nodal normals. (default). EQ.1: Normal vectors are interpolated from segment normals. This is sometimes a little more robust for sharp Lagrangian corners, and folds.
DAMP	Damping factor for penalty coupling. This is a coupling-damping scaling factor. Typically, it may be between 0.0 and 1.0 (see Remark 8).

Card 3	1	2	3	4	5	6	7	8
Variable	K	HMIN	HMAX	ILEAK	PLEAK	LCIDPOR	NVENT	IBLOCK
Type	F	F	F	I	F	I	I	I
Default	0.0	none	none	0	0.1	0	0	0

VARIABLE	DESCRIPTION
K	Thermal conductivity of a virtual fluid between the Lagrangian structure surface and the ALE material. See Remark 9.
HMIN	The absolute value is minimum air gap in heat transfer, $h_{\min}$ (See Remark 9). LT.0.0: Turn on constraint based thermal nodal coupling between LAG structure and ALE fluids. GE.0.0: Minimum air gap. If zero, default to 10^{-6}.

VARIABLE	DESCRIPTION
HMAX	Maximum air gap in heat transfer, h_{max}. There is no heat transfer above this value. See Remark 9.
ILEAK	Coupling leakage control flag (Remark 10): EQ.0: None (default), EQ.1: Weak, leakage control is turned off if penetrating volume fraction > FRCMIN + 0.2 EQ.2: Strong, with improved energy consideration. Leakage control is turned off if penetrating volume fraction > FRCMIN + 0.4
PLEAK	Leakage control penalty factor, 0 < PLEAK < 0.2 is recommended. This factor influences the additional coupling force magnitude to prevent leakage. It is conceptually similar to PFAC. Almost always, the default value (0.1) is adequate.
LCIDPOR	If this is a positive integer, a load curve ID (LCID) defining porous flow through coupling segment: Abscissa = x = ($P_{up} - P_{down}$) Ordinate = y = relative porous fluid velocity Where P_{up} and P_{down} are, respectively, the upstream and downstream pressures across of the porous coupling segment. The relative porous velocity is the ALE fluid velocity relative to the moving Lagrangian segment. This experimental data curve must be provided by the user. If LCIDPOR is a negative integer: The porous flow is controlled by the parameters FLC, FAC, ELA under *MAT_FABRIC card. **CAUTION:** The pressure under the FAC load curve is "absolute upstream pressure" (see Remark 11). Abscissa = x = absolute upstream pressure Ordinate = y = relative porous fluid velocity For CTYPE = 11 or CTYPE = 12 and POREINI = 0.0: LT.0: The load curve |LCIDPOR| is a factor versus time of the porous force computed by the Ergun equation. See Remark 14.

VARIABLE	DESCRIPTION
	GT.0: The load curve LCIDPOR is a porous force versus velocity, which replaces the force computed by the Ergun equation. See Remark 14.
	For CTYPE = 11 or CTYPE = 12 and POREINI > 0.0:
	NE.0: The load curve |LCIDPOR| is a factor versus time of the porous force computed by the Ergun equation. See Remark 14.
NVENT	The number of vent surface areas to be defined. Each venting flow surface is represented by one or more Lagrangian segments (or surfaces). For airbag applications, this may be referred to as "isentropic" venting where the isentropic flow equation is used to compute the mass flow rate based on the ratio of the upstream and downstream pressures P_{up}/P_{down}. For each of the NVENT vent surfaces, an additional card of format 6 defining the geometrical and flow properties for each vent surface will be read in. The vented mass will simply be deleted from the system and cannot be visualized as in the case of physical venting. See Remark 12.
IBLOCK	Flag to control the venting (or porous) flow blockage due to Lagrangian contact during ALE computation. EQ.0: Off EQ.1: On The venting definition is defined in this command. However, the venting flow may be defined through either the LCIDPOR parameter in this command or via the *MAT_FABRIC parameters (FLC, FAC, ELA). However, note that FVOPT (blocking) parameter under *MAT_FABRIC applies only to CV computation.

Card 4. This card is required for CTYPE 11 & 12 but is otherwise optional.

Card 4	1	2	3	4	5	6	7	8
Variable	IBOXID	IPENCHK	INTFORC	IALESOF	LAGMUL	PFACMM	THKF	
Type	I	I	I	I	F	I	F	
Default	0	0	0	0	0.0	0	0.0	

VARIABLE **DESCRIPTION**

IBOXID A box ID defining a box region in space in which ALE coupling is activated.

GT.0: At time = 0.0, the Lagrangian segments inside this box are remembered. In subsequent coupling computation steps, there is no need to search for the Lagrangian segments again.

LT.0: At each FSI bucket sort, the Lagrangian segments inside this box are marked as active coupling segments. This makes the coupling operate more efficiently when a structured mesh is approaching ALE domain, such as hydroplaning and bird strike.

IPENCHK Only for CTYPE = 4

Initial penetration check flag (See Remark 13):

EQ.0: Do not check for initial penetration.

EQ.1: Check and save initial ALE material penetration across a Lagrangian surface (d_0), but do not activate coupling at $t = 0$. In subsequent steps ($t > 0$) the actual penetration is computed as follows:

$$\underbrace{\text{Actual Penetration}}_{d_a} = \underbrace{\text{Total Penetration}}_{d_T} - \underbrace{\text{Initial Penetration}}_{d_0}$$

INTFORC A flag to turn on or off the output of ALE coupling pressure and forces on the Lagrangian segments (or surfaces).

EQ.0: Off

EQ.1: On

VARIABLE	DESCRIPTION
	Note that the coupling pressures and forces are computed based on the coupling stiffness response to the ALE fluid penetration. When INFORC = 1 and a *DATABASE_BINARY_FSIFOR (DBF) card is defined, LS-DYNA writes out the segment coupling pressure and forces to the binary interface force file for contour plotting. This interface force file must be given a name on the execution line, for example: ls-dyna i=**inputfilename.k** … h=**interfaceforcefilename** The time interval between output is defined by "dt" in the DBF card. To plot the binary data in this file: ls-prepost **interfaceforcefilename**
IALESOF	An integer flag to turn ON/OFF a supplemental Lagrange multiplier FSI constraint which provides a coupling force in addition to the basic penalty coupling contribution. This is a hybrid coupling method. EQ.0: OFF (default). EQ.1: Turn ON the hybrid Lagrange-multiplier method. LAGMUL multiplier factor is read.
LAGMUL	A Lagrange multiplier factor with a range between 0.0 and 0.05 may be defined. A typical value may be 0.01. This should never be greater than 0.1. EQ.0: OFF (default). GT.0: Turn ON the Lagrange-multiplier method and use LAGMUL as a coefficient for scaling the penalty factor.
PFACMM	Mass-based penalty stiffness factor computational options. This works in conjunction with PFAC = constant (not a load curve). The coupling penalty stiffness (CPS) is computed based on an estimated effective coupling mass. EQ.0: CPS $\propto$ PFAC $\times \min(m_{\text{Lagrangian}}, m_{\text{ALE}})$, default. EQ.1: CPS $\propto$ PFAC $\times \max(m_{\text{Lagrangian}}, m_{\text{ALE}})$. EQ.2: CPS $\propto$ PFAC $\times \sqrt{m_{\text{Lagrangian}} m_{\text{ALE}}}$, geometric-mean of the masses.

VARIABLE	DESCRIPTION
	EQ.3: CPS $\propto$ PFAC $\times K_{\text{Lagrangian}}$ where K is the bulk modulus of the Lagrangian part
THKF	For all CTYPE choices except 11: A flag to account for the coupling thickness of the Lagrangian shell part. LT.0: Use positive value of |THKF| for coupling segment thickness. EQ.0: Do not consider coupling segment thickness. GT.0: Coupling segment thickness scale factor. For CTYPE = 11: This thickness is required for volume calculation. GT.0: (Fabric) Thickness scale factor. The base shell thickness is taken from the *PART definition. LT.0: User-defined (Fabric) thickness. The fabric thickness is set to |THKF|.

Porous Coupling. This card applies only to CTYPE 11 & 12. If this card is defined, Card 4 must be defined before this card.

Card 5	1	2	3	4	5	6	7	8
Variable	A1	B1	A2	B2	A3	B3		POREINI
Type	F	F	F	F	F	F		F
Default	0.0	0.0	0.0	0.0	0.0	0.0		0.0

VARIABLE	DESCRIPTION
A1	Viscous coefficient for the porous flow Ergun equation (see Remark 14). GT.0: For CTYPE = 11 $\text{A1} = A_n$ which is the coefficient for normal-to-segment direction.

VARIABLE	DESCRIPTION
	For CTYPE = 12 $A1 = A_x$ which is the coefficient for the x-direction in the coordinate system specified by DIREC. LT.0: If POREINI = 0.0, the coefficient is time dependent through a load curve ID defined by |A1|. If POREINI > 0.0, the coefficient is porosity dependent through a load curve ID defined by |A1|. The porosity is defined by PORE (see POREINI).
B1	Inertial coefficient for the porous flow Ergun equation (see Remark 14). GT.0: For CTYPE = 11 $B1 = B_n$ which is the coefficient for normal-to-segment direction. For CTYPE = 12 $B1 = B_x$ which is the coefficient for the x-direction of a coordinate system specified by DIREC. LT.0: If POREINI = 0.0, the coefficient is time dependent through a load curve id defined by |B1|. If POREINI > 0.0, the coefficient is porosity dependent through a load curve ID defined by |B1|. The porosity is defined by PORE (see POREINI).
A2	For CTYPE = 12 Viscous coefficient for the porous flow Ergun equation (see Remark 15). GT.0: Coefficient for the y-direction of a coordinate systems specified by DIREC. $A2 = A_y$ LT.0: If POREINI = 0.0, the coefficient is time dependent through a load curve ID defined by |A1|. If POREINI > 0.0, the coefficient is porosity dependent through a load curve ID defined by |A2|. The porosity is defined by PORE (see POREINI).

VARIABLE	DESCRIPTION
B2	For CTYPE = 12 Inertial coefficient for the porous flow Ergun equation (see Remark 15). GT.0: Coefficient for the y-direction of a coordinate system specified by DIREC. $$B2 = B_y$$ LT.0: If POREINI = 0.0 and B2 < 0, the coefficient is time dependent through a load curve ID defined by |B2|. If POREINI > 0.0 and B2 < 0, the coefficient is porosity dependent through a load curve ID defined by |B2|. The porosity is defined by PORE (see POREINI).
A3	For CTYPE = 12 Viscous coefficient for the porous flow Ergun equation (see Remark 15). GT.0: Coefficient for the z-direction of a coordinate system specified by DIREC. $$A3 = A_z$$ LT.0: If POREINI = 0.0 and A3 < 0, the coefficient is time dependent through a load curve ID defined by |A3|. If POREINI > 0.0 and A3 < 0, the coefficient is porosity dependent through a load curve ID defined by |A3|. The porosity is defined by PORE (see POREINI).
B3	For CTYPE = 12 Inertial coefficient for the porous flow Ergun equation (see Remark 15). GT.0: Coefficient for the z-direction of a coordinate system specified by DIREC. $$B3 = B_z$$ LT.0: If POREINI = 0.0 and B3 < 0, the coefficient is time dependent through a load curve ID defined by |B3|. If POREINI > 0.0 and B3 < 0, the coefficient is porosity dependent through a load curve ID defined by |B3|. The porosity is defined by PORE (see POREINI).

VARIABLE	DESCRIPTION
POREINI	For CTYPE = 11 or CTYPE = 12 POREINI is the initial volume of pores in an element. The current volume is $$\text{PORE} = \text{POREINI} \times \frac{v(t)}{v(t_0)}$$ where $v(t)$ and $v(t_0)$ are the current and initial element volumes respectively.

Venting Geometry Card(s). These card(s) set venting geometry. It is repeated NVENT times (one card for defining each vent hole). It is defined only if NVENT > 0 on Card 3. The last NVENT cards for *CONSTRAINED_LAGRANGE_IN_SOLID are taken to be Card(s) 6; therefore, Cards 4 and 5 are not mandatory when Card(s) 6 are defined.

Card 6	1	2	3	4	5	6	7	8
Variable	VENTSID	VENTYP	VTCOEF	POPPRES	COEFLC			
Type	I	I	I	F	I			
Default	0	0	0	0.0	0			

VARIABLE	DESCRIPTION
VENTSID	Set ID of the vent hole shape.
VENTYP	Vent surface area set ID type: EQ.0: Part set ID (PSID). EQ.1: Part ID (PID). EQ.2: Segment set ID (SGSID).
VTCOEF	Flow coefficient for each vent surface area.
POPPRES	Venting pop pressure limit. If the pressure inside the airbag is lower than this pressure, then nothing is vented. Only when the pressure inside the airbag is greater than POPPRES that venting can begin.
COEFLC	A time-dependent multiplier load curve for correcting the vent flow coefficient, with values ranging from 0.0 to 1.0.

Guidelines:

Due to the complexity of this card, some comments on simple, efficient and robust coupling approach are given here. These are not rigid guidelines, but simply some experience-based observations.

1. **Definition (fluid and structure).** The term *fluid*, in the Fluid-Structure Interaction (FSI), refers to materials with ALE element formulation, not indicating the phase (solid, liquid or gas) of those materials. In fact, solid, liquid, and gas can all be modeled by the ALE formulation. The term *structure* refers to materials with Lagrangian element formulation.

2. **Default values (CTYPE and MCOUP).** In general, penalty coupling (CTYPE 4 & 5) is recommended, and MCOUP = negative integer is a better choice to define a specific ALE multi-material group (AMMG) to be coupled to the Lagrangian surface. At the minimum, all parameters on Card 1 are to be specified, and the default values for most are good starting choices (except MCOUP).

3. **How to correct leakage.** If there is leakage, PFAC, FRCMIN, NORMTYPE, and ILEAK are the 4 parameters that can be adjusted.

   a) For hard structure (steel) and very compressible fluid (air), PFAC may be set to 0.1 (or higher). PFAC = constant value.

   b) Next, keeping PFAC = constant, set PFACMM = 3 (optional Card 4a). This option scales the penalty factor by the bulk modulus of the Lagrangian structure. This new approach has also shown to be effective for some airbag applications.

   c) The next approach may be switching from constant PFAC to a load curve approach, that is, PFAC = load curve and PFACMM = 0. By looking at the pressure in the system near leakage original location, we can get a feel for the pressure required to stop it.

   d) If leakage persists after some iterations on the coupling force controls, one can subsequently try to set ILEAK = 2 in combination with the other controls to prevent leakage.

   e) If the modifications fail to stop the leakage, maybe the meshes have to be redesigned to allow better interactions between the Lagrangian and Ale materials.

   In the example below, the underlined parameters are usually defined parameters. A full card definition is shown for reference.

```
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|....8
*CONSTRAINED_LAGRANGE_IN_SOLID
$  LSTRSID    ALESID  LSTRSTYP   ALESTYP     NQUAD     CTYPE     DIREC     MCOUP
         1        11         0         0         4         4         2      -123
```

```
$     START       END      PFAC      FRIC    FRCMIN      NORM  NORMTYPE      DAMP
        0.0       0.0       0.1      0.00       0.3         0         0       0.0
$        CQ      HMIN      HMAX     ILEAK     PLEAK   LCIDPOR     NVENT    IBLOCK
         0         0         0         0       0.0         0         0         0
$4A IBOXID   IPENCHK   INTFORC   IALESOF    LAGMUL    PFACMM      THKF
$        0         0         0         0         0         0         0
$4B     A1        B1        A2        B2        A3        B3
$      0.0       0.0       0.0       0.0       0.0       0.0
$4C VNTSID   VENTYPE  VENTCOEF   POPPRES  COEFLCID  (STYPE:0=PSID;1=PID;2=SGSID)
$        0         0         0       0.0         0
$...|....1....|....2....|....3....|....4....|....5....|....6....|....7....|...8
```

Remarks:

1. **Meshing.** In order for a fluid-structure interaction (FSI) to occur, a Lagrangian (structure) mesh must spatially overlap with an ALE (fluid) mesh. Each mesh should be defined with independent node IDs. LS-DYNA searches for the spatial intersection of between the Lagrangian and ALE meshes. Where the meshes overlap, there is a possibility that interaction may occur. In general, LSTRSID, ALESID, LSTRSTYP and ALESTYP are required definitions for specifying overlapping-domains coupling search.

2. **Number of coupling points.** The number of coupling points, NQUAD × NQUAD, is distributed over the surface of each Lagrangian segment. Generally, 2 or 3 coupling points per each Eulerian/ALE element width is adequate. Consequently, the appropriate NQUAD values must be estimated based on the *relative resolutions* between the Lagrangian and ALE meshes.

   For example, if one Lagrangian shell element spans two ALE elements, then NQUAD for each Lagrangian segment should be 4 or 6. Alternatively, if two or three Lagrangian segments span one ALE element, then maybe NQUAD = 1 would be adequate.

   If either mesh compresses or expands during the interaction, the number of coupling points per ALE element will also change. The user must account for this and try to maintain at least two coupling points per each ALE element side length during the whole process to prevent leakage. Too many coupling points can result in instability, and not enough can result in leakage.

3. **The constraint method.** The constraint method violates kinetic energy balance. The penalty method is therefore recommended. Historically, CTYPE = 2 was sometimes used to couple Lagrangian beam nodes to ALE or Lagrangian solids, such as for modeling rebar in concrete or tire cords in rubber. For such constraint-based coupling of beams in solids, *CONSTRAINED_BEAM_IN_SOLID and *DEFINE_BEAM_SOLID_COUPLING are now preferred.

4. **Coupling direction.** DIREC = 2 (compression only) may be generally a more stable and robust choice for coupling direction. However, the physics of the problem should dictate the coupling direction. DIREC = 1 couples under both

tension and compression. This is sometimes useful; for example, in the case of a suddenly accelerating liquid in a container. DIREC = 3 is rarely appropriate because it models an extremely sticky fluid.

5. **Multi-material coupling option.** When MCOUP is a negative integer, such as MCOUP = -123, then an ALE multi-material set-ID (AMMSID) of 123 must exist. This is an ID defined by a *SET_MULTI-MATERIAL_GROUP_LIST card. This generally seems to be a better approach to couple to a specific set of AMMGs, and have a clearly defined fluid interface interacting with a Lagrangian surface. That way, any leakage may be visualized and the penalty force can be computed more precisely.

   The *Couple to all materials* option as activated by MCOUP = 0 is generally not recommended. LS-DYNA calculates the fluid coupling interface as the surface where the sum of coupled ALE materials occupies a volume fraction equal to 50%. Since MCOUP = 0 couples to all materials, the sum of all coupled ALE materials is, in this case, trivially 100%. Consequently, when MCOUP = 0, there will not be a fluid interface with which to track leakage.

6. **PFAC < 0 for CTYPE = 4, 5, or 6.** PFAC < 0 refers to a load curve. This curve may be constructed with the x-axis as the penetration distance (in absolute length unit) and the y-axis as the maximum estimated coupling pressure. This curve requires only 2 points. The first point is (0.0,0.0) because no FSI pressure applies when no penetration occurs. The x-value for the second point is 10% of the minimum ALE element width, and the y-value is the to be determined maximum observable pressure in the ALE domain. To determine this y-value, run the model once with PFAC = 0.1 (the default). Then, select only the ALE domain and plot its pressure fringes. Next, step through time to identify the maximum pressure in the ALE domain, selecting the pressure peaks at or near the FSI surfaces. This maximum pressure becomes the y-value of the second point.

7. **Normal vector direction.** The normal vectors (NV) of a Lagrangian shell part are defined by the order of the nodes in *ELEMENT definitions, using the right hand rule, and for a segment set, the order of nodes defined in *SET_SEGMENT. Let the side pointed to by NV be "positive". The penalty method measure penetration as the distance the ALE fluid penetrates from the positive side to the negative side of the Lagrangian segment. Only fluid on the positive side will be "seen" and coupled to.

   Therefore, all normal vectors of the Lagrangian segments should point uniformly toward the ALE fluid(s), AMMGs, to be coupled to. If NV point uniformly away from the fluid, coupling is not activated. In this case, coupling can be activated by setting NORM = 1. Sometimes a shell part or mesh is generated such that its normal vectors do not point uniformly in a consistent direction (all toward the inside or outside of a container, etc.) You should always check for

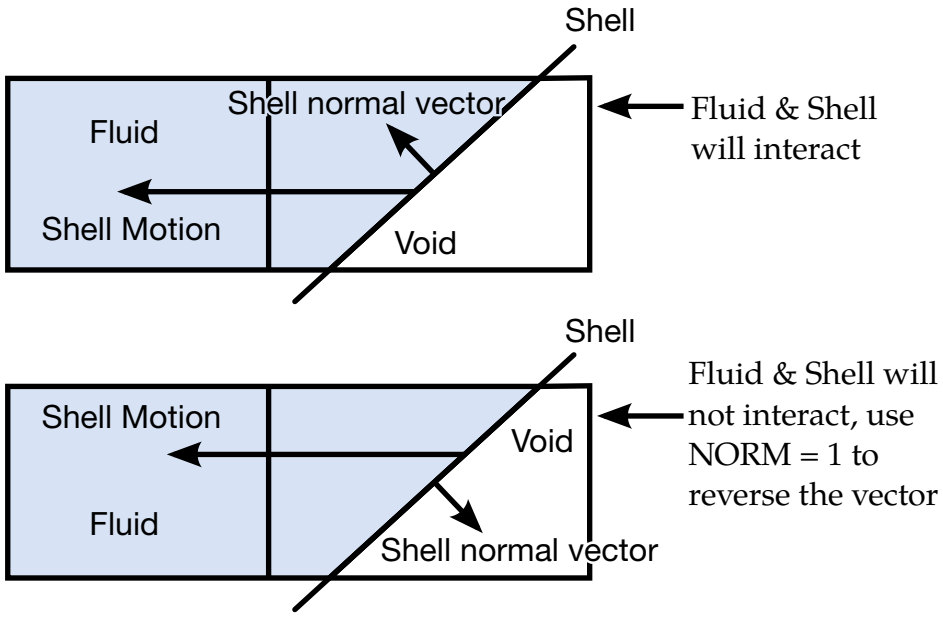


Figure 10-35. Shell Motion

the normal vectors of any Lagrangian shell part interacting with any fluid. The NORM parameter may be used to flip the normal direction of all the segments included in the Lagrangian structure set. See Figure 10-35.

8. **Coupling-damping factor.** The user-input coupling-damping factor (DAMP) is used to scale down the critical-damping force (~ damper constant × velocity). For a mass-to-rigid-wall system connected by a parallel-spring-damper connector, we can obtain solution for a critically-damped case. DAMP is a factor for scaling down the amount of damping, with DAMP = 1 being a critically-damped case.

9. **Heat transfer.** The method used is similar to that done by *CONTACT_…_THERMAL_… card, except radiation heat transfer is not considered. A gap, l, is assumed to exist between the two materials undergoing heat transfer (one is Lagrangian and the other ALE). The convection heat transfer in the gap is assumed to approach simple conduction across the medium in the gap.

$$q = K\frac{dT}{dx} \sim h\Delta T \Rightarrow h \sim \frac{K}{l}$$

The heat flux is typically defined as an energy transfer rate per unit area, $q \sim \frac{[\,J/s\,]}{m^2}$. The constant K is the thermal conductivity of the material in the gap; h, is the equivalent convection heat transfer coefficient; and ΔT is the temperature difference between the ALE and Lagrangian structure sides. There are 3 possible scenarios:

$$h \sim \begin{cases} 0 & \text{HMAX} < l \\ K/l & \text{HMIN} \le l \le \text{HMAX} \\ K/\text{HMIN} & 0 < l < \text{HMIN} \end{cases}$$

The ALE fluid must be modelled using the ALE single material with void element formulation (ELFORM = 12) because the LS-DYNA thermal solver supports only one temperature per node. However, a workaround enables partial support for ELFORM = 11. Rather than using the thermal solver's nodal temperature field, the ALE temperature is derived from element's internal energy using the heat capacity. The heat is then extracted from or added to the internal energy of ALE elements. This feature was implemented to calculate the heat exchange between a gas mixture, modeled with *MAT_GAS_MIXTURE and ALE multi-material formulation ELFORM = 11, and a Lagrangian container.

HMIN < 0 turns on constraint-based thermal nodal coupling between the Lagrangian surface nodes and ALE fluid nodes. This option only works with ALE single material with void element formulation (ELFORM = 12). Once a Lagrangian surface node is in contact with ALE fluid (gap = 0), the heat transfer described above is turned off. Instead the Lagrangian surface node temperature is constrained to the ALE fluid temperature field.

10. **Leakage control.** The dominate force preventing leakage across a coupled Lagrangian surface should be the penalty associated with the coupling. Forces from the leakage control algorithm feature should be secondary. The *DATABASE_FSI keyword controls the "dbfsi" file, which reports both the coupling forces and the leakage control force contribution. It is useful for debugging and fine-tuning.

    ILEAK = 2 conserves energy; thus, it is better for airbag applications. Leakage control should only be enabled when (1) coupling to a specific AMMG (MCOUP as a negative integer) is activated, and (2) the fluid interface is clearly defined and tracked through the *ALE_MULTI-MATERIAL_GROUP card.

11. **Pressure definition in porous flow.** There are currently two methods to model porous flow across a Lagrangian shell structure. Both methods involve defining an empirical data curve of relative porous gas velocity as a function of system pressure. The pressure definitions, however, are slightly different depending on the choice of parameter defined:

    a) When porous flow is modelled using the LCIDPOR parameter (part of *this* keyword), the velocity response curve expected to be given in terms of the pressure difference: $P_{\text{upstream}} - P_{\text{downstream}}$.

b) When LCIDPOR is negative, porous flow is modelled using the *MAT_-FABRIC material model. The FAC field in *MAT_FABRIC contains a load curve ID given in terms of absolute upstream pressure, rather than in terms of the pressure difference.

The *AIRBAG_ALE keyword assumes that the curve referenced by FAC in *MAT_FABRIC is given in terms of absolute upstream pressure. These absolute pressure data are *required* for the CV phase. During the ALE phase, LS-DYNA automatically shifts the FAC curve left (negative) by 1 atmospheric pressure for the porous coupling calculation, which uses gauge pressure, rather than absolute pressure.

The mass flowing across a porous Lagrangian surface can be tracked by the "mout" parameter of the optional "dbfsi" ASCII output file, which may be enabled with the *DATABASE_FSI keyword.

12. **Venting.** There are 2 methods to model (airbag) venting. The accumulated mass output of both may be tracked using the *DATABASE_FSI card ("mout" parameter in the "dbfsi" ASCII output file).

a) **Isentropic venting.** In isentropic venting, (define NVENT on Card 3) the flow crossing the vent hole surface is estimated from the isentropic equation. All airbag shell normal vectors should point uniformly in the same direction: typically, inward. The shell elements for the vent holes, included in the Lagrangian coupling set, should also point in the same direction as the airbag meaning usually inward. For more details on isentropic venting, see *AIRBAG_WANG_NEFSKE mass flow rate equation for field OPT = 1 and 2.

b) **Physical venting.** Physical venting models involve holes in the Lagrangian structure (usually airbags). The shell parts representing the vent holes may be either excluded from the Lagrangian coupling set, or, if included, have normal vectors reversed from the rest of the airbag. Typically, this means the holes having outward facing normal vectors, since the rest of the airbag has inward pointing normal vectors. With either approach the holes produce no coupling force to stop fluid leakage.

When a particular AMMG is present on both sides of the same Lagrangian shell surface, penalty coupling can break down. Therefore, it is recommended that *ALE_FSI_SWITCH_MMG_ID be used to switch the AMMG ID of the vented gas so that the vented gas outside the bag does not lead to leakage.

13. **Initial penetration check.** Typically, penetration check (IPENCHK) should only be used if there is high coupling force applied at $t = 0$. For example, consider a Lagrangian container, filled with non-gaseous fluid, such as ALE liquid

or solid, using the *INITIAL_VOLUME_FRACTON_GEOMETRY command. Sometimes due to mesh resolution or complex container geometry, there is initial penetration of the fluid across the container surface. This can give rise to a sharp and immediate coupling force on the fluid at $t = 0$. Turning on IPENCHK may help eliminate this spike in coupling force.

14. **Porous flow for shell elements.** For shell elements and CTYPE = 11, the Ergun-type empirical porous flow equation is applied to the normal flow direction across the porous surface. The pressure gradient along the segment normal direction is

$$\frac{dP}{dx_n} = A_n(\varepsilon,\mu)V_n + B_n(\varepsilon,\rho)|V_n|V_n$$

where the subscript n refers to the direction normal to the porous Lagrangian shell surface and where,

a) V_n is the relative normal-to-porous-shell-surface fluid velocity component.

b) $A_n(\varepsilon,\mu) = A_1(\varepsilon,\mu)$ is a viscous coefficient of the Ergun-type porous flow equation. As applied here, it should contain the fluid dynamic viscosity, μ, and shell porosity, ε, information.

c) $B_n(\varepsilon,\rho) = B_1(\varepsilon,\rho)$ is an inertial coefficient of the Ergun-type porous flow equation. As applied here, it should contain the fluid density, ρ, and shell porosity, ε, information.

The force increment applied per segment is

$$F_n = \frac{d\rho}{dx_n} \times \text{THKF} \times S,$$

where S is the segment surface area.

If *DEFINE_POROUS_LAGRANGIAN defines the porous properties of a Lagrangian structure element, the porous forces are computed with an equation similar to the one used in *LOAD_BODY_POROUS

> **NOTE:** $A_i(\varepsilon,\mu)$, $B_i(\varepsilon,\rho)$, and THKF are required input for porous shell coupling.

15. **Porous flow for solid elements.** For porous solid, CTYPE = 12, the pressure gradient along each global direction (i) can be computed similarly.

$$\frac{dP}{dx_i} = A_i(\varepsilon,\mu)V_i + B_i(\varepsilon,\rho)|V_i|V_i \text{ for } i = 1,2,3$$

where,

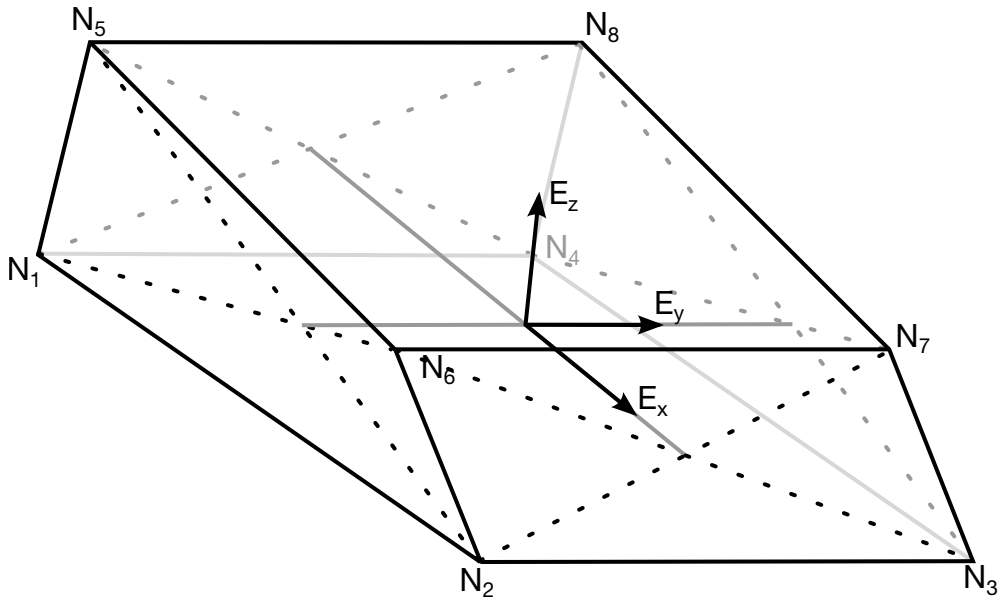


Figure 10-36. The E_x direction is aligned along the line segment connecting the centers of the 2-3-6-7 and the 1-4-8-5 faces. The E_y direction is orthogonal to the E_x direction and in the plane containing both E_x and the segment connecting the centers of the 1-2-6-5 and 3-4-8-7 faces. The E_z is normal to this plane.

a) V_i is the relative fluid velocity component through the porous solid in the 3 global directions.

b) $A_i(\varepsilon, \mu)$ is a viscous coefficient of the Ergun-type porous flow equation in the i[th] direction. As applied here, it should contain the fluid dynamic viscosity, μ, and shell porosity, ε, information.

c) $B_i(\varepsilon, \rho)$ is an inertial coefficient of the Ergun-type porous flow equation in the i[th] direction. As applied here, it should contain the fluid density (ρ) and solid porosity (ε) information.

> **NOTE:** $A_i(\varepsilon, \mu)$, and $B_i(\varepsilon, \rho)$ are required input for porous solid coupling.

If DIREC = 1, the pressure gradient in a solid is applied in a local reference coordinate system defined in Figure 10-36. If *DEFINE_POROUS_LAGRANGIAN defines the porous properties of a Lagrangian structure element, the local system can be adapted and the porous forces are computed with an equation similar to the one used in *LOAD_BODY_POROUS.

16. **Output.** To obtain the sliding energy output for each FSI, specify a *DATABASE_SLEOUT in the input deck. Including this keyword causes a file called slefsi to be output. The format of slefsi follows sleout exactly.

*CONSTRAINED_LINEAR_GLOBAL

Purpose: Define linear constraint equations between displacements and rotations, which can be defined in global coordinate systems.

> **WARNING:** For the explicit solver, nodes of a nodal constraint equation cannot be members of another constraint equation, a constraint set that contains the same degrees-of-freedom, tied interface, or rigid bodies. Nodes must not be subject to multiple, independent, and possibly conflicting constraints. Furthermore, care must be taken to ensure that single point constraints applied to nodes in a constraint equation do not conflict with the degrees-of-freedom constrained by the constraint equation.
>
> The same applies for the implicit solver; however multiple nodal constraint equations may be applied to the various degrees-of-freedom of the same node, provided that there are no conflicting constraints.

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

DOF Card. Define one card for each constrained degree-of-freedom. Input is terminated at the next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	NID	DOF	COEF					
Type	I	I	F					
Default	none	0	0					

VARIABLE	DESCRIPTION
LCID	Linear constraint definition ID. This ID can be used to identify a set to which this constraint is a member.
NID	Node ID
DOF	Degree of freedom in the global coordinate system: EQ.1: Displacement along global x-direction EQ.2: Displacement along global y-direction EQ.3: Displacement along global z-direction EQ.4: Global rotation about global x-axis EQ.5: Global rotation about global y-axis EQ.6: Global rotation about global z-axis EQ.7: Nodal electric voltage of piezoelectric material; see *MAT_ADD_PZELECTRIC. The voltage of the 1st node can only be defined as a linear combination of the voltage of other nodes, meaning all DOFs must be 7 for such an application.
COEF	Nonzero coefficient, C_k

Remarks:

In this section, linear constraint equations of the form:

$$\sum_{k=1}^{n} C_k u_k = C_0$$

can be defined, where u_k are the displacements and C_k are user defined coefficients. Unless LS-DYNA is initialized by linking to an implicit code to satisfy this equation at the beginning of the calculation, the constant C_0 is assumed to be zero. The first constrained degree-of-freedom is eliminated from the equations-of-motion:

$$u_1 = C_0 - \sum_{k=2}^{n} \frac{C_k}{C_1} u_k \ .$$

Its velocities and accelerations are given by

$$\dot{u}_1 = -\sum_{k=2}^{n} \frac{C_k}{C_1} \dot{u}_k$$

$$\ddot{u}_1 = -\sum_{k=2}^{n} \frac{C_k}{C_1} \ddot{u}_k,$$

respectively. A transformation matrix, **L**, is constructed relating the unconstrained, **u**, and constrained, $\mathbf{u}_c$, degrees-of-freedom. The constrained accelerations used in the above equation are given by:

$$\ddot{\mathbf{u}}_c = \left[\mathbf{L}^{\mathrm{T}}\mathbf{M}\mathbf{L}\right]^{-1}\mathbf{L}^{\mathrm{T}}\mathbf{F}$$

where **M** is the diagonal lumped mass matrix and **F** is the right hand side force vector. This requires the inversion of the condensed mass matrix which is equal in size to the number of constrained degrees-of-freedom minus one.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_LINEAR_GLOBAL
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Constrain nodes 40 and 42 to move identically in the z-direction.
$
$  When the linear constraint equation is applied, it goes like this:
$
$     0 = C40uz40 + C42uz42
$
$       = uz40 - uz42
$
$     uz40 = uz42
$
$   where,
$     C40  =  1.00  coefficient for node 40
$     C42  = -1.00  coefficient for node 42
$     uz40 = displacement of node 40 in z-direction
$     uz42 = displacement of node 42 in z-direction
$
$
*CONSTRAINED_LINEAR
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$        i
$       id
         2
$
$      nid       dof     coef
        40         3     1.00
        42         3    -1.00
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
```

*CONSTRAINED_LINEAR_LOCAL

Purpose: Define linear constraint equations between displacements and rotations which can be defined in a local coordinate system. Each node may have a unique coordinate ID.

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

DOF Cards. Define one card for each constrained degree-of-freedom. Input is terminated at next keyword ("*") card.

Card 2	1	2	3	4	5	6	7	8
Variable	NID	DOF	CID	COEF				
Type	I	I	I	F				
Default	none	0	global	0				
Remarks	1							

VARIABLE	DESCRIPTION
LCID	LCID for linear constraint definition. This ID can be used to identify a set to which this constraint is a member.
NID	Node ID
DOF	Degree-of-freedom in the local coordinate system; EQ.1: Displacement along local x-direction EQ.2: Displacement along local y-direction EQ.3: Displacement along local z-direction EQ.4: Local rotation about local x-axis

VARIABLE	DESCRIPTION
	EQ.5: Local rotation about local y-axis
	EQ.6: Local rotation about local z-axis
CID	Local coordinate system ID number. If the number is zero, the global coordinate system is used.
COEF	Nonzero coefficient, C_k

Remarks:

> **WARNING:** For the explicit solver, nodes of a nodal constraint equation cannot be members of another constraint equation, a constraint set that contains the same degrees-of-freedom, tied interface, or rigid bodies. Nodes must not be subject to multiple, independent, and possibly conflicting constraints. Furthermore, care must be taken to ensure that single point constraints applied to nodes in a constraint equation do not conflict with the degrees-of-freedom constrained by the constraint equation.
>
> The same applies for the implicit solver; however multiple nodal constraint equations may be applied to the various degrees-of-freedom of the same node, provided that there are no conflicting constraints.

With this keyword, you can define linear constraint equations of the following form:

$$\sum_{k=1}^{n} C_k \, u_k^L = C_0 \ ,$$

where u_k^L are the displacements in the local coordinate systems and C_k are user-defined coefficients. Unless you initialize LS-DYNA by linking to an implicit code to satisfy this equation at the beginning of the calculation, LS-DYNA assumes the constant C_0 is zero. The first constrained degree-of-freedom is eliminated from the equations-of-motion:

$$u_1^L = C_0 - \sum_{k=2}^{n} \frac{C_k}{C_1} u_k^L \ .$$

Its velocities and accelerations are given by

$$\dot{u}_1^L = -\sum_{k=2}^{n} \frac{C_k}{C_1} \dot{u}_k^L$$
$$\ddot{u}_1^L = -\sum_{k=2}^{n} \frac{C_k}{C_1} \ddot{u}_k^L$$

respectively. The local displacements are calculated every time step using the local coordinate systems defined by the user. More than one degree-of-freedom for a node can be constrained by specifying a card for each degree-of-freedom.

***CONSTRAINED_LOCAL**_*{OPTION}*

OPTION allows an optional ID to be given:

ID

Purpose: Define a local boundary constraint plane.

ID Card. Additional card for the ID keyword option.

Optional	1	2	3	4	5	6	7	8
Variable	ID	HEADING						
Type	I	A70						

Card 1	1	2	3	4	5	6	7	8
Variable	TC	RC	DIR	X	Y	Z	CID	TOL
Type	1	1	1	F	F	F	1	F
Default	0	0	none	0.0	0.0	0.0	none	0.0

VARIABLE	DESCRIPTION
ID	Optional ID which can be referred to by *SENSOR_CONTROL. This ID must be unique and cannot be shared with *BOUNDARY_SPC.
HEADING	An optional descriptor that will be written into the d3hsp file and the spcforc file.
TC	Translational constraint in local system: EQ.0: no translational constraints EQ.1: constrained x translation EQ.2: constrained y translation EQ.3: constrained z translation EQ.4: constrained x and y translations

VARIABLE	DESCRIPTION
	EQ.5: constrained y and z translations
	EQ.6: constrained x and z translations
	EQ.7: constrained x, y, and z translations
RC	Rotational constraint in local system:
	EQ.0: no rotational constraints
	EQ.1: constrained x-rotation
	EQ.2: constrained y-rotation
	EQ.3: constrained z-rotation
	EQ.4: constrained x and y rotations
	EQ.5: constrained y and z rotations
	EQ.6: constrained z and x rotations
	EQ.7: constrained x, y, and z rotations
DIR	Direction of normal for local constraint plane:
	EQ.1: local x,
	EQ.2: local y,
	EQ.3: local z.
X	Local x-coordinate of a point on the local constraint plane
Y	Local y-coordinate of a point on the local constraint plane
Z	Local z-coordinate of a point on the local constraint plane
CID	Coordinate system ID for orientation of the local coordinate system
TOL	User-defined tolerance in length units. If non-zero, the internal mesh-size dependent tolerance gets replaced by this value.

Remarks:

Nodes within a mesh-size-dependent tolerance are constrained on a local plane. This option is recommended for use with *r*-method adaptive remeshing where nodal constraints are lost during the remeshing phase.

*CONSTRAINED_MULTIPLE_GLOBAL

Purpose: Define global multi-point constraints for imposing periodic boundary condition in displacement field.

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

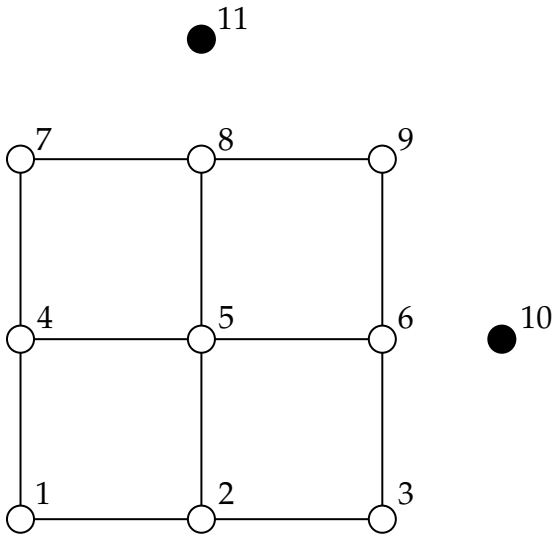
NOTE: For each constraint equation include a set of cards consisting of (1) a C*onstraint Equation Definition Card* and (2) NMP *Coefficient Cards*.

Constraint Equation Definition Card.

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

Coefficient Cards. The next NMP cards adhere to this format. Each card sets a single coefficient in the constraint equation.

Card 3	1	2	3	4	5	6	7	8
Variable	NID	DIR	COEF					
Type	I	I	F					
Default								




```
*CONSTRAINED_MULTIPLE_GLOBAL
1
3
 3, 1, 1.0
 1, 1,-1.0
10, 1,-1.0
3
 8, 1, 1.0
 2, 1,-1.0
11, 1,-1.0

*CONSTRAINED_MULTIPLE_GLOBAL
2
3
 3, 2, 1.0
 1, 2,-1.0
10, 2,-1.0
...
```

$$u_1^{(3)} - u_1^{(1)} - u_1^{(10)} = 0$$

$$u_1^{(8)} - u_1^{(2)} - u_1^{(11)} = 0$$

$$u_2^{(3)} - u_2^{(1)} - u_2^{(10)} = 0$$

Figure 10-37. Simple example.

VARIABLE	DESCRIPTION
ID	Constraint set identification. All constraint sets should have a unique set ID.
NMP	Number of nodes to be constrained mutually.
NID	Nodal ID
DIR	Direction in three-dimensional space to be constrained EQ.1: x direction EQ.2: y direction EQ.3: z direction

VARIABLE	DESCRIPTION
	LT.0: Extra DOFs for user defined element formulation (e.g. -1: the 1st extra DOF; -2: the 2nd extra DOF; …)
COEF	Coefficient α_{NID} in constraint equation: $\sum_{\text{NID}} \alpha_{\text{NID}} u_{\text{DIR}}^{(\text{NID})} = 0.$

Remarks:

1. Defining multi-point constraints by this keyword can be demonstrated by the following example: a two-dimensional unit square with four quadrilateral elements and 11 nodes as shown in Figure 10-37, where the nodes #10 and #11 are two dummy nodes serving as control points.

*CONSTRAINED_NODAL_RIGID_BODY_{OPTION}_{OPTION}_{OPTION}_{OPTION}

Purpose: Define a nodal rigid body which is a rigid body that consists of defined nodes. Unlike *CONSTRAINED_NODE_SET, which permits only constraints on translational motion, here we use the equations of rigid body dynamics to update the motion of the nodes, and therefore, rotations of the nodal sets are admissible.

Include any combination of the following available options:

<BLANK>

INERTIA

OVERRIDE

SPC

THERMAL

TITLE

The order of the options in the keyword name is arbitrary.

If *not* using the INERTIA option, LS-DYNA computes the inertia tensor from the nodal masses. The nodal masses and coordinates determine the mass properties. This rigid body may have arbitrary motion. When using the INERTIA option, constant translational and rotational velocities can be defined in a global or local coordinate system.

When using the OVERRIDE option, LS-DYNA automatically turns off all conflicting rigid body or constraint definitions until *SENSOR_CONTROL turns off the overriding nodal rigid body constraint defined with this keyword. When the overriding rigid body constraint is turned off, all conflicting rigid bodies and constraints are turned back on automatically. For example, you can use this feature to rigidize the whole model when the entire model moves like a rigid body involving no deformation.

If the center of mass is constrained, use the SPC option.

The THERMAL option allows imposing a constraint on the temperature distribution for the nodal rigid body. With this constraint, LS-DYNA sets the temperature of a particular node to the average temperature of the other nodes in the nodal rigid body.

The TITLE option enables providing a description for the nodal rigid body.

Card Summary:

Card Title. Include this keyword for the TITLE keyword option.

TITLE

Card 1. This card is required.

PID	CID	NSID	PNODE	IPRT	DRFLAG	RRFLAG	

Card 2. Include this card if using the SPC keyword option.

CMO	CON1	CON2	SPCNID	XSPC	YSPC	ZSPC	

Card 3. Include this card for the INERTIA keyword option.

XC	YC	ZC	TM	IRCS	NODEID		

Card 4. Include this card for the INERTIA keyword option.

IXX	IXY	IXZ	IYY	IYZ	IZZ		

Card 5. Include this card for the INERTIA keyword option.

VTX	VTY	VTZ	VRX	VRY	VRZ		

Card 6. Include this card if IRCS = 1. See Card 3.

XL	YL	ZL	XLIP	YLIP	ZLIP	CID2	

Card 7. This card is read when using the OVERRIDE keyword option. It is optional.

ICNT	IBAG	IPSM					

Card 8. Include this card for the THERMAL keyword option..

IDTHRM							

Data Card Definitions:

Title Card. Additional card for the TITLE keyword option.

Card Title	1	2	3	4	5	6	7	8
Variable	TITLE							
Type	A80							

VARIABLE	DESCRIPTION
TITLE	Description for the nodal rigid body

Card 1	1	2	3	4	5	6	7	8
Variable	PID	CID	NSID	PNODE	IPRT	DRFLAG	RRFLAG	
Type	I	I	I	I	I	I	I	
Default	none	0	PID	0	↓	0	0	

VARIABLE	DESCRIPTION
PID	Part ID of the nodal rigid body
CID	Optional coordinate system ID for the rigid body local system; see *DEFINE_COORDINATE_OPTION. The rigid body data and the degree-of-freedom releases are output in this local system. This local system rotates with the rigid body.
NSID	Nodal set ID, see *SET_NODE_OPTION. This nodal set defines the rigid body. EQ.0: NSID = PID; that is, the node set ID and the part ID are assumed to be identical.
PNODE	An optional node (a massless node is allowed) used for post-processing rigid body data. If PNODE is not located at the rigid body's center of mass, LS-DYNA resets the initial coordinates of PNODE to the center of mass. If CID is defined, LS-DYNA outputs the velocities and accelerations of PNODE in the local system to the d3plot and d3thdt files unless you specify PNODE as a negative

VARIABLE	DESCRIPTION
	number, in which case the global system is used.
IPRT	Print flag. For nodal rigid bodies, the following values apply: EQ.1: Write data into rbdout. EQ.2: Do not write data into rbdout. Except for two-noded rigid bodies, IPRT (if 0 or unset) defaults to the value of IPRTF in *CONTROL_OUTPUT. For two-noded rigid bodies, printing is suppressed (IPRT = 2) unless IPRT is set to 1. This is to avoid excessively large rbdout files when the model contains many two-noded welds.
DRFLAG	Displacement release flag for all nodes except the first node in the definition (see Remark 3). EQ.-7: Release the x, y, and z displacements in the global system EQ.-6: Release the z and x displacements in the global system EQ.-5: Release the y and z displacements in the global system EQ.-4: Release the x and y displacements in the global system EQ.-3: Release the z displacement in the global system EQ.-2: Release the y displacement in the global system EQ.-1: Release the x displacement in the global system EQ.0: Off for rigid body behavior EQ.1: Release the x displacement in the rigid body's local system EQ.2: Release the y displacement in the rigid body's local system EQ.3: Release the z displacement in the rigid body's local system EQ.4: Release the x and y displacements in the rigid body's local system EQ.5: Release the y and z displacements in the rigid body's local system EQ.6: Release the z and x displacements in the rigid body's local system EQ.7: Release the x, y, and z displacements in the rigid body's local system

VARIABLE	DESCRIPTION
RRFLAG	Rotation release flag for all nodes except the first node in the definition (see Remark 3). EQ.-7: Release the x, y, and z rotations in the global system EQ.-6: Release the z and x rotations in the global system EQ.-5: Release the y and z rotations in the global system EQ.-4: Release the x and y rotations in the global system EQ.-3: Release the z rotation in the global system EQ.-2: Release the y rotation in the global system EQ.-1: Release the x rotation in the global system EQ.0: Off for rigid body behavior EQ.1: Release the x rotation in the rigid body's local system EQ.2: Release the y rotation in the rigid body's local system EQ.3: Release z rotation in the rigid body's local system EQ.4: Release the x and y rotations in the rigid body's local system EQ.5: Release the y and z rotations in the rigid body's local system EQ.6: Release the z and x rotations in the rigid body's local system EQ.7: Release the x, y, and z rotations in the rigid body's local system

Constraint Card. Additional card for the SPC keyword option.

Card 2	1	2	3	4	5	6	7	8
Variable	CMO	CON1	CON2	SPCNID	XSPC	YSPC	ZSPC	
Type	F	I	I	I	F	F	F	
Default	0.0	0	0	0	0.0	0.0	0.0	

VARIABLE	DESCRIPTION
CMO	Constraint option, CMO (see Remark 4):

VARIABLE	DESCRIPTION
	EQ.+2.0: Constraints applied in global directions at the coordinates given by XSPC, YSPC, and ZSPC or the initial coordinates of node SPCNID. Unless prescribed motion is applied to the rigid body, the constraint coordinates are fixed in time. EQ.+1.0: Constraints applied in global directions, EQ.0.0: No constraints, EQ.-1.0: Constraints applied in local directions (SPC constraint). EQ.-2.0: Constraints applied in local directions (SPC constraint) at the coordinates given by XSPC, YSPC, and ZSPC or the initial coordinates of node SPCNID. Unless prescribed motion is applied to the rigid body, the constraint coordinates are fixed in time.
CON1	First constraint parameter. If CMO > 0.0, then specify the global translational constraint: EQ.0: No constraints, EQ.1: Constrained x displacement, EQ.2: Constrained y displacement, EQ.3: Constrained z displacement, EQ.4: Constrained x and y displacements, EQ.5: Constrained y and z displacements, EQ.6: Constrained z and x displacements, EQ.7: Constrained x, y, and z displacements. If CMO < 0.0, then specify the local coordinate system ID. See *DEFINE_COORDINATE_OPTION. This coordinate system is fixed in time.
CON2	Second constraint parameter: If CMO > 0.0, then specify the global rotational constraint: EQ.0: No constraints, EQ.1: Constrained x rotation, EQ.2: Constrained y rotation,

VARIABLE	DESCRIPTION
	EQ.3: Constrained z rotation, EQ.4: Constrained x and y rotations, EQ.5: Constrained y and z rotations, EQ.6: Constrained z and x rotations, EQ.7: Constrained x, y, and z rotations. If CMO < 0.0, then specify the local (SPC) constraint: EQ.000000: No constraint, EQ.100000: Constrained x translation, EQ.010000: Constrained y translation, EQ.001000: Constrained z translation, EQ.000100: Constrained x rotation, EQ.000010: Constrained y rotation, EQ.000001: Constrained z rotation. To specify a combination of local constraints, input the sum of the desired constraints.
SPCNID	For |CMO| = 2.0, the constraint coordinates (see below) are the (initial) coordinates of the node with this ID.
XSPC,YSPC,ZSPC	Coordinates where the constraints act. Superseded by SPC-NID.

Inertia Card 1. Additional card for the INERTIA keyword option.

Card 3	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	TM	IRCS	NODEID		
Type	F	F	F	F	I	I		
Default	0.	0.	0.	0.	0	0		

VARIABLE	DESCRIPTION
XC	x-coordinate of center of mass. If nodal point, NODEID, is defined, XC, YC, and ZC are ignored and the coordinates of the nodal point, NODEID, are taken as the center of mass.
YC	y-coordinate of center of mass
ZC	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 is given in a system defined by the orientation vectors given in Card 6.
NODEID	Optional nodal point defining the CG of the rigid body. If this node is not a member of the set NSID above, its motion will not be updated to correspond with the nodal rigid body after the calculation begins. PNODE and NODEID can be identical if and only if PNODE physically lies at the mass center at time zero.

Inertia Card 2. Second additional card for the INERTIA keyword 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		
Default	none	0	0	none	0	0		

VARIABLE	DESCRIPTION
IXX	I_{xx}, xx component of inertia tensor
IXY	I_{xy}, xy component of inertia tensor
IXZ	I_{xz}, xz component of inertia tensor
IYY	I_{yy}, yy component of inertia tensor
IYZ	I_{yz}, yz component of inertia tensor

VARIABLE	DESCRIPTION
IZZ	I_{zz}, zz component of inertia tensor

Inertia Card 3. Third additional card for the INERTIA keyword option. *INITIAL_VELOCITY can overwrite the velocities defined below.

Card 5	1	2	3	4	5	6	7	8
Variable	VTX	VTY	VTZ	VRX	VRY	VRZ		
Type	F	F	F	F	F	F		
Default	0	0	0	0	0	0		

VARIABLE	DESCRIPTION
VTX	x-rigid body initial translational velocity in global coordinate system.
VTY	y-rigid body initial translational velocity in global coordinate system.
VTZ	z-rigid body initial translational velocity in global coordinate system.
VRX	x-rigid body initial rotational velocity in global coordinate system.
VRY	y-rigid body initial rotational velocity in global coordinate system.
VRZ	z-rigid body initial rotational velocity in global coordinate system.

Local Inertia Tensor Card. Additional card required for IRCS = 1 (see Inertia Card 1). 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	CID2	
Type	F	F	F	F	F	F	I	
Default	CID2	CID2	CID2	CID2	CID2	CID2	0	

VARIABLE	DESCRIPTION
XL	x-coordinate of local x-axis. Origin lies at (0,0,0).
YL	y-coordinate of local x-axis
ZL	z-coordinate of local x-axis
XLIP	x-coordinate of local in-plane vector
YLIP	y-coordinate of local in-plane vector
ZLIP	z-coordinate of local in-plane vector
CID2	Local coordinate system ID, see *DEFINE_COORDINATE_… With this option, leave fields 1-6 blank.

Optional synchronization card for OVERRIDE. Optional card for OVERRIDE option.

Card 7	1	2	3	4	5	6	7	8
Variable	ICNT	IBAG	IPSM					
Type	I	I	I					
Default	0	0	0					

VARIABLE	DESCRIPTION
ICNT	Flag for contact synchronization: EQ.0: No synchronization EQ.1: Since no contact exists when both the tracked and reference sides belong to the same rigid body, setting ICNT = 1 will turn off/on all contact definitions of which the tracked and reference sides belong to the same nodal rigid body PID when PID is turned on/off by *SENSOR_CONTROL. With ICNT = 1, all related contact definitions need the optional ID; see *CONTACT.

VARIABLE	DESCRIPTION
IBAG	Flag for control volume airbag synchronization: EQ.0: No synchronization EQ.1: Since airbag pressure will not change when all segments constituting the airbag belong to the same rigid body, setting IBAG = 1 will skip the control volume airbag calculations for airbags that belong to PID when PID is on. The airbag calculation will resume when *SENSOR_CONTROL turns off PID. The related airbag time-dependent curves will be offset to not include the time that PID is on. With IBAG = 1, all associated airbags need the optional ID; see *AIRBAG.
IPSM	Flag for prescribed-motion synchronization: EQ.0: No synchronization EQ.1: Prescribed boundary conditions (*BOUNDARY_PRESCRIBED_MOTION) for PID are turned off automatically when *SENSOR_CONTROL turns off PID. A prescribed boundary condition not for PID and not for all nodes belonging to PID, that is, boundary conditions that only apply to some of the nodes belonging to PID, are turned off when PID is active to avoid boundary condition conflict. Those boundary conditions are turned on when *SENSOR_CONTROL turns off PID. The related time-dependent curves will be offset to not include the time that PID is on. With IPSM > 0, all related boundary prescribed motion cards need the optional ID; see *BOUNDARY_PRESCRIBED. EQ.2: Same as IPSM = 1, however, without time offset when those boundary conditions not for PID are turned on.

Thermal Constraint Card. Required card for the THERMAL keyword option.

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

VARIABLE	DESCRIPTION
IDTHRM	Flag for the thermal constraint option: EQ.1: Set the temperature of the first node defined in node set NSID as the average of the other nodal temperatures in the node set. LT.0: Set the temperature of node |IDTHRM| as the average of the nodal temperatures for the nodes in node set NSID. The average excludes node |IDTHERM| if it is a member of this node set.

Remarks:

1. **Local coordinate system.** The local coordinate system is set up in the following way. After the local x-axis is defined, the local z-axis is computed from the cross-product of the local x-axis vector with the given in-plane vector. Finally, the local y-axis is determined from the cross-product of the local z-axis with the local x-axis. The local coordinate system defined by CID has the advantage that the local system can be defined by nodes in the rigid body which makes repositioning of the rigid body in a preprocessor much easier since the local system moves with the nodal points.

2. **Nodal rigid body visualization.** Use PLOTEL (*CONTROL_RIGID) to visualize the nodal rigid bodies.

3. **Release conditions.** The first node in the nodal rigid body definition is treated as the lead for the case where DRFLAG and RRFLAG are nonzero. The first node always has six degrees-of-freedom. The release conditions in the global system are sometimes convenient in small displacement linear analysis. Otherwise, they are not recommended. We strongly recommend that release conditions are only used for a two-noded nodal rigid body, especially for implicit calculations.

4. **Constraints.** Constraint directions for rigid bodies (CMO ≠ 0) are fixed, that is, not updated, with time. To impose a constraint on a rigid body such that the constraint direction is updated as the rigid body rotates, use *BOUNDARY_PRESCRIBED_MOTION_RIGID_LOCAL. The constraint defined therein refers to the local system CID, which is updated with time.

   We strongly advise not applying nodal constraints, such as by *BOUNDARY_SPC_*OPTION*, to nodes of a rigid body as doing so may compromise the

intended constraints in the case of an explicit simulation. Such SPCs will be skipped in an implicit simulation and a warning issued.

If the intended constraints are not with respect to the calculated center of mass of the rigid body, the following alternative approaches can be used:

a) Set |CMO| = 2 and choose the point at which the constraints shall act. These coordinates are referenced both by the constraints given on this card and those on *BOUNDARY_PRESCRIBED_MOTION_*OPTION*.

b) *CONSTRAINED_JOINT_*OPTION* may often be used to obtain the desired effect. This approach typically entails defining a second rigid body that is fully constrained and then defining a joint between the two rigid bodies.

c) Another alternative for defining rigid body constraints that are not with respect to the calculated center of mass of the rigid body is to manually specify the initial center of mass location using *PART_INERTIA. When using *PART_INERTIA, a full set of mass properties must be specified. Note that changing its mass properties will affect the rigid body's dynamic behavior.

Setting |CMO| = 2 not only allows for a constraint point other than the center of mass. The motion prescribed by *BOUNDARY_PRESCRIBED_MOTION also acts on this point. In addition, setting |CMO| = 2 treats the constraints (including those from *BOUNDARY_PRESCRIBED_MOTION) differently from |CMO| = 1. To allow for an arbitrary constraint point, the constraints are applied and solved during the kinematic update of the rigid body. Since the inertia does not need to be modified and no joints are involved, setting |CMO| = 2 is more accurate compared to options b and c above. No time penalty is to be expected.

To obtain reaction forces from constraints, see the SPC2BND flag of *CONTROL_OUTPUT.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_NODAL_RIGID_BODY
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a rigid body consisting of the nodes in nodal set 61.
$
$  This particular example was used to connect three separate deformable
$  parts.  Physically, these parts were welded together.  Modeling-wise,
$  however, this joint is quite messy and is most conveniently modeled
$  by making a rigid body using several of the nodes in the area.  Physically,
$  this joint was so strong that weld failure was never of concern.
$
*CONSTRAINED_NODAL_RIGID_BODY
```

```
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$       pid        cid       nsid
         45                    61
$
$      nsid = 61   nodal set ID number, requires a *SET_NODE_option
$       cid        not used in this example; output will be in global coordinates
$
$
*SET_NODE_LIST
$       sid
         61
$      nid1       nid2       nid3       nid4       nid5       nid6       nid7       nid8
         823       1057       1174       1931       2124       1961       2101
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

*CONSTRAINED_NODE_INTERPOLATION

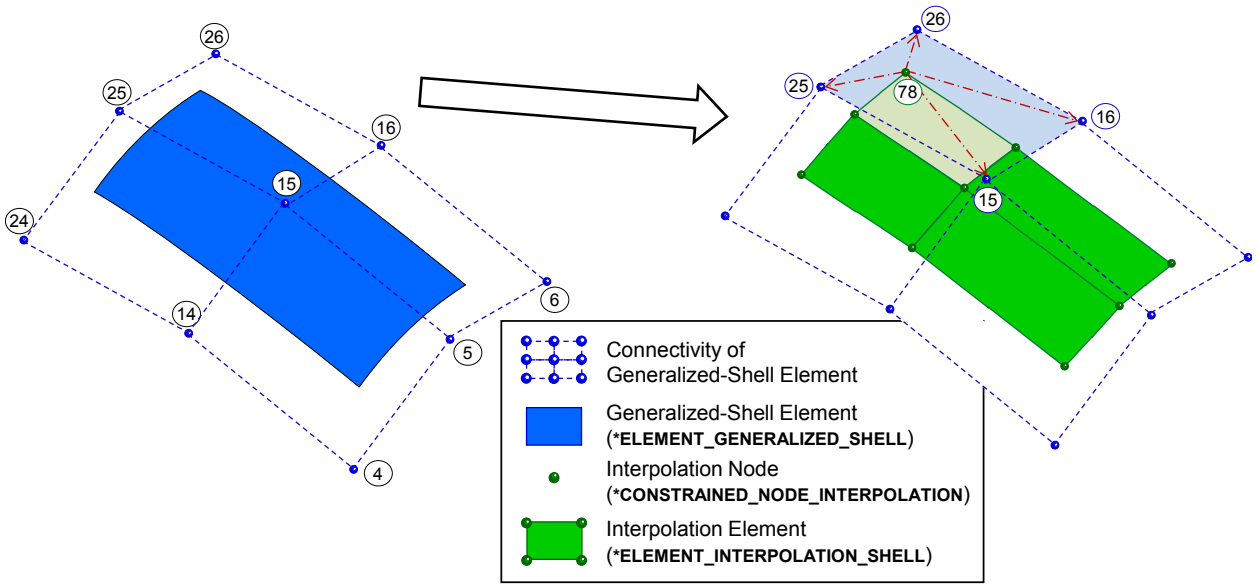
Purpose: Define constrained nodes for the use of *ELEMENT_INTERPOLATION_-SHELL and *ELEMENT_INTERPOLATION_SOLID to model contact and to visualize the results of generalized elements (see *ELEMENT_GENERALIZED_SHELL/SOLID). The displacements of these nodes are dependent on their corresponding controlling nodes.

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

Weighting Factor Cards. Define NUMCN controlling nodes and weights (one controlling node per CN*i* and W*i* pair) to constrain NID. See Remark 2. See *ELEMENT_-GENERALIZED_SHELL/SOLID. Each Weighting Factor Card can accommodate four master nodes. Include as many Weighting Factor Cards as needed.

Card 2	1	2	3	4	5	6	7	8
Variable	CN1	W1	CN2	W2	CN3	W3	CN4	W4
Type	I	F	I	F	I	F	I	F
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
NID	Node ID of the interpolation node as defined in *NODE (see Remark 1)
NUMCN	Number of nodes controlling the interpolation node
CN*i*	Node ID of controlling node *i*
W*i*	Weighting factor of controlling node *i*




```
*CONSTRAINED_NODE_INTERPOLATION
$---+--NID----+NUMCN----+----3----+----4----+----5----+----6----+----7----+----8
        78          4
$---+--CN1----+---W1----+--CN2----+---W2----+--CN3----+---W3----+--CN4----+---W4
        26       0.35         25       0.32         15       0.18         16       0.15
```

Figure 10-38. Example of a *CONSTRAINED_NODE_INTERPOLATION card

Remarks:

1. **Nodal Definition.** The coordinates of an interpolation node must be defined in *NODE. The fields TC and RC must be both be set to 7 for the interpolation node in *NODE.

2. **Interpolation Model.** The displacements of the interpolation node, $\mathbf{d}_{\text{IN}}$, are interpolated based on the displacements of the corresponding controlling nodes, $\mathbf{d}_i$, and the appropriate weighting factors, w_i. The interpolation is computed as follows:

$$\mathbf{d}_{\text{IN}} = \sum_{i=1}^{\text{NUMCN}} w_i \mathbf{d}_i.$$

*CONSTRAINED_NODE_SET_{OPTION}

To define an ID for the constrained node set the following option is available:

<BLANK>

ID

If the ID keyword option is used, an additional card is required.

Purpose: Define nodal constraint sets for translational motion in global coordinates. No rotational coupling. See Figure 10-39. Nodal points included in the sets should not be subjected to any other constraints including prescribed motion, such as motions prescribed with the *BOUNDARY_PRESCRIBED_MOTION options.

ID Card. Additional card for ID keyword option.

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

Card 2	1	2	3	4	5	6	7	8
Variable	NSID	DOF	TF					
Type	I	I	F					
Default	none	none	10^{20}					
Remarks	1, 2		3					

VARIABLE	DESCRIPTION
CNSID	Optional constrained node set ID
NSID	Nodal set ID; see *SET_NODE_*OPTION*.

***CONTRAINED_NODE_SET**

Since no rotation is permitted, this option should *not* be used to model rigid body behavior involving rotations

***CONSTRAINED_NODAL_RIGID_BODY**
***CONSTRAINED_SPOTWELD**

Behavior is like a rigid beam. These options *may* be used to model spotwelds.

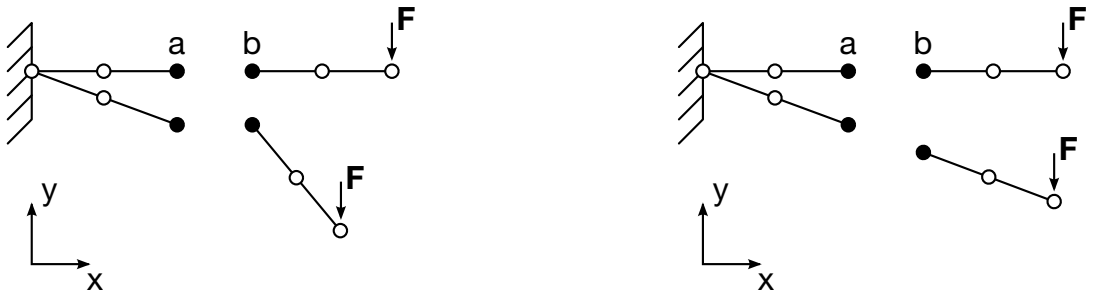


Figure 10-39. Two different ways to constrain node *a* and *b*. For rigid-body type situations this card, *CONSTRAINED_NODE_SET, may lead to un-physical results.

VARIABLE	DESCRIPTION
DOF	Applicable degrees-of-freedom: EQ.1: *x*-translational degree-of-freedom EQ.2: *y*-translational degree-of-freedom EQ.3: *z*-translational degree-of-freedom EQ.4: *x* and *y*-translational degrees-of-freedom EQ.5: *y* and *z*-translational degrees-of-freedom EQ.6: *z* and *x*-translational degrees-of-freedom EQ.7: *x*, *y*, and *z*-translational degrees-of-freedom EQ.8: Electric potential of piezoelectric material
TF	Failure time for nodal constraint set

Remarks:

1. **Mass.** The masses of the nodes are summed up to determine the total mass of the constrained set.

2. **Nodal Rigid Body.** Note that the definition of a nodal rigid body is not possible with this input. For nodal rigid bodies the keyword input *CONSTRAINED_-NODAL_RIGID_BODY_*OPTION* must be used.

3. **Failure Time.** When the failure time, TF, is reached the nodal constraint becomes inactive, and the constrained nodes may move freely.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_NODE_SET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Constrain all the nodes in a nodal set to move equivalently
$  in the z-direction.
$
*CONSTRAINED_NODE_SET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
      nsid       dof        tf
         7         3      10.0
$
$     nsid = 7    nodal set ID number, requires a *SET_NODE_option
$      dof = 3    nodal motions are equivalent in z-translation
$      tf  = 3    at time=10. the nodal constraint is removed
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

*CONSTRAINED_NODE_TO_NURBS_PATCH_{OPTION}

Purpose: To add additional massless nodes to the surface of a NURBS patch. The motion of the nodes is governed by the NURBS patch. Forces applied to the nodes are distributed to the NURBS patch. Penalty method is used to handle the displacement boundary conditions CON (see below) on the specified nodes.

To specify node sets instead of individual nodes use the option:

SET

Card 1	1	2	3	4	5	6	7	8
Variable	PATCHID	NSID	CON	CID	SF	DBFLG		
Type	I	I	I	I	F	I		
Default	none	none	0	none	1.0	0		

VARIABLE	DESCRIPTION
PATCHID	Patch ID.
NSID	Nodal set ID or node ID depending on the *OPTION*.
CON	Constraint parameter for extra node(s) of NSID. EQ.000000: no constraint EQ.100000: constrained x translation EQ.010000: constrained y translation EQ.001000: constrained z translation EQ.000100: constrained x rotation EQ.000010: constrained y rotation EQ.000001: constrained z rotation Any combination of local constraints can be specified by adding the number 1 into the corresponding column. For example, "1110" means constrained z-translation, x-rotation and y-rotation.
CID	Coordinate system ID for constraint
SF	Penalty force scale factor for the penalty-based constraint

VARIABLE	DESCRIPTION
DBFLG	Discrete beam flag. If CON = 0 and displacement boundary conditions are applied to nodes specified in NSID, then this flag must be set to 1. When DBFLG = 1, discrete beam elements are created to connect nodes in NSID to the patch.

*CONSTRAINED_POINTS

Purpose: Constrain two points with the specified coordinates connecting two shell elements at locations other than nodal points. In this option, the penalty method is used to constrain the translational and rotational degrees-of-freedom of the points. Force resultants are written into the swforc ASCII file for post-processing.

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

Card 2	1	2	3	4	5	6	7	8	9	10
Variable	EID1	X1		Y1		Z1				
Type	I	F		F		F				
Default	none	0.		0.		0.				

Card 3	1	2	3	4	5	6	7	8	9	10
Variable	EID2	X2		Y2		Z2				
Type	I	F		F		F				
Default	none	0.		0.		0.				

Card 4	1	2	3	4	5	6	7	8
Variable	PSF	FAILA	FAILS	FAILM				
Type	F	F	F	F				
Default	1.0	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
CID	Constrained points ID.
EID*i*	Shell element ID, *i* = 1, 2.
X*i*, Y*i*, Z*i*	Coordinates of the constrained points, *i* = 1, 2.
PSF	Penalty scale factor (Default = 1.0).
FAILA	Axial force resultant failure value, no failure if zero.
FAILS	Shear force resultant failure value, no failure if zero.
FAILM	Moment resultant failure value, no failure if zero.

***CONSTRAINED_RIGID_BODIES**_{OPTION}

Available options include:

<BLANK>

SET

For SET option, PIDC refers to a part set.

Purpose: Merge two rigid bodies. One rigid body, called the constrained rigid body, is merged into another one, called the lead rigid body. This command applies to parts comprised of *MAT_RIGID but not to nodal rigid bodies (*CONSTRAINED_NODAL_-RIGID_BODY).

Card 1	1	2	3	4	5	6	7	8
Variable	PIDL	PIDC	IFLAG					
Type	I	I	I					
Default	none	none	0					

VARIABLE	DESCRIPTION
PIDL	Lead rigid body part ID; see *PART.
PIDC	Constrained rigid body part ID (see *PART) or constrained rigid body part set ID for the SET keyword option (see *SET_PART)
IFLAG	This flag is meaningful *if and only if* the inertia properties of the lead part, PIDL, are defined in *PART_INERTIA. See Remark 1. EQ.1: Update the center-of-gravity, the translational mass, and the inertia matrix of PIDL to reflect its merging with the constrained rigid body (PIDC). EQ.0: The merged PIDC will not affect the properties defined in *PART_INERTIA for PIDL since the properties are assumed to already account for merged parts. If the properties are not defined in a *PART_INERTIA definition, the inertia properties of PIDC will be computed from its nodal masses.

Remarks:

1. **Inertial Properties.** The constrained rigid body is merged to the lead rigid body. Unless the inertial properties of the lead rigid body are defined with *PART_INERTIA, the inertial properties computed by LS-DYNA are based on the combination of the lead rigid body plus all the rigid bodies which are constrained to it. If the inertial properties of the lead are specified with *PART_INERTIA, then the treatment of those properties depends on IFLAG. Note that a lead rigid body may have many rigid bodies constrained to it, but it may not be constrained to another rigid body.

2. **Common Nodes.** Independent rigid bodies must not share common nodes since each rigid body updates the motion of its nodes independently of the other rigid bodies. If common nodes exist between rigid bodies, the rigid bodies sharing the nodes must be merged.

3. **Completely Separated Rigid Bodies.** Completely separated rigid bodies that do not share any common nodal points or boundaries may also be merged. All actions valid for the lead rigid body, such as constraints and given velocity, are now also valid for the newly-created rigid body.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_RIGID_BODIES
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Rigidly connect parts 35, 70, 71, and 72 to part 12.
$  All parts must be defined as rigid.
$
$  This example is used to make a single rigid body out of the five parts
$  that compose the back end of a vehicle.  This was done to save cpu time
$  and was determined to be valid because the application was a frontal
$  impact with insignificant rear end deformations.  (The cpu time saved
$  was from making the parts rigid, not from merging them - merging was
$  more of a convenience in this case for post processing, for checking
$  inertial properties, and for joining the parts.)
$
*CONSTRAINED_RIGID_BODIES
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     pidl      pidc
        12        35
        12        70
        12        71
        12        72
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

*CONSTRAINED_RIGID_BODY_INSERT

Purpose: This keyword is for modeling die inserts. One rigid body, called the constrained rigid body, is constrained to move with another rigid body, called the lead rigid body, in all directions except for one.

Card 1	1	2	3	4	5	6	7	8
Variable	ID	PIDL	PIDC	COORDID	IDIR			
Type	I	I	I	I	I			
Default	none	none	none	none	3			

Card 2	1	2	3	4	5	6	7	8
Variable	MFLAG	MCID	DEATHM					
Type	I	I	F					
Default	0	0	↓					

Card 3	1	2	3	4	5	6	7	8
Variable	PARTB	DEATHB						
Type	I	F						
Default	0	0.0						

VARIABLE	DESCRIPTION
ID	Insert ID
PIDL	Lead (die) rigid body part ID (see *PART)
PIDC	Constrained (die insert) rigid body part ID (see *PART)

VARIABLE	DESCRIPTION
COORDID	Coordinate system ID
IDIR	Direction in which the insert moves independently of the die: EQ.1: Local *x*-direction EQ.2: Local *y*-direction EQ.3: Local *z*-direction (default)
MFLAG	Motion flag: EQ.0: Relative motion is unconstrained. EQ.1: The displacement of the insert relative to the die is imposed. EQ.2: The velocity of the insert relative to the die is imposed. EQ.3: The acceleration of the insert relative to the die is imposed.
MCID	Curve defining the motion of the die insert relative to the die
DEATHM	Death time of the imposed motion. If it is equal to 0.0, the motion is imposed for the entire analysis.
PARTB	Part ID for a discrete beam connected between the insert and die
DEATHB	Death time for the discrete beam specified by PARTB

Remarks:

1. **Time Integrators.** This capability is supported by both the implicit and explicit time integrators; however, the joint death time, DEATHM, feature works only for explicit integration with the penalty method.

2. **Joint Reaction Forces.** The translational joint constraining the die and the die insert are automatically generated. The joint reaction forces will appear in the jntforc output file.

3. **Translational Degree of Freedom.** The translational motor constraining the remaining translational degree of freedom is also automatically generated, and its reaction forces also appear in the jntforc output file.

4. **Discrete Beam.** The automatically generated beam has its data written to the d3plot file and all of the optional appropriate output files.

*CONSTRAINED_RIGID_BODY_STOPPERS

Purpose: Rigid body stoppers provide a convenient way of controlling the motion of rigid tooling in metal forming applications. The motion of a "lead" rigid body is limited by load curves. This option will stop the motion based on a time dependent constraint. The stopper overrides prescribed motion boundary conditions (except relative displacement) operating in the same direction for both the lead and constrained rigid bodies. See Figure 10-40.

Card 1	1	2	3	4	5	6	7	8
Variable	PID	LCMAX	LCMIN	PSIDMX	PSIDMN	LCVMNX	DIR	VID
Type	I	I	I	I	I	I	I	I
Default	none	0	0	0	0	0	none	0

Card 2	1	2	3	4	5	6	7	8
Variable	TB	TD		STIFF				
Type	F	F		F				
Default	0	1021		0.0				

VARIABLE	DESCRIPTION
PID	Part ID of lead rigid body (see *PART)
LCMAX	Load curve ID defining the maximum coordinate or displacement as a function of time (see *DEFINE_CURVE): LT.0: Load curve ID |LCMAX| provides an upper bound for the displacement of the rigid body. EQ.0: No limitation of the maximum displacement. GT.0: Load curve ID LCMAX provides an upper bound for the position of the rigid body center of mass.
LCMIN	Load curve ID defining the minimum coordinate or displacement as a function of time (see *DEFINE_CURVE):

VARIABLE	DESCRIPTION
	LT.0: Load curve ID |LCMIN| defines a lower bound for the displacement of the rigid body. EQ.0: No limitation of the minimum displacement. GT.0: Load curve ID LCMIN defines a lower bound for the position of the rigid body center of mass.
PSIDMX	Optional part set ID of rigid bodies that are constrained in the maximum coordinate direction to the lead rigid body. The part set definition (see *SET_PART_COLUMN) may be used to define the closure distance (D_1 and D_2in Figure 10-40) which activates the constraint. The constraint does not begin to act until the lead rigid body stops. If the distance between the lead rigid body is greater than or equal to the closure distance, the constrained rigid body motion away from the lead rigid body also stops. However, the constrained rigid body is free to move towards the lead rigid body. If the closure distance is input as zero (0.0), then the constrained rigid body stops when the lead stops.
PSIDMN	Optional part set ID of rigid bodies that are constrained in the minimum coordinate direction to the lead rigid body. The part set definition, (see *SET_PART_COLUMN) may be used to define the closure distance (D_1 and D_2 in Figure 10-40) which activates the constraint. The constraint does not begin to act until the lead rigid body stops. If the distance between the lead rigid body is less than or equal to the closure distance, the constrained rigid body motion towards the lead rigid body also stops. However, the constrained rigid body is free to move away from the lead rigid part. If the closure distance is input as zero (0.0), then the constrained rigid body stops when the lead stops.
LCVMX	Load curve ID which defines the maximum absolute value of the velocity as a function of time that is allowed for the lead rigid body. See *DEFINE_CURVE: EQ.0: no limitation on the velocity.
DIR	Direction stopper acts in: EQ.1: x-translation, EQ.2: y-translation, EQ.3: z-translation, EQ.4: Arbitrary, defined by vector VID (see below),

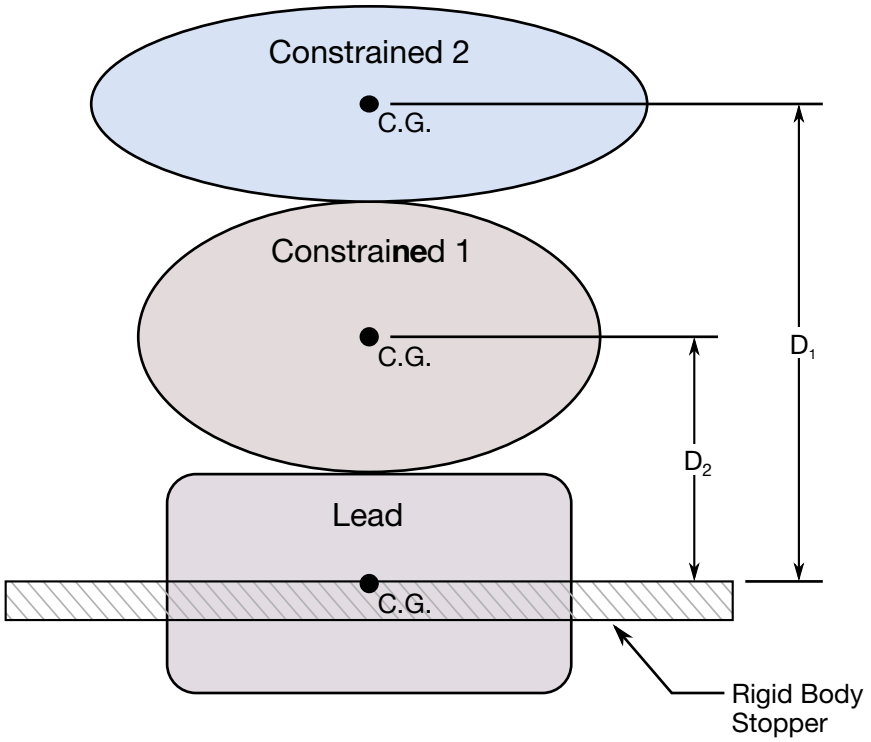


Figure 10-40. When the lead rigid body reaches the rigid body stopper, the velocity component into the stopper is set to zero. Constrained rigid bodies 1 and 2 also stop if the distance between their mass centers and the lead rigid body is less than or equal to the input values D_1 and D_2, respectively.

VARIABLE	DESCRIPTION
	EQ.5: x-axis rotation, EQ.6: y-axis rotation, EQ.7: z-axis rotation, EQ.8: Arbitrary, defined by vector VID (see below).
VID	Vector for arbitrary orientation of stopper, see *DEFINE_VECTOR.
TB	Time at which stopper is activated.
TD	Time at which stopper is deactivated.
STIFF	Augmentation stiffness for implicit; see Remark 2.

Remarks:

1. **Controlling the motion in an arbitrary direction.** By defining the optional part sets in the minimum or maximum coordinate direction (PSIDMX or PSIDMN), the motion can be controlled in an arbitrary direction.

2. **Augmentation stiffness.** For implicit, inequality constraints are imposed by using an augmented Lagrangian multiplier technique, where the augmentation stiffness can be specified with field STIFF. If STIFF is zero, then it is computed internally based on the loads and displacements associated with the stoppers. If this results in bad behavior, the stiffness may be selected so that STIFF = FORCE/DISP, where FORCE is a characteristic load and DISP is a characteristic length. A too large value may lead to inaccuracies and a too small value may lead to convergence problems. It is worth mentioning that this approach renders an indefinite matrix, and the number of negative eigenvalues seen by the linear solver is 2 (one constraint in positive and one in negative direction) per rigid body stopper.

***CONSTRAINED_RIVET**_{OPTION}

To define an ID for the rivet, the following option is available:

<BLANK>

ID

If the ID is defined, an additional card is required.

Purpose: Define massless rivets between non-contiguous nodal pairs. The nodes must not have the same coordinates. The action is such that the distance between the two nodes is kept constant throughout any motion. No failure can be specified.

ID Card. Additional card for the ID keyword option.

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

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	TF					
Type	I	I	F					
Default	none	none	10^{20}					
Remarks	1		2					

VARIABLE	DESCRIPTION
RID	Optional rivet ID.
N1	Node ID
N2	Node ID

VARIABLE	DESCRIPTION
TF	Failure time for nodal constraint set.

Remarks:

1. **Multiple Constraints.** Nodes connected by a rivet cannot be members of another constraint set that constrain the same degrees-of-freedom, a tied interface, or a rigid body, that is, nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also, care must be taken to ensure that single point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.

2. **Failure Time.** When the failure time, TF, is reached the rivet becomes inactive and the constrained nodes may move freely.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_RIVET
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Connect node 382 to node 88471 with a massless rivet.
$
*CONSTRAINED_RIVET
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$       n1         n2         tf
       382     88471        0.0
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

***CONSTRAINED_SHELL_IN_SOLID**_*{OPTION1}_{OPTION2}*

This keyword can take the following two forms:

*CONSTRAINED_SHELL_IN_SOLID

*CONSTRAINED_SHELL_IN_SOLID_PENALTY

To define a coupling ID and heading for this keyword, the following options are available:

ID

TITLE

Purpose: Provide either constraint-based or penalty-based coupling between shells and the solids/thick shells in which the shells are embedded.

To use the constraint-based coupling, this keyword takes the form of *CONSTRAINED_SHELL_IN_SOLID. It constrains shell structures to move with Lagrangian solids/thick shells. This keyword constrains both acceleration and velocity. This keyword, together with *CONSTRAINED_BEAM_IN_SOLID, intends to sidestep certain limitations of the CTYPE = 2 implementation in *CONSTRAINED_LAGRANGE_IN_SOLID. Notable features of this keyword include:

1. **Tetrahedral and pentahedral solid elements are supported.** They are no longer treated as degenerated hexahedra as in the CTYPE = 2 implementation.

2. **Velocity/Fixed boundary condition.** The CTYPE = 2 implementation failed to constrain shell nodes that were buried inside elements whose nodes had velocity/fixed boundary conditions prescribed.

3. **Optimized Sorting.** Sorting subroutine is optimized for larger problems to achieve better performance and less memory usage.

To use the penalty-based coupling, this keyword takes the form of *CONSTRAINED_SHELL_IN_SOLID_PENALTY. A penalty spring is attached between coupling points on the shell and in the solid/thick shell element. Penalty spring stiffness is calculated based on the geometric mean of shell and solid's bulk modulus. The magnitude of this coupling force can be controlled using PSSF (penalty spring stiffness scale factor). This penalty coupling conserves kinetic energy much better in transient problems such as blast loading.

If a title is not defined, LS-DYNA will automatically create an internal title for this coupling definition.

Title Card. Additional card for TITLE and ID keyword options.

Card ID	1	2	3	4	5	6	7	8
Variable	COUPID	TITLE						
Type	I	A70						

Card 1	1	2	3	4	5	6	7	8
Variable	SHSID	SSID	SHSTYP	SSTYP				
Type	I	I	I	I				
Default	none	none	0	0				

Card 2	1	2	3	4	5	6	7	8
Variable	START	END				PSSF		
Type	F	F				F		
Default	0.	10^{20}				0.1		

VARIABLE	DESCRIPTION
COUPID	Coupling (card) ID number (I10). If not defined, LS-DYNA will assign an internal coupling ID based on the order of appearance in the input deck.
TITLE	A description of this coupling definition.
SHSID	Set ID defining a part or part set ID of the Lagrangian shell structure (see *PART, *SET_PART).
SSID	Set ID defining a part or part set ID of the Lagrangian solid elements or thick shell elements (see *PART or *SET_PART).

VARIABLE	DESCRIPTION
SHSTYP	Set type of SHSID: EQ.0: Part set ID (PSID) EQ.1: Part ID (PID)
SSTYP	Set type of SSID: EQ.0: Part set ID (PSID) EQ.1: Part ID (PID)
START	Start time to activate the coupling: LT.0: Start time is set to |START|. When negative, start time is followed during the dynamic relaxation phase of the calculation. After the completion of dynamic relaxation, coupling is activated regardless of the value of END. EQ.0: Start time is inactive, meaning coupling is always active GT.0: If END = -9999, START is interpreted as the curve or table ID defining multiple pairs of start-time and end-time. Otherwise, if END > 0, start time applies both during and after dynamic relaxation.
END	End time to deactivate the coupling: LT.0: If END = -9999, START is interpreted as the curve or table ID defining multiple pairs of start-time and end-time. Otherwise, negative END indicates that coupling is inactive during dynamic relaxation. After dynamic relaxation the start and end times are followed and set to |START| and |END|, respectively. EQ.0: END defaults to 10^{20}. GT.0: END sets the time at which the coupling is deactivated.
PSSF	Penalty spring stiffness scale factor. Only available in penalty form.

*CONSTRAINED_SHELL_TO_SOLID

Purpose: Define a tie between a shell edge and solid elements. Nodal rigid bodies can perform the same function and may also be used.

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

VARIABLE	DESCRIPTION
NID	Shell node ID
NSID	Solid nodal set ID, see *SET_NODE_*OPTION*.

Remarks:

The shell-brick interface, an extension of the tied surface capability, ties regions of hexahedron elements to regions of shell elements. A shell node may be tied to up to nine brick nodes lying along the tangent vector to the nodal fiber. See Figure 10-41. During the

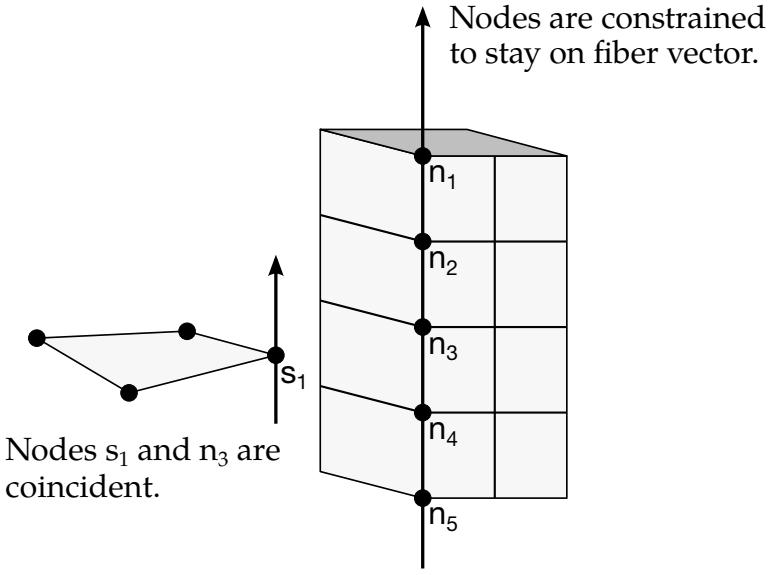


Figure 10-41. The interface between shell elements and solids ties a shell node, s_1, to a line of nodes on the solid elements, n_1 to n_5. It is very important for the nodes to be aligned.

calculation, the brick nodes thus constrained, must lie along the fiber but can move relative to each other in the fiber direction. The shell node stays on the fiber at the same relative spacing between the first and last brick node. The brick nodes must be input in the order in which they occur, in either the plus or minus direction, as one moves along the shell node fiber.

This feature is intended to tie four node shells to eight node shells or solids; it is not intended for tying eight node shells to eight node solids.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_SHELL_TO_SOLID
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Tie shell element, at node 329, to a solid element at node 203.
$    - nodes 329 and 203 are coincident
$
$  Additionally, define a line of nodes on the solids elements, containing
$  node 203, that must remain in the same direction as the fiber of the shell
$  containing node 329.  In other words:
$
$    - Nodes 119, 161, 203, 245 and 287 are nodes on a solid part that
$       define a line on that solid part.
$    - This line of nodes will be constrained to remain linear throughout
$       the simulation.
$    - The direction of this line will be kept the same as the fiber of the
$       of the shell containing node 329.
$
*CONSTRAINED_SHELL_TO_SOLID
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$      nid      nsid
       329         4
$
*SET_NODE_LIST
$      sid
         4
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       119       161       203       245       287
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

*CONSTRAINED_SOIL_PILE_{OPTION1}_{OPTION2}

Purpose: Define penalty-based coupling between 1D beam elements and 3D solid elements. The beam elements must not share nodes with the solid elements, and the beam element mesh does not have to align with the solid element mesh. The functionality is similar to *CONSTRAINED_BEAM_IN_SOLID, except that *CONSTRAINED_SOIL_-PILE is specific to geotechnical applications, such as piles in soil or anchors in rock. For general applications in which beam elements must be coupled with solids (such as reinforcement bars in concrete), *CONSTRAINED_BEAM_IN_SOLID is recommended due to its simpler input. See the general modeling requirements for *CONSTRAINED_SOIL_-PILE in Remarks 1 through 4.

The data cards described below specify the nonlinear coupling characteristics, expressing the resisting stress on the outer surface of the pile as a function of displacement of the pile relative to the soil. Movement in the axial (sliding) direction has a different coupling characteristic from movement perpendicular to the axis of the pile. In geotechnical terminology, the perpendicular coupling characteristics are sometimes described as "P-y springs" while the axial characteristics are described as "P-z springs". Additionally, a coupling characteristic is required for axial movement of the base of the pile, sometimes referred to as the "end-bearing" or the "toe" of the pile. Thus, the overall coupling depends on a set of three nonlinear coupling characteristics (Base, Axial and Perpendicular).

Where piles pass through layers of soils with different material properties, different coupling properties may be appropriate for each soil layer. To model this, include one *CONSTRAINED_SOIL_PILE keyword containing multiple sets of coupling properties. Each set of coupling properties is associated with a particular soil part ID or part set ID. LS-DYNA will then select the coupling properties for each node of the pile according to the part ID of the soil element within which that pile node is situated. LS-DYNA will automatically find the node at the free end of each pile for allocation of Base coupling properties (see description of LOCAL below for which free end will be taken as the Base).

OPTION1 specifies the format in which the coupling properties are defined on Cards 3 through 6. Available options are:

<BLANK>

CONSTANTS

CURVES

Selecting CURVES allows you to specify the properties with load curves while CONSTANTS allows you to specify the coupling with constant values. The input for the CURVES option is simpler than that of the CONSTANTS option, but the CONSTANTS option reduces the need to create load curves. If *OPTION1* is unset (<BLANK>), then the input is the same as for CONSTANTS, except that Card 2 is omitted. Card 2 allows you to add damping to the axial coupling shear stress and to pick which side of the pile is the

Base. We recommend using CONSTANTS or CURVES for *OPTION1* instead of not setting it. <BLANK> is included solely for backwards compatibility.

OPTION2 specifies whether the soil parts to which the coupling properties apply are defined by part ID (see *PART) or by part set ID (see *SET_PART). Available options are:

<BLANK>

SET

If unset (<BLANK>), part IDs are assumed.

Card Summary:

Card Sets. Include one of Cards 1 and 2 (unless *OPTION1* is unset) total. Then for each soil part/part set include one set of either Cards 3a through 6a (or possibly 7a) or Cards 3b through 6b depending on *OPTION1*. Include as many sets as required to define the coupling properties for all the relevant soil layers. This input ends with the next keyword ("*") card.

Card 1. This card is required.

PBSID	DIAM		PIDNS	PIDNB	ERROR	NRING	NRINGB

Card 2. Include this card unless OPTION1 is unset (<BLANK>).

DAMP	LOCAL	INSTRF	TIMSTR	ICD7A			

Card 3a. Include this card when OPTION1 is set to CONSTANTS or not used (<BLANK>). Include as many sets of Card 3a through Card 6a (or possibly 7a) as needed.

PID/PSID	ACU	BCU	LCCU	ASTIFFS	BSTIFFS	ASTIFFB	ZREF

Card 4a. Include this card when OPTION1 is set to CONSTANTS or not used.

KBCON	KBCU	KBSX	KBSY	KBSZ	BSTFAC	BHYPER	BLC

Card 5a. Include this card when OPTION1 is set to CONSTANTS or not used.

KVCON	KVCU	KVSX	KVSY	KVSZ	VSTFAC	VHYPER	VLC

Card 6a. Include this card when OPTION1 is set to CONSTANTS or not used.

KHCON	KHCU	KHSX	KHSY	KHSZ	HSTFAC	HHYPER	HLC

Card 7a. Include this card only when OPTION1 is set to CONSTANTS and ICD7A on Card 2 is nonzero.

KHFCON	KHFCU	KHFSX	KHFSY	KHFSZ	FRATIO		

Card 3b. Include this card when OPTION1 is set to CURVES. Include as many sets of Card 3b through Card 6b as needed.

PID/PSID	ZREF						

Card 4b. Include this card when OPTION1 is set to CURVES.

BLCZ	BLC	BLCSH	BLCSV				

Card 5b. Include this card when OPTION1 is set to CURVES.

VLCZ	VLC	VLCSH	VLCSV				

Card 6b. Include this card when OPTION1 is set to CURVES.

HLCZ	HLC	HLCSH	HLCSV	HLCFD			FRATIO

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	PBSID	DIAM		PIDNS	PIDNB	ERROR	NRING	NRINGB
Type	I	F		I	I	I	I	I
Default	none	0.0		auto	auto	0	1	↓

VARIABLE	DESCRIPTION
PBSID	Part set ID containing beam elements for coupling (the piles). See Remarks 2 and 3.
DIAM	Pile diameter (optional). If zero or blank, the pile diameter will be taken automatically from the section properties of the beam element. See Remarks 3 and 13.
PIDNS	ID for automatically generated part containing visualization elements for perpendicular and axial coupling. If not specified, LS-

VARIABLE	DESCRIPTION
	DYNA will assign a part ID. See Remarks 14 and 15.
PIDNB	ID for automatically generated part containing visualization elements for base coupling. If not specified, LS-DYNA will assign a part ID. See Remarks 14 and 15.
ERROR	Action taken if any coupling point is not constrained within a soil element: EQ.0: Stop with an error message. EQ.1: Warn and continue.
NRING	Number of coupling points around circumference at each pile node (see Remarks 11, 12 and 13): EQ.1: One coupling point coincident with pile node GT.1: NRING coupling points equally spaced around the circumference of the pile
NRINGB	Number of extra rings of coupling points on base, in addition to those around the pile circumference. By default, NRINGB is chosen automatically to distribute the base stress as uniformly as possible (see Remarks 11 and 12).

This card is included unless *OPTION1* is unset (<BLANK>).

Card 2	1	2	3	4	5	6	7	8
Variable	DAMP	LOCAL	INSTRF	TIMSTR	ICD7A			
Type	F	I	I	F	I			
Default	0.0	1	0	0.0	0			

VARIABLE	DESCRIPTION
DAMP	Optional damping coefficient for Axial coupling (stress/velocity units). An additional axial coupling shear stress equal to DAMP times the axial velocity of the pile relative to the soil will be generated. See Remark 17.
LOCAL	Flag to identify which free end of a pile is treated as the Base:

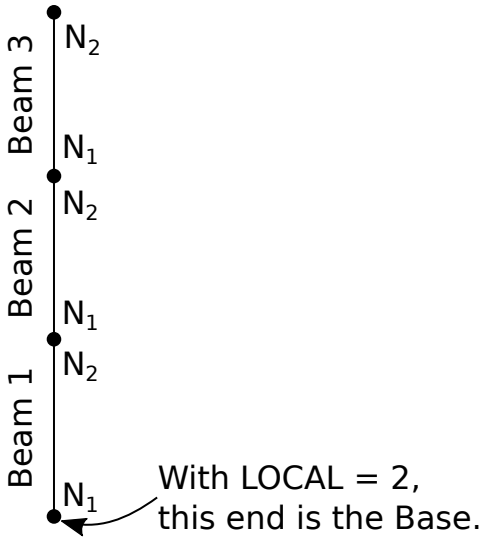


Figure 10-42. Example of a beam element topology in a pile. We recommend each beam having the same order for N_1 and N_2, so that the Base is not ambiguous for LOCAL = 2.

VARIABLE	DESCRIPTION
	EQ.1: End with the most negative global Z-coordinate EQ.2: End which is Node 1 of the attached beam element topology. See Figure 10-42.
INSTRF	Flag to control definition of soil stress for Cards 4a through 6a and Cards 4b through 6b: EQ.0: Use time-varying soil stress. TIMSTR is ignored. EQ.1: Use time-varying soil stress until time TIMSTR. Then use the soil stress that existed at time TIMSTR for the remainder of the analysis.
TIMSTR	Time at which the effect of soil stress is frozen
ICD7A	If nonzero and *OPTION1* is CONSTANTS, Card 7a will be read.

CONSTANTS General Coupling Properties Card. Include a set of Cards 3a through 6a for each soil part/part set when *OPTION1* is CONSTANTS or unset. Include as many sets as needed. See Remark 6.

Card 3a	1	2	3	4	5	6	7	8
Variable	PID/PSID	ACU	BCU	LCCU	ASTIFFS	BSTIFFS	ASTIFFB	ZREF
Type	I	F	F	I	F	F	F	F
Default	none	0.0	0.0	optional	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION
PID/PSID	Part ID or part set ID (depending on *OPTION2*) containing solid elements for coupling (the soil). See Remarks 4 and 16.
ACU	Constant term in depth-dependence formula. Units of stress.
BCU	Coefficient on relative Z-coordinate in depth-dependence formula. Units of stress/length. Note that soil strengths (and therefore coupling properties) generally increase with depth, meaning they increase with an increasingly negative Z-coordinate. Therefore, this term is usually negative.
LCCU	Optional load curve ID giving stress (stress units) as a function of relative Z-coordinate (length units). If defined, LCCU overrides ACU and BCU. Note that "increasing depth" corresponds to "increasingly negative relative Z-coordinate".
ASTIFFS	Generic stiffness term. Units of stress / length.
BSTIFFS	Generic Z-coordinate-dependent stiffness term. Units of stress / length^2
ASTIFFB	Base stiffness. Units of stress / length.
ZREF	Reference Z-coordinate to calculate "relative Z-coordinate". See Remark 16.

CONSTANTS Base Coupling Properties Card. Include a set of Cards 3a through 6a for each soil part/part set when *OPTION1* is CONSTANTS or unset. See Remarks 6 and 8.

Card 4a	1	2	3	4	5	6	7	8
Variable	KBCON	KBCU	KBSX	KBSY	KBSZ	BSTFAC	BHYPER	BLC
Type	F	F	F	F	F	F	F	I
Default	0.0	0.0	0.0	0.0	0.0	1.0	0.0	0

VARIABLE	DESCRIPTION
KBCON	Base coupling, constant term (stress units)
KBCU	Base coupling, coefficient for Cu (dimensionless)
KBSX	Base coupling, coefficient for effective global *X*-stress (dimensionless)
KBSY	Base coupling, coefficient for effective global *Y*-stress (dimensionless)
KBSZ	Base coupling, coefficient for effective global *Z*-stress (dimensionless)
BSTFAC	Base coupling, factor on elastic stiffness (dimensionless)
BHYPER	Base coupling, hyperbolic curve limit (dimensionless)
BLC	Base coupling, load curve ID for dimensionless factor on stress as a function of displacement

CONSTANTS Axial Coupling Properties Card. Include a set of Cards 3a through 6a for each soil part/part set when *OPTION1* is CONSTANTS or unset. See Remarks 6 and 9.

Card 5a	1	2	3	4	5	6	7	8
Variable	KVCON	KVCU	KVSX	KVSY	KVSZ	VSTFAC	VHYPER	VLC
Type	F	F	F	F	F	F	F	I
Default	0.0	0.0	0.0	0.0	0.0	1.0	0.0	0

VARIABLE	DESCRIPTION
KVCON	Axial coupling, constant term (stress units)
KVCU	Axial coupling, coefficient for Cu (dimensionless)
KVSX	Axial coupling, coefficient for effective global *X*-stress (dimensionless)
KVSY	Axial coupling, coefficient for effective global *Y*-stress (dimensionless)
KVSZ	Axial coupling, coefficient for effective global *Z*-stress (dimensionless)
VSTFAC	Axial coupling, factor on elastic stiffness (dimensionless)
VHYPER	Axial coupling, hyperbolic curve limit (dimensionless)
VLC	Axial coupling, load curve ID for dimensionless factor on stress as a function of displacement

CONSTANTS Perpendicular Coupling Properties Card. Include a set of Cards 3a through 6a for each soil part/part set when *OPTION1* is CONSTANTS or unset. See Remarks 6 and 10.

Card 6a	1	2	3	4	5	6	7	8
Variable	KHCON	KHCU	KHSX	KHSY	KHSZ	HSTFAC	HHYPER	HLC
Type	F	F	F	F	F	F	F	I
Default	0.0	0.0	0.0	0.0	0.0	1.0	0.0	none

VARIABLE	DESCRIPTION
KHCON	Perpendicular coupling, constant term (stress units)
KHCU	Perpendicular coupling, coefficient for Cu (dimensionless)
KHSX	Perpendicular coupling, coefficient for effective global *X*-stress (dimensionless)
KHSY	Perpendicular coupling, coefficient for effective global *Y*-stress (dimensionless)
KHSZ	Perpendicular coupling, coefficient for effective global *Z*-stress (dimensionless)
HSTFAC	Perpendicular coupling, factor on elastic stiffness (dimensionless)
HHYPER	Perpendicular coupling, hyperbolic curve limit (dimensionless)
HLC	Perpendicular coupling, load curve ID for dimensionless factor on stress as a function of displacement

CONSTANTS Perpendicular Coupling Friction Properties Card. This card defines resistance to perpendicular motion during unload/reload ("post-hole" behavior), described as "friction stress" below. Include a Card 7a after each set of Cards 3a through 6a when *OPTION1* is CONSTANTS and ICD7A on Card 2 is nonzero. See Remarks 6 and 10.

Card 7a	1	2	3	4	5	6	7	8
Variable	KHFCON	KHFCU	KHFSX	KHFSY	KHFSZ	FRATIO		
Type	F	F	F	F	F	F		
Default	0.0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
KHFCON	Perpendicular coupling friction stress, constant term (stress units)
KHFCU	Perpendicular coupling friction stress, coefficient for Cu (dimensionless)
KHFSX	Perpendicular coupling friction stress, coefficient for effective global *X*-stress (dimensionless)
KHFSY	Perpendicular coupling friction stress, coefficient for effective global *Y*-stress (dimensionless)
KHFSZ	Perpendicular coupling friction stress, coefficient for effective global *Z*-stress (dimensionless)
FRATIO	Perpendicular coupling friction stress as a ratio of the maximum perpendicular coupling stress derived from KHCON through KHSZ on Card 6a. If defined, FRATIO overrides KHFCON through KHFSZ above. FRATIO provides an alternative way to define the friction stress which is simpler than KHFCON through KHFSZ

CURVES General Coupling Properties Card. Include a set of Cards 3b through 6b for each soil part/part set when *OPTION1* is CURVES. Include as many sets as needed.

Card 3b	1	2	3	4	5	6	7	8
Variable	PID/PSID	ZREF						
Type	I	F						
Default	none	0.0						

VARIABLE	DESCRIPTION
PID/PSID	Part ID or part set ID (depending on *OPTION2*) containing solid elements for coupling (the soil). See Remarks 4 and 16.
ZREF	Reference *Z*-coordinate, used in calculation of "relative *z*-coordinate". For example, ZREF may be located at the soil surface. See Remarks 7 and 16.

CURVES Base Coupling Properties Card. Include a set of Cards 3b through 6b for each soil part/part set when *OPTION1* is CURVES. See Remarks 7 and 8.

Card 4b	1	2	3	4	5	6	7	8
Variable	BLCZ	BLC	BLCSH	BLCSV				
Type	I	I	I	I				
Default	0	0	optional	optional				

VARIABLE	DESCRIPTION
BLCZ	For base coupling, load curve ID defining ultimate strength (stress units) as a function of relative *Z*-coordinate (length units)
BLC	For base coupling, load curve ID containing normalized mobilization curve: dimensionless factor on stress as a function of displacement
BLCSH	For base coupling, optional load curve ID containing coefficient for initial effective horizontal soil stress (dimensionless) as a function

VARIABLE	DESCRIPTION
	of relative Z-coordinate
BLCSV	For base coupling, optional load curve ID containing coefficient for initial effective vertical soil stress (dimensionless) as a function of relative Z-coordinate

CURVES Axial Coupling Properties Card. Include a set of Cards 3b through 6b for each soil part/part set when *OPTION1* is CURVES. See Remarks 7 and 9.

Card 5b	1	2	3	4	5	6	7	8
Variable	VLCZ	VLC	VLCSH	VLCSV				
Type	I	I	I	I				
Default	0	0	optional	optional				

VARIABLE	DESCRIPTION
VLCZ	For axial coupling, load curve ID defining ultimate strength (stress units) as a function of relative Z-coordinate (length units)
VLC	For axial coupling, load curve ID containing normalized mobilization curve: dimensionless factor on stress as a function of displacement
VLCSH	For axial coupling, optional load curve ID containing coefficient for initial effective horizontal soil stress (dimensionless) as a function of relative Z-coordinate
VLCSV	For axial coupling, optional load curve ID containing coefficient for initial effective vertical soil stress (dimensionless) as a function of relative Z-coordinate

CURVES Perpendicular Coupling Properties Card. Include a set of Cards 3b through 6b for each soil part/part set when *OPTION1* is CURVES. See Remarks 7 and 10.

Card 6b	1	2	3	4	5	6	7	8
Variable	HLCZ	HLC	HLCSH	HLCSV	HLCFD			FRATIO
Type	I	I	I	I	I			F
Default	0	0	optional	optional	optional			0.0

VARIABLE	DESCRIPTION
HLCZ	For perpendicular coupling, load curve ID defining ultimate strength (stress units) as a function of relative Z-coordinate (length units)
HLC	For perpendicular coupling, load curve ID containing normalized mobilization curve: dimensionless factor on stress as a function of displacement
HLCSH	For perpendicular coupling, optional load curve ID containing coefficient for initial effective horizontal soil stress (dimensionless) as a function of relative Z-coordinate
HSCSV	For perpendicular coupling, optional load curve ID containing coefficient for initial effective vertical soil stress (dimensionless) as a function of relative Z-coordinate
HLCFD	For perpendicular coupling, optional load curve ID containing frictional resistance (stress units) during "post-hole" unload/reload as a function of relative Z-coordinate. See Remark 10. With HLCFD there is no way to make the frictional resistance depend on soil stress.
FRATIO	Alternative method of defining frictional resistance for perpendicular coupling during "post-hole" unload/reload. FRATIO is the stress as a ratio of the maximum perpendicular coupling stress derived from HLCZ, HLCSH and HLCSV. If defined, FRATIO overrides HLCFD.

Remarks:

1. **General Requirements.** This keyword is available only for 3D models with the explicit solver. The model's global Z-axis should be vertically upwards. If this is not the case, then it will not be possible to define depth-dependent coupling characteristics or to use the default of the LOCAL field for identifying the Base of piles.

2. **Mesh Requirements.** The beam elements must not share nodes with the solid elements. The beam element mesh does not have to align with the solid element mesh. Each pile must consist of contiguously meshed beam elements. One *CONSTRAINED_SOIL_PILE definition can connect multiple piles to multiple soil layers. Although piles are generally aligned vertically, there is no requirement for this to be the case in the model. We recommend having consistent beam element local axes along a pile. Thus, neighboring elements should have Node 1 and Node 2 the same way around, such as Node 1 always being below Node 2.

3. **Beam Element Properties.** Beam elements must be ELFORM 1 or 2. *CONSTRAINED_SOIL_PILE treats the beams as if they have a solid circular cross-section with diameter DIAM. If DIAM is nonzero, the same value is used for all the beam elements included in the coupling. If DIAM is zero or blank, the diameter of the assumed circular cross-section is calculated for each beam element separately from the area of the cross-section given on the *SECTION card. The actual section shape may be non-circular and/or non-solid, but then the surface area of the pile assumed by *CONSTRAINED_SOIL_PILE will be different from the true surface area. This difference will result in the coupling stress being multiplied by an incorrect area to obtain the force. In these cases, DIAM should be defined such that $\pi \times$ DIAM equals the perimeter of the pile cross-section. For the beam elements, rigid material is not permitted and constraint-based features such as Nodal Rigid Bodies or SPC boundary conditions must not be present on their nodes.

4. **Solid Element Properties.** The solid elements must not have more than 8 nodes. There is no other restriction on element type or material type. Rigid material type is permitted, and there is no restriction regarding constraints or other features present on the nodes of the solids.

5. **Coupling Characteristics: General.** The coupling characteristics are defined in terms of stress on the outer surface of the pile as nonlinear functions of displacement of the pile relative to the soil. Coupling stress is calculated separately at each coupling point (coupling point locations are discussed in Remark 11). The response under monotonically increasing displacement is defined by a "backbone curve", which is defined differently according to whether *OPTION1* is CONSTANTS (Remark 6) or CURVES (Remark 7). Backbone curves for all

three coupling characteristics (Base, Axial and Perpendicular) follow the same pattern but with different input parameters. Unload/reload behavior is described in Remarks 8 through 10.

6. **Backbone Coupling Curves when *OPTION1* is CONSTANTS or Unset.** The coupling follows an elastic/plastic relation with an initial elastic stiffness and a yield stress. The elastic stiffnesses (stress/displacement) for Base, Axial and Perpendicular coupling (S_{Base}, S_{Ax} and S_{Perp}, respectively) are defined as follows:

$$S_{\text{Base}} = \text{ASTIFFB} \times \text{BSTFAC}$$

$$S_{\text{Ax}} = (\text{ASTIFFB} + \text{BSTIFF} \times z_r) \times \text{VSTFAC}$$

$$S_{\text{Perp}} = (\text{ASTIFF} + \text{BSTIFF} \times z_r) \times \text{HSTFAC}$$

Here z_r is the relative Z-coordinate defined as $Z_0 - \text{ZREF}$. Z_0 is the initial Z-coordinate of the pile node.

The yield stresses for Base, Axial and Perpendicular coupling ($\sigma_{Y,\,\text{Base}}$, $\tau_{Y,\,\text{Ax}}$ and $\sigma_{Y,\,\text{Perp}}$, respectively) are defined as follows:

$$\sigma_{Y,\,\text{Base}} = f(d) \times \left\{\text{KBCON} + \text{KBCU} \times \text{Cu}(z_r) + \text{KBSX} \times \sigma'_x + \text{KBSY} \times \sigma'_y + \text{KBSZ} \times \sigma'_z\right\}$$

$$\tau_{Y,\,\text{Ax}} = f(d) \times \left\{\text{KVCON} + \text{KVCU} \times \text{Cu}(z_r) + \text{KVSX} \times \sigma'_x + +\text{KVSY} \times \sigma'_y + \text{KVSZ} \times \sigma'_z\right\}$$

$$\sigma_{Y,\text{Perp}} = f(d) \times \left\{\text{KHCON} + \text{KHCU} \times \text{Cu}(z_r) + \text{KHSX} \times \sigma'_x + \text{KHSY} \times \sigma'_y + \text{KHSZ} \times \sigma'_z\right\}$$

Here d is relative displacement (see Remark 12), z_r is the relative Z-coordinate as defined above, and σ'_x, σ'_y and σ'_z are the soil stresses in the global X, Y and Z directions, respectively. Depending on INSTR and TIMSTR (see Card 2), either time-varying soil stresses or the soil stresses at time TIMSTR are used in this calculation. Note that INSTR and TIMSTR were introduced in R14. In R13, the CONSTANTS keyword option always used current (time-varying) soil stress. If pore water is modeled (see *CONTROL_PORE_FLUID), then these are "effective stresses", meaning the stress generated by the material model excluding any pore pressure. Note that *CONSTRAINED_SOIL_PILE does not require pore pressure to be modelled.

The depth-dependence function, $\text{Cu}(z_r)$, is defined as follows: If LCCU is zero or blank, $\text{Cu}(z_r) = \text{ACU} + \text{BCU} \times z_r$. Otherwise, $\text{Cu}(z_r) = \text{LCCU}(z_r)$.

The function $f(d)$ has three possible forms, illustrated here for Base coupling (input parameters BLC and BHYPER), but the same rules apply for Axial coupling (VLC and VHYPER) and Perpendicular coupling (HLC and HHYPER).

a) If BHYPER and BLC are both zero, $f(d)$ is unity, resulting in a bilinear elastic-perfectly-plastic coupling characteristic.

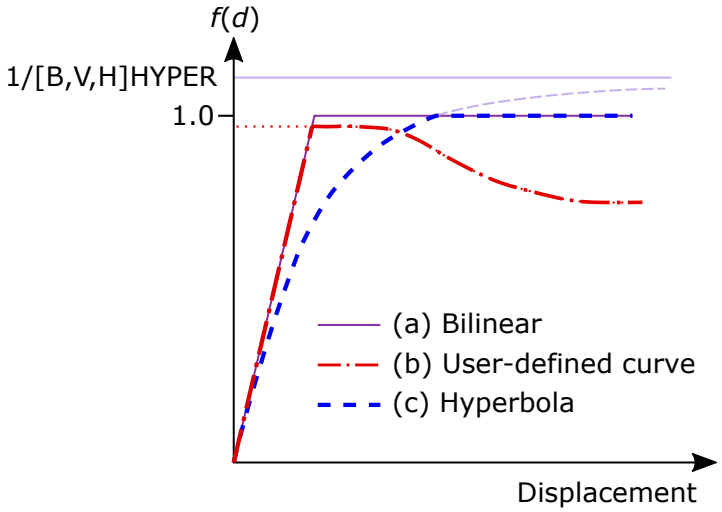


Figure 10-43. Forms of $f(d)$ available when *OPTION1* is CONSTANTS or unset.

b) If the load curve BLC is nonzero, $f(d)$ is the value of the load curve. BLC is an optional load curve (see *DEFINE_CURVE) giving a non-dimensional factor on yield stress as a function of displacement.

c) If BHYPER is nonzero (typical value: 0.95), the stress as a function of displacement follows a hyperbolic curve:

$$f(d) = \min\left[1.0, \quad {d_{\max}}\Big/{\text{BHYPER} \times (d_{\text{elastic}} + d_{\max})}\right] .$$

Here $d_{\max}$ is the maximum displacement that has occurred so far, and d_{elastic} is the elastic displacement calculated (for the example of Base coupling) from $\sigma_{\max}/S_{\text{Base}}$. $\sigma_{\max}$ is the stress term in curly brackets in the equation for $\sigma_{Y,Base}$ above.

The three options for $f(d)$ are illustrated in Figure 10-43.

7. **Backbone Coupling Curves when *OPTION1* is CURVES.** The coupling stresses for Base, Axial and Perpendicular coupling (σ_{Base}, τ_{Ax}, and σ_{Perp}, respectively) are defined as follows:

$$\sigma_{\text{Base}} = \left\{\text{BLCZ}(z_r) + 0.5\left(\sigma'_x + \sigma'_y\right) \times \text{BLCSH}(z_r) + \sigma'_z \times \text{BLCSV}(z_r)\right\} \times \text{BLC}(d)$$
$$\tau_{\text{Ax}} = \left\{\text{VLCZ}(z_r) + 0.5\left(\sigma'_x + \sigma'_y\right) \times \text{VLCSH}(z_r) + \sigma'_z \times \text{VLCSV}(z_r)\right\} \times \text{VLC}(d)$$
$$\sigma_{\text{Perp}} = \left\{\text{HLCZ}(z_r) + 0.5\left(\sigma'_x + \sigma'_y\right) \times \text{HLCSH}(z_r) + \sigma'_z \times \text{HLCSV}(z_r)\right\} \times \text{HLC}(d)$$

Here d is relative displacement (see Remark 12). z_r is the relative Z-coordinate defined as $Z_0 - \text{ZREF}$. Z_0 is the initial Z-coordinate of the pile node). σ'_x, σ'_y and σ'_z are the *initial* stresses in the global X, Y and Z directions. Depending on INSTRF and TIMSTR (see Card 2), either time-varying soil stresses or the soil stresses at time TIMSTR are used in this calculation. Note that INSTR and TIMSTR were introduced in R14. In R13, the CURVES option always used initial soil

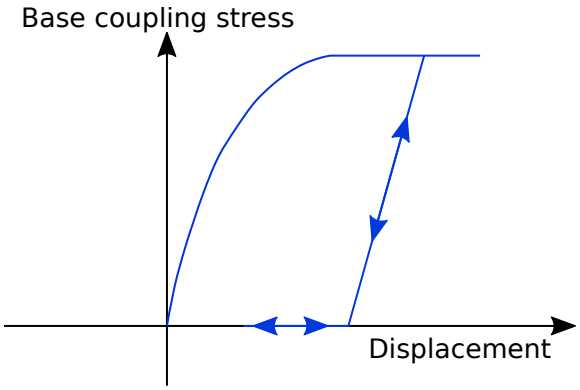


Figure 10-44. Base coupling unload/reload behavior

stress (that is, the soil stress at time zero). If pore water is modeled (see *CONTROL_PORE_FLUID), then these are "effective stress" meaning the stress generated by the material model excluding any pore pressure. Note that *CONSTRAINED_SOIL_PILE does not require pore pressure to be modelled.

The "normalized mobilization curves" BLC, VLC, HLC, if defined, must have a first point at (0,0). The initial elastic coupling stiffness is determined from the above equations using the gradient of the line between the first and second points of the normalized mobilization curve. If any of the curves (BLCZ, BLC, VLCZ, HLCSH, etc.) are left blank, the corresponding term in the above equations is set to zero. For example, if BLC is left blank, then the Base coupling stress will always be zero, meaning no Base coupling.

8. **Directionality of loading and unload/reload behavior: Base coupling.** Base coupling resists relative movement only in the axial direction corresponding to a pile being pressed downwards into the soil. This coupling results in a compressive axial load on the pile. Uplift is not resisted and results in a gap between the base of the pile and the soil. On reloading, any gap must close before compressive loading of the pile can resume. Figure 10-44 illustrates this unloading/reloading behavior for when BHYPER on Card 4a specifies the shape of the coupling.

9. **Directionality of loading and unload/reload behavior: Axial coupling.** Axial coupling shear stresses resist sliding of the pile in both axial directions. An elasto-plastic approach is adopted. Thus, if the direction of axial movement is reversed, stresses also reverse. Figure 10-45 illustrates a possible response when a user-defined mobilization curve VLC is specified.

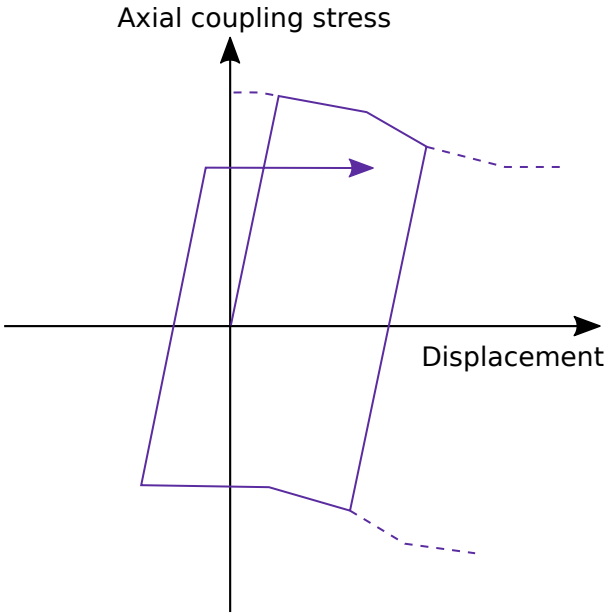


Figure 10-45. Axial coupling unload/reload behavior

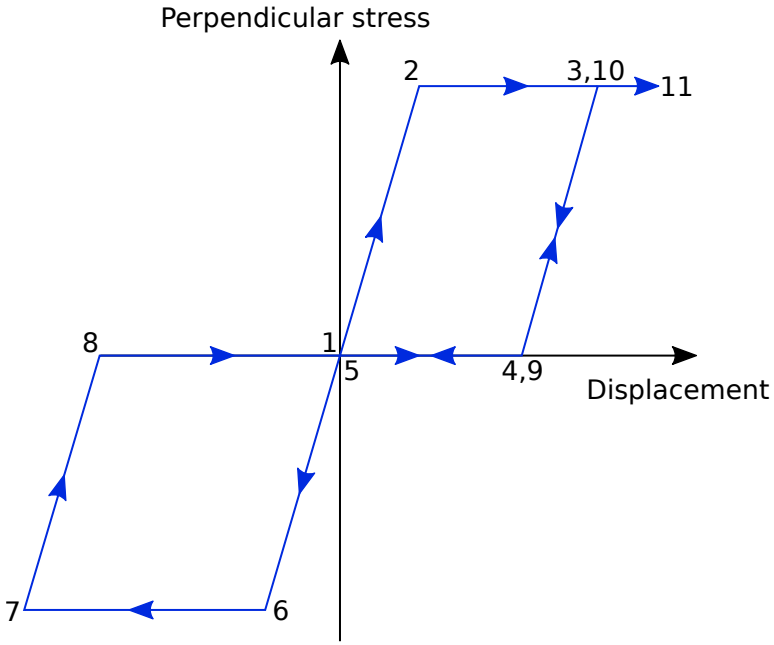


Figure 10-46. Perpendicular coupling unload/reload behavior

10. **Directionality of loading and unload/reload behavior: Perpendicular coupling.** Perpendicular coupling stresses are calculated such that, if a cyclical back-and-forth motion is applied to a pile, the effect is that of an elongating hole (sometimes called "post-hole" behavior named for the observed behavior of the hole in the ground when a fence-post is repeatedly bent in one direction and then the other). As an illustration, in Figure 10-46, HLC and HHYPER are left blank, and a cycling motion is applied. If additional frictional resistance is

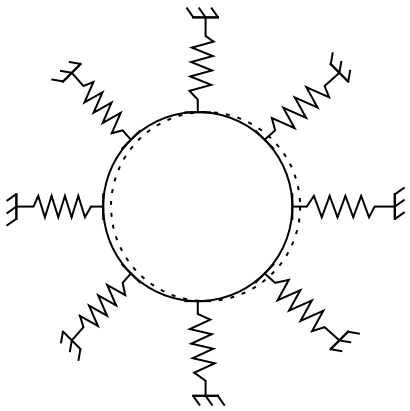


Figure 10-47. Example of how perpendicular motion in a pile is tracked.

defined (Card 7a for the CONSTANTS option; input parameters HLCFD or FRATIO for CURVES option), the friction stress is applied as an offset in the *y*-axis direction to the sections of the curve between points 4 and 5 and between points 8 and 9.

Movement could potentially occur at any angle in the plane perpendicular to the pile axis. LS-DYNA tracks the movement and gaps in 8 directions at 45-degree intervals around the pile circumference. For these directions, LS-DYNA applies the input coupling characteristics. For intermediate angles of movement, an interpolation method is used resulting in small differences between the input stress-displacement coupling characteristic and the one achieved by the model. For instance, the resistance to movement at 20 degrees would be a combination of that at 0-degrees and 45-degrees. See Figure 10-47.

11. **Location of coupling points.** LS-DYNA calculates the coupling stresses at each coupling point using the input coupling characteristics. By default (NRING = 0 or 1), one coupling point is coincident with each pile node. We recommend the default when the solid element size is greater than or equal to the pile diameter. Where the soil elements are smaller than the pile diameter, we recommend distributing the coupling stresses onto all the soil elements through which the outer surface of the pile passes. To do this, specify a value of NRING > 1 to define a ring of coupling points level with each pile node around the circumference of the pile. Typically, you should select a value of NRING such that about 1 or 2 coupling points are present in each soil element on the circumference of the pile. Irrespective of the setting of NRING, we recommend NRINGB = 0, meaning that the coupling points on the base of the pile are automatically chosen to obtain an approximately uniform distribution. NRING and NRINGB influence the coupling point mesh only in the plane perpendicular to the pile axis. There is no option to refine the coupling point mesh in the axial direction. See Figure 10-48 for illustration.

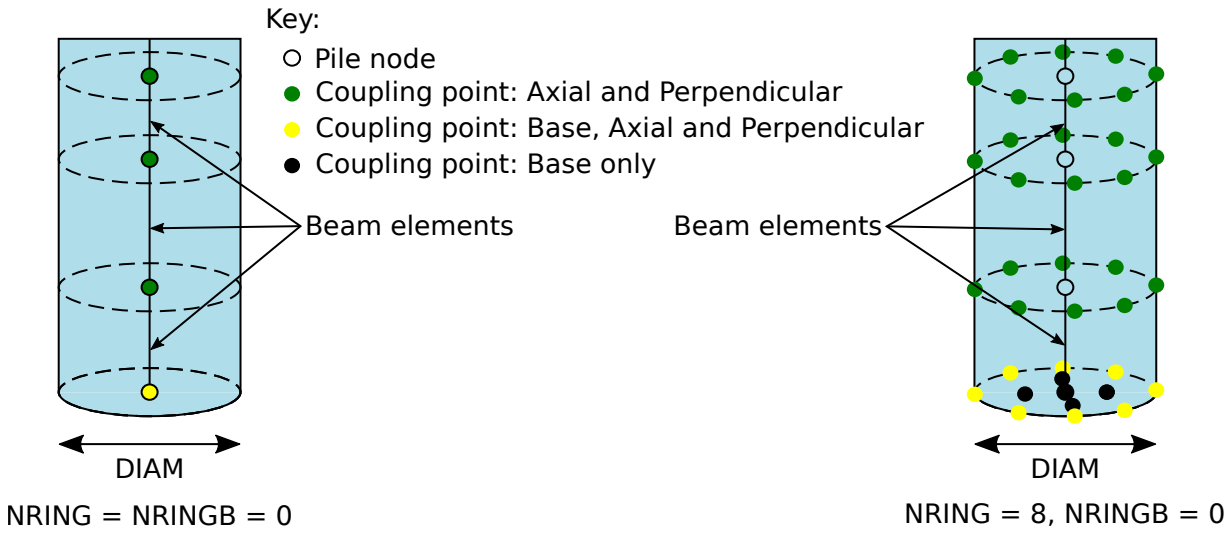


Figure 10-48. Location of coupling points

12. **Definition of displacement of the pile relative to the soil.** Relative displacement at a given coupling point is the difference of displacement between a pile point and a soil point. The pile point displacement is calculated as if a rigid link were present between the pile node and the coupling point. Therefore, when NRING > 1, the rotation of the pile node times the radial offset between pile node and coupling point contributes to the displacement. The soil point retains a constant relative position (iso-parametric coordinates) within the soil element. These calculations do not create any additional constraints on the pile node or soil nodes.

13. **Relation between coupling stress and force on the pile.** The Base, Axial and Perpendicular coupling forces are calculated at each coupling point from the coupling stresses as follows:

$$F_{\text{Base}} = \sigma_{\text{Base}} A_{\text{Bp}}$$
$$F_{\text{Ax}} = \tau_{\text{Ax}} \times \pi D \times 0.5(L_1 + L_2)/\text{NRING}$$
$$F_{\text{Perp}} = \sigma_{\text{Perp}} \times D \times 0.5(L_1 + L_2)/\text{NRING}$$

Here A_{Bp} is the area associated with a Base coupling point, D is the pile diameter (DIAM), and L_1 and L_2 are the lengths of the two pile beam elements that meet at the pile node in question. Note that the axial force is based on the whole perimeter being loaded by the shear stress, while the perpendicular force is based on the stress times the area of the pile projected in the perpendicular direction.

The coupling forces are applied to the pile node and reacted on the nodes of the soil element containing the coupling point. If NRING > 1, moments arising from the coupling force times the radial offset between the pile node and the coupling point are applied to the pile node.

14. **Coupling visualization elements.** LS-DYNA automatically creates a beam element at each coupling point. These are for visualizing the coupling stresses only; they do not create additional forces in the solution. LS-DYNA automatically creates *PART, *SECTION_BEAM and *MAT_NULL cards for the visualization elements. If desired, the IDs of the automatically generated *PART cards may be specified with input fields PIDNS and PIDNB.

    It is permitted, but not necessary, to define the *PART cards with IDs PIDNS and PIDNB in the keyword file. In this case, the two *PARTs should reference a *SECTION_BEAM with ELFORM = 6 and a *MAT_NULL. the density in the material keyword input, the volume in the section input, and the inertia in the section input should have small nonzero values.

15. **Viewing the coupling stresses.** In the d3plot output file, the coupling visualization beam elements contain fake forces which are actually the coupling stresses. For the elements in part PIDNS, Axial force (or X-Force) is really the Perpendicular coupling stress in the local x direction; Y-Shear (or Y-Force) is really the Perpendicular coupling stress in the local y direction; and, Z-Shear (or Z-Force) is really the Axial coupling stress.

    For the elements in part PIDNB, the Axial force (or X-Force) and Y-Shear (or Y-Force) are zero, and the Z-Shear (or Z-Force) is really the Base coupling stress.

    The nodes of the visualization beam elements stay coincident with the pile and soil coupling points mentioned in Remark 11 throughout the analysis, enabling visualization of the pile/soil relative displacements (see Remark 12).

16. **Depth-dependent properties.** Soil stiffness and strength generally increases with depth. Soil-pile coupling properties are related to soil properties and therefore they also tend to increase with depth. *CONSTRAINED_SOIL_PILE offers options for defining coupling properties that vary with depth (expressed via functions of Z-coordinate relative to ZREF), but depth-dependent coupling could also be achieved by defining multiple layers of soil each with a different part ID, and then defining unique but non-depth-dependent properties for each layer.

17. **Damping.** Damping of the axial response may be defined using DAMP. This is not usually required for quasi-static simulations for which other damping methods are available, so DAMP is usually set to zero. However, damping may sometimes be desirable in some dynamic simulations.

18. **Mass-scaling.** LS-DYNA checks the numerical stability of the coupling, based on the coupling stiffness and the masses of the pile and soil nodes. If the model uses mass-scaling (negative DT2MS on *CONTROL_TIMESTEP), and if a coupling point would not otherwise be stable at the current timestep, LS-DYNA will automatically add mass to the pile and soil nodes to ensure stability. The mass

added by *CONSTRAINED_SOIL_PILE during the first timestep is printed to the d3hsp file; search for "added mass for *CONSTRAINED_SOIL_PILE". If the amount of added mass seems excessive, consider reducing the initial elastic coupling stiffness (described in Remarks 6 and 7).

19. **Calibration of coupling properties.** The nonlinear coupling properties in *CONSTRAINED_SOIL_PILE account for the part of the deformation of the soil that is not captured explicitly by the soil mesh. The finer the soil mesh, the more localized deformation around the pile will be captured by the soil elements, and therefore the less deformation needs to be accounted for by the coupling properties. Thus, realistic input properties for *CONSTRAINED_SOIL_PILE depend on soil mesh size. It is advisable to calibrate the input properties against a detailed model in which the pile (or part of a pile) is represented by solid elements embedded in finely-meshed soil and subjected to axial and perpendicular loading cases.

20. **Staged construction and the dynain file.** The dynain files written during a Staged Construction analysis (see *CONTROL_STAGED_CONSTRUCTION) contain a keyword called *INITIAL_SOIL_PILE_DATA which enables the *CONSTRAINED_SOIL_PILE coupling stresses and history data to be carried forward into further analyses.

*CONSTRAINED_SOLID_IN_SOLID_{OPTION1}_{OPTION2}

This keyword can take the following two forms:

*CONSTRAINED_SOLID_IN_SOLID

*CONSTRAINED_SOLID_IN_SOLID_PENALTY

To define an ID and heading, the following options are available:

ID

TITLE

Purpose: Provide either constraint-based or penalty-based coupling between solids and the solids/thick shells in which the solids are embedded. It is one of the keywords in the *CONSTRAINED_BEAM/SHELL/SOLID_IN_SOLID family.

To use the constraint-based coupling, this keyword takes the form of *CONSTRAINED_-SOLID_IN_SOLID. It constrains solid structures to move with Lagrangian solids/thick shells. This keyword constrains both acceleration and velocity. This keyword, together with *CONSTRAINED_BEAM_IN_SOLID and *CONSTRAINED_SHELL_IN_SOLID, intends to sidestep certain limitations of the CTYPE = 2 implementation in *CON-STRAINED_LAGRANGE_IN_SOLID. Notable features of this keyword include:

1. **Tetrahedral and pentahedral solid elements are supported.** They are no longer treated as degenerated hexahedra as in the CTYPE = 2 implementation.

2. **Velocity/Fixed Boundary Condition.** The CTYPE = 2 implementation fails to constrain solid nodes that are buried inside elements whose nodes have pre-scribed velocity/fixed boundary conditions.

3. **Optimized Sorting.** The sorting subroutine is optimized for larger problems to achieve better performance and less memory usage.

To use the penalty-based coupling, this keyword takes the form of *CONSTRAINED_-SOLID_IN_SOLID_PENALTY. A penalty spring is attached between coupling points on the solid and in the solid/thick shell element. Penalty spring stiffness is calculated based on the geometric mean of the two bulk moduli of the interacting elements. The magni-tude of this coupling force can be controlled using PSSF (penalty spring stiffness scale factor). This penalty coupling conserves kinetic energy much better in transient prob-lems, such as blast loading.

If a title is not defined, LS-DYNA will automatically create an internal title for this cou-pling definition.

Title Card. Additional card for TITLE and ID keyword options

Card ID	1	2	3	4	5	6	7	8
Variable	COUPID	TITLE						
Type	I	A						

Card 1	1	2	3	4	5	6	7	8
Variable	SSIDA	SSIDB	SSTYPA	SSTYPB				
Type	I	I	I	I				
Default	none	none	0	0				

Card 2	1	2	3	4	5	6	7	8
Variable	START	END				PSSF		
Type	F	F				F		
Default	0.	10^{20}				0.1		

VARIABLE	DESCRIPTION
COUPID	Coupling (card) ID number. If not defined, LS-DYNA will assign an internal coupling ID based on the order of appearance in the input deck.
TITLE	A description of this coupling definition
SSIDA	Set ID defining a part or part set ID of the Lagrangian solid structure constrained to move with solid or thick shell elements specified with SSIDB (see *PART and *SET_PART).
SSIDB	Set ID defining a part or part set ID of the Lagrangian solid elements or thick shell elements which constrain SSIDA (see *PART and *SET_PART).

VARIABLE	DESCRIPTION
SSTYPA	Set type of SSIDA: EQ.0: Part set ID (PSID) EQ.1: Part ID (PID)
SSTYPB	Set type of SSIDB: EQ.0: Part set ID (PSID) EQ.1: Part ID (PID)
START	Start time to activate the coupling: LT.0.0: Start time is set to |START|. When negative, start time is followed during the dynamic relaxation phase of the calculation. After the completion of dynamic relaxation, coupling is active regardless of the value of END. EQ.0.0: Start time is inactive, meaning coupling is always active GT.0.0: If END = -9999, START is interpreted as the curve or table ID defining multiple pairs of start-time and end-time. Otherwise, if END > 0, start time applies both during and after dynamic relaxation.
END	End time to deactivate the coupling: LT.0.0: If END = -9999, START is interpreted as the curve or table ID defining multiple pairs of start-time and end-time. Otherwise, negative END indicates that coupling is inactive during dynamic relaxation. After dynamic relaxation, the start and end times are followed and set to |START| and |END|, respectively. EQ.0.0: END defaults to 10^{20}. GT.0.0: END sets the time at which the coupling is deactivated.
PSSF	Penalty spring stiffness scale factor. Only available in penalty form.

*CONSTRAINED_SPLINE

Purpose: Define an elastic cubic spline interpolation constraint. The displacements and slopes at the end points are continuous. The first and last nodes, which define the constraint, must be independent. The degrees-of-freedom of interior nodes may be either dependent or independent.

Card 1	1	2	3	4	5	6	7	8
Variable	SPLID	DLRATIO						
Type	I	I						
Default	0	0.10						

Node Cards. Include one card per independent/dependent node. The first and last nodes must be independent. The next keyword ("*") card terminates this input.

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

VARIABLE	DESCRIPTION
SPLID	Spline constraint ID
DLRATIO	Ratio of bending to torsional stiffness for an elastic tubular beam which connects the independent degrees-of-freedom. The default value is 0.10.
NID	Independent/dependent node ID. For explicit problems this node should not be a member of a rigid body or elsewhere constrained in the input.
DOF	Degrees-of-freedom. The list of dependent degrees-of-freedom consists of a number with up to six digits, with each digit representing a degree of freedom. For example, the value 1356 indicates

VARIABLE	DESCRIPTION
	that degrees of freedom 1, 3, 5, and 6 are controlled by the constraint. The default is 123456. Digit degree of freedom IDs: EQ.1: x EQ.2: y EQ.3: z EQ.4: Rotation about x-axis EQ.5: Rotation about y-axis EQ.6: Rotation about z-axis

***CONSTRAINED_SPOTWELD**_*{OPTION}*_*{OPTION}*

To use a time-filtered force calculation for the forced-based failure criterion, the following option is available:

FILTERED_FORCE

To define an ID for the spot weld, the following option is available:

ID

The order of the options in the keyword name is arbitrary. Both options are optional.

Purpose: Define massless spot welds between non-contiguous nodal pairs.

The spot weld is a rigid beam that connects the nodal points of the nodal pairs; thus, nodal rotations and displacements are coupled. The spot welds must be connected to nodes having rotary inertias, that is, beams or shells. If this is not the case, for example, if the nodes belong to solid elements, use the option: *CONSTRAINED_RIVET. During implicit calculations this case is treated like a rivet, constraining only the displacements. Note that shell elements do not have rotary stiffness in the normal direction and, therefore, this component cannot be transmitted.

Spot welded nodes must not have the same coordinates. Coincident nodes in a spot weld can be handled by the *CONSTRAINED_NODAL_RIGID_BODY option. Brittle and ductile failures are supported by this model. Brittle failure is based on the resultant forces acting on the weld, and ductile failure is based on the average plastic strain value of the shell elements which include the spot welded node. Spot welds, which are connected to massless nodes, are automatically deleted in the initialization phase and a warning message is printed in the messag file and the d3hsp file.

WARNING: The accelerations of spot welded nodes are output as zero into the various databases, but if the accelerations of spot welded nodes are required, use either the *CONSTRAINED_GENERALIZED_WELD or the *CONSTRAINED_NODAL_RIGID_BODY input. However, if the output interval is frequent enough, accurate acceleration time histories can be obtained from the velocity time history by differentiation in the post-processing phase.

ID Card. Additional card for the ID keyword option.

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

Card 1	1	2	3	4	5	6	7	8
Variable	N1	N2	SN	SS	N	M	TF	EP
Type	I	I	F	F	F	F	F	F
Default	none	none	0.0	0.0	none	none	10^{20}	10^{20}

Filter Card. Additional card for the FILTERED_FORCE keyword option.

Card 2	1	2	3	4	5	6	7	8
Variable	NF	TW						
Type	I	F						
Default	none	none						

VARIABLE	DESCRIPTION
WID	Optional weld ID.
N1	Node ID. See Remarks 1 and 5.
N2	Node ID. See Remarks 1 and 5.
SN	Normal force, S_n, at spot weld failure (see Remark 2 below). EQ.0.0: The failure criterion is disabled. GT.0.0: Normal force at spot weld failure

VARIABLE	DESCRIPTION
	LT.0.0: Curve ID which specifies the normal force at spot weld failure as a function of the nodal temperature
SS	Shear force, S_s, at spot weld failure (see Remark 2 below). EQ.0.0: The failure criterion is disabled. GT.0.0: Shear force at spot weld failure LT.0.0: Curve ID which specifies the shear force at spot weld failure as a function of the nodal temperature
N	Exponent for normal spot weld force (see Remark 2 below).
M	Exponent for shear spot weld force (see Remark 2 below).
TF	Failure time for nodal constraint set. See Remark 3.
EP	Effective plastic strain at failure. See Remark 4.
NF	Number of force vectors stored for filtering.
TW	Time window for filtering.

Remarks:

1. **Spot Weld Nodes and Constraints.** Nodes connected by a spot weld cannot be members of another constraint set that constrains the same degrees-of-freedom, a tied interface, or a rigid body, i.e., nodes cannot be subjected to multiple, independent, and possibly conflicting constraints. Also, care must be taken to ensure that single-point constraints applied to nodes in a constraint set do not conflict with the constraint sets constrained degrees-of-freedom.

2. **Spot Weld Failure Criteria.** Failure of the spot welds occurs when:

$$\left(\frac{|f_n|}{S_n}\right)^n + \left(\frac{|f_s|}{S_s}\right)^m \geq 1 \ ,$$

where f_n and f_s are the normal and shear interface forces. Component f_n is non-zero for tensile values only. When SN and SM are specified by curve IDs, failure of the spot welds occurs when:

$$\left(\frac{|f_n|}{S_n(T)}\right)^n + \left(\frac{|f_s|}{S_s(T)}\right)^m \geq 1 \ ,$$

where T is the nodal temperature.

3. **Failure Time.** When the failure time, TF, is reached the spot weld becomes inactive and the constrained nodes may move freely.

4. **Material Plastic Strain Failure.** Spot weld failure due to plastic straining occurs when the effective nodal plastic strain exceeds the input value, $\varepsilon^p_{\text{fail}}$. This option can model the tearing out of a spotweld from the sheet metal since the plasticity is in the material that surrounds the spot weld, not the spot weld itself. A least squares algorithm is used to generate the nodal values of plastic strains at the nodes from the element integration point values. The plastic strain is integrated through the element and the average value is projected to the nodes via a least square fit. This option should only be used for the material models related to metallic plasticity and can result is slightly increased run times. Failures can include both the plastic and brittle failures.

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_SPOTWELD
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Spotweld two nodes (34574 and 34383) with the approximate strength
$  of a 3/8" SAE Grade No 3 bolt.
$
*CONSTRAINED_SPOTWELD
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$       n1        n2        sn        sf         n         m        tf        ps
     34574     34383      36.0      18.0       2.0       2.0       10.       1.0
$
$
$       sn = 36.0  normal failure force is 36 kN
$       sf = 18.0  shear failure force is 18 kN
$        n = 2.0   normal failure criteria is raised to the power of 2
$        m = 2.0   shear failure criteria is raised to the power of 2
$       tf = 10.0  failure occurs at time 10 unless strain failure occurs
$       ps = 2.0   plastic strain at failure
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```

5. **Thermal Problem.** The 2 nodes identified by this keyword will be constrained to the same temperature in a thermal problem or in a coupled thermal-mechanical problem.

***CONSTRAINED_SPR2**_*{OPTION}*

Available options include:

<BLANK>

TITLE

Purpose: Define a self-piercing rivet with failure. This self-piercing rivet (SPR2) model includes a plastic-like damage model that reduces the force and moment resultants to zero as the rivet fails. A diameter approximately equal to the rivet's diameter gives the domain of influence.

The location of the rivet is defined by a single node at the center of two riveted sheets. The algorithm performs a normal projection from the upper and lower sheets to the rivet node and locates all nodes within the user-defined diameter of influence. L. Olovsson of Impetus Afea developed the numerical implementation of this rivet model based on research on SPR point connector models originally carried out by SIMLab (NTNU) and SINTEF; see references by Porcaro, Hanssen, et al. [2006, 2006, 2007].

Initially, only two sheets (upper and lower) could be connected with one SPR2 node. But since release R9, up to 6 sheets can be connected with one SPR2 node by defining additional parts on optional Card 4. The following stacking sequence should be used: UPID – XPID1 – XPID2 – XPID3 – XPID4 – LPID. Omitted parts can be left blank. For instance, for a 3-sheet connection, the extra part lies between the upper and lower parts, and for a regular 2-sheet connection, Card 4 can be omitted entirely.

Starting with R13, not only shell element parts can be connected using this approach, but also solid elements. However, the new capability is still experimental and should be treated carefully. For instance, the solid element parts still having some kind of obvious planar structure is advantageous.

Title Card. Additional card for the TITLE keyword option.

Card Title	1	2	3	4	5	6	7	8
Variable	TITLE							
Type	A80							

Card 1	1	2	3	4	5	6	7	8
Variable	UPID	LPID	NSID	THICK	D	FN	FT	DN
Type	I	I	I	F	F	F	F	F
Default	none	none	none	none	none	none	none	none

Card 2	1	2	3	4	5	6	7	8
Variable	DT	XIN	XIT	ALPHA1	ALPHA2	ALPHA3	DENS	INTP
Type	F	F	F	F	F	F	F	F
Default	none	none	none	none	none	none	none	0.0

Card 3 is optional.

Card 3	1	2	3	4	5	6	7	8
Variable	EXPN	EXPT	PIDVB					
Type	F	F	F					
Default	8.0	8.0	0					

Card 4 is optional.

Card 4	1	2	3	4	5	6	7	8
Variable	XPID1	XPID2	XPID3	XPID4				
Type	I	I	I	I				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
TITLE	Name or description of the SPR2 defined in this keyword. This name is used as part title for the visualization beams, such as in the d3plot database. If undefined, the part title is "SPR2_NSID_...".
UPID	Upper sheet part ID
LPID	Lower sheet part ID
NSID	Node set ID of rivet location nodes
THICK	Total thickness of upper and lower sheets
D	Rivet diameter
FN	Rivet strength in tension (pull-out): GT.0: Constant value LT.0: Material data from an instantiation of *MAT_CONSTRAINED_SPR2 (*MAT_265) with MID of absolute value |FN|
FT	Rivet strength in pure shear
DN	Failure displacement in normal direction
DT	Failure displacement in tangential direction
XIN	Fraction of failure displacement at maximum normal force
XIT	Fraction of failure displacement at maximum tangential force
ALPHA1	Dimensionless parameter scaling the effective displacement
ALPHA2	Dimensionless parameter scaling the effective displacement
ALPHA3	Dimensionless parameter scaling the effective displacement. The sign of ALPHA3 can be used to choose the normal update procedure: GT.0: Incremental update (default) LT.0: Total update (recommended)
DENS	Rivet density (necessary for time step calculation)

VARIABLE	DESCRIPTION
INTP	Flag for interpolation: EQ.0: Linear (default), EQ.1: Uniform (not recommended for asymmetrical arrangement of the upper and lower nodes), EQ.2: Inverse distance weighting.
EXPN	Exponent value for load function in normal direction
EXPT	Exponent value for load function in tangential direction
PIDVB	Part ID for visualization beams representing SPR2 in post-processing. EQ.0: Part ID automatically set (default). GT.0: PIDVB defines part ID.
XPID1	Extra part ID 1 for multi-sheet connection
XPID2	Extra part ID 2 for multi-sheet connection
XPID3	Extra part ID 3 for multi-sheet connection
XPID4	Extra part ID 4 for multi-sheet connection

Remarks:

Self-piercing rivets are a type of fastener that is sometimes used in place of spot welds to join sheet metal of similar or dissimilar materials. The rivet penetrates the upper sheet and expands to interlock with the lower sheet without penetration. The strength and fatigue characteristics of self-piercing rivets can meet or even exceed that of spot welds; consequently, their practical applications are expanding.

In the local description of the underlying model, all considerations are done in the plane-of-maximum opening defined by

$$\hat{\mathbf{n}}_o = \hat{\mathbf{n}}_l \times \hat{\mathbf{n}}_u \quad .$$

The unit normal vectors of the lower and upper sheets are $\hat{\mathbf{n}}_l$ and $\hat{\mathbf{n}}_u$, respectively (see Figure 10-49). The tangential unit normal vector of the rivet is

$$\hat{\mathbf{n}}_t = \hat{\mathbf{n}}_o \times \hat{\mathbf{n}}_u \quad .$$

A single-sheet rivet system is assumed, meaning the rivet translation and rotation follow the motion of the upper sheet. The opening appears on the lower sheet.

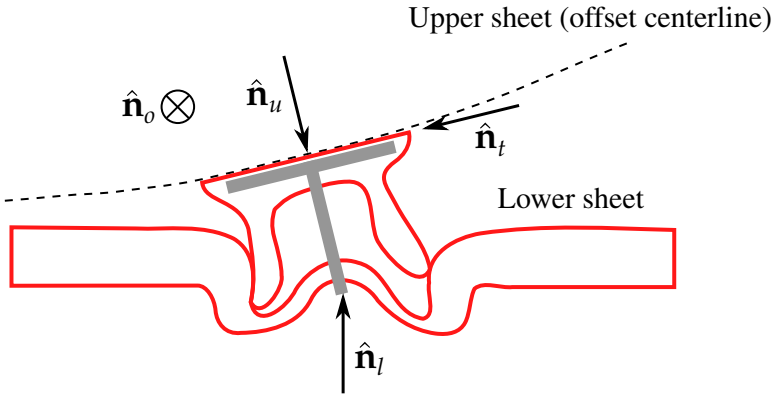


Figure 10-49. Plane of maximum opening.

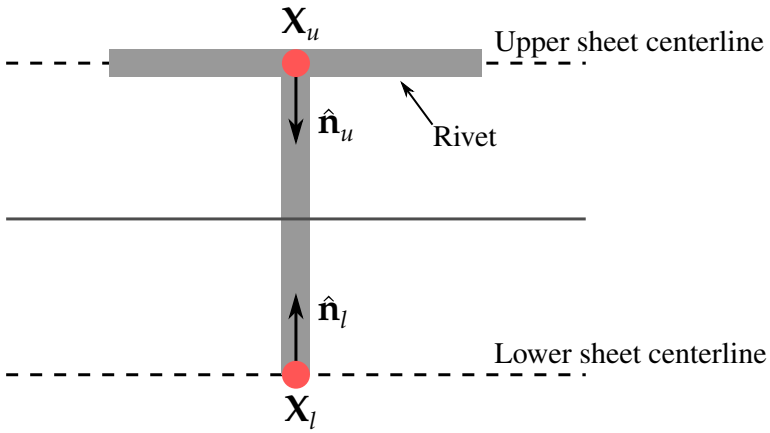


Figure 10-50. Single-sheet rivet system.

The local deformation is defined by normal stretch vector $\boldsymbol{\delta}_n$, tangential stretch $\boldsymbol{\delta}_t$ and total stretch $\boldsymbol{\delta} = \boldsymbol{\delta}_n + \boldsymbol{\delta}_t$ (see Figure 10-51). At any given time the total stretch is computed from the position vectors: $\boldsymbol{\delta} = \mathbf{X}_l^r - \mathbf{X}_l^l$ so that the scalar measures of normal stretch and tangential stretch are $\delta_n = \boldsymbol{\delta} \cdot \hat{\mathbf{n}}_n$ and $\delta_t = \boldsymbol{\delta} \cdot \hat{\mathbf{n}}_t$. The normal and tangential forces f_n and f_t are then determined by the material model, which will be explained next. The moments on the rivet always satisfy,

$$M_u + M_l = \frac{(h_1 + h_2) f_t}{2} \ .$$

The motion, the forces, and the moments are then distributed to the nodes within the radius of influence by a weighting function, which is, by default, linear (see parameter INTP).

The force-deformation relationship is defined by a nonlinear damage model for arbitrary mixed-mode loading conditions (combination of tension and shear). For pure tensile and pure shear loading, the behavior is given by,

$$f_n = \frac{f_n^{\max} \delta_n}{\eta_{\max} \delta_n^{\text{fail}}} \hat{f}_n(\eta_{\max}), \qquad f_t = \frac{f_t^{\max} \delta_t}{\eta_{\max} \delta_t^{\text{fail}}} \hat{f}_t(\eta_{\max}) \tag{1}$$

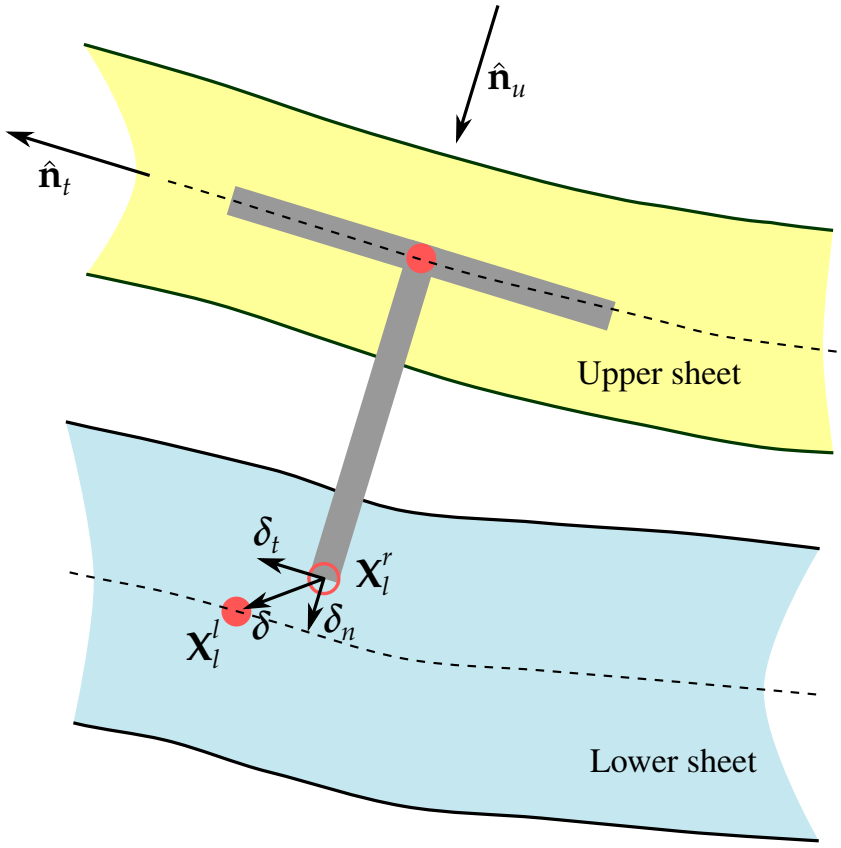


Figure 10-51. Local kinematics

respectively where,

$$\hat{f}_n(\eta_{\max}) = \begin{cases} 1 - \left(\dfrac{\xi_n - \eta_{\max}}{\xi_n}\right)^{\text{EXPN}} & \eta_{\max} \le \xi_n \\ 1 - \dfrac{\eta_{\max} - \xi_n}{1 - \xi_n} & \eta_{max} > \xi_n \end{cases}$$
$$\hat{f}_t(\eta_{\max}) = \begin{cases} 1 - \left(\dfrac{\xi_t - \eta_{\max}}{\xi_t}\right)^{\text{EXPT}} & \eta_{\max} \le \xi_t \\ 1 - \dfrac{\eta_{\max} - \xi_t}{1 - \xi_t} & \eta_{max} > \xi_t \end{cases} \tag{2}$$

In pure tension and pure shear the damage measure, $\eta_{\max}(t)$, defined in (3), simplifies to coincide with strain as indicated in Figure 10-53.

Usually, the material parameters $f_n^{\max}$, $f_t^{\max}$, δ_n^{fail}, and δ_t^{fail} can be determined directly from experiments, whereas material parameters ξ_n and ξ_t can be found by reverse engineering. For mixed-mode behavior, an effective displacement measure, $\eta(\theta)$, is given by

$$\eta(\theta, \eta_{\max}, t\ ) = \left[\xi(\theta) + \frac{1 - \xi(\theta)}{\alpha(\eta_{\max})}\right] \sqrt{\left[\frac{\delta_n(t)}{\delta_n^{\text{fail}}}\right]^2 + \left[\frac{\delta_t(t)}{\delta_t^{\text{fail}}}\right]^2}, \tag{3}$$

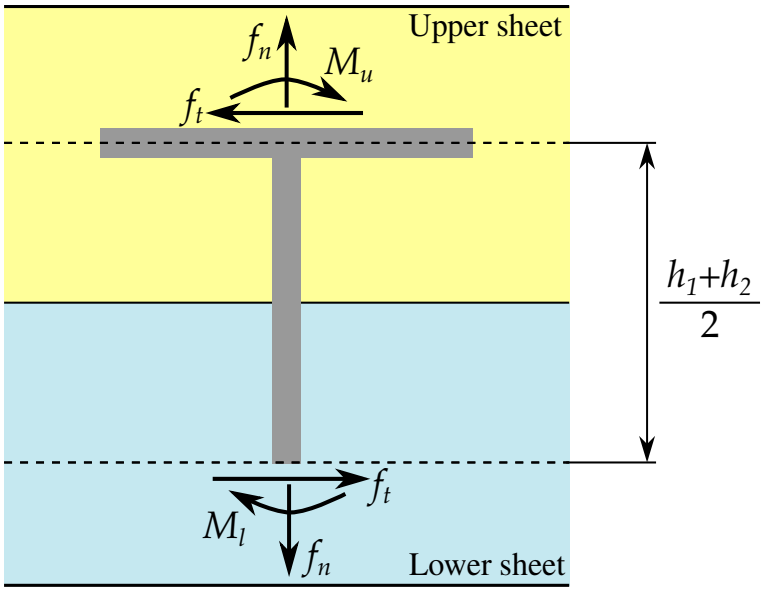


Figure 10-52. Local forces/moments

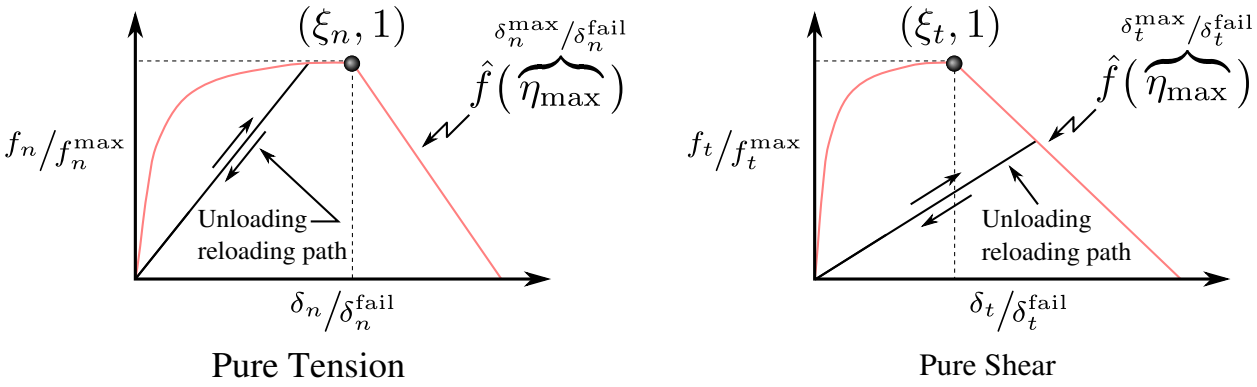


Figure 10-53. Force response of self penetrating rivet.

where,

$$\theta = \arctan\left(\frac{\delta_n}{\delta_t}\right)$$
$$\eta_{\max}(t) = \max[\eta(t)].$$

The parameter $\xi(\theta)$ which ranges from 0 to 1 scales the effective displacement as a function of the direction of the displacement vector in the δ_n-δ_t-plane according to,

$$\xi(\theta) = 1 - \frac{27}{4}\left(\frac{2\theta}{\pi}\right)^2 + \frac{27}{4}\left(\frac{2\theta}{\pi}\right)^3. \quad (4)$$

The directional scaling of the effective displacement is allowed to change as damage develops, which is characterized by the shape coefficient $\alpha(\eta_{\max})$ defined as

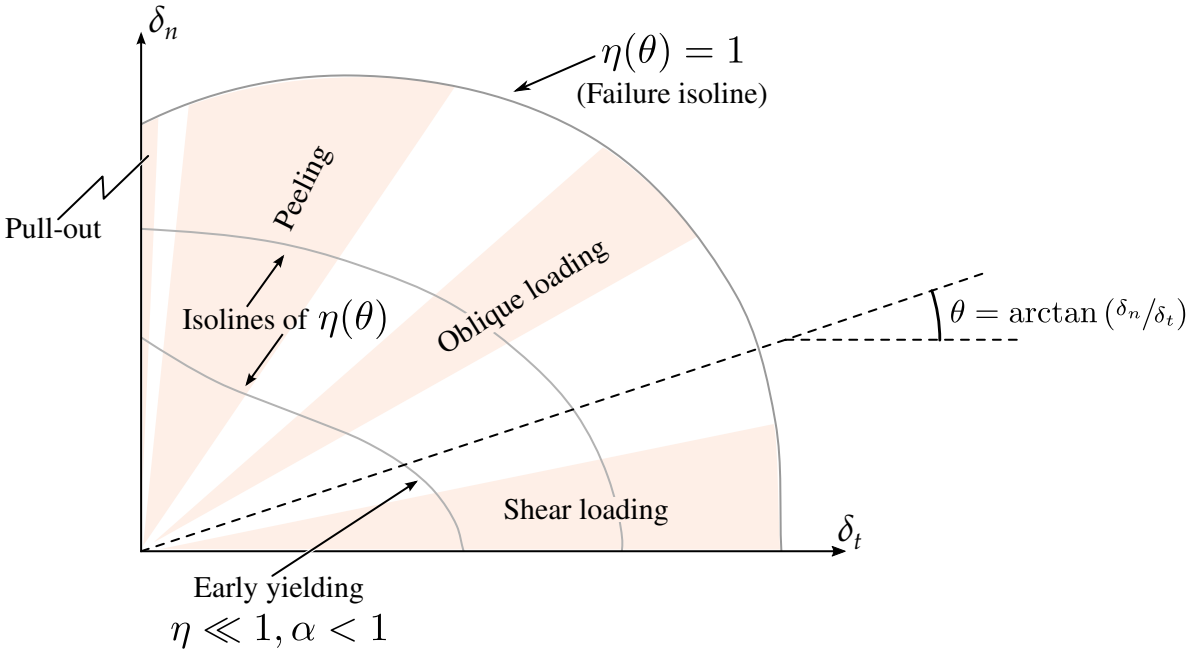


Figure 10-54. Isosurfaces of $\eta(\theta)$

$$\alpha(\eta_{\max}) = \begin{cases} \dfrac{\xi_t - \eta_{\max}}{\xi_t}\alpha_1 + \dfrac{\eta_{\max}}{\xi_t}\alpha_2 & \eta_{\max} < \xi_t \\ \dfrac{1-\eta_{\max}}{1-\xi_t}\alpha_2 + \dfrac{\eta_{\max}-\xi_t}{1-\xi_t}\alpha_3 & \eta_{\max} \geq \xi_t \end{cases}, \tag{5}$$

where α_1, α_2, and α_3 are material parameters.

The directional dependency of the effective displacement is necessary for an accurate force-displacement response in different loading directions. The coefficients α_1, and α_2 decrease the forces in the peeling and oblique loading cases to the correct levels. Both parameters are usually less than 1; whereas α_3 is typically larger than 1 as its main purpose is to moderate the failure displacement in oblique loading directions. Several qualitative features captured by this model are illustrated in Figure 10-54.

For the moment distribution, the difference between the upper sheet (stronger side where the rivet is entered) and the lower sheet (weaker side) is accounted for by a gradual transfer from the lower to the upper side as damage grows:

$$M_u = \frac{h_1 + h_2}{4}\left(1 + \frac{\eta_{\max} - \xi_1}{1-\xi_1}\right)f_1, \qquad M_l = \frac{h_1 + h_2}{4}\left(1 - \frac{\eta_{\max} - \xi_1}{1-\xi}\right)f_1 \tag{6}$$

Eventually the connection to the lower sheet becomes a moment free hinge.

We recommend using the drilling rotation constraint method for the connected components in explicit analysis. To use this method, field DRCPSID of *CONTROL_SHELL should refer to all shell parts involved in SPR2 connections.

*CONSTRAINED_TIE-BREAK

Purpose: Define a tied shell edge to shell edge interface that can release locally as a function of plastic strain of the shells surrounding the interface nodes. A rather ductile failure is achieved.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID1	NSID2	EPPF					
Type	I	I	F					
Default	none	none	0.					
Remarks		1, 2	3, 4					

VARIABLE	DESCRIPTION
NSID1	Node set ID for nodes on one side of the tied shell edge to shell edge interface; see *SET_NODE_*OPTION*.
NSID2	Node set ID for nodes on the other side of the tied shell edge to shell edge interface
EPPF	Plastic strain at failure

Remarks:

1. **Node Ordering.** Nodes in NSID2 must be given in the order they appear as one moves along the edge of the surface.

2. **Restrictions.** Tie-breaks may not cross.

3. **Failure Criterion.** Tie-breaks may be used to tie shell edges together with a failure criterion on the joint. If the average volume-weighted effective plastic strain in the shell elements adjacent to a node exceeds the specified plastic strain at failure, the node is released. The default plastic strain at failure is defined for the entire tie-break but can be overridden in node set NSID1 to define a unique failure plastic strain for each node.

4. **Model Applications.** Tie-breaks may be used to simulate the effect of failure along a predetermined line, such as a seam or structural joint. When the failure

criterion is reached in the adjoining elements, nodes along the slideline will begin to separate. As this effect propagates, the tie-breaks will appear to "unzip," thus simulating failure of the connection.

*CONSTRAINED_TIED_NODES_FAILURE

Purpose: Define a tied node set with failure based on plastic strain. The nodes must be coincident.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	EPPF	ETYPE					
Type	I	F	I					
Default	none	0.	0					
Remarks	1, 2, 4							

VARIABLE	DESCRIPTION
NSID	Nodal set ID, see *SET_NODE_*OPTION*.
EPPF	Plastic strain, volumetric strain, or damage (MAT_107, MAT_110, MAT_224, or GISSMO) at failure.
ETYPE	Element type for nodal group: EQ.0: shell, EQ.1: solid element

Remarks:

1. **Materials and Failure.** This feature applies to solid and shell elements using plasticity material models and to solid elements using the honeycomb material *MAT_HONEYCOMB (EPPF = plastic volume strain). The failure variable is the volume strain for materials 26, 126, and 201. The failure variable is the damage for materials 107, 110, 224, or GISSMO, and the equivalent plastic strain is used for all other plasticity models. The specified nodes are tied together until the average volume weighted value of the failure variable exceeds the specified value. Entire regions of individual shell elements may be tied together unlike the tie-breaking shell slidelines. The tied nodes are coincident until failure. When the volume weighted average of the failure value is reached for a group of constrained nodes, the nodes of the elements that exceed the failure value are released to simulate the formation of a crack.

2. **Coincident Nodes.** To use this feature to simulate failure, each element in the failure region should be generated with unique node numbers that are coincident in space with those of adjacent elements. Rather than merging these coincident nodes, the *CONSTRAINED_TIED_NODES_FAILURE option ties the nodal points together. A separate keyword definition must be used for each cluster of coincident nodes. As plastic strain develops and exceeds the failure strain, cracks will form and propagate through the mesh.

3. **Similar Keywords.** Entire regions of individual elements may be tied together, unlike the *CONSTRAINED_TIE-BREAK option. This latter option is recommended when the location of failure is known, e.g., as in the plastic covers which hide airbags in automotive structures.

4. **Contact Algorithms.** When using surfaces of shell elements defined using the *CONSTRAINED_TIED_NODES_FAILURE option in contact, it is best to define each node in the surface as a tracked node with the NODE_TO_SURFACE contact options. If this is not possible, the automatic contact algorithms beginning with *CONTACT_AUTOMATIC_... all of which include thickness offsets are recommended.

Example:

```
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONSTRAINED_TIED_NODES_FAILURE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Tie shell elements together at the nodes specified in nodal set 101.  The
$  constraint will be broken when the plastic strain at the nodes exceeds 0.085.
$
$  In this example, four shell elements come together at a common point.
$  The four corners of the shells are tied together with failure as opposed
$  to the more common method of merging the nodes in the pre-processing stage.
$
*CONSTRAINED_TIED_NODES_FAILURE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$     nsid      eppf
       101     0.085
$
$
*SET_NODE_LIST
$       sid
       101
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
       775       778       896       897
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
```