*CONTACT

The keyword *CONTACT provides a way of treating interaction between disjoint parts. Different types of contact may be defined:

> *CONTACT_*OPTION1_{OPTION2}_{OPTION3}_{OPTION4}_{OPTION5}*
>
> *CONTACT_ADD_WEAR
>
> *CONTACT_AUTO_MOVE
>
> *CONTACT_COUPLING
>
> *CONTACT_ENTITY
>
> *CONTACT_EXCLUDE_INTERACTION
>
> *CONTACT_FORCE_TRANSDUCER_*OPTION1_{OPTION2}*
>
> *CONTACT_GEBOD_*OPTION*
>
> *CONTACT_GUIDED_CABLE
>
> *CONTACT_INTERIOR
>
> *CONTACT_RIGID_SURFACE
>
> *CONTACT_SPG
>
> *CONTACT_1D
>
> *CONTACT_2D_*OPTION1_{OPTION2}_{OPTION3}*

CONTACT_*OPTION1*_... is the general form for defining a 3D contact algorithm. Wear models can be associated with a contact interface using *CONTACT_ADD_WEAR. *CONTACT_AUTO_MOVE is used for sheet metal forming applications and moves a surfb surface in a contact to close a gap between the surfa and surfb at a specified time. *CONTACT_COUPLING provides a means of coupling to deformable surfaces to MADYMO. *CONTACT_ENTITY treats contact using mathematical functions to describe the surface geometry for the reference surface. Portions of the 3D contact interface can be excluded from interacting with *CONTACT_EXCLUDE_INTERACTION. *CONTACT_-FORCE_TRANSDUCER measures the contact forces for 3D contacts. It is needed for measuring the forces in self-contacts since LS-DYNA does not automatically extract this data in that case.

*CONTACT

*CONTACT_GEBOD is a specialized form of the contact entity for use with the rigid body dummies (see *COMPONENT_GEBOD). *CONTACT_GUIDED_GABLE is a sliding contact for guiding 1D elements. *CONTACT_INTERIOR is used with soft foams where element inversion is sometimes a problem. Contact between layers of brick elements is treated to eliminate negative volumes. *CONTACT_RIGID_SURFACE is for modeling road surfaces for durability and NVH calculations. *CONTACT_SPG is for contact between SPG parts. *CONTACT_1D remains in LS-DYNA for historical reasons and is sometimes still used to model rebars which run along edges of brick elements. Lastly, *CONTACT_2D is the general 2D contact algorithm based on those used previously in LS-DYNA2D.

*CONTACT_OPTION1_{OPTION2}_{OPTION3}_{OPTION4}_{OPTION5}_{OPTION6}

Purpose: Define a contact interface in a 3D model. For contact in 2D models, see *CON-
TACT_2D_*OPTION*.

Introduction:

The *CONTACT keyword creates a *contact definition*. A contact definition associates sur-
faces with a *contact algorithm*. A contact algorithm is built from characteristics specified
over a small number of fields in the *CONTACT data set. The number of distinct contact
algorithms in LS-DYNA equals the number of allowed combinations of characteristics.
Not all data fields equally impact the nature of the contact algorithm. The most important
characteristics are controlled by:

1.   **OPTION1**. For some values of *OPTION1*, the contact algorithm is completely
      specified. For other values of *OPTION1*, it is not. *OPTION1* is often referred to
      as the *contact type* in this document.

2.   **The SOFT field of Optional Card A.** This field does not lend itself to a concise
      summary, but in general, it is accurate to say that the SOFT field of Optional
      Card A can change the qualitative behavior of a contact algorithm. The most
      popular values of SOFT, namely, 0, 1, and 2, specify the kind of penalty algo-
      rithm invoked.

3.   **OPTION4.** This option controls the behavior of tied contact algorithms.

The broadest categories of contact characteristics are whether a contact algorithm is pen-
alty or constraint-based, how penetration is measured (nodes-to-segments or segments-
to-segments), and whether the contact is nonsymmetric or symmetric:

1.   **Penalty-based and constraint-based contact algorithms.** Penalty-based
      contact places normal interface springs between penetrating nodes/segments
      and the contact surface. These springs apply a contact force to reduce penetra-
      tion. In constraint-based contact, nodes penetrating the contact surface are
      forced to the surface and often controlled with kinematic constraints.

2.   **How penetration is measured.** Except for segment-to-segment contacts (SOFT
      = 2 on Optional Card A and Mortar contact), all LS-DYNA contact algorithms
      look for the nodes of one surface penetrating the segments of another surface;
      that is, they are node-to-segment contacts. In contrast, the segment-to-segment
      contact algorithms search for the segments of one surface penetrating the seg-
      ments of another surface (see Note on segment-to-segment contact algorithms
   for a brief discussion).
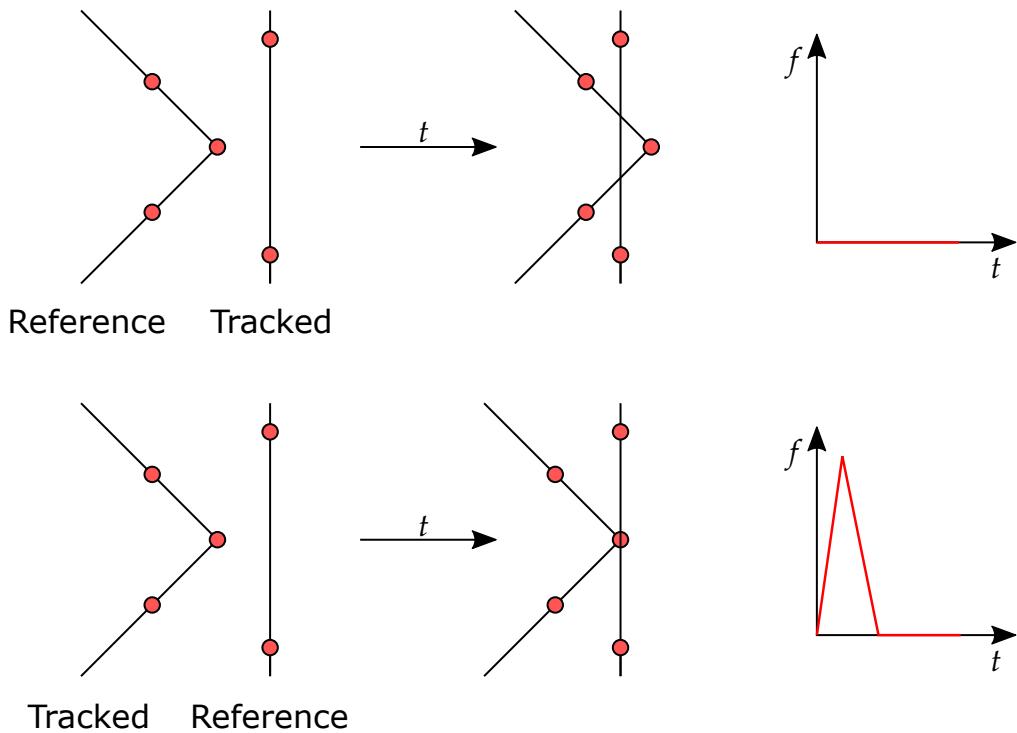
Figure 11-1. Illustration of the difference between the tracked and reference surfaces in nonsymmetric contact. In this example, nodes of the tracked surface are checked to see if they penetrate segments of the reference surface.

3. **Nonsymmetric and symmetric contact.** In nonsymmetric contact, penetration is only checked in one direction. The nodes/segments of one surface (the *tracked surface*) are checked to see if they penetrate the segments of another surface (the *reference surface*), but the nodes/segments of the other surface are not checked to see if they penetrate the first surface. See Figure 11-1. Symmetric contact checks both surfaces for penetration, usually by running the contact algorithm twice. Thus, both surfaces are the tracked and reference surfaces depending on the context. Previously, the tracked surface was known as the *slave*, and the reference surface was known as the *master*.

Setting the Contact Interface

Another essential consideration influenced by the contact characteristics is specifying the two sides of the contact interface. SURFA and SURFB set the two sides of the contact interface on Mandatory Card 1. SURFA is equivalent to *SSID*, and SURFB is equivalent to *MSID*. For nonsymmetric contact, SURFA specifies the *tracked* surface, while SURFB specifies the *reference* surface, meaning that nodes or segments of SURFA are checked to see if they penetrate SURFB. The nodes/segments of SURFB can penetrate the segments of SURFA for these kinds of contact. See Figure 11-1.

For symmetric contacts, both sides of the contact interface are checked for penetration, and contact forces are applied accordingly. The choice of SURFA and SURFB is arbitrary for symmetric contact. Depending on the context, each surface acts as the reference and tracked surfaces. For single-surface contact, only SURFA is input, and penetration is checked on both sides of the self-contact interface.

For tied contacts, SURFA nodes are tied to SURFB segments, usually at the beginning of the simulation. Note that there are some single-surface tied contacts where only SURFA is input, and the surface is tied to itself.

Note on segment-to-segment contact algorithms
We have implemented two kinds of segment-to-segment contact algorithms: SOFT = 2 and Mortar. Note that in this manual, we will use segment-to-segment to refer to SOFT = 2 and will always mention Mortar when referring to Mortar segment-to-segment to reduce confusion. Major differences between these algorithms include that Mortar is devised to work well for the implicit solver and is nonsymmetric. In contrast, segment-to-segment is generally symmetric (with some exceptions) and was designed to improve contact behavior compared to nodes-to-segment contacts in certain situations. About SOFT = 2 under Optional Card A briefly discusses segment-to-segment contact. See Remark 14 in the General Remarks section for a discussion of Mortar contact.

Note that previously, SOFT = 2 was called segment-based contact. We have changed this to segment-to-segment contact because it is more descriptive and matches the literature.

Data cards for *CONTACT keyword
Cards *must* appear in the *exact* order listed below.

CARD	DESCRIPTION
ID	Card required when *OPTION3* is set to ID; otherwise, this card is omitted.
MPP 1	Card required when *OPTION5* is set to MPP.
MPP 2	Optional card that can *only* be included if *OPTION5* is set to MPP.
Card 1	Always required.
Card 2	Always required.
Card 3	Always required.
Card 4	Required for the following permutations of *CONTACT.

> **NOTE:** The format of Card 4 (which can include multiple cards) is **different** for each option listed.

AUTOMATIC_..._TIEBREAK

AUTOMATIC_..._TIEBREAK_USER

AUTOMATIC_SURFACE_..._COMPOSITE/LUBRICATION

AUTOMATIC_SINGLE_SURFACE_TIED

AUTOMATIC_..._TIED_WELD

CONSTRAINT_*type*

DRAWBEAD

ERODING_*type*

*type*_INTERFERENCE

RIGID_*type*

TIEBREAK_NODES_...

TIEBREAK_SURFACE_...

SURFACE_TO_SURFACE_CONTRACTION_JOINT

CARD	DESCRIPTION
THRM 1	Required if *OPTION2* is set to either THERMAL or THERMAL_-FRICTION. Otherwise omit.
THRM 2	Required if *OPTION2* is set to THERMAL_FRICTION. Otherwise omit.
THRM 2.1	Inclusion of this card depends on the input on THRM 2.
ORFR 1, ORFR 2, ORFR 3, ORFR 4	Required if *OPTION6* is set. Otherwise omit. Contains friction co-efficients.
Optional Card A	Optional parameters.

> **NOTE:** Default values are highly optimized.
>
> **NOTE:** *Required* if including Optional Card B. If Optional Card A is a blank line, then values are set to their defaults, and Optional Card B may follow.

CARD	DESCRIPTION
Optional Card B	Optional parameters. *Required* if including Optional Card C. (See Optional Card A note; similar logic applies)
Optional Card C	Optional parameters. *Required* if including Optional Card D. (See Optional Card A note; similar logic applies.)
Optional Card D	Optional parameters. *Required* if including Optional Card E. (See Optional Card A note; similar logic applies.)
Optional Card E	Optional parameters. *Required* if including Optional Card F. (See Optional Card A note; similar logic applies.)
Optional Card F	Optional parameters. *Required* if including Optional Card G. (See Optional Card A note; similar logic applies.)
Optional Card G	Optional parameters

OPTIONS FOR *CONTACT KEYWORD

OPTION	REQUIRED	DESCRIPTION
OPTION1	Yes	Specifies contact type
OPTION2	No	Flag for thermal
OPTION3	No	Flag indicating ID cards follow
OPTION4	No	Offset options
OPTION5	No	Flag for MPP
OPTION6	No	Flag for orthotropic friction

Allowed values for *OPTION1*:

The following list covers the unique terminology used to describe LS-DYNA's contact types. Additional notes on contact types and a few examples are provided at the end of this manual page in "General Remarks: *CONTACT." The LS-DYNA Theoretical Manual covers the algorithms at a higher level.

Following this list are all the available values for *OPTION1*. All contact types are available for explicit and implicit calculations.

1. **Categories of contact types.** Generally, we can split the contact types specified with *OPTION1* into seven basic categories: one-way, two-way, single surface, tied, Mortar, constraint, and sliding only.

    a) One-way contact is nonsymmetric. The nodes (or segments) of SURFA cannot penetrate the segments of SURFB, but SURFB nodes and segments may penetrate SURFA. Thus, deciding on the surfaces for SURFA and SURFB is vital for this type of contact. Generally, SURFA should have a finer mesh. Compressive loads are transferred between the nodes (or segments) of SURFA and segments of SURFB when they are in contact.

    b) Two-way contact is like one-way contact, except it is symmetric. In this case, LS-DYNA checks for SURFA penetrating SURFB and then SURFB penetrating SURFA. As a result, the definitions of SURFA and SURFB are arbitrary. Using this type of contact costs roughly double that of one-way.

    c) Single surface contact requires only the input of SURFA. Unlike the other contact types, LS-DYNA checks the single surface for self-penetration. Its algorithms are based on contacts with SURFACE_TO_SURFACE in the name (see the theory manual for details).

d) Tied contact is for "gluing" two surfaces together, meaning no sliding or separation. It is particularly good for tying parts with incompatible meshes. For this contact, the nodes of SURFA (tracked) are tied to and, thus, constrained to move with the segments of SURFB (reference). No sliding is allowed, except for some TIEBREAK contacts. We also have some single-surface tied contacts for which only SURFA is input. In this case, the surface is tied to itself (AUTOMATIC_SINGLE_SURFACE_TIED, AUTOMATIC_-GENERAL_TIEBREAK, and AUTOMATIC_SINGLE_SURFACE_TIE-BREAK).

Tying only occurs at the beginning of the simulation unless the contact type is some of the TIEBREAK versions. Thus, if a tracked node enters a reference segment's vicinity during the simulation, the node will not be tied to the segment. For tying to occur, the tracked node must lie within the orthogonal projection of a reference segment, and the gap between the node and segment must be less than a certain tolerance (see Remark 4 in General Remarks). If *OPTION4* (offsets) is used, the tied tracked node can have an offset from the reference segment. Otherwise, during initialization, the node is moved to the surface. We recommend specifying tied contacts with node or segment sets instead of parts or part sets since they give you more control over what is tied. Identifying the surfaces this way prevents unintended tying.

Some tied contacts constrain only translational degrees of freedom, while others additionally constrain rotational degrees of freedom. Some types of this contact allow for failure. TIEBREAK is a special case of a tied contact allowing failure in which the contact usually becomes a regular one-way, two-way, or single surface version after failure. How a TIEBREAK contact behaves after failure depends on what the contact type would be if TIEBREAK were excluded. Note that even for automatic tiebreak segment normals should be oriented properly for these types of contact (see Remark 6 under Card 4: AUTOMATIC_...TIEBREAK).

Tied contacts with TIEBREAK or PENALTY in the name or ones with OFFSET and BEAM_OFFSET appended to the name using *OPTION4* are penalty-based. All others are constraint-based.

e) Mortar contact is a segment-to-segment-based contact with its own algorithms. It is designated with MORTAR in the name. It is especially well-suited for the implicit solver. It is not symmetric since segments on SURFA are not treated the same as SURFB. Mortar contact includes a single surface version (AUTOMATIC_SINGLE_SURFACE_MORTAR) where only SURFA is input to specify the contact interface. For more details about Mortar contact, see Remark 14 in the General Remarks section and the theory manual.

f) For this discussion, constraint category contacts have CONSTRAINT in the name. These contacts are constraint-based. They can be symmetric or non-symmetric depending on the setting of KPF on Card 4. See Remark 11 in General Remarks.

g) Sliding-only contact is one of the oldest types. LS-DYNA contains two versions: SLIDING_ONLY and SLIDING_ONLY_PENALTY. The first is constraint-based, while the second is penalty-based. Sliding-only contact restricts the nodes of SURFA to slide along SURFB. No separation is allowed with these methods. These contact types help treat interfaces where the gaseous detonation products of a high explosive act on a solid material. See the theory manual for more details. This type of contact is only available for the explicit solver.

2. **NODES_TO_SURFACE.** Contacts with NODES_TO_SURFACE in the name are one-way or tied contacts. SURFA can be defined with a node set for these contacts, and contact is only nodes to segments (neither segment-to-segment nor Mortar contact is available).

3. **SURFACE_TO_SURFACE.** Contacts with SURFACE_TO_SURFACE (also called surface-to-surface contact) in the name can have SURFA defined with segment sets and shell sets. SURFA cannot be represented with node sets. Note that being able to input as segments does not mean that the contact formulation is segment-to-segment (SOFT = 2 or Mortar). The contact formulation for most penalty contacts depends on the value of SOFT on Optional Card A. Unless the contact formulation is segment-to-segment (SOFT = 2) or the contact type includes MORTAR, the formulation looks at nodal penetrations into segments, not segment penetration into segments.

SURFACE_TO_SURFACE contacts have special, advanced features compared to NODES_TO_SURFACE contact, partly due to how they are defined. For instance, SURFACE_TO_SURFACE contacts support thermal transfer, while NODES_TO_SURFACE generally contacts do not. Note that the algorithms for SINGLE_SURFACE are based on SURFACE_TO_SURFACE. Thus, parameters that apply to surface-to-surface contact apply to SINGLE_SURFACE.

4. **AUTOMATIC**. Automatic contacts are designated with AUTOMATIC in the type. The algorithms for this type are better than those for non-automatic contacts for disjoint meshes. Automatic contacts are two-sided in that the algorithms detect penetration on either side of a shell element. In contrast, non-automatic contacts are one-sided. Thus, segment orientation does not matter for automatic contact but is crucial for non-automatic contact. Therefore, automatic contacts are advantageous for crash analysis.

To detect penetration on both sides, automatic contacts include a contact thickness of half the thickness on each side of the shell midplane (see Remark 13 in

the General Remarks). They also include a contact surface on the exterior edges of a shell surface with a radius of half the contact thickness. Because of how these projections are performed, the shell surface has a continuous contact surface. Similarly, for beam elements where beam contact is considered, the contact surface is offset from the centerline of the beam element by an equivalent radius of the beam cross-section (see the theory manual). Because of the contact surfaces, modeling appropriate gaps between parts is critical for automatic contact with beam and shell parts. See also Remark 3 in the General Remarks.

5. **ERODING.** Contact types with ERODING in the name are suitable for when elements in the contact interface fail and are deleted. These types allow for updating the contact surface due to element deletion.

6. **SMOOTH.** For SMOOTH contact, a smooth curve-fitted surface represents the reference surface segments to provide a more accurate representation of the actual surface, reduce the contact noise, and produce smoother results with coarser meshes.

   SMOOTH contact is unavailable in SMP for FORMING_SURFACE_TO_SURFACE and SURFACE_TO_SURFACE contacts.

   All contact options that include SMOOTH are available for MPP. Furthermore, for SURFACE_TO_SURFACE and SINGLE_SURFACE contacts with the SMOOTH option, both sides of the contact interface are smoothed every cycle, thereby slowing the contact treatment considerably.

   The SMOOTH option does not apply to segment-to-segment (SOFT = 2) contacts.

7. **DRAWBEAD.** DRAWBEAD contacts are a particular type of one-way contact for simulating draw beads. SURFA should be the nodes defining the draw bead, while SURFB is the blank. See Card 4: DRAWBEAD and the theory manual for a complete description.

8. **FORMING.** Contact types that include FORMING in the name are mainly used for metal forming applications. A connected mesh is not required for the SURFB (tooling) side, but the orientation of the mesh *must* be in the same direction. These contact types are based on AUTOMATIC-type contacts, and consequently, the performance is better than the original two surface contacts.

9. **INTERFERENCE.** Contact types with INTERFERENCE are intended for modeling parts with an interference fit. Therefore, LS-DYNA does not check for initial penetrations for this contact. The overlap is instead removed by contact forces, causing stress and deformation in the interference fit parts.

Single Surface or Self-Contact

AIRBAG_SINGLE_SURFACE

AUTOMATIC_GENERAL (see Remark 2 in General Remarks)

AUTOMATIC_GENERAL_EDGEONLY (see Remark 2 in General Remarks)

AUTOMATIC_GENERAL_INTERIOR (see Remark 2 in General Remarks)

AUTOMATIC_SINGLE_SURFACE (see Remark 2 in General Remarks)

AUTOMATIC_SINGLE_SURFACE_SMOOTH

ERODING_SINGLE_SURFACE

SINGLE_EDGE (see Remark 3 in General Remarks)

SINGLE_SURFACE

One-Way Contact

AUTOMATIC_BEAMS_TO_SURFACE

AUTOMATIC_NODES_TO_SURFACE

AUTOMATIC_NODES_TO_SURFACE_SMOOTH

AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE

AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH

DRAWBEAD

DRAWBEAD_BENDING

DRAWBEAD_INITIALIZE

ERODING_NODES_TO_SURFACE

FORMING_NODES_TO_SURFACE

FORMING_NODES_TO_SURFACE_SMOOTH

FORMING_ONE_WAY_SURFACE_TO_SURFACE

FORMING_ONE_WAY_SURFACE_TO_SURFACE_SMOOTH

NODES_TO_SURFACE

NODES_TO_SURFACE_INTERFERENCE

NODES_TO_SURFACE_SMOOTH

ONE_WAY_SURFACE_TO_SURFACE

ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE

ONE_WAY_SURFACE_TO_SURFACE_SMOOTH

RIGID_NODES_TO_RIGID_BODY

RIGID_BODY_ONE_WAY_TO_RIGID_BODY

Two-Way Contact

AUTOMATIC_SURFACE_TO_SURFACE

AUTOMATIC_SURFACE_TO_SURFACE_SMOOTH

ERODING_SURFACE_TO_SURFACE

FORMING_SURFACE_TO_SURFACE

FORMING_SURFACE_TO_SURFACE_SMOOTH

RIGID_BODY_TWO_WAY_TO_RIGID_BODY

SURFACE_TO_SURFACE

SURFACE_TO_SURFACE_INTERFERENCE

SURFACE_TO_SURFACE_SMOOTH

SURFACE_TO_SURFACE_CONTRACTION_JOINT

Tied Contact

The following types are tied contacts (not tiebreak). Remark 7 in the General Remarks section discusses which tied contacts can be used with rigid bodies. Remark 4 in General Remarks concerns the tying criterion. For a discussion of which TIED contact to use in which situation, see Remark 5 in General Remarks. For using TIED contacts with the implicit solver, see Remark 6 in General Remarks.

AUTOMATIC_SINGLE_SURFACE_TIED

AUTOMATIC_SURFACE_TO_SURFACE_TIED_WELD

SPOTWELD

SPOTWELD_WITH_TORSION (see Remark 8 in General Remarks)

SPOTWELD_WITH_TORSION_PENALTY

TIED_NODES_TO_SURFACE

TIED_SHELL_EDGE_TO_SURFACE

TIED_SHELL_EDGE_TO_SOLID

TIED_SURFACE_TO_SURFACE (see Remark 3 under Mandatory Card 1)

TIED_SURFACE_TO_SURFACE_FAILURE (SMP only)

The following contacts are tiebreak contacts, which means that these contacts generally turn into regular sliding contacts after failure. See the Card 4 remarks for each type for the failure criteria and the behavior after failure. They use penalty-based contact formulations.

AUTOMATIC_GENERAL_TIEBREAK

AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK

AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_DAMPING

AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_USER

AUTOMATIC_SINGLE_SURFACE_TIEBREAK

AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK

AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_USER

TIEBREAK_NODES_TO_SURFACE

TIEBREAK_NODES_ONLY

TIEBREAK_SURFACE_TO_SURFACE

TIEBREAK_SURFACE_TO_SURFACE_ONLY

Mortar Contact

AUTOMATIC_SINGLE_SURFACE_MORTAR

AUTOMATIC_SURFACE_TO_SURFACE_MORTAR

AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED

AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED_WELD

AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_MORTAR

AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_USER_MORTAR

FORMING_SURFACE_TO_SURFACE_MORTAR

Constraint Contacts

CONSTRAINT_NODES_TO_SURFACE

CONSTRAINT_SURFACE_TO_SURFACE

Sliding Only Contacts

SLIDING_ONLY

SLIDING_ONLY_PENALTY

Allowed values for *OPTION2*:

THERMAL

THERMAL_FRICTION

> **NOTE:** For sliding contacts, the *THERMAL* and *THERMAL_-FRICTION* options are restricted to contact types having *"SURFACE_TO_SURFACE"* in *OPTION1*. For tied contacts, both *"TIED_NODES_TO_SURFACE"* and *"TIED_SHELL_EDGE_TO_SURFACE"* are supported only if SURFA is *not* a node set.

Allowed value for *OPTION3*:

ID

Allowed values for *OPTION4*:

OPTION4 specifies that offsets may be used with the tied contact types. If one of these three offset options is set, then offsets are permitted for these contact types. If not, the nodes are projected back to the contact surface during the initialization phase, and a constraint formulation is used. Note that in a constraint formulation, the nodes of rigid bodies are not permitted in the definition.

OFFSET

BEAM_OFFSET

CONSTRAINED_OFFSET

OFFSET keyword option

The OFFSET option switches the formulation from a constraint-type formulation to one that is penalty-based, where discrete spring elements between the tracked nodes and reference segments transfer the force and moment (if applicable) resultants. Rigid bodies can be used with the OFFSET option.

OFFSET is available when *OPTION1* is:

TIED_NODES_TO_SURFACE

TIED_SHELL_EDGE_TO_SURFACE

TIED_SURFACE_TO_SURFACE

With OFFSET, no coupling occurs between the transmitted forces and moments; thus, equilibrium is not enforced. For TIED_NODES_TO_SURFACE, the moment equilibrium is enforced by setting FTORQ = 2 on Card E. For TIED_SHELL_EDGE_TO_SURFACE contact, the BEAM_OFFSET option may be preferred since corresponding moments accompany transmitted forces.

BEAM_OFFSET keyword option

The BEAM_OFFSET option switches the formulation from a constraint-type formulation to one that is penalty-based. Beam-like springs transfer force and moment resultants between the tracked nodes and the reference segments. Rigid bodies can be used with this option.

BEAM_OFFSET is available when *OPTION1* is:

TIED_SHELL_EDGE_TO_SURFACE

SPOTWELD

BEAM_OFFSET is also available when *OPTION1* is:

AUTOMATIC_SINGLE_SURFACE_TIEBREAK

AUTOMATIC_GENERAL_TIEBREAK

but moments are transferred only when the variable FTORQ is invoked (see Optional Card E). Those moments are transmitted to the reference surface segments through nodal forces applied to the nodes of these segments.

CONSTRAINED_OFFSET keyword option

The CONSTRAINED_OFFSET option is a constraint-type formulation. CON-STRAINED_OFFSET is available when *OPTION1* is:

TIED_NODES_TO_SURFACE

TIED_SHELL_EDGE_TO_SURFACE

TIED_SURFACE_TO_SURFACE

SPOTWELD

Allowed value *OPTION5*:

MPP

Allowed value for *OPTION6*:

ORTHO_FRICTION

ORTHO_FRICTION is available when *OPTION1* is:

AUTOMATIC_SURFACE_TO_SURFACE(_SMOOTH /_MORTAR)

AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE(_SMOOTH)

FORMING_ONE_WAY_SURFACE_TO_SURFACE(_SMOOTH)

ID Card:

Additional card for ID keyword option

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

The contact ID is needed during full deck restarts for contact initialization. If the contact ID is undefined, the default ID is determined by the sequence of the contact definitions; that is, the first contact definition has an ID of 1, the second, 2, and so forth. For a successful run in a full deck restart without contact IDs, no contact interfaces can be deleted, and added contact interfaces must be placed after the last definition in the previous run. Some of the peripheral LS-DYNA codes pick up the ID and headings to aid in post-processing.

VARIABLE	DESCRIPTION
CID	Contact interface ID. This must be a unique number.
HEADING	Interface descriptor. We suggest using unique descriptions.

MPP Cards:

Variables set with these cards are only active when using MPP LS-DYNA.

MPP Card 1. Additional card for the MPP option. When SOFT = 2 on Optional Card A, this card is ignored but still read in.

MPP 1	1	2	3	4	5	6	7	8
Variable	IGNORE	BCKT	LCBCKT	NS2TRK	INITITR	PARMAX		CPARM8
Type	I	I	I	I	I	F		I
Default	0	200	none	3	2	↓		0

MPP Card 2. The keyword reader will interpret the card following MPP Card 1 as MPP Card 2 if the first column of the card is occupied by an ampersand. Otherwise, it is interpreted as Card 1. When SOFT = 2 on Optional Card A, GRPABLE is the *only* field on this card that is not ignored.

MPP 2	1	2	3	4	5	6	7	8
Variable	&	CHKSEGS	PENSF	GRPABLE		IGTOL		
Type		I	F	I		F		
Default		0	1.0	0		0.0		

VARIABLE	DESCRIPTION
IGNORE	For automatic and non-automatic contacts, setting this variable to 1 turns on the "ignore initial penetrations" for this contact. Alternatively, for automatic contacts only, setting IGNORE = 1 on Card 4 of *CONTROL_CONTACT or on Optional Card C of *CONTACT turns on this option. In other words, for automatic contacts, if IGNORE is set to 1 in any of three places, initial penetrations are tracked. For non-automatic contacts, setting IGNORE to 1 here causes initial penetrations to be tracked.
BCKT	Bucket sort frequency. This parameter does not apply when SOFT = 2 on Optional Card A or to Mortar contacts. For these two exceptions, the BSORT option on Optional Card A applies instead.

VARIABLE	DESCRIPTION
	Note that NSBCS on *CONTROL_CONTACT does not apply to SOFT = 0 and SOFT = 1 for MPP.
LCBCKT	Load curve for bucket sort frequency. This parameter does not apply when SOFT = 2 on Optional Card A or to Mortar contacts. For the two exceptions, the negative BSORT option on Optional Card A applies instead.
NS2TRK	Number of potential contacts to track for each tracked node. The normal input for this (DEPTH on Optional Card A) is ignored.
INITITR	Number of iterations to perform when trying to eliminate initial penetrations. Note that an input of 0 means 0, not the default value (which is 2). Leaving this field blank will set INITITR to 2.
PARMAX	The parametric extension distance for contact segments. The MAXPAR parameter on Optional Card A is not used for MPP. For *CONTACT_TIEBREAK_..., PARMAX is hardcoded as 1.1. For all other tied contacts, the default is 1.0005. For tied contacts the default is 1.035 and, the actual extension used is computed as follows:

$$\text{PARMAX}_{\text{computed}} = \begin{cases} 1.0 + \text{PARMAX} & 0.0 < \text{PARMAX} < 0.5 \\ \text{PARMAX} & 1.0 \leq \text{PARMAX} \leq 1.0004 \\ \max(\text{PARMAX}, 1.035) & \text{otherwise} \end{cases}$$

CPARM8	Flag for behavior of AUTOMATIC_GENERAL contacts. CPARM8's value is interpreted as two separate flags: OPT1 and OPT2 according to the rule,

<div align="center">

CPARM8 = OPT1 + OPT2.

</div>

When OPT1 and OPT2 are *both* set, *both* options are active.

OPT1. Flag to exclude beam-to-beam contact from the same PID.

  EQ.0: Flag is not set (default).

  EQ.1: Flag is set.

  EQ.2: Flag is set. CPARM8 = 2 additionally permits contact treatment of spot weld (type 9) beams in AUTOMATIC_GENERAL contacts. See Remark 2 for details.

OPT2. Flag to shift a generated beam, affecting only shell-edge-to-shell-edge treatment. See also SRNDE in Optional Card E.

  EQ.10: A beam generated on an exterior shell edge will be shifted into the shell by half the shell thickness.

VARIABLE	DESCRIPTION
	Therefore, the shell-edge-to-shell-edge contact starts right at the shell edge and not at an extension of the shell edge.
CHKSEGS	If this value is nonzero, then for node-to-surface and surface-to-surface contacts LS-DYNA performs a special check at time 0 for elements that are inverted (or nearly so). These elements are removed from contact. These poorly formed elements have been known to occur on the tooling in metal forming problems, which allows these problems to run. It should not normally be needed for reasonable meshes.
PENSF	This option is used together with IGNORE for 3D forging problems. If nonzero, the IGNORE penetration distance is multiplied by this value each cycle, effectively pushing the tracked node back out to the surface. This is useful for nodes that might get generated below the reference surface during 3D remeshing. Care should be exercised, as energy may be generated, and stability may be affected for values lower than 0.95. A value in the range of 0.98 to 0.99 or higher (but < 1.0) is recommended.
GRPABLE	Set to 1 to invoke an alternate MPP communication algorithm for various SINGLE_SURFACE (including AUTOMATIC_GENERAL), NODES_TO_SURFACE, SURFACE_TO_SURFACE, ERODING, and SOFT = 2 contacts. This groupable algorithm does not support all contact options, including MORTAR. It is still under development. It can be significantly faster and scale better than the normal algorithm when there are more than two or three applicable contact types defined in the model. It is intended to speed up the contact processing without changing the behavior of the contact. See also *CONTROL_MPP_CONTACT_GROUPABLE.

      EQ.-1: Do not use the groupable algorithm for this contact.

      EQ.0: Do not use the groupable algorithm for the contact. However, GRP on *CONTROL_MPP_CONTACT_GROUPABLE overrides this setting.

      EQ.1: Use the groupable algorithm for this contact if supported. However, GRP on *CONTROL_MPP_CONTACT_GROUPABLE overrides this setting.

IGTOL	Scale factor for determining the contact exclusion distance. This option is available for MPP only. It applies to *CONTACT_AUTOMATIC_GENERAL and *CONTACT_AUTOMATIC_SIN-

VARIABLE	DESCRIPTION
	GLE_SURFACE with SOFT = 0 or 1. See Remark 4 for details.

Remarks:

1. **SOFT = 2.** Except for the GRPABLE field, the MPP cards are ignored by segment-to-segment (SOFT = 2 on Optional Card A) contact. When SOFT = 2, the BSORT parameter on Optional Card A can override the default bucket sort frequency. See About SOFT = 2 for more details about segment-to-segment contact

2. **Spot welds in contacts.** In general, parts with spot-weld material are ignored by the contacts. Setting OPT1 on CPARM8 can change this behavior. Depending on the contact type and type of ID in SURFA / SURFB, spot-weld parts can be included in the contact.

   a) For SURFATYP / SURFBTYP = 2 (part set), we have:

      i) For AUTOMATIC_GENERAL contacts, exclude all spot-weld parts. However, if OPT1 = 2, the spot-weld beams in the set are included.

      ii) For all other contacts, include those spot-weld parts that are in the set.

   b) For SURFATYP / SURFBTYP = 5 (include all), all spot-weld parts are excluded. However, if OPT1 = 2, spot-weld beams are included (regardless of contact type).

   c) For SURFATYP / SURFBTYP = 6 (exempted part set), all spot-weld parts are excluded (regardless of contact type or OPT1 value).

3. **Shell reference surface for tied contacts in MPP.** Unless the MPP contact is groupable (see GRPABLE here and GRP on *CONTROL_MPP_CONTACT_-GROUPABLE), MPP tied contacts ignore the influence of NLOC on *SECTION_-SHELL, OFFSET on *ELEMENT_SHELL_OFFSET, CNTCO on *CONTROL_-SHELL, and SHLOFF on Optional Card G on the shell reference surface. Non-groupable tied contacts always use the plane determined by the nodes for the shell reference surface. Thus, these fields for non-groupable tied contacts have no effect on determining what is tied or not. Groupable tied contacts honor the settings of these fields.

4. **IGTOL.** Setting IGTOL > 0.0 causes the generation of a list of nodes excluded from contact for each segment. Each list contains nodes within IGTOL*(segment thickness + node thickness) as measured by traveling along the edges of the contact segments (see Figures 11-2 and 11-3). Note that LS-DYNA builds the

exclusion list for each segment once during the first cycle. A value of 1.0 for IGTOL should be good for most reasonable meshes. Setting IGTOL > 0.0 enables the behavior of SSTHK = 1 on *CONTROL_CONTACT automatically for this contact.

This option will improve the behavior for cases when SOFT = 0 or 1, SSTHK = 1 on *CONTROL_CONTACT, and the mesh size is equal to or smaller than the thickness. Without it, spurious stresses/deformation might be observed depending on the geometry (see Figure 11-2).
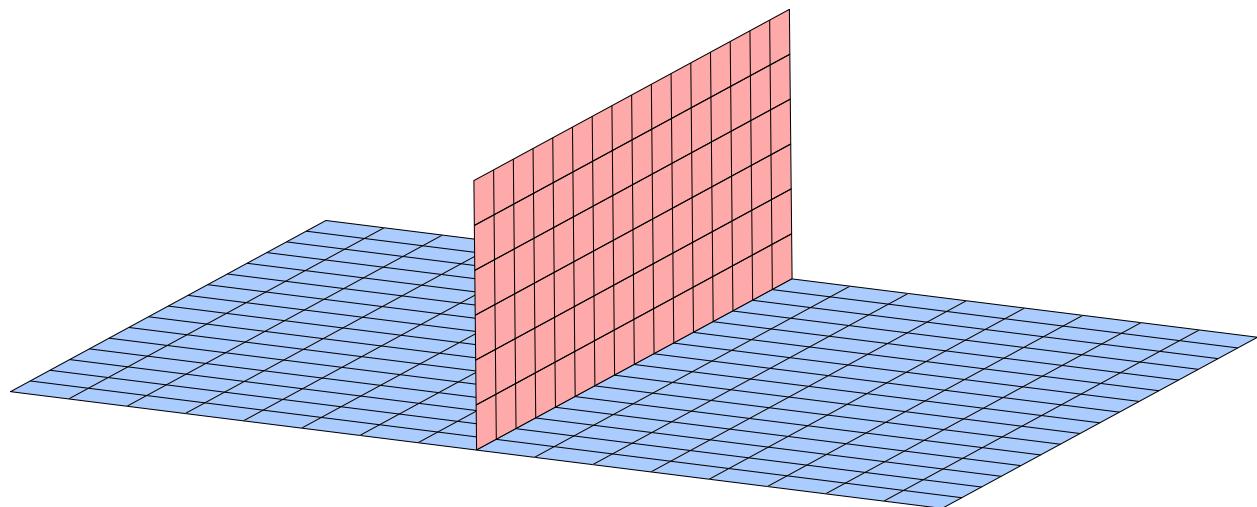
Figure 11-2. Example of a geometry that may require setting IGTOL > 0.0 when the mesh size is equal to or smaller than the thickness.
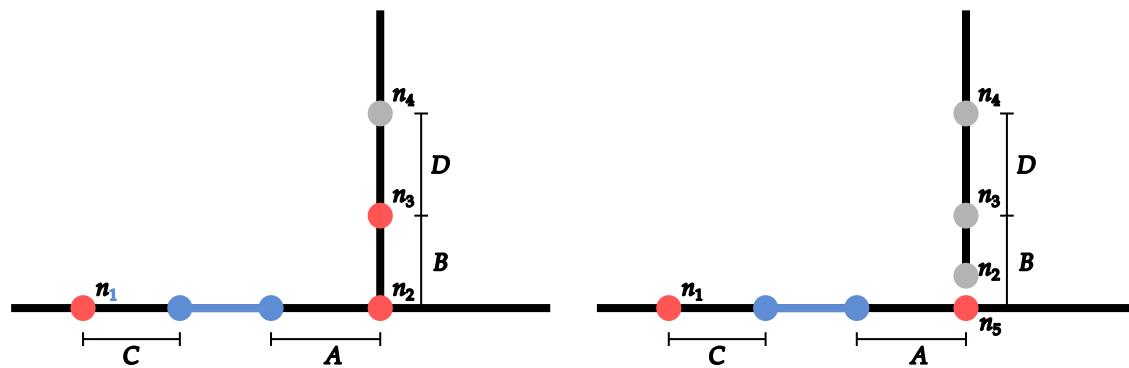
Figure 11-3. This figure illustrates two different scenarios for excluding nodes from contact with the segment in blue. In both scenarios red indicates nodes excluded from contact and grey nodes not excluded. We measure the distance by moving along the edges of the contact segments. In the left figure, n_3 is at a distance of $A + B$ from the segment and n_1 is at a distance C from the segment. In the right figure, n_2 and n_3 are not excluded because they do not connect to the segment through the mesh.

Mandatory Card 1:

> **NOTE:** SURFA is equivalent to the side of contact specified
> previously SSID, and SURFB is equivalent to the side
> of the contact specified previously with MSID.

Card 1	1	2	3	4	5	6	7	8
Variable	SURFA	SURFB	SURFATYP	SURFBTYP	SABOXID	SBBOXID	SAPR	SBPR
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none			0	0
Remarks	1	2			optional	optional		

VARIABLE	DESCRIPTION
SURFA	Segment set ID, node set ID, part set ID, part ID, shell element set ID, or branch ID for specifying the SURFA side of the contact interface (see Setting the Contact Interface). See *SET_SEGMENT, *SET_NODE_*OPTION*, *PART, *SET_PART or *SET_SHELL_*OPTION*. For ERODING_SINGLE_SURFACE and ERODING_SURFACE_TO_SURFACE contact types, use either a part ID or a part set ID. For ERODING_NODES_TO_SURFACE contact, use a node set which includes all nodes that may be exposed to contact as element erosion occurs.

> EQ.0: Includes all parts in the case of single surface contact types.

SURFB	Segment set ID, part set ID, part ID, shell element set ID, or branch ID for the SURFB side of the contact (see Setting the Contact Interface).

> EQ.0: SURFB side is not applicable for single surface contact types.

SURFATYP	ID type of SURFA:

> EQ.0: Segment set ID for surface-to-surface contact
>
> EQ.1: Shell element set ID for surface-to-surface contact

VARIABLE	DESCRIPTION
	EQ.2: Part set ID
	EQ.3: Part ID
	EQ.4: Node set ID for nodes-to-surface contact
	EQ.5: Include all non-spot-weld parts (SURFA field is ignored). See also CPARM8.
	EQ.6: Part set ID for exempted parts. All non-exempted parts (except spot-weld parts) are included in the contact.
	EQ.7: Branch ID; see *SET_PART_TREE.
	For AUTOMATIC_BEAMS_TO_SURFACE contact, either a part set ID or a part ID can be specified.
SURFBTYP	ID type of SURFB:
	EQ.0: Segment set ID
	EQ.1: Shell element set ID
	EQ.2: Part set ID
	EQ.3: Part ID
	EQ.5: Include all non-spot-weld parts (SURFB field is ignored). See also CPARM8.
	EQ.6: Part set ID for exempted parts. All non-exempted parts (except spot-weld parts) are included in the contact.
	EQ.7: Branch ID; see *SET_PART_TREE.
SABOXID	Include in contact definition only those SURFA nodes/segments within box SABOXID (corresponding to BOXID in *DEFINE_BOX), or if SABOXID is negative, only those SURFA nodes/segments within contact volume |SABOXID| (corresponding to CVID in *DEFINE_CONTACT_VOLUME). SABOXID can be used only if SURFATYP is set to 2, 3, 5, or 6, that is, SURFA is a part ID or part set ID. SABOXID is not available for ERODING contact types.
SBBOXID	Include in contact definition only those SURFB segments within box SBBOXID (corresponding to BOXID in *DEFINE_BOX), or if SBBOXID is negative, only those SURFB segments within contact volume |SBBOXID| (corresponding to CVID in *DEFINE_CONTACT_VOLUME). SBBOXID can be used only if SURFBTYP is set to 2, 3, 5, or 6, that is, SURFB is a part ID or part set ID. SBBOXID is not available for ERODING contact types.

VARIABLE	DESCRIPTION
SAPR	Include the SURFA side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files, and optionally in the dynain file for wear:

> EQ.0: Do not include.
>
> EQ.1: SURFA side forces included.
>
> EQ.2: Same as 1 but also allows for SURFA nodes to be written as *INITIAL_CONTACT_WEAR to dynain; see NCYC on *INTERFACE_SPRINGBACK_LSDYNA.

VARIABLE	DESCRIPTION
SBPR	Include the SURFB side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files, and optionally in the dynain file for wear:

> EQ.0: Do not include:
>
> EQ.1: SURFB side forces included.
>
> EQ.2: Same as 1, but also allows for SURFB nodes to be written as *INITIAL_CONTACT_WEAR to dynain; see NCYC on *INTERFACE_SPRINGBACK_LSDYNA.

Remarks:

1. **SURFA Set and Single Surface Contacts.** Setting the SURFA set ID equal to zero is valid only for the single surface contact algorithms, meaning the options:

   SINGLE_SURFACE

   AUTOMATIC_GENERAL_...

   AUTOMATIC_SINGLE_SURFACE_...

   AIRBAG_…

   ERODING_SINGLE_SURFACE

2. **SURFB Set with Single Surface Contact** A SURFB set ID is not defined for the single surface contact algorithms (including AUTOMATIC_GENERAL).

3. **Selecting Sides of Contact Interface for TIED_SURFACE_TO_SURFACE.** For *CONTACT_TIED_SURFACE_TO_SURFACE, we generally recommend making SURFA the more finely meshed part. This recommendation has exceptions. For instance, if the coarsely meshed SURFB side deforms, then the finer SURFA side will deform with high stress localizations in the folding lines.

Mandatory Card 2:

Card 2	1	2	3	4	5	6	7	8
Variable	FS	FD	DC	VC	VDC	PENCHK	BT	DT
Type	F	F	F	F	F	I	F	F
Default	0.	0.	0.	0.	0.	0	0.	10^{20}

VARIABLE	DESCRIPTION

If *OPTION1* is TIED_SURFACE_TO_SURFACE_FAILURE, then

FS   Normal tensile stress at failure. Failure occurs if

$$\left[\frac{\max(0.0, \sigma_{\text{normal}})}{FS}\right]^2 + \left[\frac{\sigma_{\text{shear}}}{FD}\right]^2 > 1$$

where σ_{normal} and σ_{shear} are the interface normal and shear stresses.

FD   Shear stress at failure. See FS.

Else

FS   Static coefficient of friction. If FS is > 0 and not equal to 2. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact according to,

$$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-\text{DC}|v_{\text{rel}}|} \ .$$

The three other possibilities are:

EQ.2:   For a subset of SURFACE_TO_SURFACE type contacts (see Remark 1 below), FD is a table ID (see *DEFINE_TABLE). That table specifies two or more values of contact pressure, with each pressure value in the table corresponding to a curve of friction coefficient as a function of relative velocity. Thus, the friction coefficient becomes a function of pressure and relative velocity. See Figure 11-4.

EQ.-2:   If only one friction table is defined using *DEFINE_FRICTION, it will be used and there is no need to define parameter FD. If more than one friction table is defined, then the friction table ID is defined by FD

VARIABLE	DESCRIPTION
	below.
	EQ.-1: If the frictional coefficients defined in the *PART section are to be used, set FS to -1.0.
	WARNING: The FS = -1.0 and F = -2.0 options apply only to contact types:

> SINGLE_SURFACE
>
> AIRBAG_SINGLE_SURFACE
>
> AUTOMATIC_GENERAL
>
> AUTOMATIC_SINGLE_SURFACE
>
> AUTOMATIC_SINGLE_SURFACE_MORTAR
>
> AUTOMATIC_NODES_TO_SURFACE
>
> AUTOMATIC_SURFACE_TO_SURFACE
>
> AUTOMATIC_SURFACE_TO_SURFACE_MORTAR
>
> AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
>
> ERODING_SINGLE_SURFACE

VARIABLE	DESCRIPTION
FD	Dynamic coefficient of friction. If FS > 0 and not equal to 2, the frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact according to, $$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-DC|v_{\text{rel}}|}$$ Otherwise:
	FS.EQ.-2: Friction table ID if more than one friction table is defined
	FS.EQ.2: Table ID for table that specifies two or more values of contact pressure, with each pressure value in the table corresponding to a curve of friction coefficient as a function of relative velocity.

End If

VARIABLE	DESCRIPTION
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-\text{DC}|v_{\text{rel}}|}.$$
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed as $F_{\text{lim}} = \text{VC} \times A_{\text{cont}}$ with A_{cont} being the area of the segment contacted by the node in contact. The suggested value for VC is the yield stress in shear $\text{VC} = \sigma_o / \sqrt{3}$ where σ_0 is the yield stress of the contacted material.
VDC	Viscous damping coefficient in percent of critical or the coefficient of restitution expressed as percentage (see ICOR on Optional Card E). In order to avoid undesirable oscillation in contact, such as for sheet forming simulation, a contact damping perpendicular to the contacting surfaces is applied. When ICOR, the 6th column of Optional Card E, is not defined or 0, the applied damping coefficient is given by $$\xi = \frac{\text{VDC}}{100}\xi_{\text{crit}} \ ,$$ where VDC is an integer (in units of percent) between 0 and 100. The formula for critical damping is $$\xi_{\text{crit}} = 2m\omega,$$ where m is determined by nodal masses as $$m = \min(m_{\text{tracked}}, m_{\text{reference}}) \ ,$$ and ω is determined from k, the interface stiffness, according to $$\omega = \sqrt{k\frac{m_{\text{tracked}} + m_{\text{reference}}}{m_{\text{tracked}}m_{\text{reference}}}} \ .$$
PENCHK	Small penetration in contact search option. If the tracked node penetrates more than the segment thickness times the factor XPENE (see *CONTROL_CONTACT), the penetration is ignored, and the tracked node is set free. The thickness is taken as the shell thickness if the segment belongs to a shell element, or it is taken as 1/20 of its shortest diagonal if the segment belongs to a solid element. This option applies to the surface-to-surface contact algorithms. See Table 11-2 for contact types and more details.

VARIABLE	DESCRIPTION

BT   Birth time (contact surface becomes active at this time):

- LT.0: Birth time is set to |BT|. When negative, birth time is followed during the dynamic relaxation phase of the calculation. After dynamic relaxation has completed, contact is activated regardless of the value of BT.

- EQ.0: Birth time is inactive, meaning contact is always active

- GT.0: If DT = -9999, BT is interpreted as the curve or table ID defining multiple pairs of birth-time/death-time; see Remark 2 below. Otherwise, if DT > 0, birth time applies both during and after dynamic relaxation.

DT   Death time (contact surface is deactivated at this time):

- LT.0: If DT = -9999, BT is interpreted as the curve or table ID defining multiple pairs of birth-time/death-time. Otherwise, negative DT indicates that contact is inactive during dynamic relaxation. After dynamic relaxation the birth and death times are followed and set to |BT| and |DT|, respectively.

- EQ.0: DT defaults to 10^{20}.

- GT.0: DT sets the time at which the contact is deactivated.
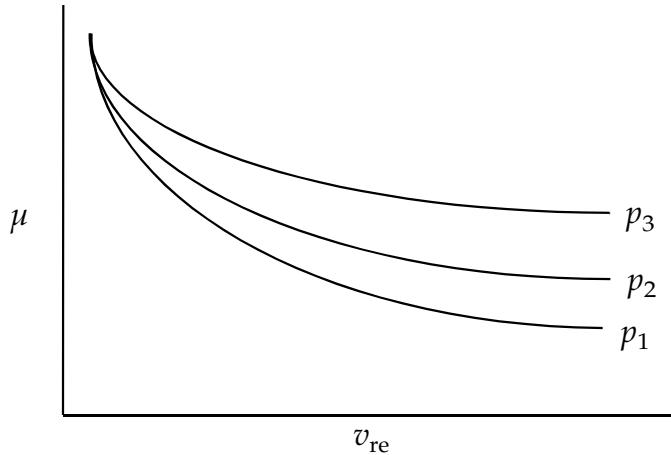
Figure 11-4. Friction coefficient, μ, can be a function of relative velocity and pressure. See Remark 1 for FS = 2.0.

Remarks:

1. **Implemented Contacts for FS = 2.** The FS = 2 method of specifying the friction coefficient as a function of pressure and relative velocity is implemented for all contact types for which SOFT = 2 (see When segment-to-segment contact is available in the Optional Card A section) and for all Mortar contacts. When FS = 2 and SOFT = 2, we recommend setting FNLSCL to a value in the range of 0.5 to 1.0 and setting DNLSCL to 0 (refer to Remark 5 under the description of Optional Card D for *CONTACT). These recommendations can be ignored for Mortar contact as they do not apply.

   The following ONE_WAY contact types can safely be used with FS = 2 when SOFT = 0 or 1.

   ONE_WAY_SURFACE_TO_SURFACE   (SMP and MPP)

   AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE   (MPP only)

   FORMING_ONE_WAY_SURFACE_TO_SURFACE   (MPP only)

   FS = 2 with SOFT = 0 or 1 is implemented but not advised for the following contact types:

   SURFACE_TO_SURFACE   (SMP and MPP)

   AUTOMATIC_SURFACE_TO_SURFACE   (MPP only)

   FORMING_SURFACE_TO_SURFACE   (MPP only)

A caveat pertaining to the MPP contacts listed above is that the "groupable" option must *not* be invoked. See *CONTROL_MPP_CONTACT_GROUPABLE.

For SOFT = 0 or 1, FS = 2 is not implemented in SMP for non-Mortar AUTOMATIC and FORMING contact types. The static friction coefficient will literally be taken as 2.0 if FS is set to 2 for these SMP contacts.

2. **Contact Birth and Death Times.** If DT = -9999, BT is taken to be the ID of a curve (*DEFINE_CURVE) or the ID of a table (*DEFINE_TABLE(_2D)). The curve(s) define multiple birth-times and death-times as ordered (x, y) pairs, that is, each data point in the curve defines a time window during which the contact is active. To satisfy general curve input requirements, the curve(s) must have at least two data points. For example, a curve with two data points $(20, 30)$ and $(50, 70)$ activates the contact when $20 \leq \text{time} \leq 30$ and when $50 \leq \text{time} \leq 70$. To define two separate curves which apply to the dynamic relaxation phase and the subsequent normal phase of the analysis, respectively, BT should point to a table ID. That table should have only two VALUEs, 0 and 1. The curve corresponding to the table VALUE = 0 defines the time windows of active contact for the normal phase of the analysis and curve corresponding to the table VALUE = 1 defines the time windows of active contact for the dynamic relaxation phase of the analysis. An example follows.

```
*CONTACT_TIED_SURFACE_TO_SURFACE
3,3,0,3
$ DT = -9999 has a very special meaning (see User's Manual).
,,,,,, 600, -9999
*DEFINE_TABLE_2D
600
$$ value of "0"  indicates that corresponding curve is for normal phase
$$ value of "1"  indicates that corresponding curve is for DR phase
0,701
1,702
$$ Curves must have at least 2 points to satisfy general input requirements.
*DEFINE_CURVE
$ this curve gives a range of time for which contact is active during the normal phase
701
$ active from t=0 to t=1.1e-3
0,1.1e-3
$ active from t=10 to t=20
10,20
*DEFINE_CURVE
$ this curve gives a range of time for which contact is active during DR
702
$ birth contact at time=2.e-3, death at time = 1.0
2.e-3,1
$ active from t=1 to t=2\
1,2
```

Mandatory Card 3:

Card 3	1	2	3	4	5	6	7	8
Variable	SFSA	SFSB	SAST	SBST	SFSAT	SFSBT	FSF	VSF
Type	F	F	F	F	F	F	F	F
Default	1.	1.	element thickness	element thickness	1.	1.	1.	1.

VARIABLE	DESCRIPTION

SFSA   Scale factor on default SURFA penalty stiffness when SOFT = 0 or SOFT = 2; see also *CONTROL_CONTACT. For MORTAR *frictional* contact this is the stiffness scale factor for the entire contact, and SFSB does not apply.

      LT.0:   -SFSA gives the load curve ID for contact pressure as a function of penetration depth. This is supported for Mortar contact only.

SFSB   Scale factor on default SURFB penalty stiffness when SOFT = 0 or SOFT = 2; see also *CONTROL_CONTACT. For MORTAR *tied* contact, this is an additional stiffness scale factor, resulting in a total stiffness scale of SFSA × SFSB.

SAST   Optional contact thickness for SURFA surface (overrides default contact thickness). This option applies to contact with shell and beam elements. SAST has no bearing on the actual thickness of the elements; it only affects the location of the contact surface. For the *CONTACT_TIED_... options, SAST and SBST (below) can be defined as negative values, which will cause the determination of whether or not a node is tied to depend only on the separation distance relative to the absolute value of these thicknesses (see Remark 4 in General Remarks). More information is given under General Remarks: *CONTACT.

SBST   Optional contact thickness for SURFB surface (overrides default contact thickness). This option applies only to contact with shell elements. For the TIED options, see SAST above.

SFSAT   Scale factor applied to the contact thickness of the SURFA surface. This option applies to contact with shell and beam elements. SF-

VARIABLE	DESCRIPTION
	SAT has no bearing on the actual thickness of the elements; it only affects the location of the contact surface. SFSAT is ignored if SAST is nonzero except in the case of MORTAR contact (see Remark 14 in the General Remarks: *Contact section).
SFSBT	Scale factor applied to the contact thickness of the SURFB surface. This option applies only to contact with shell elements. SFSBT has no bearing on the actual thickness of the elements; it only affects the location of the contact surface. SFSBT is ignored if SBST is non-zero except in the case of MORTAR contact (see Remark 14 in the General Remarks: *Contact section).
FSF	Coulomb friction scale factor. The Coulomb friction value is scaled as $\mu_{sc} = \text{FSF} \times \mu_c$; see Mandatory Card 2.
VSF	Viscous friction scale factor. If this factor is defined, then the limiting force becomes: $F_{\lim} = \text{VSF} \times \text{VC} \times A_{\text{cont}}$; see Mandatory Card 2.

Remarks:

The fields FSF and VSF above can be overridden segment by segment on the *SET_SEGMENT or *SET_SHELL_*OPTION* cards for the **SURFA surface only** as A3 and A4, and for the **SURFB surface only** as A1 and A2. See *SET_SEGMENT and *SET_SHELL_*OPTION*.

Card 4: AUTOMATIC_…_TIEBREAK

This card 4 is mandatory for:

    CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_*{OPTION}

    CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_*{OPTION}

    *CONTACT_AUTOMATIC_SINGLE_SURFACE_TIEBREAK

    *CONTACT_AUTOMATIC_GENERAL_TIEBREAK

If the response parameter, OPTION, below is set to 9 or 11, three damping constants can be defined for the various failure modes. To do this, set the keyword option to

    DAMPING

For OPTION = -11, -9, 2, 4, 6, 7, 8, 9 and 11 of *CONTACT_AUTOMATIC_SURFACE_-TO_SURFACE_TIEBREAK, the Mortar treatment may be activated. This is primarily intended for implicit analysis. The keyword option for this is

    MORTAR

The DAMPING option cannot be combined with the MORTAR option. With the MORTAR option, the adhesive strength can be output to the intfor file by specifying NTIED on *DATABASE_EXTENT_INTFOR, which is essentially the inverse of the damage.

Card 4	1	2	3	4	5	6	7	8
Variable	OPTION	NFLS	SFLS	PARAM	ERATEN	ERATES	CT2CN	CN
Type	I	F	F	F	F	F	F	F
Default	required	required	required	↓	0.0	0.0	1.0	↓

Damping Card. Additional card for the case of OPTION = 9 or 11 with the DAMPING keyword option active.

Card 4.1a	1	2	3	4	5	6	7	8
Variable	DMP_1	DMP_2	DMP_3					
Type	F	F	F					
Default	0.0	0.0	0.0					

OPTION = 13/14 Cards. Two additional cards for the case of OPTION = 13 or 14.

Card 4.1b	1	2	3	4	5	6	7	8
Variable	G1C_0	G1C_INF	EDOT_G1	T0	T1	EDOT_T	FG1	LCG1C
Type	F	F	F	F	F	F	F	F

OPTION = 13/14 Cards. Two additional cards for the case of OPTION = 13 or 14.

Card 4.2b	1	2	3	4	5	6	7	8
Variable	G2C_0	G2C_INF	EDOT_G2	S0	S1	EDOT_S	FG2	LCG2C
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
OPTION	Response:

  EQ.-11: See 11. NFLS/SFLS/ERATEN/ERATES are functions of temperature. MORTAR option only.

  EQ.-9: See 9. NFLS/SFLS/ERATEN/ERATES are functions of temperature. MORTAR option only.

  EQ.-3: See 3. Moments are transferred. We recommend using contact type AUTOMATIC_ONE_WAY_SURFACE_- TO_SURFACE_TIEBREAK with this option.

  EQ.-2: See 2. Moments are transferred. We recommend using contact type AUTOMATIC_ONE_WAY_SURFACE_-

VARIABLE	DESCRIPTION
	TO_SURFACE_TIEBREAK with this option.
	EQ.-1: See 1. Moments are transferred. We recommend using contact type AUTOMATIC_ONE_WAY_SURFACE_-TO_SURFACE_TIEBREAK with this option.
	EQ.1: Tracked nodes in contact and those that come into contact will permanently stick. Tangential motion is inhibited.
	EQ.2: Tiebreak is active for nodes that are initially in contact. Until failure, tangential motion is inhibited. If PARAM is set to unity (1.0), shell thickness offsets are ignored, and the orientation of the shell surfaces is required such that the outward normals point to the opposing contact surface.
	EQ.3: Same as 1 above but with failure after sticking.
	EQ.4: Tiebreak is active for nodes which are initially in contact but tangential motion with frictional sliding is permitted.
	EQ.5: Tiebreak is active for nodes which are initially in contact. Stress is limited by the yield condition described in Remark 3 below. Damage behavior is modeled by a curve which defines normal stress as a function of gap (crack opening). This option can be used to represent deformable glue bonds.
	EQ.6: This option is for use with solids and thick shells only. Tiebreak is active for nodes which are initially in contact. Failure stress must be defined for tiebreak to occur. After the failure stress tiebreak criterion is met, damage is a linear function of the distance between points initially in contact. When the distance is equal to PARAM, damage is fully developed, and interface failure occurs. After failure, this option behaves as a surface-to-surface contact.
	EQ.7: Dycoss Discrete Crack Model. We recommend using contact type AUTOMATIC_ONE_WAY_SURFACE_-TO_SURFACE_TIEBREAK with this option. See Remark 3.
	EQ.8: Similar to OPTION = 6, but it works with offset shell elements. We recommend using contact type AUTO-MATIC_ONE_WAY_SURFACE_TO_SURFACE_-

VARIABLE	DESCRIPTION
	TIEBREAK with this option.
	EQ.9: Discrete Crack Model with power law and B-K damage models. We recommend using contact type AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK with this option. See Remark 3.
	EQ.10: Similar to OPTION = 7, but it works with offset shell elements. We recommend using contact type AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_-TIEBREAK with this option.
	EQ.11: Similar to OPTION = 9, but it works with offset shell elements. We recommend using contact type AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_-TIEBREAK with this option.
	EQ.13: Elastoplastic, rate-dependent damage model based on *MAT_240. We recommend using contact type AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK with this option. See Remark 7.
	EQ.14: Similar to OPTION = 13, but it works with offset shell elements. We recommend using contact type AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK with this option.
NFLS	Normal failure stress for OPTION = 2, 3, 4, 6, 7, 8, ±9, 10 or ±11. For OPTION = 5 NFLS becomes the plastic yield stress as defined in Remark 5. For OPTION = 9 or 11 and NFLS < 0, a load curve with ID |NFLS| is referenced defining normal failure stress as a function of element size. See Remark 3. For OPTION = -9 or -11 and NFLS < 0, |NFLS| is the ID of a load curve giving normal failure stress as a function of temperature; it applies to the Mortar option only.
SFLS	Shear failure stress for OPTION = 2, 3, 6, 7, 8, ±9, 10 or ±11. For OPTION = 4, SFLS is a frictional stress limit if PARAM = 1. This frictional stress limit is independent of the normal force at the tie. For OPTION = 5 SFLS becomes the curve ID which defines normal stress as a function of gap. For OPTION = 9 or 11 and SFLS < 0, |SFLS| references a load curve ID, defining shear failure stress as a function of element size. See Remark 3. For OPTION = -9 or -11 and SFLS < 0, |SFLS| is the ID of a load curve giving shear failure stress as a function of temperature; it applies to the Mortar option only.

VARIABLE	DESCRIPTION
PARAM	For OPTION = 2, setting PARAM = 1 causes the shell thickness offsets to be ignored. For OPTION = 4, setting PARAM =1 causes SFLS to be a frictional stress limit. For OPTION = 6 or 8, PARAM is the critical distance, CCRIT, at which the interface failure is complete. For OPTION = 7 or 10 PARAM is the friction angle in degrees. For OPTION = 9 or 11, it is the exponent in the damage model. A positive value invokes the power law, while a negative one, the B-K model. See *MAT_138 for additional details. For OPTION = 13 or 14, it is the thickness of the tiebreak layer; a value greater than zero is recommended. Default value is 1.0 for OPTIONs 9 and 11, but otherwise default value is 0.0.
ERATEN	For OPTION = 7, ±9, 10, ±11 only. Normal energy release rate (stress × length) used in damage calculation; see Lemmen and Meijer [2001]. For OPTION = -9 or -11, this is the ID of a load curve giving normal energy release rate as function of temperature; it applies to the Mortar option only.
ERATES	For OPTION = 7, ±9, 10, ±11 only. Shear energy release rate (stress × length) used in damage calculation; see Lemmen and Meijer [2001]. For OPTION = -9 or -11, this is the ID of a load curve giving shear energy release rate as function of temperature; it applies to the Mortar option only.
CT2CN	The ratio of the tangential stiffness to the normal stiffness for OPTION = 9, 11, 13, and 14. The default is 1.0.
CN	Normal stiffness (stress/length) for OPTION = 9, 11, 13, and 14 and for OPTION = 2, 4, 6, 7, and 8 for the MORTAR option only. If CN is not given explicitly, penalty stiffness divided by segment area is used (default). This optional stiffness should be used with care since contact stability can get affected. A warning message with a recommended time step is given initially.
DMP_1	Mode I damping force per unit velocity per unit area
DMP_2	Mode II damping force per unit velocity per unit area
DMP_3	Mode III damping force per unit velocity per unit area
G1C_0, …	All variables on Cards 4.1b and 4.2b are the same as in *MAT_240.

Remarks:

1. **Contact behavior after failure.** After failure, this contact option behaves as a surface-to-surface contact with thickness offsets. After failure, no interface tension is possible.

2. **Restrictions.** Segment-to-segment contact (SOFT = 2) is *not* implemented for the tiebreak option. *CONTACT_AUTOMATIC_SINGLE_SURFACE_TIE-BREAK and *CONTACT_AUTOMATIC_GENERAL_TIEBREAK are supported only for OPTIONs 1 through 5 and only for MPP.

3. **Tiebreak, failure, damage, and crack initiation.** The following overviews the failure criteria for the various OPTIONs.

    a) For OPTION = 2, 3, and 6, the tiebreak failure criterion has normal and shear components:

    $$\left(\frac{|\sigma_n|}{\text{NFLS}}\right)^2 + \left(\frac{|\sigma_s|}{\text{SFLS}}\right)^2 \geq 1 \ .$$

    σ_n is the tensile normal stress and is taken as zero if the normal stress is compressive. σ_s is the shear stress.

    b) For OPTION = 4, the tiebreak failure criterion has only a tensile normal stress component:

    $$\frac{|\sigma_n|}{\text{NFLS}} \geq 1 \ .$$

    c) For OPTION = 5, the stress is limited by a perfectly plastic yield condition. For ties in tension, the yield condition is

    $$\frac{\sqrt{\sigma_n^2 + 3|\sigma_s|^2}}{\text{NLFS}} \leq 1 \ .$$

    For ties in compression, the yield condition is

    $$\frac{\sqrt{3|\sigma_s|^2}}{\text{NLFS}} \leq 1 \ .$$

    The stress is also scaled by the damage function which is obtained from the load curve. For ties in tension, both normal and shear stress are scaled. For ties in compression, only shear stress is scaled.

    d) For OPTION = 6 or 8, damage initiates when the stress meets the failure criterion. The stress is then scaled by the damage function. Assuming no load reversals, the energy released due to the failure of the interface is approximately $0.5 \times S \times \text{CCRIT}$, where

$$S = \sqrt{\max(\sigma_n, 0)^2 + |\sigma_s|^2}$$

at the initiation of damage. This interface may be used for simulating crack propagation. For the energy release to be correct, the contact penalty stiffness must be much larger than

$$\frac{\min(\text{NFLF}, \text{SFLS})}{\text{CCRIT}} .$$

e) OPTION = 7 and 10 are implementations of the Dycoss Discrete Crack Model as described in Lemmen and Meijer [2001]. The relation for the crack initiation is given as

$$\left[\frac{\max(\sigma_n, 0)}{\text{NFLS}}\right]^2 + \left[\frac{\sigma_s}{\text{SFLS} - \sin(\text{PARAM})\min(0, \sigma_n)}\right]^2 = 1 .$$

f) OPTION = 9 and 11 are based on the fracture model in the cohesive material model *MAT_COHESIVE_MIXED_MODE, where the model is described in detail. Failure stresses/peak tractions NFLS and/or SFLS can be defined as functions of characteristic element length (square root of the reference segment area) using load curves. With these options, nearly the same global responses (for instance, load-displacement curve) can be obtained with coarse meshes compared to fine meshes. In general, lower peak tractions are needed for coarser meshes. See also *MAT_138.

4. **Determining state of tiebreak surface.** For OPTIONs 6 thru 11 of *CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK, the component labeled "contact gap" in the intfor database (*DATABASE_BINARY_INTFOR) indicates the condition of the tiebreak surface. The "contact gap" represents a damage value ranging from 0 (tied, no damage) to 1 (released, full damage).

5. **Automatic tiebreak tying tolerance.** Tying in the AUTOMATIC_..._TIEBREAK contacts occurs if the tracked node is within a small tolerance of the reference surface *after* taking into account contact thicknesses. For MPP, the tolerance is given by

$$\text{tol} = 0.01\sqrt{2 \times \text{reference segment area}} .$$

For SMP, the tolerance is

$$0.4(\text{tracked contact thickness} + \text{reference contact thickness}) .$$

6. **Defining SURFB and SURFA sets.** We recommend defining the SURFA and SURFB sides of tiebreak contact using segment sets rather than part or part set IDs. Doing this enables being more selective when choosing which segments to tie and ensures that contact stresses calculated from nodal contact forces are not diluted by segments not actually on the actual contact surface. It also gives more

direct control over the contact segment normal vectors when segment sets are used.

Segment normal vectors should point toward the opposing contact surface to distinguish tension from compression properly. Also, unlike regular automatic contacts, segment normal vectors need to be properly oriented in MPP because they are not automatically oriented. This issue can lead to incorrect results upon failure.

7. **Variables applicable to OPTIONs 13 and 14.** OPTIONs 13 and 14 are based on material model 240; see LS-DYNA Keyword User's Manual, Volume II. Applicable variables are PARAM, CT2CN, CN, and the variables on optional Cards 4.1b and 4.2b NFLS, SFLS, ERATEN, and ERATES are not used.

Card 4: AUTOMATIC_..._SURFACE_TO_SURFACE_TIEBREAK_USER_...

These cards, 4.1, 4.2, and 4.3, are mandatory for:

> *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_USER

> *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK_USER_MORTAR

> *CONTACT_AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK_USER

Card 4.1	1	2	3	4	5	6	7	8
Variable	OPTION	NHV	CT2CN	CN	OFFSET	NHMAT	NHWLD	
Type	I	I	F	F	I	I	I	
Default	required	0	1.0	0.0	0	0	0	

Card 4.2	1	2	3	4	5	6	7	8
Variable	UP1	UP2	UP3	UP4	UP5	UP6	UP7	UP8
Type	F	F	F	F	F	F	F	F

Card 4.3	1	2	3	4	5	6	7	8
Variable	UP9	UP10	UP11	UP12	UP13	UP14	UP15	UP16
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
OPTION	User tiebreak type (101 - 105 inclusive). A number between 101 and 105 must be chosen. Corresponding subroutine utb<OP-TION> in dyn21cnt.f will be called for the non-Mortar option while subroutine mortar_usrtbrk will be called for the Mortar option.
NHV	Number of history variables (maximum of 3 for non-Mortar, arbitrary for Mortar)

VARIABLE	DESCRIPTION
CT2CN	Ratio of the tangential stiffness to the normal stiffness
CN	Normal stiffness (stress/length). If CN is not given explicitly, penalty stiffness divided by segment area is used (default). This optional stiffness should be used with care since contact stability can be affected. A warning message with a recommended time step is given initially.
OFFSET	Flag for offset treatment (only for non-Mortar option): EQ.0: No offset EQ.1: With offset for shell elements
NHMAT	Number of material history variables to be read in the user tiebreak routine (Mortar option only)
NHWLD	Number of tied weld history variables to be read in the user tiebreak routine, assuming they have been carried over from a previous simulation (Mortar option only). See Remark 4.
UP1…UP16	User parameters

Remarks:

1. **User Defined Tiebreak Model.** This option allows the implementation of a user defined tiebreak model. Please check the comments in subroutine utb101 or mortar_usrtbrk (dyn21cnt.f) for more information.

2. **Contact Behavior after Failure.** After failure, this contact option behaves as a surface-to-surface contact with thickness offsets. After failure, no interface tension is possible.

3. **Preferred Contact Type.** We recommend using the …_ONE_WAY_... definition for this option.

4. **Mortar Option.** Essentially the same behavior can be expected for the Mortar contact as for the non-Mortar contact. The Mortar capability, however, is accompanied with some extra features. For instance, this Mortar contact may be used to assess the adhesive properties of a preceding lamination process by reading history variables from a previous simulation that used a corresponding user Mortar tied weld model (see Card 4: AUTOMATIC_SURFACE_TO_SURFACE_..._TIED_WELD). To do this, the ID of the present tiebreak contact should be the same as the ID of the earlier tied weld contact leading up to the start of the current delamination simulation, and NHWLD should state how many

history variables to read. Furthermore, *INTERFACE_SPRINGBACK_LSDYNA with FTYPE = 3 and CFLAG = 1 should be used to create a dynain.lsda in the previous simulation which will be read with *INCLUDE in the current simulation, all in accordance with standard continuation procedures.

Card 4: AUTOMATIC_SURFACE_TO_SURFACE_COMPOSITE/LUBRICATION

This contact model is designed for simulating the processing of laminated composite materials or certain types of lubrication. Surfaces in contact may support shear up to the limit defined by MODEL and be in compression or in tension up to the tensile limit σ_f defined by TFAIL. After TFAIL is reached, the contact fails in both tension and shear. If the surfaces come back into contact, the bonding heals, and the contacting surfaces may support shear and tension.

This Card 4 is mandatory for:

> *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_COMPOSITE
>
> *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_LUBRICATION

Card 4	1	2	3	4	5	6	7	8
Variable	TFAIL	MODEL	CIDMU	CIDETA	D			
Type	F	I	I	I	F			
Default	required	required	required	required	0.0			

VARIABLE	DESCRIPTION
TFAIL	Tensile traction, σ_f, required for failure
MODEL	Model for shear response (see Remark 2):

> EQ.1: Limiting shear stress depends on CIDMU in both tension and compression.
>
> EQ.2: Limiting shear stress depends on CIDETA in tension and CIDMU in compression.
>
> EQ.3: Limiting shear stress depends on CIDETA in both tension and compression.

CIDMU	Curve ID for the coefficient of friction, $\mu(H)$, as a function of the Hershey number, H
CIDETA	Curve ID for the viscosity, $\eta(T)$, as a function of temperature, T (see Remark 1)
D	Composite film thickness

Remarks:

1. **Viscosity.** The viscosity, $\eta(T)$, is defined as a function of temperature by CIDE-TA. The value of the viscosity is not extrapolated if the temperature falls outside of the temperature range defined by the curve.

2. **Shear Stress Limits.** The following lists the shear stress limits for the different values of MODEL:

    a) *MODEL = 1.* The coefficient of friction, μ, for MODEL = 1 is defined in terms of the Hershey number, $H = \eta(T)V/(p + \sigma_f)$, where p is the contact pressure (positive in compression, and negative in tension) and V the relative velocity between the surfaces. The shear stress in tension and compression is limited according to

    $$\tau \le \mu(H)(p + \sigma_f) \ .$$

    b) *MODEL = 2.* The coefficient of friction, μ, for MODEL = 2 is defined in terms of the Hershey number, $H = \eta(T)V/p$. Note the definition of the Hershey number for this model differs from MODEL = 1. In compression the shear stress is limited by

    $$\tau \le \mu(H)p \ .$$

    and in tension, the shear stress is limited according to

    $$\tau \le \frac{\eta(T)V}{d} \ .$$

    c) *MODEL = 3.* The shear stress for MODEL = 3 in tension and compression is limited according to

    $$\tau \le \frac{\eta(T)V}{d} \ .$$

Card 4: AUTOMATIC_SINGLE_SURFACE_TIED

This special feature was originally implemented to allow for the calculation of eigenvalues and eigenvectors on geometries that are connected by a contact interface using the AUTOMATIC_SINGLE_SURFACE options. It also can be used in a more general sense to tie a set of parts based only on their proximity to each other in the initial geometry. This contact type does not revert to non-tied AUTOMATIC_SINGLE_SURFACE behavior at any time during the simulation.

This Card 4 is mandatory for:

*CONTACT_AUTOMATIC_SINGLE_SURFACE_TIED

Card 4	1	2	3	4	5	6	7	8
Variable	CLOSE							
Type	F							
Default	Rem 1							

VARIABLE	DESCRIPTION
CLOSE	Tolerance for tying surfaces using a penalty formulation. Surfaces closer than CLOSE are tied. Surfaces farther apart than CLOSE are not tied.

Remarks:

1. **CLOSE Default.** If a value of CLOSE is not input, the default value of CLOSE is calculated for each tracked node and is equal to 0.0106 times the square root of the area of the nearest reference segment.

2. **Rigid Body Modes.** If there is significant separation between the tied surfaces, the rigid body modes will be opposed by the contact stiffness, and the calculated eigenvalues for rigid body rotations will not be zero.

3. **Recommended Settings.** When using this contact type, we recommend setting TIEDID on Optional Card D to 1.

Card 4: AUTOMATIC_SURFACE_TO_SURFACE_..._TIED_WELD

This special feature is implemented to allow for the simulation of welding. As regions of the surfaces are heated to the welding temperature and come into contact, the nodes are tied.

This Card 4 is mandatory for:

        *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_TIED_WELD

        *CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR_TIED_WELD

Card 4	1	2	3	4	5	6	7	8
Variable	TEMP	CLOSE	HCLOSE	NTPRM	NMHIS	NSTWH	NMTWH	TIME
Type	F	F	F	F	F	F	F	F
Default	none	0.0	0.0	0.0	0.0	0.0	0.0	0.0

VARIABLE	DESCRIPTION

TEMP   For the non-Mortar option, this is the minimum temperature required on both surfaces for tying. For the Mortar tied weld, surfaces tie when one of the surfaces reaches the specified temperature. Once the surfaces are tied, they remain tied even if the temperature drops.

        LT.0: |TEMP| represents the user tied weld ID passed to subroutine `mortar_usrtie` for defining an arbitrary condition to tie. This option is only available for MORTAR_TIED_-WELD and TEMP must be between -999 and -1.

CLOSE   Surfaces closer than CLOSE are tied. If CLOSE is left as 0.0, it defaults to one percent of the mesh characteristic length scale. Nodes that are above or below the surface will be tied if they are close enough to the surface.

HCLOSE   Thermal contact conductivity for a tied interface, in case of using the option THERMAL in the contact

NTPRM   If TEMP < 0, number of user tied weld parameters; otherwise ignored. NTPRM allocates the array `cprm(*)` in subroutine `mortar_usrtie`.

VARIABLE	DESCRIPTION
NMHIS	If TEMP < 0, number of material history variables accessible from the user tied weld interface; otherwise ignored. NMHIS allocates the arrays `shis(*)` and `mhis(*)` in subroutine `mortar_usrtie`.
NSTWH	If TEMP < 0, number of SURFA tied weld history variables for the user tied weld interface. NSTWH allocates the array `shst(*)` in subroutine `mortar_usrtie`.
NMTWH	If TEMP < 0, number of SURFB tied weld history variables for the user tied weld interface. NMTWH allocates the array `mhst(*)` in subroutine `mortar_usrtie`.
TIME	This parameter applies to the Mortar tied weld contact only. The conditions above for tying/welding must be satisfied for at least TIME consecutive time units for tying/welding to occur. This option is intended to prevent premature tying/welding which may otherwise occur if only temperature is considered.

User Tied Welds Parameters Card. Insert as many cards needed to define NTPRM tied weld parameters.

Card 4.1	1	2	3	4	5	6	7	8
Variable	TPRM1	TPRM2	TPRM3	TPRM4	TPRM5	TPRM6	TPRM7	TPRM8
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
TPRM*i*	User tied welds parameter *i*

Remarks:

1. **Separation and Rigid Body Modes.** If there is significant separation between the tied surfaces, the rigid body modes will not be opposed by the contact stiffness. In other words, the offset between the surfaces is handled like the contact with OFFSET.

2. **Contact Behavior Below Welding Temperature.** If the surfaces are below the welding temperature, the surfaces interact with the standard AUTOMATIC_-SURFACE_TO_SURFACE options.

3. **MORTAR Contact.** The MORTAR option is primarily intended for implicit and is supported for SMP and MPP. With this option, the tied segments can be viewed in the intfor file by specifying NTIED on *DATABASE_EXTENT_INTFOR. It is also possible to implement a user defined condition for tying segments, activated by TEMP < 0. See subroutine mortar_usrtie for comments and a sample code which is available in the source code of an object version of LS-DYNA. The history variables can subsequently be used in a Mortar tiebreak routine for assessing the quality of the lamination (see Remark 4 in the Card 4: AUTOMATIC_..._SURFACE_TO_SURFACE_TIEBREAK_USER_... section)._.

Card 4: CONSTRAINT_..._TO_SURFACE

This card 4 is mandatory for:

  *CONTACT_CONSTRAINT_NODES_TO_SURFACE

  *CONTACT_CONSTRAINT_SURFACE_TO_SURFACE

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

VARIABLE	DESCRIPTION

KPF   Kinematic partition factor for constraint:

  EQ.0.0: Fully symmetric treatment

  EQ.1.0: One-way treatment with SURFA nodes constrained to SURFB surface. Only the SURFA nodes are checked against contact.

  EQ.-1.0: One-way treatment with SURFB nodes constrained to SURFA surface. Only the SURFB nodes are checked against contact.

Card 4: DRAWBEAD

This card 4 is mandatory for:

       *CONTACT_DRAWBEAD

       *CONTACT_DRAWBEAD_BENDING

       *CONTACT_DRAWBEAD_INITIALIZE

Note that variables related to the automatic multiple draw bead feature, meaning NBEAD, POINT1, POINT2, WIDTH, and EFFHGT, only work with draw beads that are defined with node sets, not with beam elements. Also, NBEAD and the related variables on Card 4.4 can be used together with the option INITIALIZE or with the option BEND-ING.

The BENDING keyword option is for weakening the blank underneath the draw bead. See Weakening Effect below.

Card 4.1	1	2	3	4	5	6	7	8
Variable	LCIDRF	LCIDNF	DBDTH	DFSCL	NUMINT	DBPID	ELOFF	NBEAD
Type	I	I	F	F	I	I	I	I
Default	required	none	0.0	1.0	0	0	0	optional

Bending Card. Additional card for BENDING keyword option. This card is for modeling the draw bead weakening effect.

Card 4.2	1	2	3	4	5	6	7	8
Variable	EPM	EPSCALE	ENDING					
Type	F	F	F					
Default	none	none	10.					
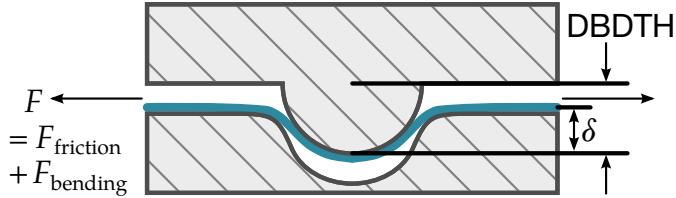
Figure 11-5. The draw bead contact model.

Initialization Card. Additional card for INITIALIZE keyword option. This card initializes the plastic strain and thickness of elements that pass under the draw bead.

Card 4.3	1	2	3	4	5	6	7	8
Variable	LCEPS	TSCALE	LCEPS2	OFFSET				
Type	I	F	I	F				
Default	required	1.0	optional	optional				

Additional card to be included if NBEAD in Card 4.1 is defined.

Card 4.4	1	2	3	4	5	6	7	8
Variable	POINT1	POINT2	WIDTH	EFFHGT				
Type	I	I	F	F				
Default	none	none	none	none				

VARIABLE	DESCRIPTION
LCIDRF	GT.0: Load curve ID giving the bending component of the re-straining force, F_{bending}, per unit draw bead length as a function of displacement, δ; see Figure 11-5. This force is due to the bending and unbending of the blank as it moves through the draw bead. The total restraining force is the sum of the bending and friction components.
	LT.0: |LCIDRF| gives the load curve ID defining maximum bead force as a function of normalized draw bead length. The abscissa values are between zero and one and are the nor-malized draw bead length. The ordinate gives the maxi-mum allowed draw bead retaining force when the bead is

VARIABLE	DESCRIPTION
	in the fully closed position. If the draw bead is not fully closed, linear interpolation is used to compute the draw bead force.
LCIDNF	Load curve ID giving the normal force per unit draw bead length as a function of displacement, δ; see Figure 11-5. This force originates from bending the blank into the draw bead as the binder closes on the die. The normal force begins to develop when the distance between the die and binder is less than the draw bead depth. As the binder and die close on the blank, this force should diminish or reach a plateau.
DBDTH	Draw bead depth; see Figure 11-5. This depth is needed to determine the correct displacement, δ, from contact displacements.
DFSCL	Scale factor for load curve (default = 1.0). This factor scales the load curve ID, LCIDRF, above.
NUMINT	Number of equally spaced integration points along the draw bead: EQ.0: Internally calculated based on element size of elements that interact with draw bead. This is necessary for the correct calculation of the restraining forces. More integration points may increase the accuracy since the force is applied more evenly along the bead.
DBPID	Optional part ID for the automatically generated truss elements used to display the draw bead during post-processing. If undefined, LS-DYNA assigns a unique part ID.
ELOFF	Option to specify an element ID offset for the truss elements that are automatically generated for displaying the draw bead during post-processing. If undefined, LS-DYNA chooses a unique offset.
NBEAD	Number of line beads. It must be an odd integer.
EPM	Maximum strain the blank will experience when it passes the bead
EPSCALE	Scale factor to weaken the stress-strain curve
ENDING	Parameter to define the length of the bead ends where the weakening effect will phase out

VARIABLE	DESCRIPTION
LCEPS	Load curve ID defining the plastic strain as a function of the parametric coordinate through the shell thickness. The parametric coordinate must be defined in the interval between -1 and 1 inclusive. The value of plastic strain at the integration point is interpolated from this load curve. If the plastic strain at an integration point exceeds the value of the load curve at the time initialization occurs, the plastic strain at the point will remain unchanged.
TSCALE	Scale factor that multiplies the shell thickness as the shell element moves under the draw bead
LCEPS2	Optional load curve ID defining the plastic strain as a function of the parametric coordinate through the shell thickness, which is used after an element has traveled a distance equal to OFFSET. The parametric coordinate should be defined in the interval between -1 and 1 inclusive. The value of plastic strain at the integration point is interpolated from this load curve. If the plastic strain at an integration point exceeds the value of the load curve at the time initialization occurs, the plastic strain at the point will remain unchanged. Input parameters LCEPS2 and OFFSET provides a way to model the case where a material moves under two draw beads. In this latter case the curve would be the sum of the plastic strains generate by moving under two consecutive beads.
OFFSET	If the center of an element has moved a distance equal to OFFSET, the load curve ID, LCEPS2, is used to reinitialize the plastic strain. The TSCALE scale factor is also applied.
POINT1	Node ID of the first node on a binder
POINT2	Node ID of a *matching* node on the opposing binder
WIDTH	Total bead width defining distance between the innermost and outermost bead walls
EFFHGT	Effective bead height. Draw bead restraining force starts to take effect when the binder gap is less than EFFHGT.

Overview:

For this draw bead model, the blank is the SURFB part, and the male part of the draw bead is SURFA. The male part of the draw bead, which moves with the punch, is input as a curve defined using a list of nodes or a part consisting of beams, as discussed below.

Associated with this curve is a *region of influence* that is characterized by the DBDTH field of Card 4.1.

As the punch comes down and the region of influence intersects the elements on the blank, forces are applied to the blank at the points of closest approach. These forces depend on the penetration distance, δ, which is geometrically defined in Figure 11-5. The draw bead force model consists of two terms:

1. a *resisting force* which is a function of δ, and is defined through the load curve specified in the LCIDRF field. This force is applied in a direction opposite to velocity.

2. a *normal force*, which pushes the male part of the draw bead away from the blank as specified by LCIDNF. This normal force, in turn, is used to model friction, which depends on the product of the friction coefficient and the normal force.

The curve representing the male part of the draw bead can be defined in three ways:

1. a *consecutive* list of *SURFA* nodes that lie along the bead

2. a part ID of a beam that lies along the draw bead

3. a part set ID of beams that lie along the draw bead

For straight draw beads, only two nodes or a single beam need to be defined, that is, one at each end. For curved beads, many nodes or beams may be required to define the curvature of the bead geometry.

When beams are used to define the bead, each node, except for the first and last, must connect with two beam elements. Therefore, the number of SURFA nodes must equal the number of beam elements plus one.

The contact algorithm checks for penetration at the integration points. Integration points are equally spaced along the draw bead and do not depend on the nodal spacing used in the definition of the draw bead. By tying extra nodes to rigid bodies (see *CONSTRAINED_EXTRA_NODES or *CONSTRAINED_RIGID_BODIES), the draw bead nodal points do not need to belong to the element connectivities of the die and binder. The blank makes up the SURFB surface.

> **NOTE:** We recommended defining a box (see *DEFINE_BOX) around the draw bead to limit the size of the SURFB surface considered for the draw bead. This will substantially reduce the cost and memory requirements.

Multiple draw beads model:

Developed in conjunction with the *Ford Motor Company Research & Advanced Engineering Laboratory*, the multiple draw bead feature provides a simple way (1) to model the neglected effects of the draw bead width and (2) to attenuate the bead forces when the distance between the upper and lower binders is more than the draw bead height.

1. **Draw Bead Width Correction.** As shown in Figure 11-6, a sheet blank edge often does not cross the draw bead's curve of definition but *does* fall within its width. When the bead is modelled as a 1-dimensional (no width) curve, no forces may be applied to a major portion of the blank. The neglect of width effects leads to excessive blank edge draw-ins resulting in either loose metal in the part or wrinkles on the draw wall or product surface.

    The multiple beads feature improves this shortcoming by replacing the single 1-dimensional bead with an equivalent set of beads distributed over the width of the physical bead. The bead force is distributed uniformly over the NBEAD sub-beads, such that the resultant force is equal to that of the original 1-dimensional bead. Note that NBEAD *must* be an odd integer.

    Figure 11-7 schematically represents the NBEAD = 3 case for which two additional line beads are automatically generated. The forces specified by the load curve, LCIDRF, is evenly distributed over the 3 beads. In Figure 11-8, bead forces are recovered from the ASCII rcforc files for both cases of NBEAD = 1 and 3, indicating the total force applied (shown on the left) on one single bead is distributed evenly among the three automatically generated beads for the case of NBEAD = 3.

    The stress distribution is also more realistic with multiple beads. In a channel draw (half model) as shown in Figure 11-12, no significant changes in mean stress values are found between NBEAD = 3 and one single line bead. In fact, the compressive stresses are more realistically and evenly distributed around the bead region, with stresses in NBEAD = 3 about 1/3 of those in a single line bead.

2. **Lower Binder Gap Correction.** As originally implemented, the draw bead contact model applies the draw bead forces, as specified in the load curve, when the upper binder reaches the blank, regardless of the lower binder's position. If the lower binder is not in contact with the blank, LS-DYNA still applies draw bead forces, even though it is unphysical to do so. The EFFHGT, POINT1 and POINT2 fields together provide a simple model to avoid these unphysical forces. The POINT1 and POINT2 fields are taken as nodes on the opposing binders. The draw bead contact is disabled when the Euclidean distance between POINT1 and POINT2 is greater than EFFHGT; consequently, the two nodes *must* be chosen so they converge to a single point as the draw bead closes.

As shown in Figure 11-9, a simple model was built to verify the effectiveness of the variable EFFHGT. The upper binder is pushed down to close with the lower binder while a strip of sheet blank is being pulled in the direction indicated. The distance between the binders is 12 mm initially, as shown in Figure 11-10. The closing gap and pulling force in the *x*-direction were recovered throughout the simulation. With EFFHGT set to 8 mm, the pulling force history indicates the bead forces starting to take effect after the upper binder has traveled for 4 mm as expected (see Figure 11-11).

Weakening Effect:

Without BENDING, the contact algorithm ignores the bending/unbending effect and the huge normal stress when the blank goes through the draw beads. As a result, material stretching during forming simulations cannot be predicted. The algorithm invoked with BENDING weakens the blank as it travels through the bead and stretches it when it passes through the bead.

Example:

The following partial keyword example shows how to use NBEAD with the option INITIALIZE. A total of 3 beads will be generated, including the original bead. The total bead width between the inner and outer most beads is 10.0 mm (WIDTH). The effective bead height EFFHGT is 1.11 mm, which is the gap between upper die and lower binder upon closing. Note the feature NBEAD only works with draw beads that are defined using a node set (#1001, which is used as SURFB in *CONTACT…), and the node set is constrained together with the lower binder (PID 8).

```
*CONTACT_DRAWBEAD_INITIALIZE_ID
$#     cid                                                          title
      1001        DRAW DBEAD #1
$#   surfa       surfb  surfatyp  surfbtyp    saboxid   sbboxid       sapr      sbpr
$     Draw bead node set ID is 1001, blank PID is 11.
      1001          11         4         3          0      1001          0         0
$#      fs          fd        dc        vc        vdc     penchk         bt        dt
     0.000       0.000     0.000     0.000 20.000000          0        0.0 1.000E+20
$#    sfsa        sfsb      sast      sbst      sfsat      sfsbt        fsf       vsf
   0.200000    0.200000     0.000     0.000   1.000000   1.000000   1.000000   1.000000
$#  lcidrf      lcidnf     dbdth     dfscl     numint      dbpid      eloff     NBEAD
     90905       90906    &dbdth  0.500000          0          0          0         3
$    LCEPS      TSCALE    LCEPS2    OFFSET
       212        0.95
$   POINT1      POINT2     WIDTH    EFFHGT
     49668       50595      10.0      1.11
*SET_NODE_LIST
$#     sid         da1       da2       da3       da4
      1001
$#    nid1        nid2      nid3      nid4      nid5       nid6       nid7      nid8
     50820       50821     50822     50823     50824      50825      50826     50827
*CONSTRAINED_EXTRA_NODES_SET
$      PID        NSID
         8        1001
*DEFINE_CURVE
```

```
212
-1.0,0.05
1.0,0.05
```

Inner bead wall

Blank edge flow direction

Blank drawn edge

Outer bead wall

Location of a single line bead force
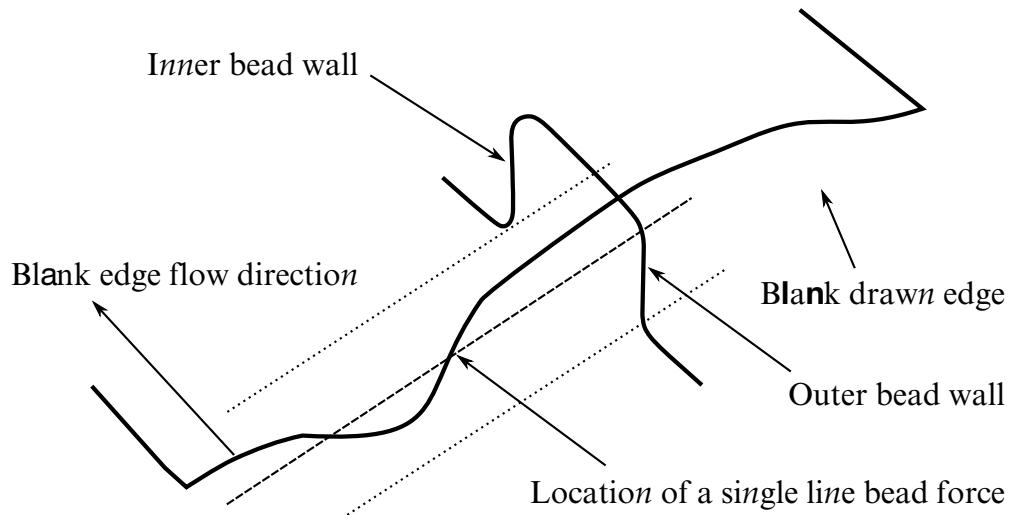
Figure 11-6. A possible scenario of sheet blank edge draw-in condition

WIDTH

Auto-created beads

Defined node set/beam

5.25

5.25
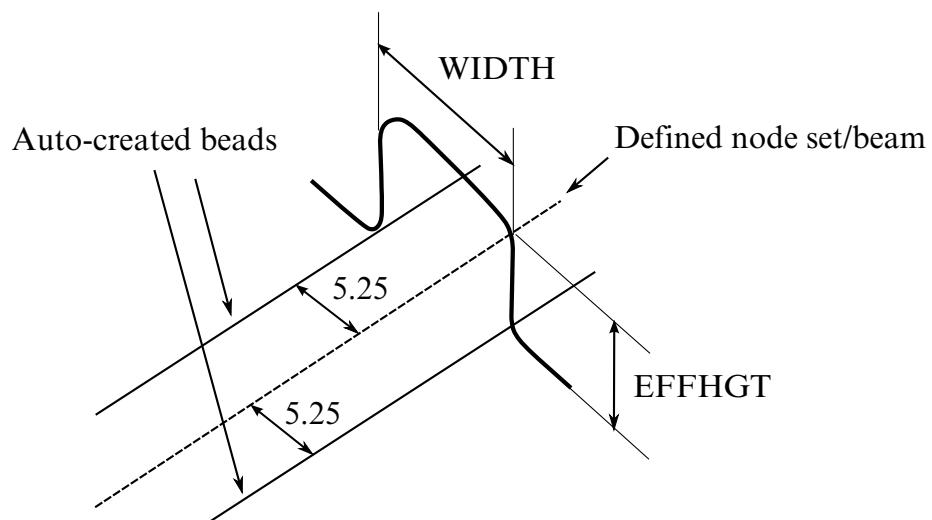
EFFHGT

Figure 11-7. Definition of multiple draw beads
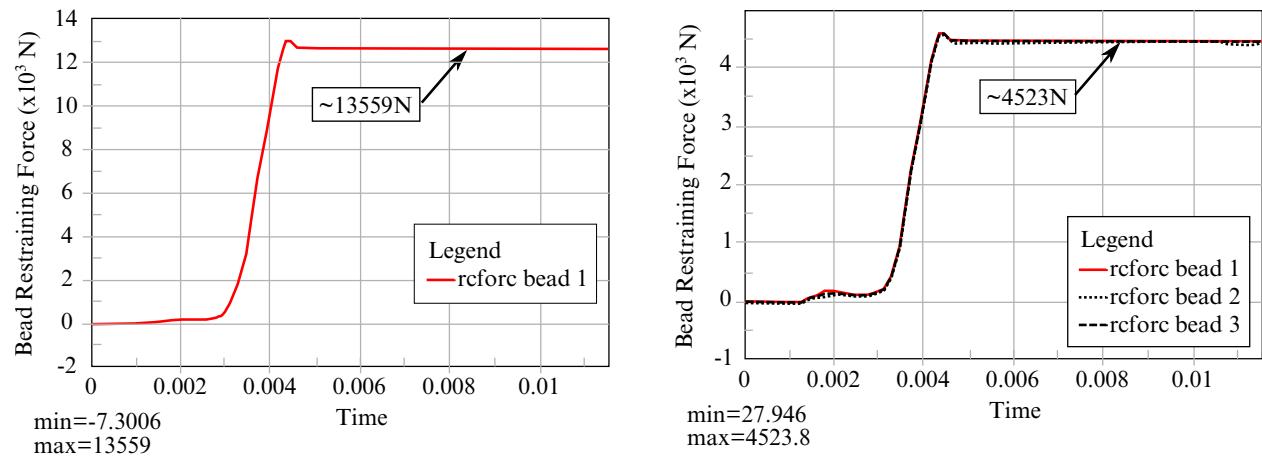
Figure 11-8. Bead force verification between NBEAD = 1 (left) and 3
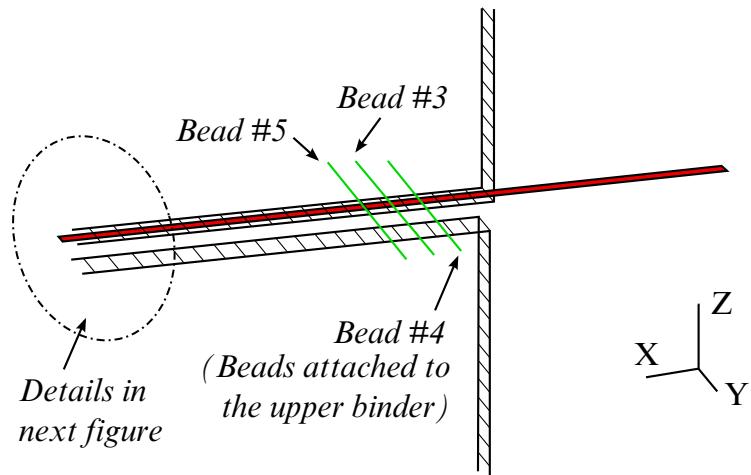
Figure 11-9. A verification model for the variable EFFHGT
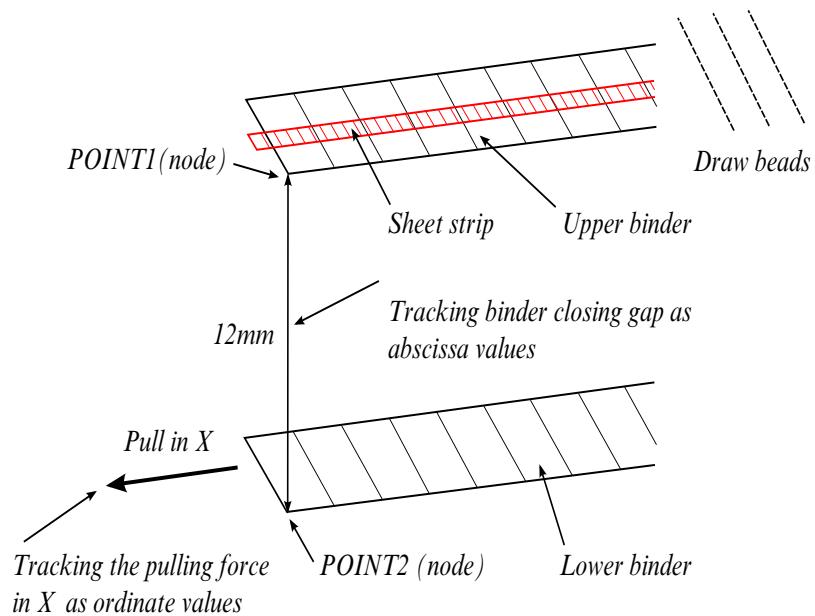
Figure 11-10. Tracking the closing gap and pulling distance
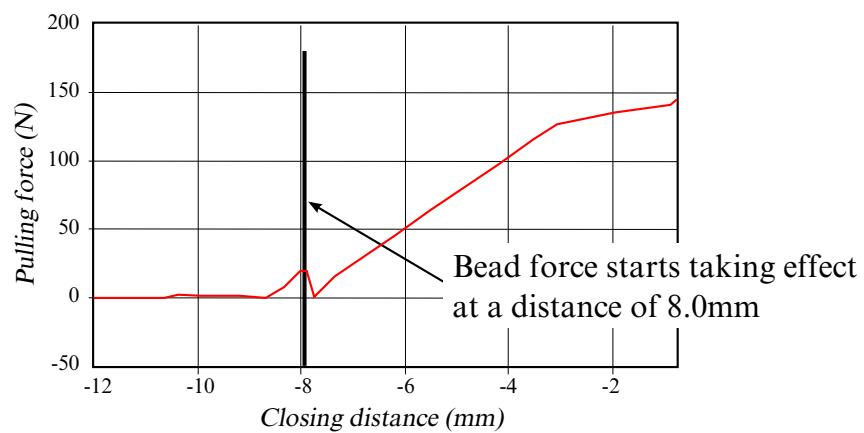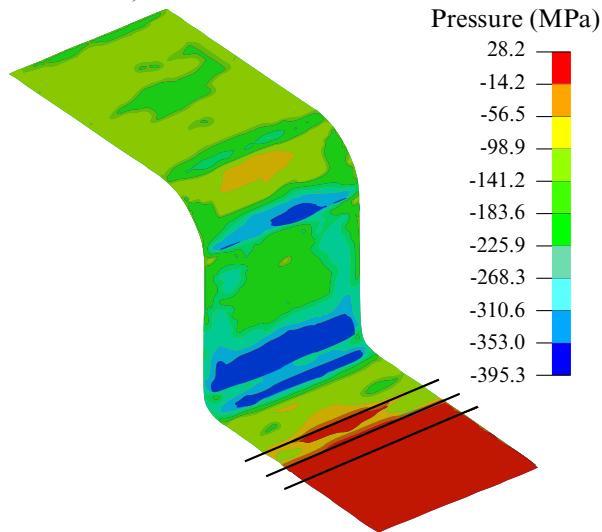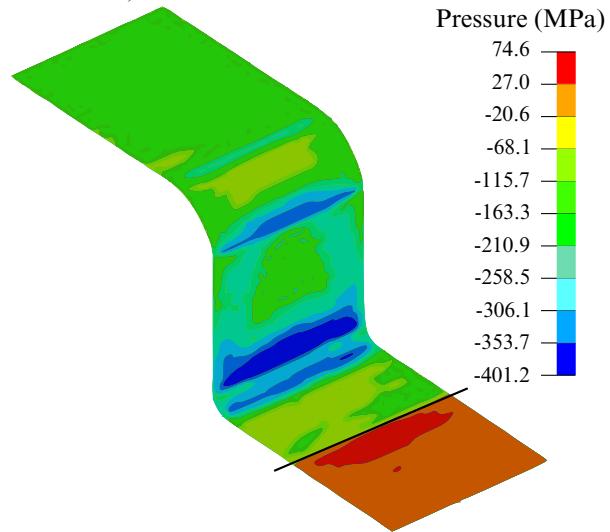
Figure 11-11. Pulling force (NODFOR) as a function of closure distance

Time=0.0152, #nodes=7005, #elem=6503
Contours of pressure (mid-plane)
min=-395.339, at elem# 15423
max=28.1887, at elem# 13317

Time=0.0152, #nodes=6976, #elem=6476
Contours of pressure (mid-plane)
min=-401.24, at elem# 15280
max=74.6163, at elem# 13016

Pressure (MPa)

| 28.2 |
| -14.2 |
| -56.5 |
| -98.9 |
| -141.2 |
| -183.6 |
| -225.9 |
| -268.3 |
| -310.6 |
| -353.0 |
| -395.3 |

Pressure (MPa)

| 74.6 |
| 27.0 |
| -20.6 |
| -68.1 |
| -115.7 |
| -163.3 |
| -210.9 |
| -258.5 |
| -306.1 |
| -353.7 |
| -401.2 |

Mean stresses of a channel draw
(NBEAD=3)

Mean stresses of a channel draw
on one line bead

Figure 11-12. Mean stress comparison between NBEAD = 3 and 1

Card 4: ERODING_..._SURFACE

ERODING contacts update the active contact segments to account for erosion (deletion) of elements on the contact surface. Segments that could potentially become exposed to contact during the simulation are determined during initialization and that number of segments is reported in the contact description in d3hsp. However, at any given point in time during the simulation, only the subset consisting of currently exposed (external) segments is active in the contact evaluation.

This Card 4 is mandatory for:

        *CONTACT_ERODING_NODES_TO_SURFACE

        *CONTACT_ERODING_SINGLE_SURFACE

        *CONTACT_ERODING_SURFACE_TO_SURFACE

Card 4	1	2	3	4	5	6	7	8
Variable	ISYM	EROSOP	IADJ					
Type	I	I	I					
Default	0	1	↓					

VARIABLE	DESCRIPTION
ISYM	Symmetry plane option:

        EQ.0: Off (default)

        EQ.1: Do not include faces with normal boundary constraints (such as segments of brick elements on a symmetry plane).

This option is important for retaining the correct boundary conditions in the model with symmetry.

EROSOP   Erosion/interior node option (EROSOP is hardcoded to 1 for both SMP and MPP):

        EQ.0: Only exterior boundary information is saved. (This option is no longer supported.)

        EQ.1: Storage is allocated so that eroding contact can occur.

IADJ   Adjacent material treatment for solid elements (in the case of MPP,

VARIABLE	DESCRIPTION

IADJ is hardcoded to 1):

EQ.0: Solid element faces are included only for free boundaries (default for SMP).

EQ.1: Solid element faces are included if they are on the boundary of the material subset. This option also allows for erosion within a body and the subsequent treatment of contact.

Remarks:

1. **Time step.** Eroding contact may control the time step (see ECDT in *CONTROL_CONTACT).

2. **SURFA type.** For ERODING_NODES_TO_SURFACE, define the SURFA side using a node set, not a part ID or part set ID.

3. **Negative volume failure criterion.** ERODING contact automatically invokes a negative volume failure criterion for all solid elements in the model, except as overridden by PSFAIL in *CONTROL_SOLID. PSFAIL will limit the negative volume failure criterion to a set of solid parts. A negative volume failure criterion circumvents an error termination due to negative volume by deleting solid elements that develop negative volume.

4. **Contact friction.** Contact friction is not considered by SMP LS-DYNA for *CONTACT_ERODING_NODES_TO_SURFACE and *CONTACT_ERODING_SURFACE_TO_SURFACE unless SOFT is set to 2 on Optional Card A. MPP LS-DYNA has no such exclusion for contact friction.

5. **SPH contact thickness.** The contact thickness (see SAST on Card 3 and ITHK = 1 on *CONTROL_SPH) of SPH nodes is not considered for *CONTACT_ERODING_NODES_TO_SURFACE in SMP. The thickness of the SPH node is set to zero for this type of contact. The thickness is considered for MPP.

Card 4: SURFACE_INTERFERENCE

This Card 4 is mandatory for:

 *CONTACT_NODES_TO_SURFACE_INTERFERENCE

 *CONTACT_ONE_WAY_SURFACE_TO_SURFACE_INTERFERENCE

 *CONTACT_SURFACE_TO_SURFACE_INTERFERENCE

Purpose: This contact option provides a means of modeling parts which are shrink fitted together and are, therefore, prestressed in the initial configuration. This option turns off the nodal interpenetration checks (which changes the geometry by moving the nodes to eliminate the interpenetration) at the start of the simulation and allows the contact forces to develop to remove the interpenetrations. The load curves defined in this section scale the interface stiffness constants such that the stiffness can increase slowly from zero to a final value with effect that the interface forces also increase gradually to remove the overlaps.

Card 4	1	2	3	4	5	6	7	8
Variable	LCID1	LCID2						
Type	I	I						
Default	0	0						

VARIABLE	DESCRIPTION
LCID1	Load curve ID which scales the interface stiffness during dynamic relaxation. This curve must originate at $(0,0)$ at time $= 0$ and gradually increase.
LCID2	Load curve ID which scales the interface stiffness during the transient calculation. This curve generally has a constant value of unity for the duration of the calculation if LCID1 is defined. If LCID1 $= 0$, this curve must originate at $(0,0)$ at time $= 0$ and gradually increase to a constant value.

Remarks:

1. **Shell thickness offsets.** In the case of MPP LS-DYNA, consideration of shell thickness offsets is controlled as in an "old" contact type, using SHLTHK, either

on *CONTROL_CONTACT or on Optional Card B of *CONTACT_…_INTER-FERENCE.

In the case of SMP, shell thickness offsets are always considered with the lone exception being when THKOPT = 2 on Optional Card B of *CONTACT_…_IN-TERFERENCE.

2. **Penetrations and orientations.** The check to fix initial penetrations is skipped. Automatic orientation of shell elements is also skipped. Furthermore, segment orientation for shell elements and interpenetration checks *are skipped*. Therefore, it is necessary in the problem setup to ensure that all contact segments which belong to shell elements are properly oriented, that is, the outward normal vector of the segment based on the right-hand rule relative to the segment numbering, must point to the opposing contact surface. Consequently, automatic contact generation should be avoided for parts composed of shell elements unless automatic generation is used on the SURFA side of a nodes to surface interface.

3. **Resolving an interference during a transient phase preceded by a dynamic relaxation phase.** If LCID1 = 0 and LS-DYNA performs dynamic relaxation (such as for initializing the stresses in other components), the INTEFERENCE contact treats initial penetrations as specified by the IGNORE parameters on *CONTACT and *CONTROL_CONTACT. To make the INTERFERENCE contact inactive during a dynamic relaxation phase and resolve the penetrations during the subsequent transient phase based on the curve LCID2, use either of the two following approaches:

    a) Set the contact birth time BT equal to a negative value (BT < 0) that is in magnitude anything larger than the expected completion time of the dynamic relaxation phase. After completing dynamic relaxation, LS-DYNA activates the contact regardless of the value of BT.

    b) Specify curve LCID1 with zero ordinates (meaning a zero contact stiffness) during the dynamic relaxation phase.

For both approaches, originate curve LCID2 at (0,0) at time = 0 and gradually increase the scale factor to a constant value.

Card 4: RIGID_TO_RIGID

This Card 4 is mandatory for:

     *CONTACT_RIGID_NODES_TO_RIGID_BODY

     *CONTACT_RIGID_BODY_ONE_WAY_TO_RIGID_BODY

     *CONTACT_RIGID_BODY_TWO_WAY_TO_RIGID_BODY

Card 4	1	2	3	4	5	6	7	8
Variable	LCID	FCM	US		LCDC	DSF	UNLCID	
Type	I	I	F		I	F	I	
Default	required	required	LCID		optional	0.0	optional	

VARIABLE	DESCRIPTION

LCID   Load curve ID giving force as a function of penetration behavior for RIGID contact. See FCM below.

FCM   Force calculation method for RIGID contact:

      EQ.1: Load curve gives total normal force on surface as a function of maximum penetration of any node (RIGID_-BODY_ONE_WAY only).

      EQ.2: Load curve gives normal force on each node as a function of the penetration of a node through the surface (all RIGID contact types).

      EQ.3: Load curve gives normal pressure as a function penetration of a node through the surface (RIGID_BODY_TWO_-WAY and RIGID_BODY_ONE_WAY only).

      EQ.4: Load curve gives total normal force as a function of maximum soft penetration (RIGID_BODY_ONE_WAY only). In this case the force will be followed based on the original penetration point.

US   Unloading stiffness for RIGID contact. The default is to unload along the loading curve. This should be equal to or greater than the maximum slope used in the loading curve.
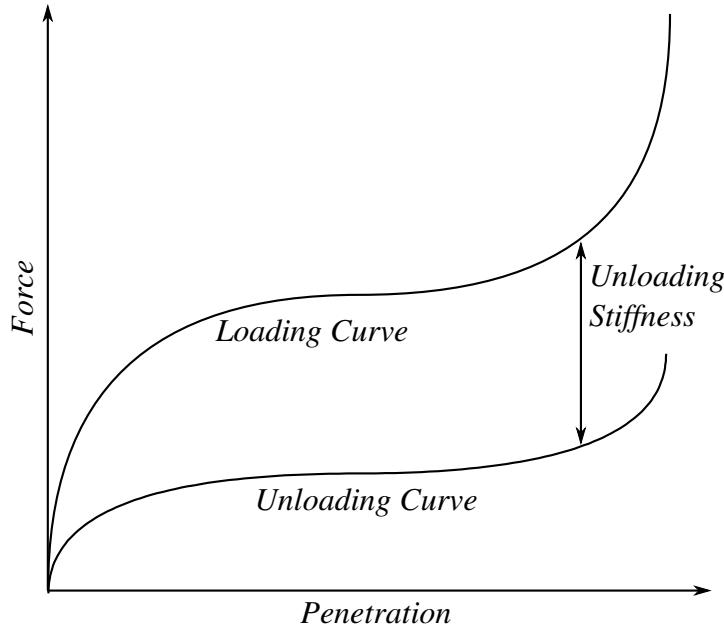
Figure 11-13. Behavior if an unloading curve is defined

VARIABLE	DESCRIPTION
LCDC	Load curve ID giving damping coefficient (DC) as a function of penetration velocity. The damping force FD is then: FD = DSF × DC × velocity.
DSF	Damping scaling factor
UNLCID	Optional load curve ID giving force as a function of penetration behavior for RIGID_BODY_ONE_WAY contact during unloading. This option requires the definition of the unloading stiffness, US. See Figure 11-13.

Card 4: TIEBREAK_NODES

This Card 4 is mandatory for:

> *CONTACT_TIEBREAK_NODES_TO_SURFACE
>
> *CONTACT_TIEBREAK_NODES_ONLY

Card 4	1	2	3	4	5	6	7	8
Variable	NFLF	SFLF	NEN	MES				
Type	F	F	F	F				
Default	required	required	2.	2.				

VARIABLE	DESCRIPTION
NFLF	Normal failure force. Only tensile failure, meaning tensile normal forces, will be considered in the failure criterion.
SFLF	Shear failure force
NEN	Exponent for normal force in the failure criterion
MES	Exponent for shear force in the failure criterion

Remarks:

1. **Node by Node Failure Attributes.** These fields can be overridden node by node with the attributes on the *SET_NODE data cards.

2. **Failure Criterion.** The tiebreak failure criterion is given as:

$$\left(\frac{|f_n|}{\text{NFLF}}\right)^{\text{NEN}} + \left(\frac{|f_s|}{\text{SFLF}}\right)^{\text{MES}} \geq 1 \ .$$

    Failure is assumed if the left side is larger than 1. f_n and f_s are the normal and shear interface force. Both NFLF and SFLF must be defined. If failure in only tension or shear is required, then set the other failure force to a large value (10^{10}).

3. **Contact Behavior after Failure.** After failure, CONTACT_TIEBREAK_-NODES_TO_SURFACE behaves as a nodes-to-surface contact with no thickness offsets (no interface tension possible) whereas the CONTACT_TIEBREAK_-

NODES_ONLY stops acting as a contact. Prior to failure, the two contact types behave identically.

Card 4: TIEBREAK_SURFACE

This Card 4 is mandatory for:

 *CONTACT_TIEBREAK_SURFACE_TO_SURFACE

 *CONTACT_TIEBREAK_SURFACE_TO_SURFACE_ONLY

Card 4	1	2	3	4	5	6	7	8
Variable	NFLS	SFLS	TBLCID	THKOFF				
Type	F	F	I	I				
Default	required	required	0	0				

VARIABLE	DESCRIPTION

NFLS Tensile failure stress

SFLS Shear failure stress

TBLCID Optional load curve ID defining the resisting tensile stress as a function of gap opening in the normal direction for the post-failure response. This option applies only to SMP and can be used to model adhesives.

THKOFF Thickness offsets are considered if THKOFF = 1. If shell offsets are included in the meshed geometry, this option is highly recommended since segment orientations can be arbitrary and the contact surfaces can be disjoint. This option is *not* available in the MPP version of LS-DYNA. It works by substituting with *CONTACT_-AUTOMATIC_SURFACE_TO_SURFACE_TIEBREAK (OPTION = 2 if TBLCID is not specified; OPTION = 5 if TBLCID is specified).

Remarks:

1. **Segment by Segment Failure Attributes.** The failure fields, NFLS and SFLS, can be overridden segment by segment on the *SET_SEGMENT or *SET_SHELL data cards for the SURFA surface with A1 and A2, respectively. These variables do not apply to the SURFB surface.

2.   **Failure Criterion.** The tiebreak contact fails when:

$$\left(\frac{|\sigma_n|}{\text{NFLS}}\right)^2 + \left(\frac{|\sigma_s|}{\text{SFLS}}\right)^2 \geq 1 \ .$$

Both NFLS and SFLS must be defined. If failure in only tension or shear is required, then set the other failure stress to a large value (10^{10}). When used with shells, contact segment normal vectors are used to establish the tension direction (as opposed to compression). Compressive stress does not contribute to the failure equation.

3.   **Contact Behavior after Failure.** After failure, *CONTACT_TIEBREAK_SURFACE_TO_SURFACE behaves as a surface-to-surface contact with no thickness offsets while *CONTACT_TIEBREAK_SURFACE_TO_SURFACE_ONLY stops acting as a contact altogether. Until failure, it ties the SURFA nodes to the SURFB segments.

4.   **Contact Stiffness**. Contact stiffness is based on the SURFB side. The variables PENOPT and SOFT have no effect on contact stiffness.

Card 4: CONTRACTION_JOINT

This Card 4 is mandatory for:

        *CONTACT_SURFACE_TO_SURFACE_CONTRACTION_JOINT

Purpose: This contact option turns on the contraction joint model designed to simulate the effects of sinusoidal joint surfaces (shear keys) in the contraction joints of arch dams and other concrete structures. The sinusoidal functions for the shear keys are defined according to the following three methods [Solberg and Noble 2002]:

    a) Method 1.

$$\hat{g} = g - A\{1 - \cos[B(s_2 - s_1)]\}$$

    b) Method 2.

$$\hat{g} = g - 2A \left| \sin\left[\frac{B(s_2 - s_1)}{2} \right] \right|$$

    c) Method 3 (default).

$$\hat{g} = g - A\cos(Bs_2) + A\cos(Bs_1)$$

Here g is a gap function for the contact surface and $\hat{g}$ is a gap function for the joint surface. A is the key amplitude parameter, and B is the key frequency parameter. s_1 and s_2 are referential surfaces:

$$s_1 = \mathbf{X}_{surface1} \cdot \mathbf{T}_{key}$$
$$s_2 = \mathbf{X}_{surface2} \cdot \mathbf{T}_{key}$$
$$\mathbf{T}_{key} = \mathbf{T}_{slide} \times \mathbf{n}$$

$\mathbf{T}_{slide}$ is the free sliding direction of the keys and $\mathbf{n}$ is the surface normal in reference.

Card 4	1	2	3	4	5	6	7	8
Variable	MTCJ	ALPHA	BETA	TSVX	TSVY	TSVZ		
Type	I	F	F	F	F	F		
Default	0	0.0	0.0	0.0	0.0	0.0		

VARIABLE	DESCRIPTION
MTCJ	The method option for the gap function, $\hat{g}$
ALPHA	Key amplitude parameter, A

VARIABLE	DESCRIPTION
BETA	Key frequency parameter, B
TSVX	x component of the free sliding direction $\mathbf{T}_{\text{slide}}$
TSVY	y component of the free sliding direction $\mathbf{T}_{\text{slide}}$
TSVZ	z component of the free sliding direction $\mathbf{T}_{\text{slide}}$

THERMAL:

THRM 1 is mandatory for the THERMAL (and THERMAL_FRICTION) option, meaning:

        CONTACT_..._THERMAL_...

> **NOTE:** If Card 4 is required, then it must go before THRM 1.
> (Card 4 is required for certain contact types.)

Thermal Card 1.

THRM 1	1	2	3	4	5	6	7	8
Variable	K	FRAD	H0	LMIN	LMAX	FTOSA	BC_FLG	ALGO
Type	F	F	F	F	F	F	I	I
Default	none	none	none	none	none	0.5	0	0

VARIABLE	DESCRIPTION

K   Thermal conductivity of fluid between the contact surfaces. If a gap with a thickness l_{gap} exists between the contact surfaces, then the conductance due to thermal conductivity between the contact surfaces is

$$h_{\text{cond}} = \frac{K}{l_{\text{gap}}} \ .$$

Note that LS-DYNA calculates l_{gap} based on deformation (see Remark 2).

FRAD   Radiation factor, f_{rad}, between the contact surfaces:

$$f_{\text{rad}} = \frac{\sigma}{\frac{1}{\varepsilon_1} + \frac{1}{\varepsilon_2} - 1} \ ,$$

where

$$\sigma = \text{Stefan-Boltman constant}$$
$$\varepsilon_1 = \text{emissivity of SURFA surface}$$
$$\varepsilon_2 = \text{emissivity of SURFB surface}$$

LS-DYNA calculates the radiant heat transfer conductance as

$$h_{\text{rad}} = f_{\text{rad}}(T_{\text{SURFA}} + T_{\text{SURFB}})(T_{\text{SURFA}}^2 + T_{\text{SURFB}}^2) \ .$$

VARIABLE	DESCRIPTION
H0	Heat transfer conductance for closed gaps. Use this heat transfer conductance for gaps in the range $$0 \leq l_{\text{gap}} \leq l_{\text{min}} \ .$$
LMIN	Minimum gap, l_{min}. The heat transfer conductance defined above (H0) is used for gap thicknesses less than this value. If $l_{\text{min}} < 0$, then $-l_{\text{min}}$ is a load curve number defining l_{min} as a function time.
LMAX	No thermal contact if the gap is greater than this value (l_{max})
FTOSA	Fraction, f, of sliding friction energy partitioned to the SURFA surface (see Remark 3). Energy partitioned to the SURFB surface is $(1 - f)$.
	EQ.0: Set to 0.5 (default): The sliding friction energy is partitioned 50% - 50% to the SURFA and SURFB surfaces in contact.
BC_FLAG	Thermal boundary condition flag:
	EQ.0: Thermal boundary conditions are on when parts are in contact.
	EQ.1: Thermal boundary conditions are off when parts are in contact.
ALGO	Thermal Contact algorithm type:
	EQ.0: Two-way contact: Both surfaces change temperature due to contact
	EQ.1: One-way contact: Surface of SURFB does not change temperature due to contact. Surface of SURFA does change temperature.
	EQ.2: Two-way edge contact
	EQ.3: One-way edge contact (no heat transfer into the reference side)
	NOTE: For ALGO = 2 and 3, shell edges must be on the SURFA side, and solid facets and shell surface need to be on the SURFB side.

Remarks:

1. **Contact Types.** Thermal contact involves heat transfer across surfaces, so the contact definition must be of a SURFACE_TO_SURFACE type. Thus, in general the NODES_TO_SURFACE contacts cannot be used, and an error termination will be the result. The exception is TIED_NODES_TO_SURFACE and TIED_-SHELL_EDGE_TO_SURFACE contacts with the SURFA side *not* defined with node sets. The TIED_NODES_TO_SURFACE and TIED_SHELL_EDGE_TO_-SURFACE contacts also allow thermal contact with beam elements. These contacts can model, for instance, heat transfer through spot weld beams used for tying opposite surfaces. Thermal contact with beam elements is otherwise only supported for Mortar contacts, by inheriting the Mortar algorithms from the mechanical side of the contact.

2. **Heat Conductance.** The heat conductance is calculated as

$$h = \begin{cases} h_0 & 0 \leq l_{\text{gap}} \leq l_{\text{min}} \\ h_{\text{cond}} + h_{\text{rad}} & l_{\text{min}} < l_{\text{gap}} \leq l_{\text{max}} \\ 0 & l_{\text{gap}} > l_{\text{max}} \end{cases}$$

LS- DYNA calculates l_{gap} based on deformation.

3. **Sliding Friction Energy Fraction.** f can be calculated using:

$$f = \frac{1}{1 + \dfrac{\sqrt{(\rho C_p k)}_{\text{SURFB side material}}}{\sqrt{(\rho C_p k)}_{\text{SURFA side material}}}}$$

THERMAL_FRICTION:

THRM 2 is required after THRM 1 if the FRICTION suffix is added to THERMAL.

*CONTACT_…_THERMAL_FRICTION_…

The blank (or work piece) must be defined as SURFA in a metal forming model.

Purpose:

1. Define the mechanical static and dynamic friction coefficients as a function of temperature.

2. Define the thermal contact conductance as a function of temperature and pressure.

THERMAL Card 2.

THRM 2	1	2	3	4	5	6	7	8
Variable	LCFST	LCFDT	FORMULA	A	B	C	D	LCH
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

THERMAL Card 3 - User Subroutine Cards. Additional cards for FORMULA < 0. Use as many cards as needed to set |FORMULA| number of parameters.

THRM 2.1	1	2	3	4	5	6	7	8
Variable	UC1	UC2	UC3	UC4	UC5	UC6	UC7	UC8
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
LCFST	Load curve ID for static coefficient of friction as a function of temperature. The load curve value multiplies the coefficient value FS. For Mortar contact, the static coefficient of friction can be given as a load curve or table. In the case of a load curve, it is a function of

VARIABLE	DESCRIPTION
	the mean temperature, $(T_{\text{SURFA}} + T_{\text{SURFB}})/2$, in the contact interface. In the case of a table, the static coefficient of friction is an arbitrary function of T_{SURFA} and T_{SURFB}; each value of T_{SURFB} should be associated with a curve having T_{SURFA} as the abscissa.
LCFDT	Load curve ID for dynamic coefficient of friction as a function of temperature. The load curve value multiplies the coefficient value FD. For Mortar contact, the dynamic coefficient of friction can be given as a load curve or table. In the case of a load curve, it is a function of the mean temperature, $(T_{\text{SURFA}} + T_{\text{SURFB}})/2$, in the contact interface. In the case of a table, the dynamic coefficient of friction is an arbitrary function of T_{SURFA} and T_{SURFB}; each value of T_{SURFB} should be associated with a curve having T_{SURFA} as the abscissa.
FORMULA	Formula that defines the contact heat conductance as a function of temperature and pressure.

> **EQ.1:** $h(P)$ is defined by load curve A, which contains data for contact conductance as a function of pressure.
>
> **EQ.2:** $h(P)$ is given by the following where A, B, C and D although defined by load curves are typically constants for use in this formula. The load curves are functions of temperature.
>
> $$h(P) = a + bP + cP^2 + dP^3$$
>
> **EQ.3:** $h(P)$ is given by the following formula from [Shvets and Dyban 1964]:
>
> $$h(P) = \frac{\pi k_{\text{gas}}}{4\lambda}\left[1. + 85\left(\frac{P}{\sigma}\right)^{0.8}\right] = \frac{a}{b}\left[1. + 85\left(\frac{P}{c}\right)^{0.8}\right]$$
>
> where,
>
> a: is evaluated from the load curve, A, for the thermal conductivity, k_{gas}, of the gas in the gap as a function of temperature.
>
> b: is evaluated from the load curve, B, for the parameter grouping $\pi/4\lambda$. Therefore, this load curve should be set to a constant value. λ is the surface roughness.
>
> c: is evaluated from the load curve, C, which specifies a stress metric for deformation (such as yield) as a function of temperature.
>
> **EQ.4:** $h(P)$ is given by the following formula from [Li and

VARIABLE	DESCRIPTION

Sellars 1996]:

$$h(P) = a\left[1 - \exp\left(-b\frac{P}{c}\right)\right]^d$$

where,

  a: is evaluated from the load curve, A, which defines a load curve as a function of temperature.

  b: is evaluated from the load curve, B, which defines a load curve as a function of temperature.

  c: is evaluated from the load curve, C, which defines a stress metric for deformation (such as yield) as a function of temperature.

  d: is evaluated from the load curve D, which is a function of temperature.

  EQ.5: $h(\text{gap})$ is defined by load curve A, which contains data for contact conductance as a function of interface gap.

  LT.0: This is equivalent to defining the keyword *USER_IN-TERFACE_CONDUCTIVITY. The user subroutine `usrhcon` will be called for this contact interface to define the contact heat transfer coefficient.

A	Load curve ID for the *a* coefficient used in the formula
B	Load curve ID for the *b* coefficient used in the formula
C	Load curve ID for the *c* coefficient used in the formula
D	Load curve ID for the *d* coefficient used in the formula
LCH	Load curve ID for *h*. If defined, this input takes precedence over any other definitions. This parameter can refer to a curve ID (see *DEFINE_CURVE) or a function ID (see *DEFINE_FUNCTION). When LCH is a curve ID (and a function ID) it is interpreted as follows:

  GT.0: The heat transfer coefficient is defined as a function of time, t, by a curve consisting of $(t, h(t))$ data pairs.

  LT.0: The heat transfer coefficient is defined as a function of temperature, T, by a curve consisting of $(T, h(T))$ data pairs.

When the reference is to a function, it is prototyped as $h =$

VARIABLE	DESCRIPTION

$h(t, T_{\text{avg}}, T_{\text{SURFA}}, T_{\text{SURFB}}, P, g)$ where:

$$t = \text{solution time}$$
$$T_{\text{avg}} = \text{average interface temperature}$$
$$T_{\text{SURFA}} = \text{SURFA segment temperature}$$
$$T_{\text{SURFB}} = \text{SURFB segment temperature}$$
$$P = \text{interface pressure}$$
$$g = \text{gap distance between SURFA and SURFB segment}$$

UCi User parameters

ORTHO_FRICTION:

Additional cards for the ORTHO_FRICTION option:

*CONTACT_..._ORTHO_FRICTION_...

ORTHO_FRIC 1.

ORFR 1	1	2	3	4	5	6	7	8
Variable	FS1_SA	FD1_SA	DC1_SA	VC1_SA	LC1_SA	OACS_SA	LCFSA	LCPSA
Type	F	F	F	F	I	I	I	I
Default	0.	0.	0.	0.	0	0	0	0

ORTHO_FRIC 2.

ORFR 2	1	2	3	4	5	6	7	8
Variable	FS2_SA	FD2_SA	DC2_SA	VC2_SA	LC2_SA			
Type	F	F	F	F	I			
Default	0.	0.	0.	0.	0			

ORTHO_FRIC 3.

ORFR 3	1	2	3	4	5	6	7	8
Variable	FS1_SB	FD1_SB	DC1_SB	VC1_SB	LC1_SB	OACS_SB	LCFSB	LCPSB
Type	F	F	F	F	I	I	I	I
Default	0.	0.	0.	0.	0	0	0	0

ORTHO_FRIC 4.

ORFR 4	1	2	3	4	5	6	7	8
Variable	FS2_SB	FD2_SB	DC2_SB	VC2_SB	LC2_SB			
Type	F	F	F	F	I			
Default	0.	0.	0.	0.	0			

VARIABLE	DESCRIPTION
FSn_SA or SB	Static coefficient of friction in the local n orthotropic direction for the SURFA (SA) or SURFB (SB) surface. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact, $$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-\text{DC}|v_{\text{rel}}|} ,$$ where the direction and surface are left off for clarity.

When contacts with ORTHO_FRICTION are defined by segment sets, each segment allows the specification of an offset angle in degrees from the 1-2 side of the segment which locates the 1-direction for the friction. The offset angle is input as the first attribute of the segment in *SET_SEGMENT. The transverse direction, or 2-direction, is in the plane of the segment and is perpendicular to the 1-direction. The offset angle is taken as zero when part (set) IDs are used in the contact definition. See *DEFINE_FRICTION_ORIENTATION for remarks regarding use of *CONTACT_FORMING_-ONE_WAY_SURFACE_TO_SURFACE_ORTHO_FRICTION.

VARIABLE	DESCRIPTION
FDI_SA or SB	Dynamic coefficient of friction in the local n orthotropic direction
DCn_SA or SB	Exponential decay coefficient for the local n direction
VCn_SA or SB	Coefficient for viscous friction in the local n direction. See the description for VC for mandatory Card 2 above.
LCn_SA or SB	Table ID of a two- or three-dimensional table (see *DEFINE_TABLE, *DEFINE_TABLE_2D, or *DEFINE_TABLE_3D) giving the friction coefficient in the local n direction as a function of the relative velocity, interface pressure, and optionally the temperature. In the 2D case, each curve in the 2D table definition defines the coefficient of friction as a function of the interface pressure

VARIABLE	DESCRIPTION
	corresponding to a particular value of the relative velocity. In the 3D case, each table in the 3D table definition corresponds to a given temperature, and that table gives the friction coefficient as a function of the relative velocity and interface pressure. Note that 3D tables are only available for Mortar contact.
OACS_SA or SB	If the default value, 0, is active, the frictional forces acting on a node sliding on a segment are based on the local directions of the segment. If OACS is set to unity, 1, the frictional forces acting on a node sliding on a segment are based on the local directions of the sliding node. No matter what the setting for OACS, the_SA coefficients are always used for SURFA nodes and the_SB coefficients for SURFB nodes.
LCFSA or SB	Optional load curve that gives the coefficient of friction as a function of the direction of relative motion, as measured in degrees from the first orthotropic direction. If this load curve is specified, the other parameters (FS, FD, DC, VC, LC) are ignored. This is currently only supported in the MPP version.
LCPSA or SB	Optional load curve that gives a scale factor for the friction coefficient as a function of interface pressure. This is only used if LCFSA (or SB) is defined.

Remarks:

1. **Orthotropic Friction for Mortar Contact.** For Mortar contact, we consider what is going on in the plane of surface *A* with normal $\mathbf{n}$, and refer to Figure 11-14. The orthotropic friction law is based on an elliptic yield surface:

$$p \geq \sqrt{\left(\frac{t_\|}{\mu_\|}\right)^2 + \left(\frac{t_\perp}{\mu_\perp}\right)^2}$$

    for the Coulomb friction law. This surface is a cone in the pressure-traction space as shown in Figure 11-14. We also mandate that the slip direction is associative

$$\mathbf{v} \sim \frac{\partial p}{\partial \mathbf{t}} \ ,$$

    resulting in an angle δ between the slip and traction. Here p is the contact pressure, $\mathbf{t} = t_\| \mathbf{a}_\| + t_\perp \mathbf{a}_\perp$ is the traction vector and $\mathbf{\mu} = \{\mu_\|, \mu_\perp\}$ are the coefficients of friction in the orthonormal system $\mathbf{A} = \{\mathbf{a}_\|, \mathbf{a}_\perp\}$ of surface *A*. Similarly, we have the orthonormal system $\mathbf{B} = \{\mathbf{b}_\|, \mathbf{b}_\perp\}$ on surface *B* which is related to $\mathbf{A}$ by
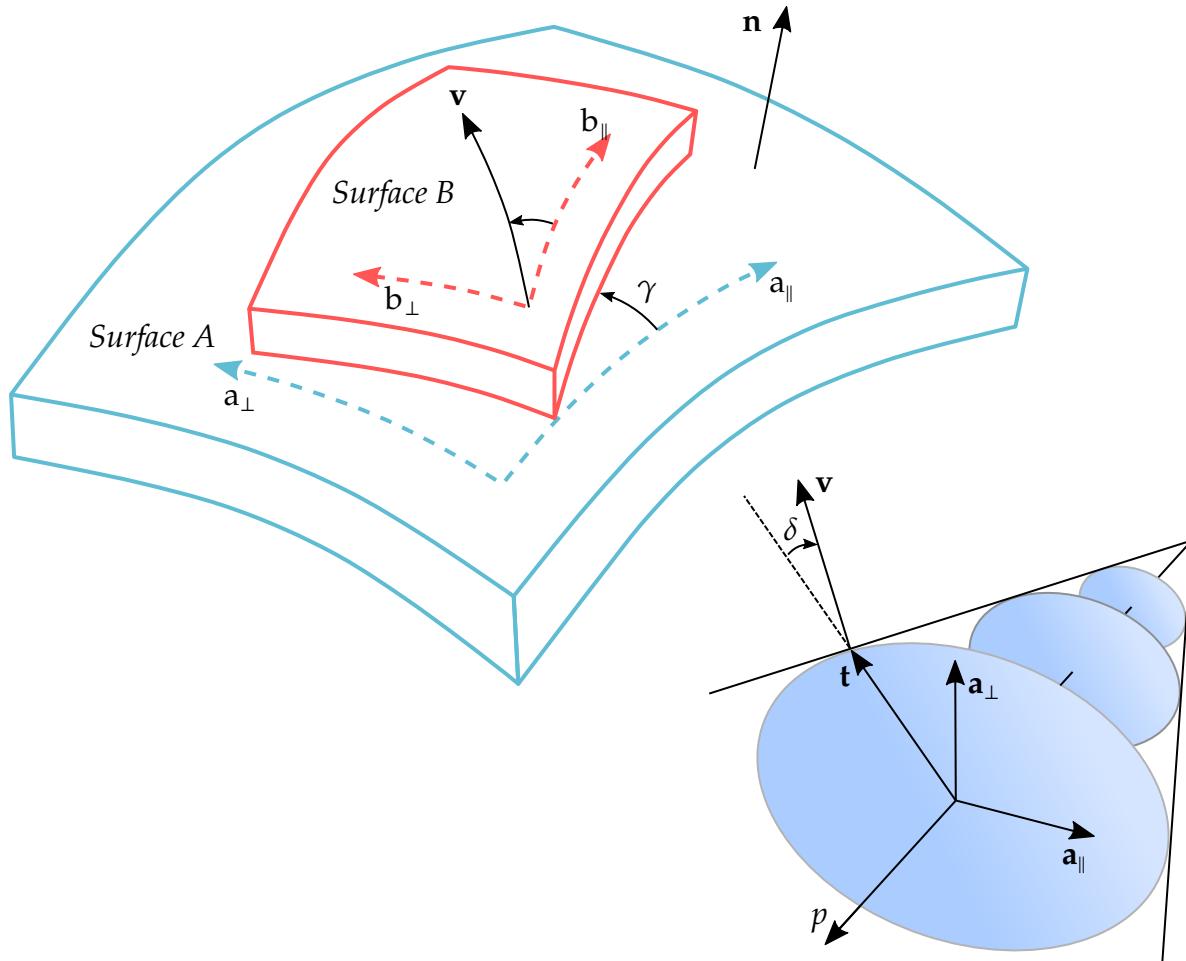
Figure 11-14. Definitions of terms for orthotropic friction in Mortar contact. Surface B with its main fibers in direction $\mathbf{b}_\parallel$ slides on surface A with its fibers in direction $\mathbf{a}_\parallel$. The contact pressure is p and the sliding velocity of surface A relative to B is in the direction $\mathbf{v}$. This results in a frictional traction, $\mathbf{t}$, on surface A directed (by and large) in the same direction as $\mathbf{v}$. The corresponding traction on surface B is in the opposite direction and of the same magnitude as $\mathbf{t}$.

rotating it an angle γ with respect to the normal $\mathbf{n}$ of surface A. First we need to mention how these systems, $\mathbf{A}$ and $\mathbf{B}$, are constructed in the first place.

We assume that the materials of the interacting surfaces are anisotropic. Then $\mathbf{a}_\parallel$ is the *first* material direction of the material constituting surface A and $\mathbf{b}_\parallel$ is likewise the corresponding first material direction on surface B. Therefore, currently the orthotropic directions for mortar contact are *not* set on the contact card but are inherited from the underlying materials. In Figure 11-14, $\mathbf{A}$ and $\mathbf{B}$ should not be confused with element specific directions, as this exposition is completely independent of finite elements.

The friction coefficients $\boldsymbol{\mu}$ are determined from the angle γ (see Figure 11-14) through direction cosines

$$\mu = \text{FS1\_A} \qquad \mu = \text{FS2\_A} \qquad \mu = \text{FS1\_B} \qquad \mu = \text{FS2\_B}$$
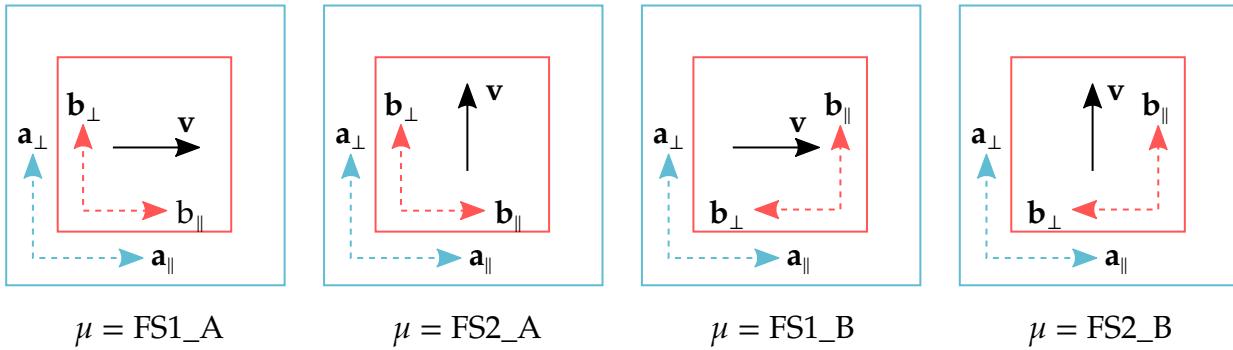
Figure 11-15. Interpretation of input parameters for Mortar orthotropic friction

$$\alpha = \cos^2\gamma = (\mathbf{a}_\parallel \cdot \mathbf{b}_\parallel)^2 = (\mathbf{a}_\perp \cdot \mathbf{b}_\perp)^2$$
$$\beta = \sin^2\gamma = (\mathbf{a}_\perp \cdot \mathbf{b}_\parallel)^2 = (\mathbf{a}_\parallel \cdot \mathbf{b}_\perp)^2$$

as

$$\mu_\parallel = \alpha\, \text{FS1\_SA} + \beta\, \text{FS1\_SB}$$
$$\mu_\perp = \alpha\, \text{FS2\_SA} + \beta\, \text{FS2\_SB}.$$

It follows that the interpretation of the input parameters that are relevant for Mortar orthotropic friction are (see Figure 11-15):

a) FS1_SA being the coefficient of friction for $\mathbf{v}$ in direction $\mathbf{a}_\parallel$ when $\mathbf{a}_\parallel$ is aligned with $\mathbf{b}_\parallel$.

b) FS1_SB being the coefficient of friction for $\mathbf{v}$ in direction $\mathbf{a}_\parallel$ when $\mathbf{a}_\parallel$ is aligned with $\mathbf{b}_\perp$.

c) FS2_SA being the coefficient of friction for $\mathbf{v}$ in direction $\mathbf{a}_\perp$ when $\mathbf{a}_\parallel$ is aligned with $\mathbf{b}_\parallel$.

d) FS2_SB being the coefficient of friction for $\mathbf{v}$ in direction $\mathbf{a}_\perp$ when $\mathbf{a}_\parallel$ is aligned with $\mathbf{b}_\perp$.

The effective friction coefficient will then be determined by the yield surface and flow rule as

$$\mu = \frac{\|\mathbf{t}\|}{p}$$

and thus, determines the relationship between contact pressure p and magnitude of sliding traction t.

The behavior for relative sliding $\mathbf{v}$ in either of the main A directions $\mathbf{a}_\parallel$ and $\mathbf{a}_\perp$ is by design quite intuitive; the effective friction coefficients will be $\mu_\parallel$ and $\mu_\perp$, respectively, and the tractions $\mathbf{t}$ are collinear with the sliding. However, from the associative slip law the traction $\mathbf{t}$ and sliding $\mathbf{v}$ are not *always* pointing in the

exact same direction, and this is explained best by considering two situations. With prescribed sliding **v** in some other direction than $\mathbf{a}_\parallel$ or $\mathbf{a}_\perp$, the resulting traction vector **t** will have its main component in the direction of sliding, but also a (nonzero) component towards the direction with the *largest* friction coefficient. The corresponding tendency will be seen with a prescribed traction **t**, for which the resulting sliding direction **v** will tend towards the direction with the *smallest* friction coefficient. See Figure 11-16.

$$\mu_\perp < \mu_\parallel \qquad\qquad\qquad \mu_\perp > \mu_\parallel$$
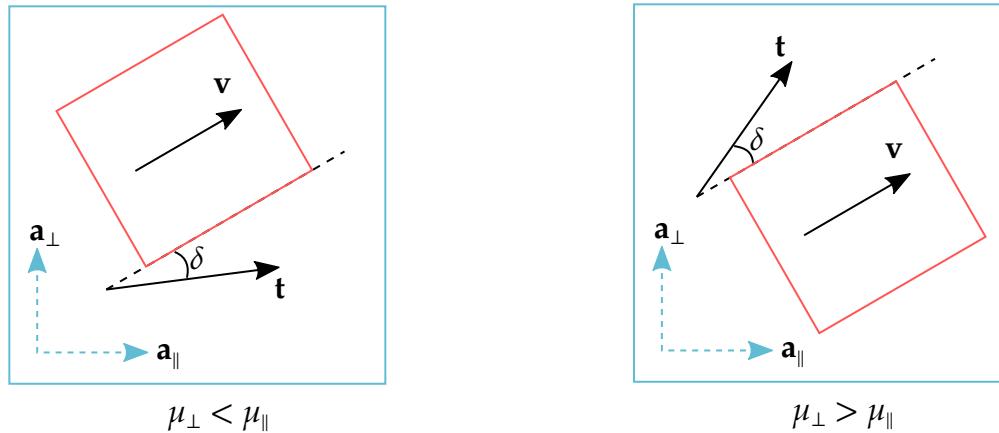
Figure 11-16. Illustration of how direction of sliding differs from the traction direction depending on the coefficients of friction

Optional Card A:

> **NOTE:** If Card 4 is required, then it must go before this card.
> (Card 4 is required for certain contact types.)

Optional Card A.

Card A	1	2	3	4	5	6	7	8
Variable	SOFT	SOFSCL	LCIDAB	MAXPAR	SBOPT	DEPTH	BSORT	FRCFRQ
Type	I	F	I	F	I	I	I	I
Default	0	.1	0	1.025	2	2	10-100	1
Remarks			type a13					

VARIABLE	DESCRIPTION
SOFT	Contact formulation:

        EQ.0: Standard penalty formulation

        EQ.1: Soft constraint penalty formulation. See About SOFT = 1.

        EQ.2: Segment-to-segment contact penalty formulation. See About SOFT = 2.

        EQ.4: Constraint approach for FORMING contacts. This formulation only applies to one-way forming contacts. You should use it when the penalty formulations result in large penetrations. The results, however, are sensitive to damping.

        EQ.6: Special contact algorithm to handle sheet blank edge (deformable) to gage pin (rigid shell) contact during implicit gravity loading. This applies to *CONTACT_FORMING_NODES_TO_SURFACE only. See remarks under About SOFT = 6.

SOFSCL   Scale factor for constraint forces of soft constraint option invoked with SOFT = 1 (default = 0.10). Values greater than 0.5 for single surface contact and 1.0 for one-way treatment are inadmissible.

VARIABLE	DESCRIPTION
LCIDAB	Load curve ID defining airbag thickness as a function of time for type a13 contact (*CONTACT_AIRBAG_SINGLE_SURFACE).
MAXPAR	Maximum parametric coordinate in segment search (values between 1.025 and 1.20 are recommended). This variable applies only to SMP; for MPP, see PARMAX on MPP 1. It does not apply to automatic contacts, except for AUTOMATIC_GENERAL. It also does not apply to segment-to-segment (SOFT = 2) and Mortar segment-to-segment contacts. Larger values can increase the cost. If zero, MAXPAR defaults to 1.025 for most contact options. The exceptions to this are listed in the table below.

Contact Option	Default
SINGLE_EDGE	1.300
SPOTWELD_BEAM_OFFSET	1.050
SPOTWELD_CONSTRAINED_OFFSET	1.006
SPOTWELD_WITH_TORSION	1.006
TIED_NODES_TO_SURFACE	1.070
TIED_NODES_..._CONSTRAINED_OFFSET	1.006
TIED_NODES_..._OFFSET	1.006
TIED_SHELL_..._BEAM_OFFSET	1.050
TIED_SHELL_..._CONSTRAINED_OFFSET	1.006
TIED_SHELL_..._OFFSET	1.006
TIED_SURFACE_TO_SURFACE(_FAILURE)	1.070
TIED_SURFACE_..._CONSTRAINED_OFFSET	1.006
TIED_SURFACE_..._OFFSET	1.006
AUTOMATIC_GENERAL(_...)	1.100

This factor allows for an increase in the size of the segments which may be useful at sharp corners. For the SPOTWELD and ..._OFFSET options larger values can sometimes lead to numerical instabilities; however, a larger value is sometimes necessary to ensure that all nodes of interest are tied.

SBOPT	Segment-to-segment (SOFT = 2) contact options (see Fields used with SOFT = 2 under About SOFT = 2):

VARIABLE	DESCRIPTION
	EQ.1: Pinball edge-edge contact (not recommended)
	EQ.2: Assume planer segments (default)
	EQ.3: Warped segment checking
	EQ.4: Sliding option (not recommended when using DEPTH = 5, 25, and 35)
	EQ.5: Do options 3 and 4 (not recommended when using DEPTH = 5, 25, and 35)
DEPTH	Search depth in automatic contact to check for nodal penetration through the closest contact segments. A value of 1 (one segment) is sufficiently accurate for most crash applications and is much less expensive. By default, this value is 2 (two segments) for improved accuracy, except for *CONTACT_AUTOMATIC_GENERAL which has a default of 3.
	LT.0: |DEPTH| is the load curve ID defining searching depth as a function of time. (not available when SOFT = 2)
	See Fields used with SOFT = 2 under About SOFT = 2 for segment-to-segment contact options controlled by DEPTH.
BSORT	Number of cycles between bucket sorts. Values of 25 and 100 are recommended for contact types 4 (SINGLE_SURFACE) and 13 (AUTOMATIC_SINGLE_SURFACE), respectively. Values of 10-15 are okay for surface-to-surface and node-to-surface contact. If zero, LS-DYNA determines the interval. BSORT applies only to SMP (see BCKT on MPP 1 for MPP) except in the case of SOFT = 2 or for Mortar contact, in which case BSORT applies to both SMP and MPP. For Mortar contact the default is the value associated with NSBCS on *CONTROL_CONTACT.
	LT.0: |BSORT| is the load curve ID defining bucket sorting frequency as a function of time.
FRCFRQ	Number of cycles between contact force updates for penalty contact formulations. This option can provide a significant speed-up of the contact treatment. If used, values exceeding 3 or 4 are dangerous. Considerable care must be exercised when using this option, as this option assumes that contact does not change FRCFRQ cycles.
	EQ.0: FRCFRQ is set to 1, and force calculations are performed each cycle (strongly recommended).

Remarks:

1. **Contact stiffness based on stability.** Setting SOFT = 1 or 2 on optional contact Card A will cause the contact stiffness to be determined based on stability considerations by taking into account the time step and nodal masses. This approach is generally more effective for contact between materials of dissimilar stiffness or dissimilar mesh densities.

2. **Switching between SOFT = 1 and SOFT = 2.** By adding "soft=2to1" on the LS-DYNA execution line, all occurrences of SOFT = 2 in a model will be changed to SOFT = 1. Conversely, by adding "soft=1to2" on the execution line, all occurrences of SOFT = 1 in a model will be changed to SOFT = 2 (other variables related to SOFT = 2, such as SBOPT and DEPTH are left unchanged).

About SOFT = 1:

The soft constraint formulation (SOFT = 1) may be helpful if the material constants of the elements that make up the surfaces in contact have a wide variation in the elastic bulk moduli because the interface stiffness is based on the nodal mass and the global time step size. This method of computing the interface stiffness will typically give a much higher stiffness value than would be obtained by using the bulk modulus. Therefore, this method is the preferred approach when soft foam materials interact with metals.

Note that with this method LS-DYNA computes two contact stiffnesses, $k_{soft=0}$ and k_{cs}. These stiffnesses are the contact stiffness calculated for SOFT = 0 and the stability contact stiffness, respectively. Generally, the maximum of these two stiffnesses is the stiffness for SOFT = 1. See the theory manuals for details about how these contact stiffnesses are calculated.

About SOFT = 2:

What is segment-to-segment contact
Traditionally, to determine contact, the nodes of one surface are checked to see if they penetrate the segments of another surface. The direction of the applied penalty force depends on the direction of the reference segment's normal vector at the contact point. Segment-to-segment contact differs from node-to-segment contact as contact forces are calculated for pairs of segments rather than a single node and a segment. It is supported by a subset of contact types (see When segment-to-segment contact is available). To calculate a penalty force and direction for each pair, the algorithm judges one segment to be the reference segment. Contact forces to oppose penetration are applied to nodes of both segments in the direction of the reference segment's normal vector. Optional frictional forces are in the plane of the reference segment. For ONE_WAY contact types, reference segments are always from SURFB. With all other types, the location and orientation of
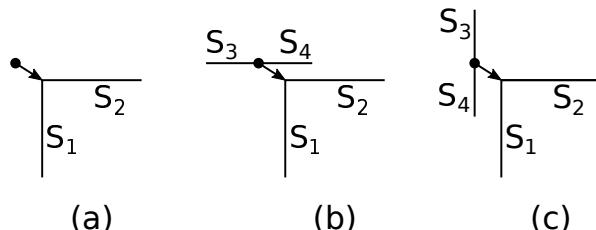
Figure 11-17. (a) illustrates the ambiguity of determing the normal vector for a node to segment contact at a corner since there is no knowledge about the shape of the surface impacting the corner. With segment-based contact, this difficulty is resolved since it looks at segments coming into contact with segments. Thus LS-DYNA can judge the difference between cases (b) and (c).
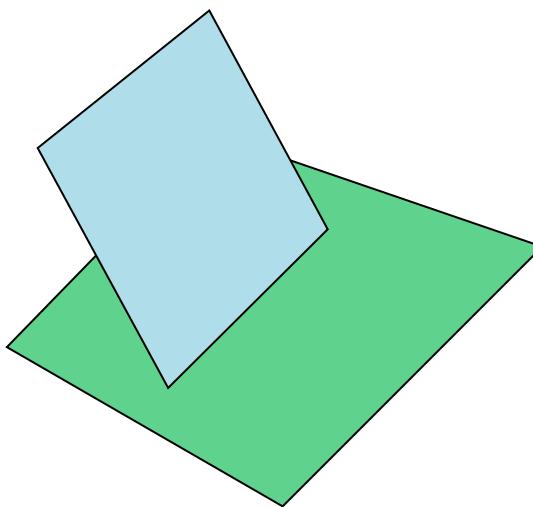
Figure 11-18. Example of two segments coming into contact at a T-intersection. The algorithms for segment-based contact would choose the direction of the contact force as the normal direction to the segment in green.

the segments in a pair are used to choose a reference segment which may be from either SURFA or SURFB.

Why use segment-to-segment contact

In certain situations, node-to-segment contact can lead to undesirable behavior. For instance, if a node of one surface contacts another surface at a corner, determining the reference segment normal is ambiguous (see Figure 11-17). Segment-to-segment contact solves this problem. It looks at segments contacting segments and the directions of the segment normal vectors, thus making the contact direction less ambiguous.

Another problem arises when two segments come into contact at a T-intersection (see Figure 11-18). In traditional contact, the tracked node would be pushed away from the reference segment in the direction normal to the reference segment. This behavior may not always be physical, even in symmetric contact, when both sides are checked for penetration. With segment-to-segment contact, excluding one-way, the contact algorithm

determines the tracked and reference segments. With Figure 11-18 as an example, the algorithm determines that the green segment is the reference segment and the blue the tracked segment. Therefore, the normal direction for the contact force is chosen to be the normal vector of the green segment. Note that for one-way contact, the direction is always the normal vector to the segments in SURFB. Thus, if the segment in blue is in SURFB, unphysical results will occur.

When segment-to-segment contact is available

Segment-to-segment contact is for general shell, solid, and beam element contact. It is available for contact types with SURFACE_TO_SURFACE, ONE_WAY_SURFACE_TO_-SURFACE, and SINGLE_SURFACE in the name that may additionally include AUTO-MATIC, ERODING, and AIRBAG. When the contact type includes AUTOMATIC, the orientation of shell segment normal vectors is automatic. Otherwise, the segment or element orientations are used as input.

Note that for beam elements, the contact surfaces, SURFA and SURFB (if needed), must be defined with parts or part sets. Additionally, only DEPTH values (see Fields used with SOFT = 2 below) that support edge-to-edge contact (see Edge treatment below) are available, namely, DEPTH = 5, 15, 25, 35, and 55. If DEPTH is not set to any of these values, the segment-based contact algorithm internally sets DEPTH to 55 for contact that includes beam elements.

Edge treatment

Edge-to-edge treatment with SOFT = 2 differs from the edge treatments with SOFT = 0 or 1. The edge treatment method for SOFT = 2 looks at penetration for each pair of segments and determines whether the penetrations look like surface or edge penetration. Thus, it is available for solid and shell elements. For SOFT = 0 or 1, edge treatment is only available for shells. SOFT = 2 also applies edge treatment to interior edges if the penetration occurs at a bend in the mesh. For SOFT = 0 or 1, interior edge penetration is not available except for with *CONTACT_AUTOMATIC_GENERAL_INTERIOR when using shell elements (see Remark 2 in General Remarks for details).

By default, we do not enable edge treatment for SOFT = 2. It depends on the setting of DEPTH. See Fields used with SOFT = 2 for which settings of DEPTH enable this feature.
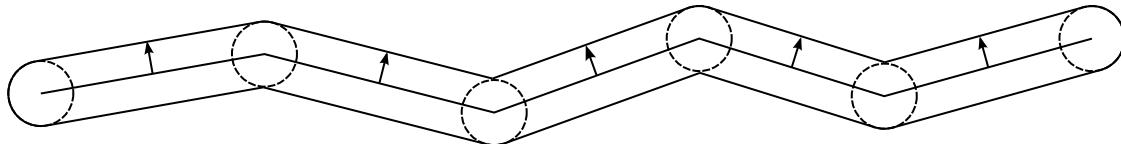
Projecting the contact surface for shell thickness

To account for shell thickness, segment-to-segment contact uses segment-based projection on both the tracked and reference segments as shown in Figure 11-19. With this projection, half the contact thickness is projected normal to the shell midplane in each direction. Note that SHLOFF on Optional Card G and CNTCO on *CONTROL_SHELL can change the location of the reference plane for projection.
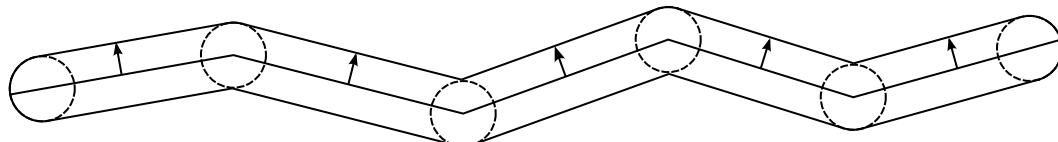
The treatment of external edges depends on SHLEDG on Optional Card D and SRNDE on Optional Card E with SHLEDG having priority. The internal edge treatment depends only on SHLEDG. If SHLEDG is 0 on *CONTROL_CONTACT and on *CONTACT_...,
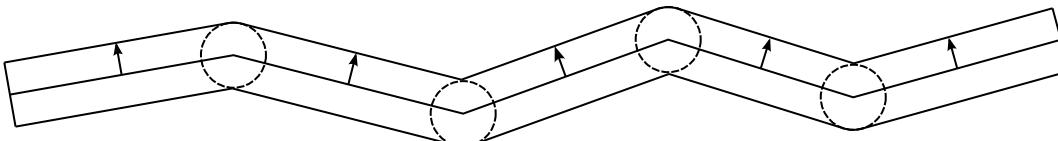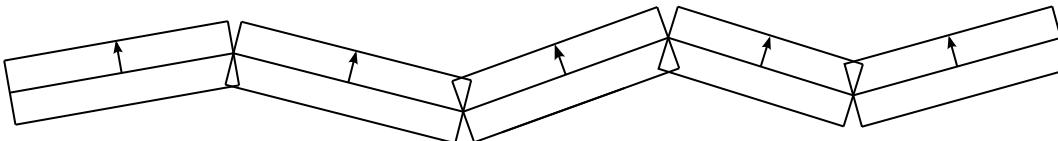
Projecting the contact surface with SHLEDG = 0 **an**d SRNDE = 0

Projecting the contact surface with SHLEDG = 0 **an**d SRNDE = 1

Projecting the contact surface with SHLEDG = 0 **an**d SRNDE = 2

Projecting the contact surface with SHLEDG = 1

Figure 11-20. Projecting the contact surface for SOFT = 2

the shell edges are rounded with a radius of half the contact thickness. However, setting SRNDE = 1 or 2 changes the behavior of the exterior edges (see Optional Card E for a description). If SHLEDG is 1, SRNDE is ignored and both exterior and interior edges are square and flush with the nodes. Note that for values of DEPTH = 23, 33, 35, and 55, the setting of SHLEDG is ignored and the behavior is like SHLEDG = 1 (see Fields used with SOFT = 2 for a discussion of DEPTH). The advantage of SHLEDG = 0 is that it better preserves energy than SHLEDG = 1 when the surfaces meet at an angle. However, as shells become smaller this advantage has become less noticeable, so SHLEDG = 1 works well in most cases.

Fields used with SOFT = 2
Fields that do *not* apply to the supported contact types are *not* used, such as FLANGL on optional Card C. Only the GRPABLE field is supported on the MPP cards. All fields on mandatory Cards 1, 2, and 3 are active, except for VSF and PENCHK. On optional Card A, some parameters have different meanings than they do for the default contact which are discussed in detail below. MAXPAR on optional Card A is not supported. PENMAX, THKOPT, and SHLTHK on optional Card B are also not used (only the ISYM, I2D3D, SLDTHK, and SLDSTF parameters are active). IGNROFF on Card F is also not supported.
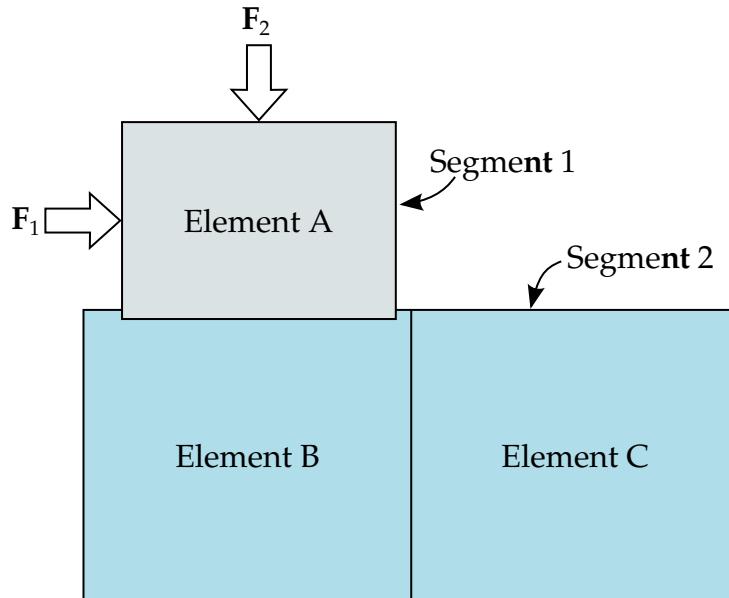
Figure 11-21. Example situation of when to use the sliding option (SBOPT = 3 or 5). Without the sliding option, the regular algorithm causes Segment 2 to become a tracked segment and Segment 1 to become a reference segment because offsets do not apply to solid elements. The sliding option prevents this unphysical contact force between those segments.

The SBOPT parameter on optional Card A controls several options. We do not recommend setting DEPTH = 1 for pinball edge-to-edge checking and only include it for backward compatibility. For edge-to-edge checking, we recommend setting DEPTH = 5 instead (see below). The warped segment option more accurately checks for penetration of warped surfaces.

The sliding option (SBOPT = 4 or 5) uses neighbor segment information to improve sliding behavior. It is useful for preventing segments from incorrectly catching nodes on a sliding surface. However, we do not recommend using it with DEPTH = 5, 25, and 35 because it can interfere with edge-to-edge checking and lead to incorrect behavior. Figure 11-21 illustrates an example situation ideal for using the sliding option. In this example, a block modeled with a solid element slides across a surface consisting of two solid elements. Because segments on solid elements do *not* have offsets, Element A must penetrate Element B for contact forces to apply. With the default algorithm, Segment 2 on Element C becomes a tracked segment, and Segment 1 on Element A becomes a reference segment as Element A moves to the right. Thus, the regular contact algorithm applies a nonphysical force in the normal direction of Segment 1. The sliding option prevents this unphysical contact force because it looks at the segments neighboring Segment 2. The orientation of the segment to the left of Segment 2 causes the algorithm to not apply a contact force in that direction.

The DEPTH parameter controls several additional options for segment-to-segment contact.

1. **DEPTH = 2 (default; but not recommended).** Surface penetrations measured at nodes are checked.

2. **DEPTH = 3.** Surface penetration is also measured at the edge. This option is more accurate than DEPTH = 2 and is good for a wide variety of simulations but does not check for edge-to-edge penetration.

3. **DEPTH = 5.** Both surface penetrations and edge-to-edge penetrations are checked.

4. **DEPTH = 13.** The penetration checking is the same as for DEPTH = 3, but the code has been tuned to better conserve energy.

5. **DEPTH = 15.** Behavior is the same as DEPTH = 5 from versions R4.2 and earlier. This option enables backward compatibility of results.

6. **DEPTH = 23.** The penetration checking is similar to DEPTH = 3 but uses different methods to try to improve robustness.

7. **DEPTH = 25 or 35.** The penetration checking is similar to DEPTH = 5 but uses different methods to try to improve robustness.

8. **DEPTH = 45.** The splitting pinball method [Belytschko and Yeh, 1993] is used. This method is more accurate at the cost of more CPU time. It is recommended when modeling complex contacts between parts comprised of shells. It does not apply to solid or thick shell parts, but such parts can be coated with null shells to make DEPTH = 45 available. It also includes edge-to-edge checking.

9. **DEPTH = 55.** The penetration checking is similar to DEPTH = 25 and 35, but it is robust in edge-to-edge checking. Thus, penetration is less likely.

10. **DEPTH = 1 or 4.** Airbag contact has two additional options, DEPTH = 1 and 4. DEPTH = 4 activates additional airbag logic that uses neighbor segment information when judging if the contact is between interior or exterior airbag surfaces. This option is not recommended and is maintained only for backward compatibility. Setting DEPTH = 1 suppresses all airbag logic.

About SOFT = 6:

SOFT = 6 contact addresses contact issues in situations where blank gage pins are narrow or small and blank meshes are coarse (Figure 11-22 left), leading to missing contact in some cases. This feature applies to gravity loading and forming of a sheet blank with an adaptive mesh. It can *only* be used with *CONTACT_FORMING_NODES_TO_SURFACE. In addition, the variable ORIENT in *CONTROL_CONTACT must be set to "4". Currently this feature is available in double precision, SMP only.
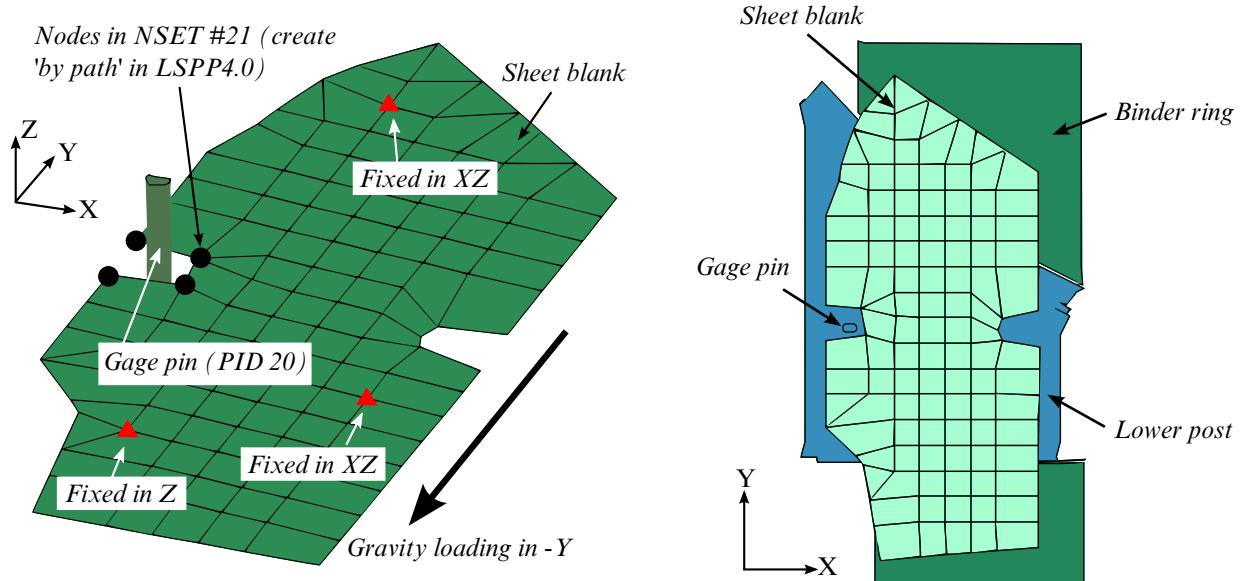
Figure 11-22. Illustrative/test model for SOFT = 6 (left) and initial blank position.

1. **SURFA type.** SURFA must be a node set that includes the nodes along the entire blank edge or the portion of the blank edge that will be in contact with gage pins (see Figure 11-22 left) or the part ID (not part set ID) of the blank. In the node set case, the nodes in the node set must be listed in consecutive order, as defined *by path* in the LS-PrePost application, under *Model → CreEnt → Cre → Set Data → *SET_NODE*. Note that the blank edge and the gage pins do not have any thickness.

2. **Example.** In the partial keyword example below, node set ID 21 (SUR-FATYP = 4) is in contact with the gage pin of part set ID 20. As shown in Figure 11-22 (left), with the boundary condition applied, a blank with a very coarse mesh is loaded with a body force. The left notch is anticipated to be in contact with the gage pins. The initial position (top view) of the test model is shown in Figure 11-22 (right) and the final gravity-loaded blank positions are shown in Figure 11-23 (left) without SOFT = 6, and in Figure 11-23 (middle and right) with SOFT = 6. Without SOFT = 6, the contact between the blank edge and the pin is missed completely.
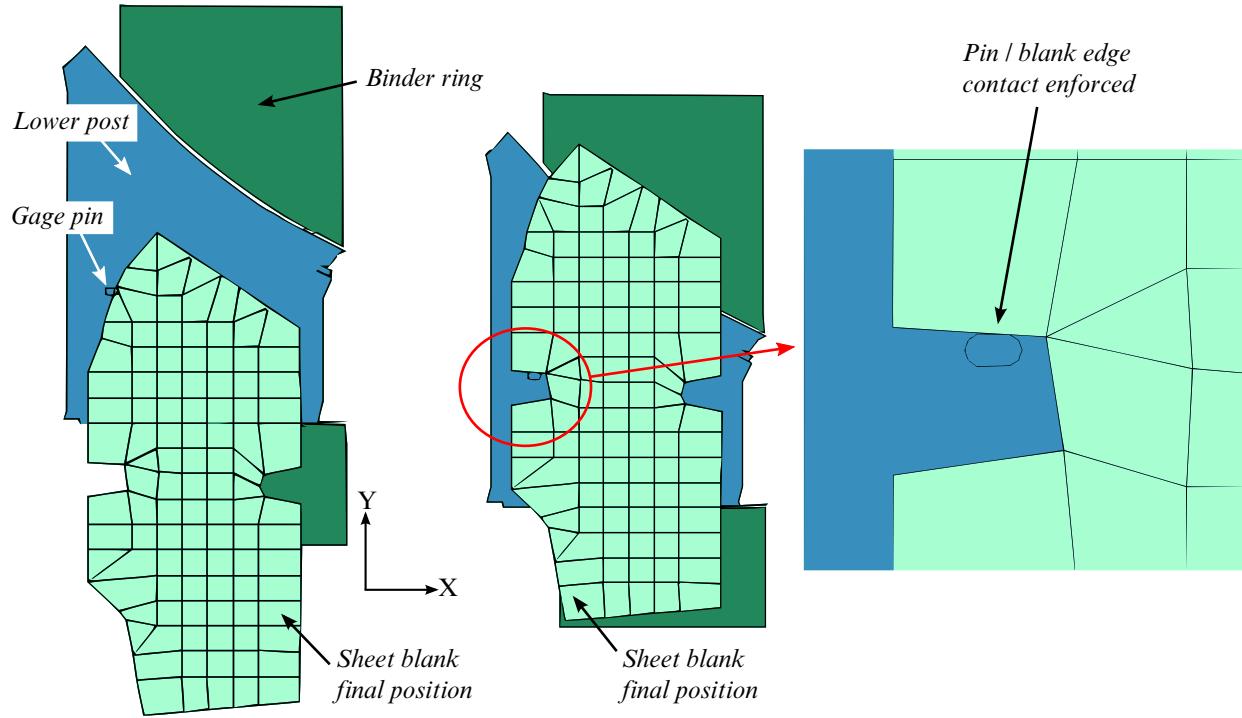
```
*CONTROL_TERMINATION
1.0
*CONTROL_IMPLICIT_FORMING
1
*CONTROL_IMPLICIT_GENERAL
1,0.2
*CONTROL_IMPLICIT_NONLINEAR
$  NSLOLVR    ILIMIT    MAXREF    DCTOL    ECTOL    RCTOL    LSTOL
        2         1      1200    0.000     0.00        0
$   dnorm   divflag   inistif
        0         2         0        1                 1
*SET_NODE_LIST
```

Gravity loading results without using   Gravity loading results using SOFT=6;
SOFT=6; Pin/blank edge contact missed.   Pin/blank edge contact sucessful.

Figure 11-23. Final blank position without (left) and with (right) SOFT = 6.

```
$ blank edge node set around the gage pin
21
1341,1342,1343,1344
*SET_PART_LIST
$ gage pin
20
20
*SET_PART_LIST
$ blank
13
13
*CONTACT_FORMING_NODES_TO_SURFACE
$     SURFA      SURFB   SURFATYP   SURFBTYP    SABOXID    SBBOXID       SAPR       SBPR
         21         20          4          2
$        FS         FD         DC          V        VDC     PENCHK         BT         DT
      0.125                                         20.          4
$      SFSA       SFSB       SAST       SBST      SFSAT      SFSBT        FSF        VSF

$      SOFT
          6
```

3. **Coarse meshes and initial contact penetration.** Sometimes a coarse blank
    mesh causes an initial contact penetration between the blank gaging hole edge
    and the gaging pin mesh. Starting with R11.0, this contact algorithm creates ad-
    ditional nodes on the blank along the hole edge and then moves the nodes to
    clear the initial penetration with the gage pin, as shown in Figure 11-24.
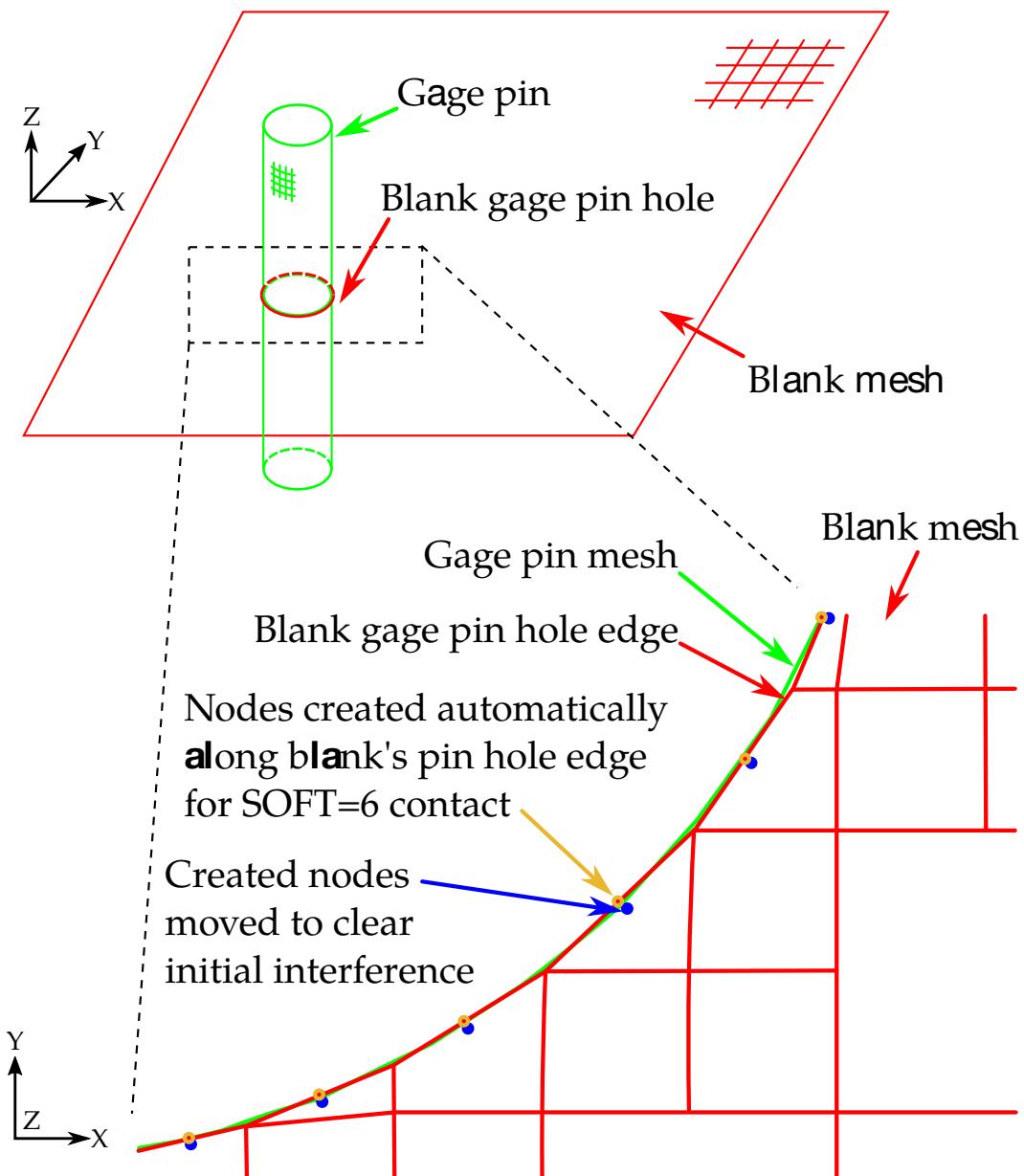
Figure 11-24. Improvement to SOFT = 6 for a coarse blank mesh

Optional Card B:

> **NOTE:** If using Optional Card B, then you must include Optional Card A. (Optional Card A may be a blank line.)

Optional Card B.

Card B	1	2	3	4	5	6	7	8
Variable	PENMAX	THKOPT	SHLTHK	SNLOG	ISYM	I2D3D	SLDTHK	SLDSTF
Type	F	I	I	I	I	I	F	F
Default	↓	0	0	0	0	0	0.0	0.0
Remarks		Old types 3, 5, 10	Old types 3, 5, 10					

VARIABLE	DESCRIPTION

PENMAX For old types 3, 5, 8, 9, 10 (see Mapping of *CONTACT keyword option to "contact type" in d3hsp at the end of General Remarks) and Mortar contact, PENMAX is the maximum penetration distance. For contact types a3, a5, a10, 13, 15, and 26, see Table 11-2 for how the maximum penetration distance is determined.

 EQ.0.0: For old type contacts 3, 5, and 10, use small penetration search and value calculated from thickness and XPENE; see *CONTROL_CONTACT.

 EQ.0.0: For contact types a3, a5, and a10, the default value is 0.4 for shells and solids in SMP, 0.5 for solids in MPP, and 0.8 for shells in MPP.

 EQ.0.0: For contact types, 13 and 15, the default is 0.4 for shells, 0.4 for solids in SMP, and 0.5 for solids in MPP.

 EQ.0.0: For contact type 26, the default value is 10.

 EQ.0.0: For Mortar contact, the default is a characteristic size of the element; see Theory manual.

THKOPT Thickness option for contact types 3, 5, and 10 (non-automatic surface-to-surface including one-way and non-automatic nodes-to-

VARIABLE	DESCRIPTION
	surface):
	EQ.0: Default is taken from the control card; see SHLTHK on *CONTROL_CONTACT.
	EQ.1: Thickness offsets are included.
	EQ.2: Thickness offsets are not included (old way).
SHLTHK	Define if and only if THKOPT above equals 1. Shell thickness is considered in non-automatic surface-to-surface and non-automatic node-to-surface type contact options, where options 1 and 2 below activate the new contact algorithms. The thickness offsets are always included in single surface, constraint method, and automatic contact types.
	EQ.0: Thickness is not considered.
	EQ.1: Thickness is considered, but rigid bodies are excluded.
	EQ.2: Thickness is considered including rigid bodies.
SNLOG	For SOFT = 0 or 1, SNLOG controls the shooting node logic in thickness offset contact. With the shooting node logic enabled, the first cycle that a tracked node penetrates a reference segment, that node is moved back to the reference surface without applying any contact force.
	EQ.0: Logic is enabled (default).
	EQ.1: Logic is skipped (sometimes recommended for metal forming calculations or for contact involving foam materials).
	For SOFT = 2, SNLOG controls the thick segment check (see Remark 1):
	EQ.0: Do a thick segment check but do not report the results.
	EQ.1: Do a thick segment check but do not report the results.
	EQ.2: Do a thick segment check and report the results to the d3hsp file.
	EQ.3: Do not do a thick segment check.
ISYM	Symmetry plane option:
	EQ.0: Off
	EQ.1: Do not include faces with normal boundary constraints (such as segments of brick elements on a symmetry

VARIABLE	DESCRIPTION
	plane).
	This option is important to retain the correct boundary conditions in the model with symmetry. For ERODING contacts this option may also be defined on Card 4.
I2D3D	Segment searching option:
	EQ.0: Search 2D elements (shells) before 3D elements (solids, thick shells) when locating segments.
	EQ.1: Search 3D (solids, thick shells) elements before 2D elements (shells) when locating segments.
SLDTHK	Optional solid element thickness. For non-Mortar contacts, a non-zero positive value will activate the contact thickness offsets in the contact algorithms where offsets apply. The contact treatment will then be equivalent to the case where null shell elements are used to cover the brick elements. The contact stiffness parameter below, SLDSTF, may also be used to override the default value.
	For Mortar contacts, SLDTHK is the offset from the solid element surface in the direction of the normal to the point where contact acts. Therefore, in this case a negative value for SLDTHK is well defined. SLDSTF is ignored for Mortar contact.
SLDSTF	Optional solid element stiffness. A nonzero positive value overrides the bulk modulus taken from the material model referenced by the solid element. For segmen-to-segment contact (SOFT = 2), SLDSTF replaces the stiffness used in the penalty equation. This parameter does not apply to Mortar contacts.

Remarks:

1. **SNLOG.** For segment-to-segment contact (invoked by setting SOFT = 2 on Card A), SNLOG controls the thick segment check. The thick segment check has always been done during initialization of single surface contact. It prevents possible unstable contact behavior that can occur when shell segments have thick offsets and segment dimensions are short relative to their thickness. When a pair of segments is found to be near each other when measured along the surface, and the segments in the pair are thick enough that their offsets could contact at a bend in the mesh, the pair is added to a list of pairs which will be skipped over during the simulation. Short and thick segments typically have enough stiffness that skipping contact between these pairs does not compromise the solution.

However, setting SNLOG = 3 will switch off this check. Setting SNLOG = 2 will leave the check active but will output to the d3hsp file a report of the pairs of elements which will not be checked for contact. Setting SNLOG = 0 or 1 will leave the check active and will not output a report.

Optional Card C:

> **NOTE:** If using Optional Card C, then you must define Optional Cards A and B. (Optional Cards A and B may be blank lines.)

Optional Card C.

Card C	1	2	3	4	5	6	7	8
Variable	IGAP	IGNORE	DPRFAC / MPAR1	DTSTIF / MPAR2	EDGEK		FLANGL	CID_RCF
Type	I	I	F	F	F		F	I
Default	1	0	0.0	0.0	0.0		0.0	0
Remarks		3	1	2	4			

VARIABLE	DESCRIPTION
IGAP	For Mortar contacts, IGAP is used to progressively increase contact stiffness for large penetrations, or use a linear relationship between penetration and contact pressure; see Remark 14 in the General Remarks section. For other contacts, IGAP can be used to improve implicit convergence at the expense of (1) creating some sticking if parts attempt to separate and (2) possibly under-reporting the contact force magnitude in the output files rcforc and ncforc. (Implicit only.)

      LT.0: Like IGAP = 1 except the maximum distance between contact surfaces at which stickiness is on is scaled by IGAP/10.

      EQ.1: Apply method to improve convergence (default).

      EQ.2: Do not apply method.

      GT.2: Set IGAP = 1 for first IGAP − 2 converged equilibrium states, then set IGAP = 2.

IGNORE	Ignore initial penetrations for the *CONTACT_AUTOMATIC options:

      LT.0: Applies only to the Mortar contact. When less than zero, the behavior is the same as for |IGNORE|, but contact between segments belonging to the same part is ignored.

VARIABLE	DESCRIPTION

The main purpose of this option is to avoid spurious contact detections that otherwise could result for complicated geometries in a single surface contact, typically, when eliminating initial penetrations by interference. See IGNORE = 3 and IGNORE = 4.

EQ.0:   Take the default value from the fourth card of the *CONTROL_CONTACT input.

EQ.1:   Allow initial penetrations to exist by tracking the initial penetrations.

EQ.2:   Allow initial penetrations to exist by tracking the initial penetrations. However, penetration warning messages are printed with the original coordinates, and the recommended coordinates of each penetrating node are given. For Mortar contact, this is the default (see Remark 14 in the General Remarks section).

EQ.3:   Applies only to the Mortar contact. With this option initial penetrations are eliminated between time zero and the time specified by MPAR1. Intended for small initial penetrations. See Remark 14 in the General Remarks section.

EQ.4:   Applies only to the Mortar contact. With this option initial penetrations are eliminated between time zero and the time specified by MPAR1. In addition, a maximum penetration distance can be given as MPAR2, intended for large initial penetrations. See Remark 14 in the General Remarks section.

DPRFAC   Depth of penetration reduction factor for SOFT = 2 contact (see Remark 1 below):

    EQ.0.0:   Initial penetrations are always ignored.

    GT.0.0.and.LT.1.0:   Initial penetrations are penalized over time.

    GE.1.0:   DPRFAC is a set ID used to limit the scope of DPRFAC.

    LE.-1.0: |DPRFAC| is the load curve ID defining DPRFAC as a function of time.

DTSTIF   Time step used in stiffness calculation for SOFT = 1 and SOFT = 2 contact (see Remark 2 below).

    EQ.0.0:   Use the initial value that is used for time integration.

VARIABLE	DESCRIPTION
	GT.0.0: Use the value specified.
	GT.-1.0.and.LT.-0.01: Use a moving average of the solution time step (SOFT = 2 only).
	LE.-1.0: |DTSTIF| is the ID of a curve that defines DTSTIF as a function of time.
MPAR1	This field only applies to Mortar contact. If IGNORE = 2, MPAR1 corresponds to the initial contact pressure in interfaces with initial penetrations. For IGNORE = 3 and 4 it corresponds to the time of closure of initial penetrations. See Remark 14 in the General Remarks section.
	LE.-1.0: |MPAR1| is the ID of a curve defining the relative penetration reduction as function of time. This is available for the IGNORE = 3 and 4 cases.
MPAR2	For Mortar contact and IGNORE = 4, MPAR2 corresponds to a penetration depth that must be at least the penetration occurring in the contact interface. See Remark 14 in the General Remarks section.
EDGEK	Scale factor for penalty stiffness of edge-to-edge contact when SOFT = 2 and DEPTH = 5, 15, 25, or 35 (see Remark 4):
	EQ.0.0: Use the default penalty stiffness.
	GT.0.0: Scale the stiffness by EDGEK.
FLANGL	Angle tolerance in radians for feature lines option in smooth contact:
	EQ.0.0: No feature line is considered for surface fitting in smooth contact.
	GT.0.0: Any edge with an angle between two contact segments bigger than this angle will be treated as feature line during surface fitting in smooth contact.
CID_RCF	Coordinate system ID to output rcforc force resultants and ncforc data in a local system.

Remarks:

1. **DPRFAC.** DPRFAC is used only by segment-to-segment contact (SOFT = 2); see About SOFT = 2 under optional Card A for more details about segment-to-segment contact. By default, SOFT = 2 contact measures the initial penetration between segment pairs that are found to be in contact and subtracts the measured

value from the total penetration for as long as a pair of segments remains in contact. The penalty force is proportional to this modified value. This approach prevents shooting nodes but may allow unacceptable penetration. DPRFAC can be used to decrease the measured value over time until the full penetration is penalized. Setting DPRFAC = 0.01 will cause ~1% reduction in the measured value each cycle. We recommend a small value, such as 0.001. DPRFAC does not apply to initial penetrations at the start of the calculation, only those that are measured at later times. This prevents nonphysical movement and energy growth at the start of the calculation.

The anticipated application for the load curve option is to reduce the initial penetrations at the end of a calculation if the final geometry is to be used for a subsequent analysis. To achieve this, the load curve should have a y-value of zero until a time near the end of the analysis and then ramp up to a positive value, such as 0.01, near the end of the analysis.

If the value of DPRFAC is greater or equal to 1.0, then the nearest integer will be used as a set ID and DPRFAC penetration reduction will only be done for pairs of segments that are both within that set. The set can be defined by either *SET_-SHELL, *SET_SEGMENT, or *SET_PART. A search for a set with that set ID will be made in that order, and the search will stop when a matching set is found. The value of the first default attribute of that set (DA1), will be used as the DPRFAC value and should be either a small positive value, or a negative value to reference a curve.

2. **DTSTIF.** DTSTIF is used only by the SOFT = 1 and SOFT = 2 contact options; see About SOFT = 1 and About SOFT = 2 under optional Card A for more details.. By default, when the SOFT option is active, the contact uses the initial solution time step to scale the contact stiffness. If the user sets DTSTIF to a nonzero value, the inputted value will be used. Because the square of the time step appears in the denominator of the stiffness calculation, a DTSTIF value larger than the initial solution time step reduces the contact stiffness and a smaller value increases the stiffness. This option could be used when one component of a larger model has been analyzed independently and validated. When the component is inserted into the larger model, the larger model may run at a smaller time step due to higher mesh frequencies. In the full model analysis, setting DTSTIF equal to the component analysis time step for the contact interface that treats the component will cause consistent contact stiffness between the analyses.

    The load curve option allows contact stiffness to be a function of time. This should be done with care as energy will not be conserved. A special case of the load curve option is when |DTSTIF| = LCTM on *CONTROL_CONTACT. LCTM sets an upper bound on the solution time step. For |DTSTIF| = LCTM, the contact stiffness time step value will track LCTM whenever the LCTM value is less than the initial solution time step. If the LCTM value is greater, the initial

solution time step is used. This option could be used to stiffen the contact at the end of an analysis. To achieve this, the LCTM curve should be defined such that it is larger than the solution time step until near the end of the analysis. Then the LCTM curve should ramp down below the solution time step causing it to decrease and the contact to stiffen. A load curve value of 0.1 of the calculated solution time step will cause penetrations to reduce by about 99%. To prevent shooting nodes, the rate at which the contact stiffness increases is automatically limited. Therefore, to achieve 99% reduction, the solution should be run for perhaps 1000 cycles with a small time step.

For segment-to-segment contact (SOFT = 2), setting DTSTIF less than or equal to -0.01 and greater than -1.0, causes the contact stiffness to be updated based on the current solution time step. Varying the contact stiffness during a simulation can cause energy growth. Thus, this option should be used with care when extra stiffness is needed to prevent penetration and the solution time step has dropped below the initial. Because quick changes in contact stiffness can cause shooting nodes, using a moving average of the solution time step can prevent this. The value of DTSTIF determines the number of terms in the moving average where $n = 100 \times (\text{-DTSTIF})$ such that $n = 1$ for DTSTIF = -0.01 and $n = 100$ for DTSTIF = -0.999. Setting DTSTIF = -1.0 triggers the load curve option described in the previous paragraph, so DTSTIF cannot be smaller than -0.999 for this option.

3. **IGNORE with SOFT = 2.** When SOFT = 2 on Optional Card A, treatment of initial penetrations is always like IGNORE = 1 in that initial penetrations are ignored when calculating penalty forces. If SOFT = 2 and IGNORE = 2, then a report of initial penetrations will be written to the messag file(s) in the first cycle.

4. **EDGEK.** When SOFT = 2 and DEPTH = 5, 15, 25, or 35 on Optional Card A, the EDGEK parameter will scale the contact penalty stiffness when contact is between segment edges if EDGEK > 0.0. This is true for both shell segments and solid element segments. Surface contact stiffness is unaffected by EDGEK.

Optional Card D:

> **NOTE:** If using Optional Card D, then you must include Optional Cards A, B, and C. (Optional Cards A, B, and C may be blank lines.)

Optional Card D.

Card D	1	2	3	4	5	6	7	8
Variable	Q2TRI	DTPCHK	SFNBR	FNLSCL	DNLSCL	TCSO	TIEDID	SHLEDG
Type	I	F	F	F	F	I	I	I
Default	0	0.0	0.0	0.0	0.0	0	0	0
Remarks	1	2	3	5	5		4	

VARIABLE	DESCRIPTION

Q2TRI   Flag to split quadrilateral contact segments into two triangles (only available when SOFT = 2):

     EQ.0: Off (default)

     EQ.1: On for all SURFA shell segments

     EQ.2: On for all SURFB shell segments

     EQ.3: On for all shell segments

     EQ.4: On for all shell segments of material type 34

DTPCHK   Time interval between shell penetration reports (only available for SOFT = 2):

     EQ.0.0: Off (default)

     GT.0.0: Check and report segment penetrations at time intervals equal to DTPCHK

     LT.0.0: Check and report segment penetrations at time intervals equal to |DTPCHK|. In addition, the calculation stops with an error at $t = 0$ if any intersections are initially present.

SFNBR   Scale factor for neighbor segment contact (only available when

VARIABLE	DESCRIPTION
	SOFT = 2)
	EQ.0.0: Off (default)
	GT.0.0: Check neighbor segments for contact.
	LT.0.0: Neighbor segment checking with improved energy balance when |SFNBR| < 1000. |SFNBR| ≥ 1000 activates a split-pinball based neighbor contact with a penalty force scale factor of |SFNBR + 1000|. For example, the force scale factor used is 2 when SFNBR = -1002.
FNLSCL	Scale factor for nonlinear force scaling, f. See Remark 5. This field only applies to segment-to-segment contact invoked with SOFT = 2 on Optional Card A (see About SOFT = 2 under optional Card A).
DNLSCL	Distance for nonlinear force scaling, d. See Remark 5. This field only applies to segment-to-segment contact invoked with SOFT = 2 on Optional Card A (see About SOFT = 2 under optional Card A).
TCSO	Flag to consider only contact segments (not all attached elements) when computing the contact thickness for a node or segment (for SURFACE_TO_SURFACE contact and shell elements only):
	EQ.0: Off (default)
	EQ.1: Only consider segments in the contact definition
TIEDID	Flag for incremental displacement update for tied contacts (see Remark 4 below):
	EQ.0: Off (default)
	EQ.1: On
SHLEDG	Flag for assuming edge shape for shells when measuring penetration. This is available for segment-to-segment contact (SOFT = 2).
	EQ.0: Default to SHLEDG on *CONTROL_CONTACT
	EQ.1: Shell edges are assumed to be square and are flush with the nodes.
	EQ.2: Shell edges are assumed to be round with a radius equal to half the shell thickness. The edge centers lie on the lines between the segment nodes and extend outward by the radius.

VARIABLE	DESCRIPTION
	Note that for DEPTH values of 23, 33, 35, and 55, the setting of SHLEDG is ignored and the behavior is like SHLEDG = 1.

Remarks:

1. **Q2TRI.** Setting Q2TRI to a nonzero value causes quadrilateral shell segments to be split into two triangles. Only the contact segments are split. The elements are not changed. This option is only available for segment-to-segment contact which is activated by setting SOFT = 2. See About SOFT = 2 under optional Card A for more details about segment-based contact.

2. **DTPCHK (penetration check).** Setting DTPCHK to a positive value causes a penetration check to be done periodically with the interval equal to DTPCHK. The check looks for shell segments that are penetrating the mid-plane of another shell segment. It does not report penetration of thickness offsets. The penetrating pairs are reported to the messag file or files for MPP. If at least one penetration is found, the total number of pairs is reported to the screen output. This option is only available for segment-to-segment contact which is activated by setting SOFT = 2. See About SOFT = 2 under optional Card A for more details about segment-to-segment contact.

3. **SFNBR.** SFNBR is a scale factor for optional neighbor segment contact checking. This is available only in segment-to-segment (SOFT = 2) contact (see About SOFT = 2 under optional Card A for more details about segment-to-segment contact). This is helpful when a mesh folds, such as during compression folding of an airbag. Only shell element segments are checked. Setting SFNBR to a negative value modifies the neighbor checking to improve energy balance. When used, a value between -0.5 and -1.0 is recommended.

4. **Round off in OFFSET and TIEBREAK.** There have been several issues with tied OFFSET contacts and AUTOMATIC_TIEBREAK contacts with offsets creating numerical round-off noise in stationary parts. By computing the interface displacements incrementally rather than using total displacements, the round-off errors that occur using single precision are eliminated. The incremental approach (TIEDID = 1) is available for the following contact types:

    TIED_SURFACE_TO_SURFACE_OFFSET

    TIED_NODES_TO_SURFACE_OFFSET

    TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET

    TIED_SHELL_EDGE_TO_SURFACE_OFFSET

AUTOMATIC_..._TIEBREAK

AUTOMATIC_SINGLE_SURFACE_TIED

5.   **FNLSCL and DNLSCL.** FNLSCL = f and DNLSCL = d invoke alternative contact stiffness scaling options. These variables apply only when SOFT = 2 (see About SOFT = 2 under optional Card A for more details about segment-to-segment contact).

When FNLSCL > 0 and DNLSCL > 0, this feature scales the stiffness by the depth of penetration to provide smoother initial contact and a larger contact force as the depth of penetration exceeds DNLSCL. The stiffness k is scaled by the relation

$$k \rightarrow kf\sqrt{\frac{\delta}{d}} \, ,$$

where δ is the depth of penetration, making the penalty force proportional to the 3/2 power of the penetration depth. Adding a small amount of surface damping (such as VDC = 10) is advised with this feature.

For FNLSCL < 0, and DNLSCL > 0, an alternative stiffness scaling scheme is used,

$$k \rightarrow k\left[\frac{0.01|f|A_o}{d(d-\delta)}\right] \, .$$

Here A_0 is the overlap area of segments in contact. For δ greater than $0.9d$, the stiffness is extrapolated to prevent it from going to infinity.

For FNLSCL > 0, and DNLSCL = 0, the contact is scaled by the overlap area:

$$k \rightarrow kf\left(\frac{A_o}{A_m}\right) \, .$$

Here A_m is the mean area of all the contact segments in the contact interface. This third option can improve friction behavior, particularly when the FS = 2 option is used.

Optional Card E:

> **NOTE:** If using Optional Card E, then you must define Optional Cards A, B, C and D. (Optional Cards A, B, C and D may be blank lines.)

Optional Card E.

Card E	1	2	3	4	5	6	7	8
Variable	SHAREC	CPARM8	IPBACK	SRNDE	FRICSF	ICOR	FTORQ	REGION
Type	I	I	I	I	F	I	I	I
Default	0	0	0	0	1.	0	0	0
Remarks	1		2	3	5			4

VARIABLE	DESCRIPTION

SHAREC   Shared constraint flag (only available for segment-to-segment contact which is activated with SOFT = 2):

    EQ.0: Segments that share constraints are not checked for contact.

    EQ.1: Segments that share constraints are checked for contact.

CPARM8   This variable is similar to CPARM8 in *CONTACT_…_MPP but applies to SMP and not to MPP. CPARM8 for SMP only controls the treatment of spot weld beams in *CONTACT_AUTOMATIC_-GENERAL.

    EQ.0: Spot weld (type 9) beams are not considered in the contact even if included in SURFA.

    EQ.2: Spot weld (type 9) beams are considered in the contact if included in SURFA.

IPBACK   If set to a nonzero value, create a "backup" penalty tied contact for this interface. This option applies to constrained tied contacts only. See Remark 2.

SRNDE   Flag for non-extended exterior shell segment edges. See Remark 3 below for further information and restrictions.

VARIABLE	DESCRIPTION
	EQ.0: Exterior shell edges have their usual treatment where the contact surface extends beyond the shell edge.
	EQ.1: The contact surface is rounded at exterior shell edges but does not extend beyond the shell edges.
	EQ.2: The shell edges are square.
FRICSF	Scale factor for frictional stiffness (available for SOFT = 2 only).
ICOR	If set to a nonzero value, VDC is the coefficient of restitution expressed as a percentage. First and foremost, this option is intended only for contact between independent, unconstrained rigid bodies, and its application to any other contact situation will not render the desired effect. When SOFT = 0 or 1, ICOR applies to AUTOMATIC_NODES_TO_SURFACE, AUTOMATIC_SURFACE_TO_SURFACE and AUTOMATIC_SINGLE_SURFACE. ICOR applies to all SOFT = 2 contacts. Two proprietary implementations are available, depending on the sign of ICOR. A positive ICOR should give satisfactory results for most applications.
FTORQ	FTORQ controls the transmittal of moments across the contact interface for some contact types. The moments addressed by FTORQ are those due to contact friction (or tangential forces in the case of tied/tiebreak contact).
	For MPP with SOFT ≠ 2, AUTOMATIC_GENERAL, AUTOMATIC_SURFACE_TO_SURFACE, AUTOMATIC_SINGLE_SURFACE, and AUTOMATIC_NODES_TO_SURFACE are supported for nonzero FTORQ. For AUTOMATIC_GENERAL, only FTORQ = 1 works and causes the transmission of moments due to beam-to-beam friction. Setting FTORQ = 1 or 2 for AUTOMATIC_SURFACE_TO_SURFACE, AUTOMATIC_SINGLE_SURFACE, and AUTOMATIC_NODES_TO_SURFACE leads to transmitting a moment due to tied/tiebreak, contact friction, or damping. In the case of AUTOMATIC_SINGLE_SURFACE, FTORQ = 1 differs only from FTORQ = 2 in that nodal moments are distributed to the nodes of the contact segment if the segment belongs to a shell element; otherwise, the moment is transmitted as self-balanced nodal forces rather than as nodal moments. For AUTOMATIC_SURFACE_TO_SURFACE and AUTOMATIC_NODES_TO_SURFACE, FTORQ = 1 is the same as FTORQ = 2.
	For SMP with SOFT ≠, 2 FTORQ = 1 and 2 are supported for AU-

VARIABLE	DESCRIPTION
	TOMATIC_SURFACE_TO_SURFACE_TIEBREAK and AUTO-MATIC_ONE_WAY_SURFACE_TO_SURFACE_TIEBREAK. FTORQ = 2 is supported for AUTOMATIC_SURFACE_TO_SUR-FACE and AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE. SOFT = 2 contacts support FTORQ = 2. FTORQ = 1 is not supported.
REGION	The ID of a *DEFINE_REGION which will delimit the volume of space where this contact is active. See Remark 4 below.

Remarks:

1. **SHAREC.** The SHAREC flag is a segment-to-segment contact (SOFT = 2) option that allows contact checking of segment pairs that share a multi-point constraint or rigid body. Sharing a constraint is defined as having at least one node of each segment that belongs to the same constraint.

2. **IPBACK.** The IPBACK flag is only applicable to constraint-based tied contacts (TIED with no options, or with CONSTRAINED_OFFSET). An identical penalty-based contact is generated with type OFFSET, except in the case of SHELL_-EDGE constrained contact which generates a BEAM_OFFSET type. The ID of the generated interface will be set to the ID of the original interface plus 1 if that ID is available, otherwise one more than the maximum used contact ID. For nodes successfully tied by the constraint interface, the extra penalty tying should not cause problems, but nodes dropped from the constraint interface due to rigid body or other conflicting constraints will be handled by the penalty contact. In MPP, nodes successfully tied by the constraint interface are skipped during the penalty contact phase.

3. **SRNDE.** Contact at shell edges is, by default, treated by adding cylindrical caps along the free edges, with the radius of the cylinder equal to half the thickness of the segment. This has the side effect of extending the segment at the free edges, which can cause problems. Setting SRNDE = 1 "rounds over" the (through the thickness) corners of the element instead of extending it. The edges of the segment are still rounded, but the overall size of the contact area is not increased. The effect is as if the free edge of the segment was moved in toward the segment by a distance equal to half the segment thickness, and then the old cylindrical treatment was performed. Setting SRNDE = 2 will treat the shell edges as square, with no extension. This variable has no effect on shell-edge-to-shell-edge interaction in AUTOMATIC_GENERAL; for that, see CPARM8 on the MPP Card.

When SOFT = 0 or SOFT = 1, SRNDE = 1 and 2 are available for the AUTOMATIC_SINGLE, AUTOMATIC_GENERAL, NODES_TO_SURFACE, and SURFACE_TO_SURFACE contacts in both SMP and MPP. However, in MPP, SRNDE = 1 and 2 cause enabling the groupable algorithm (see GRPBLE on MPP2), even if it is disabled. Contact type AUTOMATIC_SURFACE_TO_SURFACE is also supported for these values of SRNDE when SOFT = 0 and 1 but only for SMP.

When SOFT = 2, both the SRNDE = 1 and SRNDE = 2 options are available with all keywords that are supported when SOFT = 2. Both options are available for SMP, MPP, and hybrid versions, and with and without the MPP groupable option. However, for DEPTH = 23, 33, 35, and 55, SRNDE is ignored, and the behavior is like SHLEDG = 1. For all other values of DEPTH, SHLEDG takes precedence over SRNDE. SRNDE is ignored unless SHLEDG is 0 on both *CONTACT and *CONTROL_CONTACT.

When SOFT = 0 or SOFT = 1, the contact algorithm is based on the node-to-surface contact. The SRNDE options are applied to the reference segment in the contact pair, but not the tracked node. For SOFT = 2, the SRNDE options are applied to both segments in the contact pair.

4.   **REGION.** Setting a nonzero value for REGION does not limit or in any way alter the list of SURFA/SURFB nodes and segments. This option should not be used for that purpose. For efficiency, the smallest possible portion of the model should be defined as SURFA or SURFB using the normal mechanisms for specifying the SURFB and SURFA surfaces. Setting a nonzero value will, however, result in contact outside the REGION being ignored. As SURFA and SURFB nodes and segments pass into the indicated REGION, contact for them will become active. As they pass out of the REGION, they will be skipped in the contact calculation. This option is currently available in MPP for all SOFT options, and in SMP when SOFT = 2. Contacts of type AUTOMATIC_SINGLE_SURFACE, AUTOMATIC_…_TO_SURFACE, and ERODING_… can be modified using REGION.

5.   **FRICSF.** The FRICSF factor is an optional factor to scale the frictional stiffness. FRICSF is available only when SOFT = 2 on optional Card A. With penalty contact, the frictional force is a function of the stiffness, the sliding distance, and the Coulomb limit.

Optional Card F:

> **NOTE:** If using Optional Card F, then you must include Optional Cards A, B, C, D and E. (Optional Cards A, B, C, D and E may be blank lines.)

Optional Card F.

Card F	1	2	3	4	5	6	7	8
Variable	PSTIFF	IGNROFF		FSTOL	2DBINR	SSFTYP	SWTPR	TETFAC
Type	I	I		F	I	I	I	F
Default	0	0		2.0	0	0	0	0.0
Remarks	1					2		

<div align="center">VARIABLE   DESCRIPTION</div>

PSTIFF   Flag to choose the method for calculating the penalty stiffness. This is available for segment-to-segment contact activated by setting SOFT = 2 on optional Card A (see About SOFT = 2 for details).

> **EQ.0:** Default determined by the setting of PSTIFF on *CONTROL_CONTACT.

> **EQ.1:** Based on nodal masses or on material density and segment dimensions. The segment mass is taken as the larger of the two values calculated.

> **EQ.2:** Based on material density and segment dimensions

IGNROFF   Flag to ignore the thickness offset for shells in the calculation of the shell contact surface. As an example, this allows shells to be used for meshing tooling surfaces without modifying the positions of the nodes to compensate for the shell thickness. This variable applies to automatic contact types, excluding segment-to-segment (SOFT = 2) contacts. This flag only applies to SMP.

> **EQ.0:** Use default thicknesses.

> **EQ.1:** Ignore the SURFA side thickness.

> **EQ.2:** Ignore the SURFB side thickness.

> **EQ.3:** Ignore the thickness of both sides. This option is only

VARIABLE	DESCRIPTION
	supported in SMP.
FSTOL	Tolerance used with the SMOOTH option for determining which segments are considered flat. The value is in degrees and approximately represents half the angle between adjacent segments.
2DBINR	Flag to indicate that 2D belts initially inside retractors are involved in the contact. This is only available for SURFACE_TO_SURFACE contact of segment-to-segment contact (SOFT = 2).
	EQ.0: No 2D belt initially inside a retractor is involved.
	EQ.1: 2D belts initially inside retractors are involved.
SSFTYP	Flag to determine how the SSF option on *PART_CONTACT behaves when SOFT = 2 on optional card A:
	EQ.0: Use SSF from the tracked segment as determined by the SOFT = 2 algorithm (see Remark 2).
	EQ.1: Use the larger of the SSF values.
	EQ.2: Use the smaller of the SSF values.
SWTPR	Flag to use tapered shell contact segments adjacent to segments that are thinned by the SPOTHIN option on *CONTROL_CONTACT. This option is only available when SOFT = 2 on Optional Card A.
	EQ.0: Use full thickness constant segments.
	EQ.1: Use tapered segments.
TETFAC	Scale factor for the computed volume of tetrahedral solid elements for the mass calculation in SOFT = 2 contact (see Remark 3):
	EQ.0.0: Same behavior as in R11 for backward compatibility.
	GT.0.0.AND.LE.0.5: Scale factor is 0.5.
	GT.0.5: Scale factor is TETFAC.

Remarks:

1. **PSTIFF.** See Remark 6 on *CONTROL_CONTACT for an explanation of the PSTIFF option. Specifying PSTIFF here overrides the behavior defined by PSTIFF on *CONTROL_CONTACT.

2.   **SSFTYP.** SSFTYP affects how the SSF field on *PART_CONTACT is used when the contact formulation is segment-to-segment (SOFT = 2 on Optional Card A). It only applies when the segments in contact belong to different parts, and the parts have different penalty scale factors. The SSF field on *PART_CONTACT assigns a penalty scale factor to a given part. For one-way segment-to-segment contact, the SSF for SURFA is used when SSFTYP = 0. For all other segment-to-segment contacts, the contact algorithm judges for each pair of segments in contact which segment is the tracked segment and which is the reference segment. The reference segment in the pair determines the contact force direction. If SSFTYP = 0, then the SSF of the tracked segment is used. If SSFTYP = 1, then the larger of the two scale factors is used regardless of the contact type.

3.   **TETFAC.** TETFAC is a factor applied when calculating the mass using the material density and segment dimensions (see PSTIFF). By default, 3/8 of the mass of the tetrahedron is used for the contact segment when calculating the mass this way. For 0 < TETFAC ≤0.5, half the mass of a tetrahedral element is used. For hexahedrons, using half the mass (the default behavior for hexahedrons) is reasonable. However, for tetrahedrons, a larger value than 0.5 is preferable because several tets fit into one hex. Therefore, a TETFAC value around 3.0 to 5.0 should make the contact stiffness more comparable with hex meshes.

Optional Card G:

> **NOTE:** If using Optional Card G, then you must include Optional Cards A, B, C, D, E, and F. (Optional Cards A, B, C, D, E, and F may be blank lines.)

Optional Card G.

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

VARIABLE	DESCRIPTION

SHLOFF   Flag affecting the location of the contact surfaces for shells when NLOC is nonzero in *SECTION_SHELL or *PART_COMPOSITE, or when OFFSET is specified using *ELEMENT_SHELL_OFFSET. This flag is the local version of CNTCO on *CONTROL_SHELL. Thus, set this field to 1 to enable the behavior locally for this contact and leave CNTCO as 0 to disable this behavior for all contacts without this field set to 1.

       EQ.0: The setting of CNTCO on *CONTROL_SHELL determines the contact reference plane.

       EQ.1: The contact reference plane coincides with shell reference surface.

General Remarks: *CONTACT

1. **Force output.** Contact force data can be recorded with *DATABASE_RCFORC, *DATABASE_NCFORC, and *DATABASE_BINARY_INTFOR. Note that LS-DYNA only outputs data between a SURFA and SURFB surface. Self-contact, such as single surface contact, only has SURFA and thus no data is output by default. In this case, a *CONTACT_FORCE_TRANSDUCER_*OPTION* is needed to extract contact forces from contact types that may not otherwise be recordable.

2. **AUTOMATIC_GENERAL compared to AUTOMATIC_SINGLE_SURFACE.** *CONTACT_AUTOMATIC_GENERAL is a single surface contact similar to *CONTACT_AUTOMATIC_SINGLE_SURFACE. Both types are automatic, so they consider shell thickness and beam thickness by offsetting the contact interface from the shell midplane and the beam centerline (see Item 4 under Allowed Values for Option 1).

    *CONTACT_AUTOMATIC_GENERAL differs from *CONTACT_AUTOMATIC_SINGLE_SURFACE in how it handles beam contact and shell edge contact. *CONTACT_AUTOMATIC_GENERAL includes treatment of beam-to-beam contact in which it checks the entire length of the beam for penetration, not just the nodes of the beam. For shells, it checks for penetration along the entire length of the exterior (unshared) shell edges. *CONTACT_AUTOMATIC_GENERAL essentially adds null beams to the exterior edges of shell parts so that edge-to-edge treatment of the shell parts is handled by virtue of contact through the automatically generated null beams. By adding the word INTERIOR to *CONTACT_AUTOMATIC_GENERAL, the contact algorithm goes a step further by adding null beams to all the shell meshlines, both along the exterior, unshared edges and the interior, shared shell edges. The EDGEONLY option skips the node-to-surface contact and does only the edge-to-edge and beam-to-beam contact.

    Another distinction is the default value of PENMAX (see Optional Card A). The value of PENMAX helps determine the value of penetration that triggers the release of a penetrating node (see Table 11-2). For AUTOMATIC_GENERAL, the default value of PENMAX causes the penetration value to be effectively unlimited while for AUTOMATIC_SINGLE_SURFACE the default leads to a release condition of about half an element thickness.

    By default, the AUTOMATIC_GENERAL checks for nodal penetration through the three closest segments (DEPTH = 3 on Optional Card A). For AUTOMATIC_SINGLE_SURFACE, the default search depth is 2 segments. The three segment check is more expensive but may be more robust for contact in corners.

    Last, segment-to-segment contact invoked with SOFT = 2 on Optional Card A is supported for AUTOMATIC_SINGLE_SURFACE but not for AUTOMATIC_GENERAL.

3. **Edge-to-edge contact treatment.** For SOFT = 0 and 1, only AUTOMATIC_-GENERAL and SINGLE_EDGE include edge-to-edge contact treatment. This treatment is available for shell elements. AUTOMATIC_GENERAL is a single surface contact that includes edge treatment by checking for penetration along the entire length of the exterior (unshared) shell edges. Appending INTERIOR to AUTOMATIC_GENERAL, additionally causes the edge contact to include interior (meshline) edges. AUTOMATIC_GENERAL_EDGEONLY performs edge-to-edge contact on exterior edges but excludes nodes to surface contact.

    SINGLE_EDGE contact is an old contact algorithm that only treats edge-to-edge contact. In other words, it has edge contact but excludes nodes to surface contact. It also only applies to exterior edges. As it is based on single surface contact, only SURFA is defined for this contact. The edge-to-edge treatment is limited in that contact only occurs between edges whose normal vectors (that lie in the plane of their respective shells) point toward each other (see the theory manual for illustration). Thus, AUTOMATIC_GENERAL is the preferred contact for edge treatment when SOFT = 0 or 1.

    All contact using the segment-to-segment (SOFT = 2) formulation have edge-to-edge capabilities. However, these capabilities are not enabled by default. To enable these capabilities, set DEPTH to 5, 15, 25, 35, 45, or 55 (see Fields used with SOFT = 2 in About SOFT = 2 under Optional Card A for details). Edge-to-edge treatment with SOFT = 2 differs from the edge treatments with SOFT = 0 or 1. The edge treatment method for SOFT = 2 looks at penetration for each pair of segments and determines whether the penetrations look like surface or edge penetration. Thus, it is available for solid and shell elements. SOFT = 2 also applies edge treatment to interior edges if the penetration occurs at a bend in the mesh.

    For Mortar segment-to-segment contact, edge-to-edge contact is enabled for the automatic Mortar contacts for shell elements. It only has edge treatment for external edges, except in the case of *MAT_034. For *MAT_034, edge-to-edge contact on internal edges is activated if the angle between two adjacent segments exceeds an internally set value. See Remark 14 for a more detailed discussion of Mortar contact.

4. **Tying distance criterion.** Tying will only work if the surfaces are near each other. The criterion used to determine whether a tracked node is tied down is that it must be "close". For shell elements "close" is less than a distance, δ, defined as:

$$\delta_1 = 0.60 \times (\text{thickness of tracked node} + \text{thickness of reference segment})$$
$$\delta_2 = 0.05 \times \min(\text{reference segment diagonals})$$
$$\delta = \max(\delta_1, \delta_2)$$

    If a node is further away, it will not be tied, and a warning message will be printed. For solid elements, the node thickness is zero and the segment thickness

is the element volume divided by the segment area; otherwise, the same procedure is used.

If there is a large difference in element areas between the two sides of the contact interface, δ_2 may be too large, causing the unexpected projection of nodes that should not be tied. This can occur when adaptive remeshing is used. To avoid this difficulty, the SURFA and SURFB thicknesses can be specified as negative values on Card 3 in which case

$$\delta = \text{abs}(\delta_1) \ .$$

5. **Tied contact types for different situations**. For tying solids-to-solids, that is, for situations where none of the nodes have rotational degrees-of-freedom, use TIED_NODES_TO_SURFACE and TIED_SURFACE_TO_SURFACE type contacts. These contact types may include the OFFSET or CONSTRAINED_OFFSET option.

   For tying shells-to-shells and beams-to-shells, that is, for situations where all the nodes have rotational degrees-of-freedom, use TIED_SHELL_EDGE_TO_SURFACE type contacts. This contact type may include the OFFSET, CONSTRAINED_OFFSET, or BEAM_OFFSET option. For beam spot welds, you can use *CONTACT_SPOTWELD or *CONTACT_SPOTWELD_WITH_TORSION for beam-to-shell contact. When *MAT_SPOTWELD is used for the beam material, the material model can include failure for the spot weld.

   TIED_SHELL_EDGE_TO_SOLID is intended for tying shell edges to solids or beam ends to solids.

   Tied contacts with failure include TIEBREAK contacts and TIED_SURFACE_-TO_SURFACE_FAILURE. Segment orientation is important for distinguishing the direction of tension from compression when defining these contacts.

6. **Tied contact types and the implicit solver.** Nonphysical results have been observed when the implicit time integrator is used for models that combine tied contact formulations with automatic single point constraints on solid element rotational degrees of freedom (AUTOSPC on *CONTROL_IMPLICIT_-SOLVER). The following subset of tied interfaces support a *strongly objective* mode (discussed in the next paragraph) and are verified to behave correctly with the implicit time integrator:

   a) TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET
      TIED_SURFACE_TO_SURFACE_CONSTRAINED_OFFSET

   b) TIED_NODES_TO_SURFACE_OFFSET
      TIED_SURFACE_TO_SURFACE_OFFSET

   c) TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET

d) TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET

The first two of these ignore rotational degrees of freedom, while the third and fourth constrain rotations. The first and third are constraint-based; while the second and fourth are penalty-based. These four contact types are intended to cover most use scenarios.

Setting IACC = 1 on *CONTROL_ACCURACY activates the *strongly objective formulation* for the above mentioned contacts (as well as for the non-offset options as a side effect, namely, *CONTACT_TIED_NODES_TO_SURFACE, *CONTACT_TIED_SURFACE_TO_SURFACE, *CONTACT_TIED_SHELL_EDGE_-TO_SURFACE, and *CONTACT_TIED_SHELL_EDGE_TO_SOLID). When active, forces and moments transform correctly under superposed rigid body motions within a single implicit step. Additionally, this formulation applies rotational constraints consistently *when, and only when, necessary*. In particular, strong objectivity is implemented so that tracked nodes without rotational degrees of freedom are not rotationally constrained, while tracked nodes with bending and torsional rotations are rotationally constrained. Additionally, strong objectivity ensures that the constraint is physically correct.

For a reference node belonging to a *shell*, the tracked node's bending rotations (rotations in the plane of the reference segment) are constrained to match the reference segment's rotational degrees of freedom; for reference nodes *not belonging to a shell*, the tracked node's bending rotations are constrained to the reference segment rotation as determined from its individual nodal translations. The tracked node's torsional rotations (rotations with respect to the normal of the reference segment) are *always* constrained based on the reference segment's torsional rotation as determined from its individual nodal translations, thus avoiding the relatively weak drilling mode of shells. This tied contact formulation properly treats bending *and* torsional rotations. Since the tracked node's rotational degrees of freedom typically come from shell or beam elements, the most frequently used options are:

TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET

TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET

The other two classes of "non-rotational" formulations:

TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET
TIED_SURFACE_TO_SURFACE_CONSTRAINED_OFFSET

TIED_NODES_TO_SURFACE_OFFSET
TIED_SURFACE_TO_SURFACE_OFFSET

are included for situations in which rotations do not need to be constrained at all. See the LS-DYNA Theory Manual for further details.

7. **Tying to rigid bodies.** The following tied contact types are constraint-based and *will not* work with rigid bodies:

> TIED_NODES_TO_SURFACE
>
> TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET
>
> TIED_SURFACE_TO_SURFACE
>
> TIED_SURFACE_TO_SURFACE_CONSTRAINED_OFFSET
>
> TIED_SURFACE_TO_SURFACE_FAILURE (SMP only)
>
> TIED_SHELL_EDGE_TO_SURFACE
>
> TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET
>
> TIED_SHELL_EDGE_TO_SOLID
>
> SPOTWELD
>
> SPOTWELD_WITH_TORSION

The following tied contact types are penalty-based and *will* work with rigid bodies:

> SPOTWELD_WITH_TORSION_PENALTY
>
> TIED_NODES_TO_SURFACE_OFFSET
>
> TIED_SHELL_EDGE_TO_SURFACE_OFFSET
>
> TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET
>
> TIED_SURFACE_TO_SURFACE_OFFSET

Also, it may sometimes be advantageous to use the CONSTRAINED_EXTRA_-NODE_*OPTION* instead for tying deformable nodes to rigid bodies since in this latter case the tied nodes may be an arbitrary distance away from the rigid body.

8. **SPOTWELD_WITH_TORSION contact.** The contact algorithm for tying spot welds with torsion, SPOTWELD_WITH_TORSION, must be used with care. Parts that are tied by this option should be subjected to stiffness proportional damping of approximately ten percent, meaning input a coefficient of 0.10. This can be defined for each part on the *DAMPING_PART_STIFFNESS input.

Stability problems may arise with this option if damping is not used. This comment applies also to the PENALTY keyword option.

9.   **Forming contact type and surface thickness.** For "Forming" contact, the surface thickness of SURFB is ignored (SURFB should be the rigid tooling while SURFA should be the blank). Furthermore, SURFB can be offset away from the blank by setting a negative (meaning opposite of the positive normal of the SURFB surface) value for SBST on Mandatory Card 3. This offsets SURFB from the surface midplane by |SBST|/2 in the direction opposite to the SURFB positive normal direction. A tool and die can be quickly offset (virtually, not physically) with this field.

10. **Airbag interactions.** Modeling airbag interactions with structures and occupants using the actual fabric thickness, which is approximately 0.30 mm, may result in a contact breakdown that leads to inconsistent occupant behavior between different machines. Based on our experience, using a two-way automatic type contact definition, meaning AUTOMATIC_SURFACE_TO_SURFACE, between any airbag to structure/occupant interaction and setting the airbag fabric contact thickness to at least 10 times the actual fabric thickness has helped improved contact behavior and eliminates the machine inconsistencies. Due to a large stiffness difference between the airbag and the interacting materials, we recommend the soft constraint option (SOFT = 1) or the segment-to-segment option (SOFT = 2). Note that with the above contact definition, only the airbag materials should be included in any *AIRBAG_SINGLE_SURFACE definitions to avoid duplicate contact treatment that can lead to numerical instabilities.

11. **Constraint contact.** The following two contact types are constraint-based and must be used with care. The surface and the nodes which are constrained to a surface are not allowed to be used in any other CONSTRAINT_… contact definition:

CONSTRAINT_NODES_TO_SURFACE

CONSTRAINT_SURFACE_TO_SURFACE

If, however, contact must be defined from both sides as in sheet metal forming, one of these contact definitions can be a CONSTRAINT type; the other one could be a standard penalty type such as SURFACE_TO_SURFACE or NODES_TO_-SURFACE.

12. **Penalty-based contact types and shell thickness.** The following penalty-based contact types consider shell thickness, that is, the contact surfaces for shells are offset from the shell midplane. This offset distance can be modified using variables on Card 3 of *CONTACT. The SHLTHK option on the *CONTROL_CONTACT card is ignored for these contact types.

> AIRBAG_SINGLE_SURFACE
>
> AUTOMATIC_GENERAL
>
> AUTOMATIC_GENERAL_INTERIOR
>
> AUTOMATIC_NODES_TO_SURFACE
>
> AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
>
> AUTOMATIC_SINGLE_SURFACE
>
> AUTOMATIC_SURFACE_TO_SURFACE
>
> SINGLE_SURFACE

A thickness that is too small may result in loss of contact while an unrealistically large thickness may result in a degradation in speed during the bucket sorts as well as nonphysical behavior. To address the latter behavior, the default setting of SSTHK in *CONTROL_CONTACT may sometimes cause the contact thickness to be reduced for certain single surface contacts (see Remark 2 of *CONTROL_CONTACT).

13. **Projecting the contact surface for shell thicknesses.** For SOFT = 0 and 1, the method for projecting the contact surface to account for shell thickness depends on the choice of contact and the choice between SMP and MPP. The two implemented methods for projection are nodal normal projection and segment-based projection. For a discussion of projecting the contact thickness for SOFT = 2, see Projecting the contact surface for shell thickness under About SOFT = 2 in the remarks following Optional Card A.

For SMP, segment-based projection is used in contact types:

> AIRBAG_SINGLE_SURFACE
>
> AUTOMATIC_GENERAL
>
> AUTOMATIC_NODES_TO_SURFACE
>
> AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
>
> AUTOMATIC_SINGLE_SURFACE
>
> AUTOMATIC_SURFACE_TO_SURFACE
>
> FORMING_NODES_TO_SURFACE
>
> FORMING_ONE_WAY_SURFACE_TO_SURFACE

Nodal normal projection

Segment-based projection for non-automatic contacts in SMP

Segment-based projection for automatic contacts in SMP and MPP
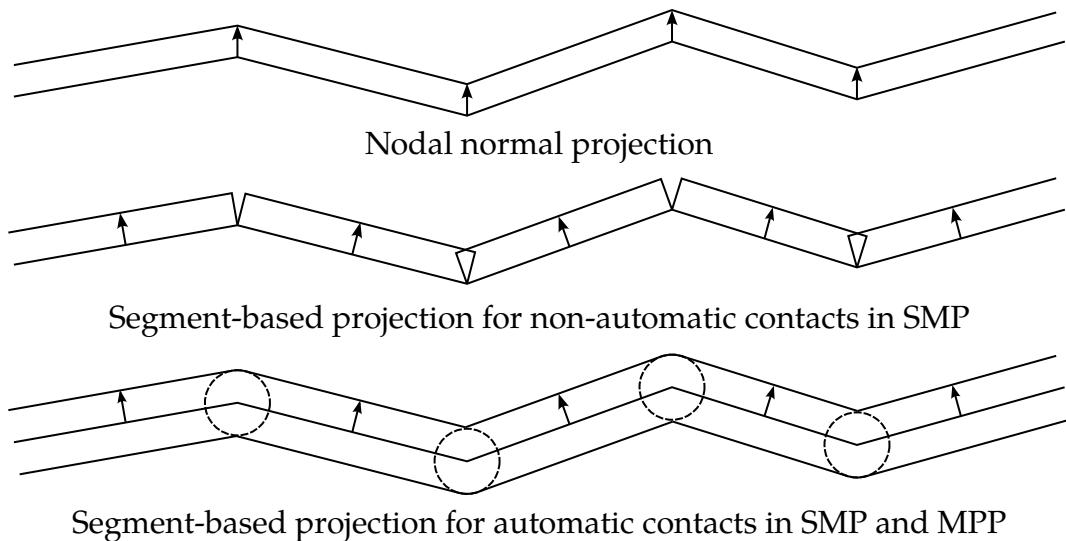
Figure 11-25. Nodal normal projection and segment-based projection used in contacts (SMP). MPP contacts use nodal normal projection exclusively.

FORMING_SURFACE_TO_SURFACE

The remaining contact types in SMP use nodal normal projections if projections are used (see THKOPT and SHLTHK on Optional Card B). For MPP, all automatic contacts use segment-based projection, while all the others use nodal normal projections (if projections are used).

Figure 11-25 illustrates the different types of projection. Note that for the MPP version of nodal normal projection, the vectors are computed such that the surface looks like a segment-based projection with the surfaces extended to remove the gaps. Segment-based projection is used for automatic contacts in both SMP and MPP, and for this kind of contact, the projection occurs in both directions automatically. The gaps are automatically closed for these contacts using circles tangent to the projections. In SMP, the gaps for non-automatic contacts projected with segment-based projection can be closed with MAXPAR on Optional Card A.

The method used can influence the accuracy and cost of the calculation. The main advantage of nodal projections for non-automatic contacts is that a continuous contact surface is obtained, which is much more accurate in applications such as metal forming. The disadvantages of nodal projections are the higher costs due to the nodal normal calculations, difficulties in treating T-intersections and other geometric complications, and the need for consistent orientation of contact surface segments.

For the treatment of the free edges of shell surfaces, see SRNDE on Optional Card E.
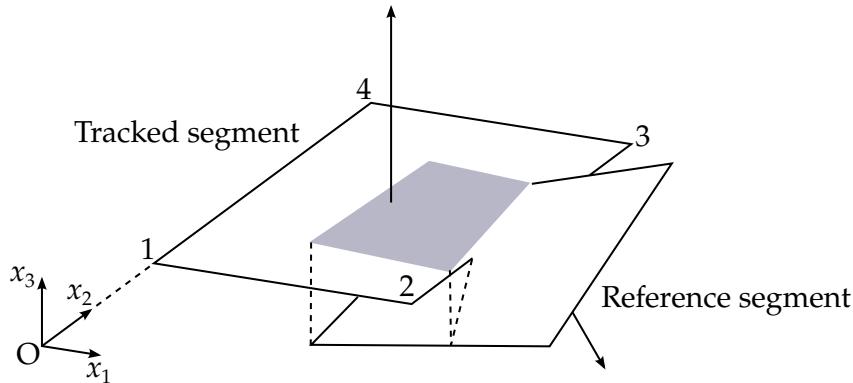
Figure 11-26. Illustration of Mortar segment-to-segment contact

Note that SHLOFF on Optional Card G and CNTCO on *CONTROL_SHELL can change the location of the reference plane for the projection.

14. **Overview of Mortar Contact.** Mortar contact, invoked by appending the suffix MORTAR to either FORMING_SURFACE_TO_SURFACE, AUTOMATIC_SUR-FACE_TO_SURFACE or AUTOMATIC_SINGLE_SURFACE is a segment-to-segment penalty-based contact. For two segments on each side of the contact interface that are overlapping and penetrating, a consistent nodal force assembly taking into account the individual shape functions of the segments is performed; see Figure 11-26 for an illustration. A *CONTACT_FORCE_TRANSDUCER_-PENALTY can extract forces from Mortar contacts, *but the SURFA and SURFB sides must then be defined through parts or part sets.* For the automatic Mortar contacts, support for eroded solid and shell elements is automatically invoked without the need of an ERODING option. It will also create new edge segments exposed to the exterior as shell elements erode, but beam elements are not yet supported. As for transducers, treatment of eroding requires the SURFA and SURFB sides to be defined through parts or part sets. In this respect, the results with this contact may be more accurate, especially when considering contact with elements of higher order.

By appending the suffix TIED to the *CONTACT_AUTOMATIC_SURFACE_-TO_SURFACE_MORTAR keyword or the suffix TIEBREAK_MORTAR (only OPTION = 2, 4, 6, 7, 8 and 9 are supported) to the *CONTACT_AUTOMATIC_-SURFACE_TO_SURFACE keyword, the contact is treated as a tied contact interface with optional failure in the latter case. For MORTAR_TIED_WELD, parts can be welded together to create a tied contact at the welded zones; segment pairs are dynamically created based on a weld criterion to make up a tied contact element. For all these Mortar tied contacts, the tied SURFA segments can be viewed in the intfor file by specifying NTIED on *DATABASE_EXTENT_INT-FOR, a convenient way to know which segments. This contact is intended for implicit analysis in particular but is nevertheless supported for explicit analysis as well. For explicit analysis, the bucket sort frequency is 100 if not specified.
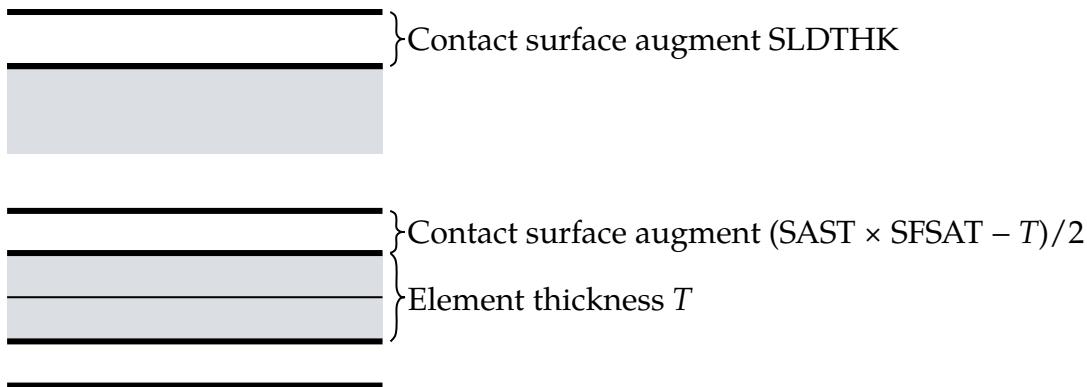
Figure 11-27. Illustration of contact surface location for automatic Mortar contact, solids on top and shells below.

FORMING *Mortar* contact, in contrast to other forming contacts, does *not* assume a rigid SURFB side, but, if this side consists of shell elements, the normal vector should be oriented towards the SURFA side. Furthermore, no shell thickness is considered on the SURFB side. The SURFA side is assumed to be a deformable part, and the orientation of the elements does not matter. However, each FORMING contact definition should be such that contact occurs with ONE deformable SURFA side only, which obviously leads to multiple contact definitions if two-sided contact is presumed.

AUTOMATIC contact is supported for solids, shells, thick shells, and beams. Here the thicknesses are considered both for rigid and deformable parts. Flat edge contact is supported for shell elements and contact with beams occurs on the lateral surface area as well as on the end tip. The contact assumes that the beam has a cylindrical shape[1] with a cross sectional area coinciding with that of the underlying beam element. The contact surface can be augmented with the aid of parameters SAST and SFSAT for shells and beams, while SLDTHK is used for solids and thick shells. For shells/beams SAST corresponds to the contact thickness of the element (SBST likewise for the SURFB side); by default, this is the same as the element thickness. This parameter can be scaled with aid of SFSAT (SFSBT for the SURFB side) to adjust the location of the contact surface; see Figure 11-27.

For all unilateral Mortar contacts (meaning non-tied contacts), FS and FD on Mandatory Card 2 are supported and work like similar non-Mortar contacts. If implicit statics is used, the dynamic friction coefficient is set equal to the static coefficient.

For solids PENMAX on Optional Card B can be used to determine the maximum penetration. It also determines the search depth for finding contact pairs. If set,

---

[1] Internally the cylinder is faceted as described in the Theory Manual, chapter Mortar Contact.

it should correspond to a characteristic thickness in the model. Also, the contact surface can be adjusted with the aid of SLDTHK if it is of importance to reduce the gap between parts; see Figure 11-27. This may be of interest if initial gaps result in free objects undergoing rigid body motion and thus preventing convergence in implicit. SLDTHK may be set to a negative number, meaning that the contact surface will be offset in the negative direction of the normal by the amount specified. This option may be useful for instance if mesh coarseness prevents concentric cylinders from rotating freely with respect to each other.

For the TIED option, the criterion for tying two contact surfaces is by default that the distance should be less than $0.05 \times T$, that is, within 5% of the element thickness (characteristic size for solids). In this case PENMAX can be used to set the tying distance, meaning if PENMAX is positive then segments are tied if the distance is less than PENMAX.

If initial penetrations are detected (reported in the messag file), then by default these penetrations will be given zero contact stress. This treatment is due to IGNORE = 2 being the default for Mortar contact. IGNORE = 2 behaves differently for Mortar contacts than for other contacts. For this option, the penetrations are not tracked, but the contact surface is fixed at its initial location. In addition, for IGNORE = 2, an initial contact pressure can be imposed on the interface by setting MPAR1 to the desired contact pressure. Any rigid body motion due to initial contact gaps can be properly eliminated with these settings. If you want initial penetrations to result in contact pressure (the "old" IGNORE = 0 behavior), use IGNORE = 2 and set MPAR1 to a large number; the contact stress cannot be higher than that allowed for the given penetration. IGNORE = 1 is similar to IGNORE = 2, except the penetrations will be tracked. Thus, if the surfaces separate, the contact surface locations are updated accordingly. In the event of surfaces separating enough to form gaps, the contact is restored to as if there were no initial penetrations in the first place.

A third option is IGNORE = 3, for which prestress can be applied. This allows initial penetrations to exist which are closed during the time between zero and the value given by MPAR1. It works like the INTERFERENCE option, except that the closure is linear in time. Initial penetrations, however, must be small enough for the contact algorithm to detect them. For this option MPAR1 can be chosen negative, meaning that the reduction factor is specified as a load curve as function of time. This load curve should increase from 0 to 1, where 1 means that all initial penetrations have been removed.

For large penetrations IGNORE = 4 is recommended (this can only be used if the SURFA side consists of solid elements). This option does pretty much the same thing as IGNORE = 3, but you may provide a penetration depth with MPAR2. This depth must be at least as large as (and preferably in the order of) the maximum initial penetration in the contact interface; otherwise, an error termination
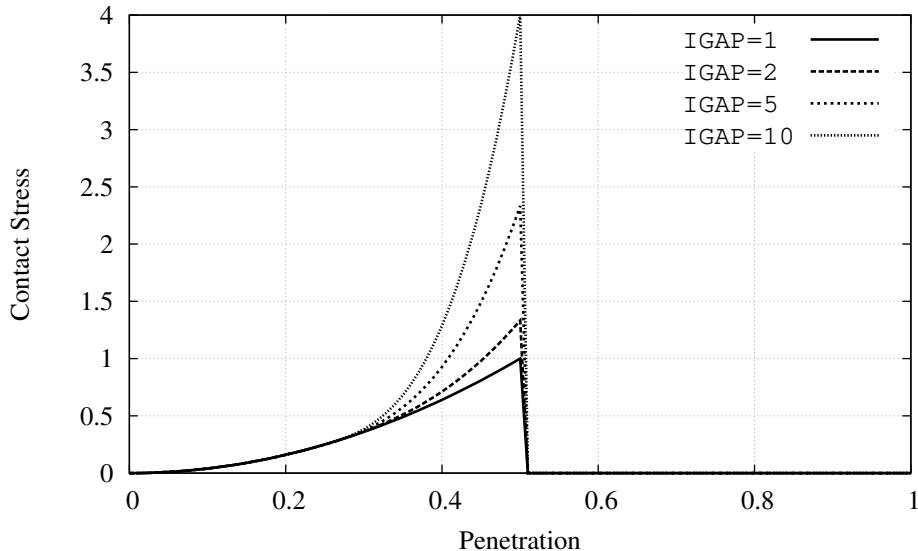
Figure 11-28. Mortar contact stress as function of penetration

will be the result. This parameter helps the contact algorithm locate the contact surface and thus estimate the initial penetration. With this option the contact surfaces are pushed back and placed in incident contact at places where initial penetrations are present which can be done for (more or less) arbitrary initial penetration depths. Like IGNORE = 3, the contact surfaces will be restored linearly in the time given by MPAR1, but here the load curve option is not applicable.

Mortar contacts in implicit analysis can cause the contact pressure to locally be very high which leads to the release of large enough penetrations in subsequent steps. Penetration information can be requested on MINFO on *CONTROL_-OUTPUT which issues a warning if there is a danger of this happening. To prevent contact release, you may increase IGAP which penalizes large penetrations without affecting small penetration behavior and thereby overall implicit performance. Figure 11-28 shows the contact pressure as function of penetration for the Mortar contact, including the effect of increasing IGAP. It also shows that for sufficiently large penetrations the contact is not detected in subsequent steps which is something to avoid. In addition, a negative IGAP can be used to render a linear relationship between the penetration and contact pressure. This might be helpful if relatively large penetrations are observed for small contact pressures.

Table 11-1 lists fields on the mandatory and optional cards that apply to MORTAR contact. Any parameter not mentioned in this list is ignored. See a similar table on the manual page for *CONTROL_CONTACT.

Table 11-1. Fields related to Mortar contact

Data Card	Comment
Mandatory Card 1	Apply as for any other contact. Parts or part sets are recommended. Contact stiffness is taken from the SURFA side of the contact.
Mandatory Card 2	Except for PENCHK, these fields also apply as for any other contact. PENCHK is ignored.
Mandatory Card 3	Except for SFSB, FSF and VSF, these also apply as for any other contact. SFSB, FSF and VSF are ignored.
Optional Card A	Only BSORT applies. All others are ignored. BSORT is the bucket sort frequency.
Optional Card B	Only PENMAX and SLDTHK apply. All others are ignored. PENMAX is the contact search depth for solid element segments, and SLDTHK is the distance by which solid element segments are offset in the direction of the segment normal.
Optional Card C	First four parameters apply, others are ignored. IGAP is a stiffness parameter for large penetrations which is usually not needed. IGNORE deals with initial penetrations as described above, and MPAR1/MPAR2 are parameters associated with the choice of IGNORE.
Optional Cards D, E, F, and G	No parameter applies. Shell edges are always flat.

Table 11-2. Criterion for node release for nodal points which have penetrated too far. This criterion does not apply to SOFT = 2 contact. Larger penalty stiffnesses are recommended for the contact interface which allows nodes to be released. For node-to-surface type contacts (5, 5a), the element thicknesses which contain the node determines the nodal thickness. The parameter is defined on the *CONTROL_CONTACT input.

INTERFACE TYPE ID	PENCHK	ELEMENT TYPE	FORMULA FOR RELEASE OF PENETRATING NODAL POINT
1, 2, 6, 7	–		
3, 5, 8, 9, 10 (without thickness)	0	solid	d = PENMAX if PENMAX > 0 d = 10^{10} if PENMAX = 0
		shell	d = PENMAX if PENMAX > 0 d = 10^{10} if PENMAX = 0
	1	solid	d = XPENE × thickness of solid element
		shell	d = XPENE thickness of shell element
	2	solid	d = 0.05 × minimum diagonal length
		shell	d = 0.05 × minimum diagonal length
3, 5, 10 (thickness), 17 and 18	–	solid	d = XPENE × thickness of solid element
		shell	d = XPENE × thickness of shell element
a3, a5, a10	–	solid	d = PENMAX × thickness of solid element *[default: PENMAX = 0.4 in SMP and 0.5 in MPP]*
		shell (SMP)	d = PENMAX × (tracked thickness + reference thickness) for PENMAX ≥ 0.4. d = 0.4 × (tracked thickness + reference thickness) for PENMAX < 0.4. *[default: PENMAX = 0.4]*
		shell (MPP)	d = ½ × (PENMAX + 1) × (tracked thickness + reference thickness) *[default: PENMAX = 0.8]*
13, 15	–	solid (SMP)	d = PENMAX × thickness of solid element *[default: PENMAX = 0.4]*
		solid (MPP)	d = PENMAX × thickness of solid element if PENMAX ≥ 0.5 d = 0.5 × thickness of solid element if PENMAX < 0.5 *[default: PENMAX = 0.5]*
		shell	d = PENMAX × (tracked thickness + reference thickness) if PENMAX ≥ 0.4 d = 0.4 × (tracked thickness + reference thickness) if PENMAX < 0.4 *[default: PENMAX = 0.4]*
4	–	solid	d = 0.5 × thickness of solid element

INTERFACE TYPE ID	PENCHK	ELEMENT TYPE	FORMULA FOR RELEASE OF PENETRATING NODAL POINT
		shell	d = 0.4 × (tracked thickness + reference thickness)
26	———	solid	d = PENMAX × thickness of solid element *[default: PENMAX = 10.0]*
		shell	d = PENMAX × (tracked thickness + reference thickness) *[default: PENMAX = 10.0]*

Mapping of *CONTACT keyword option to "contact type" in d3hsp:

Structured Input Type ID	Keyword Name
a 13	AIRBAG_SINGLE_SURFACE
26	AUTOMATIC_GENERAL
i 26	AUTOMATIC_GENERAL_INTERIOR
a 5	AUTOMATIC_NODES_TO_SURFACE
t 5	AUTOMATIC_NODES_TO_SURFACE_TIEBREAK
a 10	AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE
t 10	AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE_TIE-BREAK
13	AUTOMATIC_SINGLE_SURFACE
a 3	AUTOMATIC_SURFACE_TO_SURFACE
18	CONSTRAINT_NODES_TO_SURFACE
17	CONSTRAINT_SURFACE_TO_SURFACE
23	DRAWBEAD
16	ERODING_NODES_TO_SURFACE
14	ERODING_SURFACE_TO_SURFACE
15	ERODING_SINGLE_SURFACE
27	FORCE_TRANSDUCER_CONSTRAINT
25	FORCE_TRANSDUCER_PENALTY
m 5	FORMING_NODES_TO_SURFACE
m 10	FORMING_ONE_WAY_SURFACE_TO_SURFACE
m 3	FORMING_SURFACE_TO_SURFACE
5	NODES_TO_SURFACE
5	NODES_TO_SURFACE_INTERFERENCE
10	ONE_WAY_SURFACE_TO_SURFACE
20	RIGID_NODES_TO_RIGID_BODY

Structured Input Type ID	Keyword Name
21	RIGID_BODY_ONE_WAY_TO_RIGID_BODY
19	RIGID_BODY_TWO_WAY_TO_RIGID_BODY
22	SINGLE_EDGE
4	SINGLE_SURFACE
1	SLIDING_ONLY
p 1	SLIDING_ONLY_PENALTY
3	SURFACE_TO_SURFACE
3	SURFACE_TO_SURFACE_INTERFERENCE
8	TIEBREAK_NODES_TO_SURFACE
9	TIEBREAK_SURFACE_TO_SURFACE
6	TIED_NODES_TO_SURFACE
o 6	TIED_NODES_TO_SURFACE_OFFSET
c 6	TIED_NODES_TO_SURFACE_CONSTRAINED_OFFSET
7	TIED_SHELL_EDGE_TO_SURFACE or SPOTWELD
o 7	TIED_SHELL_EDGE_TO_SURFACE_OFFSET
c 7	TIED_SHELL_EDGE_TO_SURFACE_CONSTRAINED_OFFSET or SPOTWELD_CONSTRAINED_OFFSET
b 7	TIED_SHELL_EDGE_TO_SURFACE_BEAM_OFFSET or SPOTWELD_BEAM_OFFSET
s 7	SPOTWELD_WITH_TORSION
2	TIED_SURFACE_TO_SURFACE
o 2	TIED_SURFACE_TO_SURFACE_OFFSET
c 2	TIED_SURFACE_TO_SURFACE_CONSTRAINED_OFFSET

Contact Examples:

```
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONTACT_NODES_TO_SURFACE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Make a simple contact that prevents the nodes in part 2 from
$  penetrating the segments in part 3.
$
*CONTACT_NODES_TO_SURFACE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$    surfa     surfb  surfatyp  surfbtyp  saboxid   sbboxid      sapr      sbpr
         2         3         3         3
$
$       fs        fd        dc        vc       vdc    penchk        bt        dt

$
$     sfsa      sfsb      sast      sbst     sfsat     sfsbt       fsf       vsf


$
$    surfatyp, surfbtyp = 3  id's specified in surfa and surfb are parts
$      surfa = 2 use nodes in part 2 in the contact
$      surfb = 3 use segments in part 3 in the contact
$
$  Use defaults for all parameters.
$
$$$$  Optional Cards A and B not specified (default values will be used).
$
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONTACT_SINGLE_SURFACE
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Create a single surface contact between four parts: 28, 97, 88 and 92
$    - create a part set with set ID = 5, list the four parts
$    - in the *CONTACT_SINGLE_SURFACE definition specify:
$         surfatyp = 2  which means surfa is a part set
$         surfa = 5  use part set 5 for defining the contact surfaces
$
$  Additional contact specifications described below.
$
*CONTACT_SINGLE_SURFACE
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$    surfa     surfb  surfatyp  surfbtyp  saboxid   sbboxid      sapr      sbpr
         5                   2
$       fs        fd        dc        vc       vdc    penchk        bt        dt
      0.08      0.05        10                  20                          40.0
$     sfsa      sfsb      sast      sbst     sfsat     sfsbt       fsf       vsf


$
$       fs = 0.08  static coefficient of friction equals 0.08
$       fd = 0.05  dynamic coefficient of friction equals 0.05
$       dc =   10  exponential decay coefficient, helps specify the transition
$                  from a static slide to a very dynamic slide
$      vdc =   20  viscous damping of 20% critical (damps out nodal
```

```
$                       oscillations due to the contact)
$       dt = 40.0  contact will deactivate at 40 ms (assuming time unit is ms)
$
$$$$  Optional Cards A and B not specified (default values will be used).
$
$
*SET_PART_LIST
$      sid
         5
$     pid1      pid2      pid3      pid4
        28        97        88        92
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$$$$  *CONTACT_DRAWBEAD
$
$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$$
$
$  Define a draw bead contact:
$    - the draw bead is to be made from the nodes specified in node set 2
$    - the reference segments are those found in the box defined by box 2
$         that are in part 18
$    - include SURFA and SURFB forces in interface file (sapr, sbpr = 1)
$
*CONTACT_DRAWBEAD
$
$...>....1....>....2....>....3....>....4....>....5....>....6....>....7....>....8
$    surfa     surfb  surfatyp  surfbtyp   saboxid   sbboxid      sapr      sbpr
         2        18         4         3                   2         1         1
$
$       fs        fd        dc        vc       vdc    penchk        bt        dt
      0.10
$
$     sfsa      sfsb      sast      sbst     sfsat     sfsbt       fsf       vsf


$
$$$$ Card 4 required because it's a drawbead contact
$
$  lcdidrf    lcidnf     dbdth     dfscl    numint
         3              0.17436       2.0
$
$  lcdidrf =       3  load curve 3 specifies the bending component of the
$                     restraining force per unit draw bead length
$    dbdth = 0.17436  draw bead depth
$    dfscl =     2.0  scale load curve 3 (lcdidrf) by 2
$
$$$$  Optional Cards A and B not specified (default values will be used).
$
*DEFINE_BOX
$   boxid       xmm       xmx       ymn       ymx       zmn       zmx
         2 0.000E+00 6.000E+00 6.000E+00 1.000E+02-1.000E+03 1.000E+03
$
*SET_NODE_LIST
$      sid       da1       da2       da3       da4
         2
$     nid1      nid2      nid3      nid4      nid5      nid6      nid7      nid8
      2580      2581      2582      2583      2584      2585      2586      2587
      2588      2589      2590
$
*DEFINE_CURVE
$     lcid      sidr      scla      sclo      offa      offo
         3
$              a                   o
$           DEPTH           FORC/LGTH
```

```
0.000E+00          0.000E+00
1.200E-01          1.300E+02
1.500E-01          2.000E+02
1.800E-01          5.000E+02
```

*CONTACT_ADD_WEAR

Purpose: Associate a wear model to a contact interface.

Wear is associated with friction, so the frictional coefficient *must* be nonzero for the associated contact interface. This feature calculates the wear depth, sliding distance and possibly user defined wear history variables according to the specified model which are then written to the intfor database (see *DATABASE_EXTENT_INTFOR) for post-processing. Note that this data is *not* written unless the parameter NWEAR and/or NWUSR are set on the *DATABASE_EXTENT_INTFOR card. *H*-adaptive remeshing is supported with this feature. Implicit analysis is supported, for which mortar is the preferred contact.

For positive CID values, this keyword does not affect the results of a simulation and only provides wear quantities for post-processing. For negative CID values, the contact surface is perturbed internally based on the wear depth so that the contact behavior is affected by the wear on the surface.

Card 1	1	2	3	4	5	6	7	8
Variable	CID	WTYPE	P1	P2	P3	P4	P5	P6
Type	I	I	F	F	F	F	F	F
Default	none	0	none	none	↓	↓	↓	↓

User Defined Wear Parameter Cards. Define as many cards as needed to define P1 parameters if and only if WTYPE < 0.

Card 2	1	2	3	4	5	6	7	8
Variable	W1	W2	W3	W4	W5	W6	W7	W8
Type	F	F	F	F	F	F	F	F

VARIABLE	DESCRIPTION
CID	Contact interface ID, see *CONTACT_…
	LT.0: Perturb contact surface according to wear values (see Remark 5).
	GT.0: Calculate wear properties for post-processing only.

VARIABLE	DESCRIPTION
WTYPE	Wear law:

> LT.0: User defined wear law; value specifies type used in sub-routine.
>
> EQ.0: Archard's wear law

P1	First wear parameter:

> WTYPE.EQ.0: Dimensionless scale factor k. If negative, the absolute value specifies a table ID with $k = k(p, \dot{d})$ as a function of contact pressure $p \geq 0$ and relative sliding velocity $\dot{d} \geq 0$. The table should be defined such that for each specified value of $\dot{d}$, there exists a curve such that k (ordinate) is a function of p (abscissa).
>
> WTYPE.LT.0: Number of user wear parameters for this interface

P2	Second wear parameter:

> WTYPE.EQ.0: SURFA surface hardness parameter H_A. If negative, the absolute value specifies a curve ID with $H_A = H_A(T_A)$ as function of SURFA node temperature T_A.
>
> WTYPE.LT.0: Number of user wear history variables per contact node. These can be output to the intfor file; see NWUSR on *DATABASE_EXTENT_INTFOR.

P3	Third wear parameter:

> WTYPE.EQ.0: SURFB surface hardness parameter H_B. If negative, the absolute value specifies a curve ID with $H_B = H_B(T_B)$ as function of SURFB node temperature T_B.
>
> WTYPE.LT.0: Not used

P4 - P6	Not used
Wn	nth user defined wear parameter.

Remarks:

1. **Archard's Wear Law.** Archard's wear law (WTYPE = 0) states that the wear depth w at a contact point evolves with time as

$$\dot{w} = k\frac{p\dot{d}}{H}$$

where $k > 0$ is a dimensionless scale factor, $p \geq 0$ is the contact interface pressure, $\dot{d} \geq 0$ is the relative sliding velocity of the points in contact and $H > 0$ is the surface hardness (force per area). The wear depth for a node in contact is incremented in accordance with this formula, accounting for the different hardness of the SURFA and SURFB side, H_A and H_B, respectively. By using negative numbers for wear parameters P1, P2 or P3, the corresponding parameter is defined by a table or a curve. For P1, the value of k is taken from a table with contact pressure p and sliding velocity $\dot{d}$ as arguments, while for P2 or P3, the corresponding hardness H is taken from curves with the associated contact nodal temperature T as argument. That is, the SURFA side hardness will be a function of the SURFA side temperature, and vice versa.

2. **User Defined Wear Laws.** Customized wear laws may be specified as a user-defined subroutine called userwear. This subroutine is called when WTYPE < 0. This subroutine is passed wear parameters (see P1) for this interface as well as number of wear history variables (see P2) per contact node. The wear parameters are defined on additional cards (see WN) and the history variables are updated in the user subroutine. The history variables can be output to the intfor file, see NWUSR on *DATABASE_EXTENT_INTFOR. WTYPE may be used to distinguish between different wear laws, and consequently any number of different laws can be implemented within the same subroutine. For more information, we refer to the source code which contains extensive commentaries and two sample wear laws.

3. **Using Wear Laws.** Only one wear law per contact interface can be specified. The procedure for activating this feature involves

   a) Using the present keyword to associate wear to a contact interface

   b) Setting NWEAR and/or NWUSR on the *DATABASE_EXTENT_INTFOR card.

   c) Having a contact interface with friction of a type that is supported. If SOFT = 2 on optional card A of the contact data, then any valid keyword option is supported. If SOFT = 0 or SOFT = 1, then the following list is supported.

   *CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE

*CONTACT_FORMING_SURFACE_TO_SURFACE

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE

*CONTACT_FORMING_SURFACE_TO_SURFACE_MORTAR

*CONTACT_AUTOMATIC_SURFACE_TO_SURFACE_MORTAR

*CONTACT_AUTOMATIC_SINGLE_SURFACE_MORTAR

4. **Limitations.** The following are not supported with this keyword.

    a) _SMOOTH option

    b) MPP "groupable" option

5. **Negative CID Option.** For this option, the wear is accounted for by perturbing the contact surface internally (not by updating the node positions). As such, no changes to the element geometry will be observed due to wear, however increased part penetrations and stress redistributions will occur, as well as redistribution of contact pressure and wear depth in the intfor results. The method is only physically justified for relatively small wear depths. Care must therefore be taken in selecting the wear parameters to ensure that the wear depth does not exceed the maximum penetration associated with the contact. This option is available for SMP, MPP and Hybrid versions when SOFT = 2 on optional card A of the *CONTACT card but is only available for the SMP version when SOFT = 0 or 1.

*CONTACT_AUTO_MOVE

Purpose: Automatically move the surfb surface in a contact definition to close an unspecified gap between the surfa and surfb surfaces. The gap may result from loading the surfa part with an initial gravity load. The gap will be closed at a specified time to save CPU time. The surfb surface in metal forming applications is typically the upper cavity while the surfa part is the blank. This feature is only applicable to sheet metal forming application.

Cards 1	1	2	3	4	5	6	7	8
Variable	ID	CONTID	VID	LCID	ATIME	OFFSET		
Type	I	I	I	I	F	F		
Default	none	none	none	0	0.0	0.0		

VARIABLE	DESCRIPTION
ID	Move ID for this automatic move input:

  GT.0: Velocity controlled tool kinematics (the variable VAD = 0 in *BOUNDARY_PRESCRIBED_MOTION_RIGID)

  LT.0: Displacement controlled tool kinematics (VAD = 2)

CONTID	Contact ID, as in *CONTACT_FORMING_..._ID, which defines the surfa and surfb part set IDs.
VID	Vector ID of a vector oriented in the direction of movement of the surfb surface, as in *DEFINE_VECTOR. The origin of the vector is unimportant since the direction cosines of the vector are computed and used.
LCID	Load curve defining tooling kinematics, either by velocity as a function of time or by displacement as a function of time. This load curve will be adjusted automatically during a simulation to close the empty tool travel.
ATIME	Activation time specifying the moment the surfb surface (tool) will be moved

VARIABLE	DESCRIPTION
OFFSET	Time at which a surfb surface will move to close a gap distance, which may happen following the move of another surfb surface. This is useful for sequential multiple flanging or press hemming simulations. Simulation time (CPU) is much faster based on the shortened tool travel (no change to the termination time).

Example: gravity loading and closing with implicit static

Referring to the partial input deck below and Figure 11-29, a combined simulation of gravity loading and binder closing of a fender outer is demonstrated on the NUMISHEET 2002 benchmark. In this multistep implicit static set up, the blank is allocated 0.3 "time" units (3 implicit steps for DT0 = 0.1) to be loaded with gravity. At the end of gravity loading, a gap of 12mm was created between the upper die and the blank (see Figure 11-30). The upper die is set to be moved at 0.3 "time" units, closing the gap caused by the gravity effect on the blank (see Figure 11-31 left). An intermediate closing state is shown at $t = 0.743$ (see Figure 11-31 right) while the final completed closing is shown in Figure 11-32. It is noted that the upper die is controlled with displacement (VAD = 2) in the shape of a right triangular in the displacement versus "time" space as defined by load curve #201, and the ID in *CONTACT_AUTO_MOVE is set to "-1".

```
*PARAMETER
R grvtime       0.3
R endtime       1.0
R diemv      145.45
*CONTROL_TERMINATION
&endtime
*CONTROL_IMPLICIT_FORMING
2,2,100
*CONTROL_IMPLICIT_GENERAL
$   IMFLAG        DT0
        1       0.10
*CONTROL_ACCURACY
        1        2
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_ID
11
....
....
....
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*BOUNDARY_PRESCRIBED_MOTION_RIGID
$#     pid        dof        vad       lcid         sf        vid      death      birth
        2          3          2        201  -1.000000          0        0.0      0.000
*CONTACT_AUTO_MOVE
$      ID    ContID        VID       LCID      ATIME
       -1        11         89        201   &grvtime
*DEFINE_VECTOR
89,0.0,0.0,0.0,0.0,0.0,-10.0
*DEFINE_CURVE
201
0.0,0.0
&grvtime,0.0
1.0,&diemv
```

Similarly, "velocity" controlled tool kinematics is also enabled. In the example keyword below, the "velocity" profile is ramped up initially and then kept constant. Note that the variable VAD in *BOUNDARY is set to "0", and ID in *CONTACT_AUTO_MOVE is set to positive "1" indicating it is a velocity boundary condition.

```
*PARAMETER
R grvtime        0.3
R tramp         0.001
R diemv        145.45
R clsv         1000.0
*PARAMETER_EXPRESSION
R tramp1  tramp+gravtime
R endtime tramp1+(abs(diemv)-0.5*clsv*tramp)/clsv
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_ID
11
....
....
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*BOUNDARY_PRESCRIBED_MOTION_RIGID
$#     pid        dof        vad       lcid         sf        vid      death      birth
        2          3          0        201  -1.000000          0        0.0      0.000
*CONTACT_AUTO_MOVE
$       ID     ContID        VID       LCID       ATIME
        1         11         89        201     &grvtime
*DEFINE_VECTOR
89,0.0,0.0,0.0,0.0,0.0,-10.0
*DEFINE_CURVE
201
0.0,0.0
0.2,0.0
&tramp1,&clsv
&endtime,&clsv
```

Example: tool delay in sequential flanging process with explicit dynamic:

The following example demonstrates the use of the variable OFFSET. As shown in Figure 11-33 (left), a total of 5 flange steels are auto-positioned initially according to the initial blank shape. Upon closing the pressure pad, the first set of 4 flanging steels moves to home, completing the first stage of the stamping process (see Figure 11-33 right).

The gap created by the completion of the first flanging process is closed automatically at a time defined using variables ATIME/OFFSET (see Figure 11-34 left). During the second stage of the process, flanging steel *&flg5pid* moves to home completing the final flanging (see Figure 11-34 right). An excerpt from the input deck for this model can be found below. This deck was created using LS-PrePost's eZ-Setup feature (http://ftp.lstc.com/-anonymous/outgoing/lsprepost/), with two additional keywords added:*CONTACT_-AUTO_MOVE and *DEFINE_VECTOR.

Flanging steel #5 is set to move in a cam angle defined by vector #7 following the completion of the flanging (straight down) process of flanging steel #2. The variables ATIME and OFFSET in *CONTACT_AUTO_MOVE are both defined as &endtim4, which is calculated based on the automatic positioning of tools/blank using *CONTROL_FORMING_AUTOPOSITION. At a defined time, flanging steel #5 'jumps' into position to where

it just comes into contact with the partially formed down-standing flange, saving some CPU times (see Figure 11-34 left). Flanging steel #5 continues to move to its home position completing the simulation (see Figure 11-34 right). The CPU time saving is 27% in this case.

```
*KEYWORD
*PARAMETER
...
*PART

  &flg5pid  &flg5sec  &flg5mid
...
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$ Local coordinate system for flanging steel #5 move direction
*DEFINE_COORDINATE_SYSTEM
$#     cid        xo        yo        zo        xl        yl        zl
  &flg5cid  -5.09548   27.6584  -8.98238  -5.43587   26.8608  -9.48034
$#     xp        yp        zp
  -5.82509   27.5484  -8.30742
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$ Auto positioning
*CONTROL_FORMING_AUTOPOSITION_PARAMETER_SET
$     SID       CID       DIR      MPID  POSITION   PREMOVE     THICK  PARORDER
...
  &flg5sid  &flg5cid        3  &blk1sid        -1            &bthick    flg5mv
*PART_MOVE
$    PID          XMOV          YMOV          ZMOV   CID   IFSET
&flg5sid           0.0           0.0        &flg5mv&flg5cid       1
...
*MAT_RIGID
$     MID        RO         E        PR         N    COUPLE         M     ALIAS
  &flg5mid 7.830E-09 2.070E+05      0.28
$     CMO      CON1      CON2
       -1  &flg5cid    110111
$LCO or A1        A2        A3        V1        V2        V3
  &flg5cid
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*CONTACT_AUTO_MOVE
$      ID    CONTID       VID      LCID     ATIME    OFFSET
        1         7         7        10  &endtim4  &endtim4
*DEFINE_VECTOR
$     VID        XT        YT        ZT        XH        YH        ZH
        7       0.0       0.0       0.0-0.5931240 0.5930674-0.5444952
*CONTACT_FORMING_ONE_WAY_SURFACE_TO_SURFACE_ID
$     CID
        7
$   SURFA     SURFB  SURFATYP  SURFBTYP   SABOXID   SBBOXID      SAPR      SBPR
  &blk1sid  &flg5sid         2         2                           1         1
$      FS        FD        DC        VC       VDC    PENCHK        BT        DT
...
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
$                    Tool kinematics
$  -----------------------closing
*BOUNDARY_PRESCRIBED_MOTION_RIGID_local
...
  &flg5pid         3         0         4       1.0         0  &endtim4
$  -----------------------flanging
*BOUNDARY_PRESCRIBED_MOTION_RIGID_local
...
  &flg5pid         3         0        10       1.0         0            &endtim4
$---+----1----+----2----+----3----+----4----+----5----+----6----+----7----+----8
*END
```
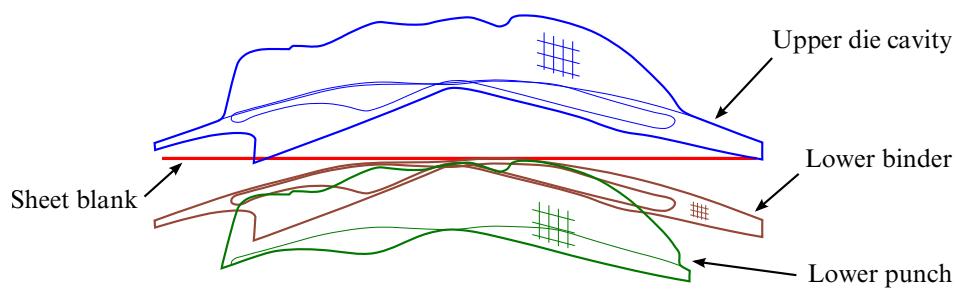
Figure 11-29. Initial parts auto-positioned at $t = 0.0$.
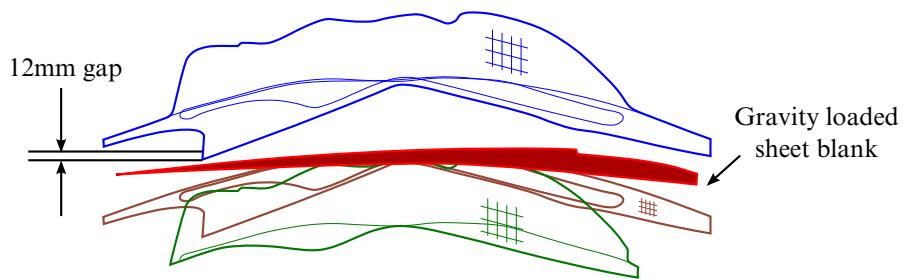
Figure 11-30. Gravity loading on blank at $t = 0.2$.
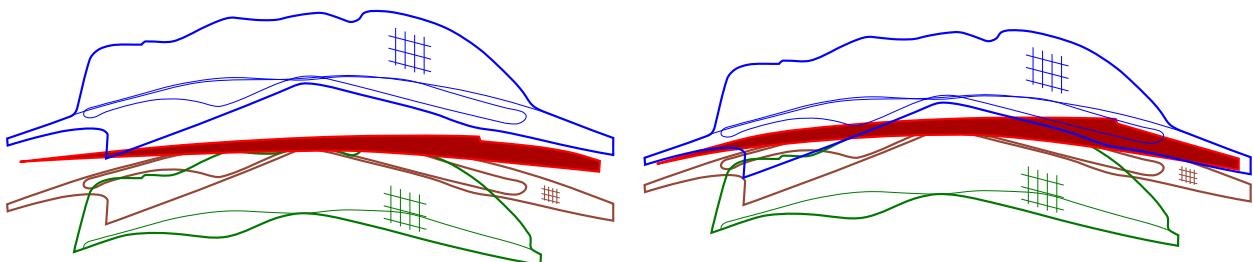
Figure 11-31. Upper die moved down at $t = 0.3$ to close the gap (left); continue closing at $t = 0.743$ (right).
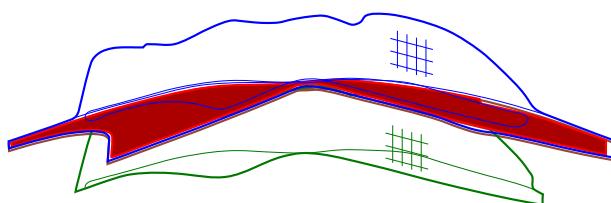
Figure 11-32. Closing complete at $t = 1.0$.

Flanging Steel #2: 0

Flanging steel #5:
&flg5pid

$t = 0$

$t = 0.00954$
ATIME, OFFSET, &endtim4
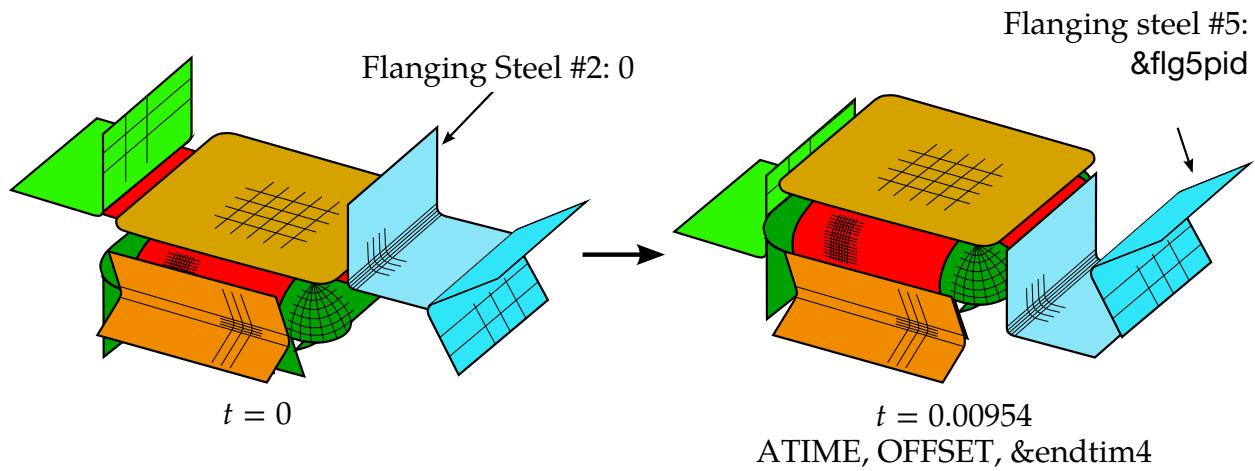
Figure 11-33. A sequential flanging process (left); first set of flanging steels reaching home (right).

$t = 0.018743$

$t = 0.026022$

Gap closes. This happens in an "instant," meaning the time step does not increment.

&flg5pid finish flanging
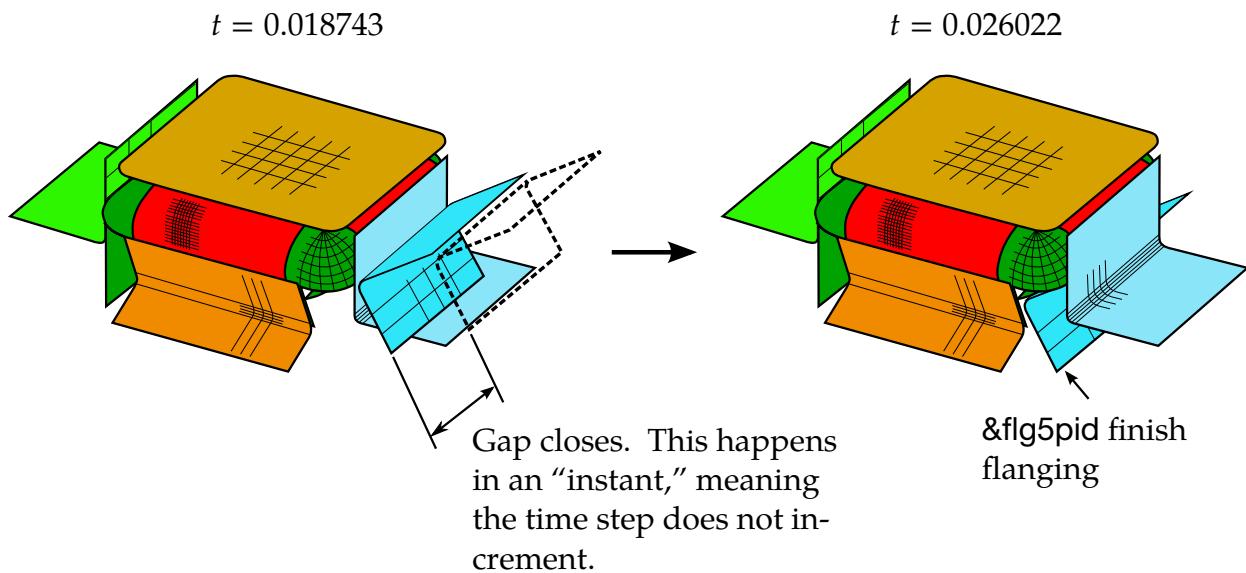
Figure 11-34. Closing the empty travel (left); flanging steel &flg5pid completes flanging process (right).

*CONTACT_COUPLING

Purpose: Define a coupling surface for MADYMO to couple LS-DYNA with deformable and rigid parts within MADYMO. In this interface, MADYMO computes the contact forces acting on the coupling surface, and LS-DYNA uses these forces in the update of the motion of the coupling surface for the next time step. Contact coupling can be used with other coupling options in LS-DYNA.

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

Set Cards. Include on card for each coupled set. The next keyword ("*") card terminates this input.

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

VARIABLE	DESCRIPTION
SID	Set ID for coupling. See Remark 1 below.
STYPE	Set type:
	EQ.0: part set
	EQ.1: shell element set
	EQ.2: solid element set
	EQ.3: thick shell element set

Remarks:

1. **Coupling Surface.** Only one coupling surface can be defined. If additional surfaces are defined, the coupling information will be added to the first definition.

2. **Units.** The units and orientation can be converted by using the *CONTROL_-COUPLING keyword. It is not necessary to use the same system of units in MADYMO and in LS-DYNA if unit conversion factors are defined.

*CONTACT_ENTITY

Purpose: Define a contact entity. Geometric contact entities treat the impact between a deformable body, defined as a set of tracked nodes or nodes in a shell part set, and a rigid body. The shape of the rigid body is determined by attaching geometric entities. Contact between these geometric entities and the tracked nodes is a penalty formulation. The penalty stiffness is optionally maximized within the constraint of the Courant criterion. As an alternative, a finite element mesh made with shells can be used as geometric entity. Also, axisymmetric entities with arbitrary shape made with multi-linear polygons are possible. The latter is particularly useful for metal forming simulations. See *DATA-BASE_GCEOUT for contact force output.

> **WARNING:** If the problem being simulated involves dynamic motion of the entity, care should be taken to ensure that the inertial properties of the entity are correct. These properties may need to be specified with *PART_IN-ERTIA.

Card Summary:

Card 1. This card is required.

PID	GEOTYP	SURFA	SURFATYP	SF	DF	CF	INTORD

Card 2. This card is required.

BT	DT	SO	GO	ITHK	SAPR		

Card 3. This card is required.

XC	YC	ZC	AX	AY	AZ		

Card 4. This card is required.
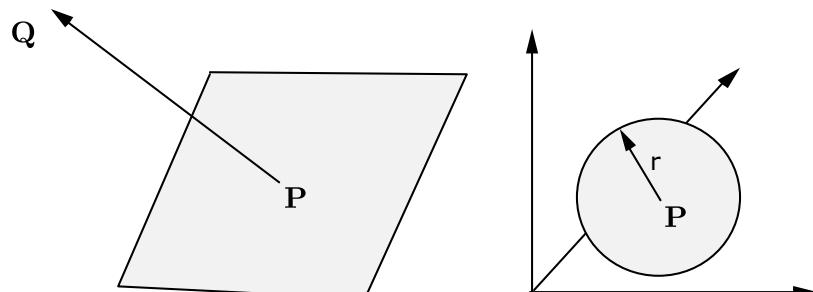
BX	BY	BZ					

Card 5. This card is required.
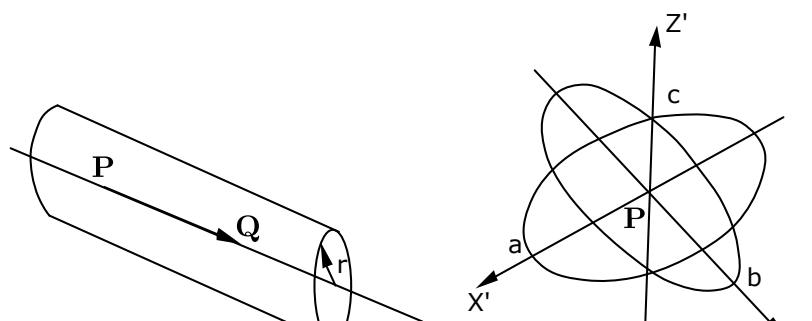
INOUT	G1	G2	G3	G4	G5	G6	G7
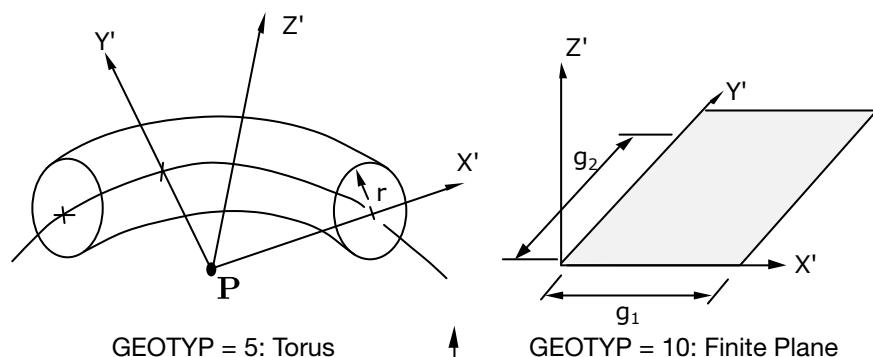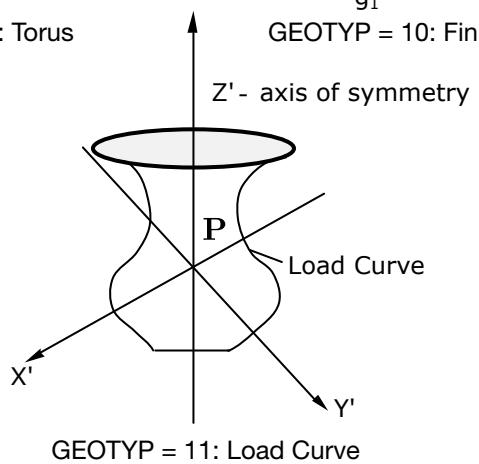
GEOTYP = 1: Infinite Plane

GEOTYP = 2: Sphere

GEOTYP = 3: Infinite Cylinder

GEOTYP = 4: Hyperellipsoid

GEOTYP = 5: Torus

GEOTYP = 10: Finite Plane

GEOTYP = 11: Load Curve

Figure 11-35. Contact Entities

Data Card Definitions:

Card 1	1	2	3	4	5	6	7	8
Variable	PID	GEOTYP	SURFA	SURFATYP	SF	DF	CF	INTORD
Type	I	I	I	I	F	F	F	I
Default	none	none	none	0	1.	0.	0.	0

VARIABLE	DESCRIPTION

PID   Part ID of the rigid body to which the geometric entity is attached, see *PART.

GEOTYP   Type of geometric entity (see Figure 11-35):

    EQ.1: Plane,

    EQ.2: Sphere,

    EQ.3: Cylinder,

    EQ.4: Ellipsoid,

    EQ.5: Torus,

    EQ.6: CAL3D/MADYMO Plane, see Appendix I,

    EQ.7: CAL3D/MADYMO Ellipsoid, see Appendix I,

    EQ.8: VDA surface, see Appendix L,

    EQ.9: Rigid body finite element mesh (shells only),

    EQ.10: Finite plane,

    EQ.11: Load curve defining line as surface profile of axisymmetric rigid bodies.

SURFA   Tracked surface set ID, see *SET_NODE_*OPTION*, *PART, or *SET_PART.

SURFATYP   Tracked surface set type:

    EQ.0: Node set

    EQ.1: Part

    EQ.2: Part set

VARIABLE	DESCRIPTION
SF	Penalty scale factor. Useful to scale maximized penalty.
DF	Damping option (see description for *CONTACT_*OPTION*):

> EQ.0: No damping,
>
> GT.0: Viscous damping in percent of critical, e.g., 20 for 20% damping,
>
> LT.0: DF must be a negative integer. -DF is the load curve ID giving the damping force as a function of relative normal velocity (see Remark 1 below).

CF	Coulomb friction coefficient. See Remark 2 below.

> EQ.0: No friction
>
> GT.0: Constant friction coefficient
>
> LT.0: CF must be a negative integer. -CF is the load curve ID giving the friction coefficient as a function of time.

INTORD	Integration order (tracked materials only). This option is not available with entity types 8 or 9 where only nodes are checked:

> EQ.0: check nodes only,
>
> EQ.1: 1 point integration over segments,
>
> EQ.2: 2×2 integration,
>
> EQ.3: 3×3 integration,
>
> EQ.4: 4×4 integration,
>
> EQ.5: 5×5 integration.

> This option allows a check of the penetration of the rigid body into the deformable (tracked) material. Then virtual nodes at the location of the integration points are checked.

Card 2	1	2	3	4	5	6	7	8
Variable	BT	DT	SO	GO	ITHK	TPR		
Type	F	F	I	I	I	I		
Default	0.	10^{20}	0	0	0	0		

VARIABLE	DESCRIPTION
BT	Birth time
DT	Death time
SO	Flag to use penalty stiffness as in surface-to-surface contact:

> EQ.0: Contact entity stiffness formulation,
>
> EQ.1: Surface to surface contact method,
>
> EQ.2: Normal force is computed via a constraint-like method. The contact entity is considered to be infinitely massive, so this is recommended only for entities with constrained motion.
>
> LT.0: SO must be an integer. -SO is the load curve ID giving the force versus the normal penetration. See Remark 1.

GO	Flag for automatic meshing of the contact entity for entity types 1 - 5 and 10 - 11. GO = 1 creates null shells for visualization of the contact entity. Note these shells have mass and will affect the mass properties of the rigid body PID unless *PART_INERTIA is used for the rigid body.

> EQ.0: Mesh is not generated.
>
> EQ.1: Mesh is generated.

ITHK	Flag for considering thickness for shell tracked nodes (applies only to entity types 1, 2, 3; SURFATYP must be set to zero).

> EQ.0: Shell thickness is not considered.
>
> EQ.1: Shell thickness is considered.

SAPR	Include the tracked side in *DATABASE_BINARY_INTFOR interface force files; valid only when SURFATYP > 0:

> EQ.1: Tracked side forces included.

Card 3	1	2	3	4	5	6	7	8
Variable	XC	YC	ZC	AX	AY	AZ		
Type	F	F	F	F	F	F		
Default	0.	0.	0.	0.	0.	0		

Card 4	1	2	3	4	5	6	7	8
Variable	BX	BY	BZ					
Type	F	F	F					
Default	0.	0.	0.					

VARIABLE	DESCRIPTION
XC	x-center, x_c. See Remark 3.
YC	y-center, y_c. See Remark 3.
ZC	z-center, z_c. See Remark 3.
AX	x-direction for local axis $\mathbf{A}$, A_x. See Remark 3.
AY	y-direction for local axis $\mathbf{A}$, A_y. See Remark 3.
AZ	z-direction for local axis $\mathbf{A}$, A_z. See Remark 3.
BX	x-direction for local axis $\mathbf{B}$, B_x. See Remark 3.
BY	y-direction for local axis $\mathbf{B}$, B_y. See Remark 3.
BZ	z-direction for local axis $\mathbf{B}$, B_z. See Remark 3.

Card 5	1	2	3	4	5	6	7	8
Variable	INOUT	G1	G2	G3	G4	G5	G6	G7
Type	I	F	F	F	F	F	F	F
Default	0	0.	0.	0.	0.	0.	0.	0.

VARIABLE	DESCRIPTION

INOUT

In-out flag. Allows contact from the inside or the outside (default) of the entity:

> EQ.0: Tracked nodes exist outside of the entity.

> EQ.1: Tracked nodes exist inside the entity.

G1

Entity coefficient g_1 (CAL3D/MADYMO plane or ellipse number) for coupled analysis (see Appendix I). See Geometric Contact Entity Coefficients below.

G2

Entity coefficient g_2. See Geometric Contact Entity Coefficients below.

G3

Entity coefficient g_3. See Geometric Contact Entity Coefficients below.

G4

Entity coefficient g_4. See Geometric Contact Entity Coefficients below.

G5

Entity coefficient g_5. See Geometric Contact Entity Coefficients below.

G6

Entity coefficient g_6. See Geometric Contact Entity Coefficients below.

G7

Entity coefficient g_7. See Geometric Contact Entity Coefficients below.

Remarks:

1. **Damping and Contact Load Curves.** The optional load curves that are defined for damping versus relative normal velocity and for force versus normal penetration should be defined in the positive quadrant. The sign for the damping

force depends on the direction of the relative velocity and the treatment is symmetric if the damping curve is in the positive quadrant. If the damping force is defined in the negative and positive quadrants, the sign of the relative velocity is used in the table look-up.

2. **Friction Coefficient.** If at any time the friction coefficient is ≥ 1.0, the force calculation is modified to a constraint like formulation which allows no sliding. This is only recommended for entities with constrained motion since the mass of the entity is assumed to be infinite.

3. **Geometric Entity Coordinate System.** The coordinates, (x_c, y_c, z_c), are the positions of the local origin of the geometric entity in global coordinates. The entity's local A-axis is determined by the vector (A_x, A_y, A_z) and the local B-axis by the vector (B_x, B_y, B_z).

   Cards 3 and 4 define a local to global transformation. The geometric contact entities are defined in a local system and transformed into the global system. For the ellipsoid, this is necessary because it has a restricted definition for the local position. For the plane, sphere, and cylinder, the entities can be defined in the global system and the transformation becomes $(x_c, y_c, z_c) = (0,0,0)$, $(A_x, A_y, A_z) = (1,0,0)$, and $(B_x, B_y, B_z) = (0,1,0)$.

Geometric Contact Entity Coefficients:

Figure 11-35 shows the definitions of the geometric contact entities. The relationships between the entity coefficients and the Figure 11-35 variables are as described below. Note that (P_x, P_y, P_z) defines a point and (Q_x, Q_y, Q_z) is a direction vector.

GEOTYP = 1

$$g_1 = P_x \qquad\qquad g_4 = Q_x$$
$$g_2 = P_y \qquad\qquad g_5 = Q_y$$
$$g_3 = P_z \qquad\qquad g_6 = Q_z$$
$$\qquad\qquad\qquad g_7 = L$$

If automatic generation is used, a square plane of length L on each edge is generated which represents the infinite plane. If generation is inactive, then g_7 may be ignored.

GEOTYP = 2

$$g_1 = P_x \qquad\qquad g_4 = r$$
$$g_2 = P_y$$
$$g_3 = P_z$$

GEOTYP = 3

$$g_1 = P_x \qquad\qquad g_4 = Q_X$$
$$g_2 = P_y \qquad\qquad g_5 = Q_y$$
$$g_3 = P_z \qquad\qquad g_6 = Q_z$$
$$\qquad\qquad\qquad\qquad g_7 = r$$

If automatic generation is used, a cylinder of length $\sqrt{Q_x^2 + Q_y^2 + Q_z^2}$ and radius r is generated which represents the infinite cylinder. The midpoint of the displayed cylinder is at point P.

GEOTYP = 4

$$g_1 = P_x \qquad\qquad g_4 = a$$
$$g_2 = P_y \qquad\qquad g_5 = b$$
$$g_3 = P_z \qquad\qquad g_6 = c$$
$$\qquad\qquad\qquad\qquad g_7 = n \text{ (order of the ellipsoid)}$$

GEOTYP = 5

$$g_1 = \text{Radius of torus}$$
$$g_2 = r$$
$$g_3 = \text{number of elements along minor circumference}$$
$$g_4 = \text{number of elements along major circumference}$$

GEOTYP = 8

$$g_1 = \text{Blank thickness (option to override true thickness)}$$
$$g_2 = \text{Scale factor for true thickness (optional)}$$
$$g_3 = \text{Load curve ID defining thickness versus time. (optional)}$$

GEOTYP = 9

$$g_1 = \text{Shell thickness (option to override true thickness).}$$
$$g_2 = \text{Scale factor for true thickness (optional)}$$
$$g_3 = \text{Load curve ID defining thickness versus time. (optional)}$$

Note: The shell thickness specification is necessary if the tracked surface is generated from solid elements.

GEOTYP = 10

$$g_1 = \text{Length of edge along } X' \text{ axis}$$
$$g_2 = \text{Length of edge along } Y' \text{ axis}$$

GEOTYP = 11

g_1 = Load curve ID defining axisymmetric surface profile about the *Z*-axis. Load curves defined by the keywords *DEFINE_CURVE or *DEFINE_CURVE_ENTITY can be used.

g_2 = Number of elements along circumference

   EQ.0: default set to 10

g_3 = Number of elements along axis

   EQ.0: default set to 20

   EQ.-1: the elements generated from points on the load curve

g_4 = Number of sub divisions on load curve used to calculate contact

   EQ.0: default set to 1000

*CONTACT_EXCLUDE_INTERACTION

Purpose: Specify portions of a contact interface to exclude from having a contact interaction. The portions of the interface are specified with sets. If a single set is given, the contact algorithm excludes all contact with segments in that set. If two sets are defined, the contact algorithm excludes only contact between segments in set A from contact with segments in set B. This option is only available for the segment-to-segment contact that is invoked by setting SOFT = 2 on optional Card A of *CONTACT.

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

Set Cards. Include as many lines as needed to specify different portions of the interface to be excluded from contact. The next keyword ("*") card terminates this input.

Card 2	1	2	3	4	5	6	7	8
Variable	SIDA	SIDB	TYPEA	TYPEB				
Type	I	I	I	I				
Default	0	0						

VARIABLE	DESCRIPTION
CEID	Contact exclusion ID for output only
CID	Contact interface ID for limiting exclusions to an interface
SIDA	Set ID of set A
SIDB	Set ID of set B
TYPEA	ID type of set SIDA:
	EQ.0: Segment set
	EQ.1: Shell element set

VARIABLE	DESCRIPTION
	EQ.2: Part set
TYPEB	ID type of set SIDB:
	EQ.0: Segment set
	EQ.1: Shell element set
	EQ.2: Part set

Remarks:

There are two ways to use this keyword. If both SIDA and SIDB are defined, interface CID excludes from contact all segment pairs where one segment is in set SIDA and the other in set SIDB. The second way is to define either SIDA or SIDB but not both. When this is done, contact interface CID excludes from contact any segment pair if either segment is in SIDA or SIDB.

During initialization, segments in SIDA or SIDB are compared with SURFA and SURFB, and a list of matches is made for exclusion. If CID is a single-surface type contact, SURFA is searched for matches with either SIDA or SIDB or both. If CID is a surface-to-surface type contact, SURFA is searched for matches with SIDA, and SURFB is searched for matches with SIDB.

*CONTACT_FORCE_TRANSDUCER_OPTION1_{OPTION2}_

Purpose: Create a force transducer to measure the contact force of a 3D contact. LS-DYNA only outputs data between a SURFA and SURFB surface. Self-contact, such as single surface contact, only has a SURFA definition. In the case of self-contact, force transducers provide a means to record the self-contact forces. Force transducers are not needed for other types of contact. However, they may be useful for extracting the contact forces for any part of the model. For instance, if you have two-way contact, you may only want to extract the forces on a section of SURFB. Force transducers do not apply any force to the model nor do they have any effect on the solution.

This keyword is *not* implemented for 2D contact.

Available options for *OPTION1* are:

      CONSTRAINT

      PENALTY

The CONSTRAINT option extracts forces from constraint-based contact types while the PENALTY option extracts forces only from penalty-based contacts.

Available options for *OPTION2* are:

      ID

In the output files, the contact ID identifies the results from the force transducer measurements. If the contact ID is undefined, the default ID is determined by the sequence of the contact definitions, that is, the first contact definition has an ID of 1, the second, 2, and so forth. The ID and heading are picked up by some of the peripheral LS-DYNA codes to aid in post-processing.

Card ID. Additional card for ID keyword option

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

Card 1	1	2	3	4	5	6	7	8
Variable	SURFA	SURFB	SURFATYP	SURFBTYP	SABOXID	SBBOXID	SAPR	SBPR
Type	I	I	I	I	I	I	I	I
Default	none	optional	none	optional	optional	optional	0	0

This card must be included as a blank line.

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

This card must be included as a blank line.

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

VARIABLE	DESCRIPTION
CID	Contact interface ID. This must be a unique number.
HEADING	Interface descriptor. We suggest using unique descriptions.
SURFA	Segment set ID, node set ID, part set ID, part ID, or shell element set ID specifying the SURFA side of the force transducer; see *SET_SEGMENT, *SET_NODE_*OPTION*, *PART, *SET_PART or *SET_SHELL_*OPTION*. EQ.0: Includes all parts.
SURFB	Segment set ID, node set ID, part set ID, part ID, or shell element set ID specifying the SURFB side of the force transducer. A SURFB set is not required for force transducers, See Remark 1.

VARIABLE	DESCRIPTION
SURFATYP	ID type of SURFA:

SURFATYP — ID type of SURFA:

    EQ.0: Segment set ID

    EQ.1: Shell element set ID

    EQ.2: Part set ID

    EQ.3: Part ID

    EQ.4: Node set ID. See Remark 2.

    EQ.5: Include all (SURFA is ignored)

    EQ.6: Part set ID for exempted parts. All non-exempted parts are included in the force transducer.

    EQ.7: Branch ID; see *SET_PART_TREE.

SURFBTYP — ID type of SURFB:

    EQ.0: Segment set ID

    EQ.1: Shell element set ID

    EQ.2: Part set ID

    EQ.3: Part ID

    EQ.4: Node set ID. See Remark 2.

    EQ.5: Include all (SURFB is ignored).

    EQ.6: Part set ID for exempted parts. All non-exempted parts are included in the force transducer.

    EQ.7: Branch ID; see *SET_PART_TREE.

SABOXID — Include in force transducer definition only those SURFA nodes/segments within box SABOXID (corresponding to BOXID in *DEFINE_BOX), or if SABOXID is negative, only those SURFA nodes/segments within contact volume |SABOXID| (corresponding to CVID in *DEFINE_CONTACT_VOLUME). SABOXID can be used only if SURFATYP is set to 2 or 3, that is, SURFA is a part ID or part set ID

SBBOXID — Include in force transducer definition only those SURFB segments within box SBBOXID (corresponding to BOXID in *DEFINE_BOX), or if SBBOXID is negative, only those SURFB segments within contact volume |SBBOXID| (corresponding to CVID in *DEFINE_CONTACT_VOLUME). SBBOXID can be used only if SURFBTYP is set to 2 or 3, that is, SURFB is a part ID or part set ID.

VARIABLE	DESCRIPTION
SAPR	Include the SURFA side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files, and optionally in the dynain file for wear: EQ.0: Do not include. EQ.1: SURFA side forces included. EQ.2: Same as 1 but also allows for SURFA nodes to be written as *INITIAL_CONTACT_WEAR to dynain; see NCYC on *INTERFACE_SPRINGBACK_LSDYNA.
SBPR	Include the SURFB side in the *DATABASE_NCFORC and the *DATABASE_BINARY_INTFOR interface force files, and optionally in the dynain file for wear: EQ.0: Do not include. EQ.1: SURFB side forces included. EQ.2: Same as 1, but also allows for SURFB nodes to be written as *INITIAL_CONTACT_WEAR to dynain; see NCYC on *INTERFACE_SPRINGBACK_LSDYNA.

Remarks:

1. **SURFB set with force transducers.** A SURFB set ID (SURFB and SURFBTYP) is optional for force transducers. If the contact forces acting between two particular surfaces are needed, a SURFB surface should be defined. In this case, only the contact forces applied between SURFA and SURFB are recorded. Otherwise, the contact forces between SURFA and any surface are output. If a transducer is used for extracting forces from Mortar contacts, the SURFA and SURFB sides *must* be defined through parts or part sets; segment and node sets will not gather the correct data.

    > **NOTE:** The SURFB surface option of FORCE_TRANSDUC-ER is only implemented for the PENALTY option. It works only in conjunction with Mortar contacts, AUTOMATIC_SURFACE_TO_SURFACE, ERODING_-SINGLE_SURFACE, AUTOMATIC_SINGLE_SUR-FACE contact types, and groupable AUTOMATIC_-GENERAL contact types (MPP only), except as noted in the next remark.

2. **SURFA/SURFB node sets.** Using node sets to specify the surfaces requires the PENALTY keyword option. Additionally, if either SURFA or SURFB is a node set, both must be node sets. Node sets allow the transducer to give correct results for eroding materials. The node sets should include all nodes that may be exposed as erosion occurs.

3. **Force output.** Contact force data can be recorded with *DATABASE_RCFORC, *DATABASE_NCFORC, and *DATABASE_BINARY_INTFOR. The contact energy data for force transducers can be recorded with *DATABASE_SLEOUT, but this data is only recorded for MPP.

*CONTACT_GEBOD_OPTION_

Purpose: Define contact interaction between the segment of a GEBOD dummy and parts or nodes of the finite element model. This implementation follows that of the contact entity, however, it is specialized for the dummies. Forces may be output using the *DATABASE_GCEOUT command. See *COMPONENT_GEBOD and Appendix N for further details.

Conventional *CONTACT_*OPTION* treatment (surface-to-surface, nodes-to-surface, etc.) can also be applied to the segments of a dummy. To use this approach, the part ID assignments must be first determined by running the model through LS-DYNA's initialization phase.

The following options are available and refer to the ellipsoids which comprise the dummy. Options involving HAND are *not* applicable for the child dummy since its lower arm and hand share a common ellipsoid.

LOWER_TORSO	RIGHT_LOWER_ARM
MIDDLE_TORSO	LEFT_HAND
UPPER_TORSO	RIGHT_HAND
NECK	LEFT_UPPER_LEG
HEAD	RIGHT_UPPER_LEG
LEFT_SHOULDER	LEFT_LOWER_LEG
RIGHT_SHOULDER	RIGHT_LOWER_LEG
LEFT_UPPER_ARM	LEFT_FOOT
RIGHT_UPPER_ARM	RIGHT_FOOT
LEFT_LOWER_ARM	

Card 1	1	2	3	4	5	6	7	8
Variable	DID	SURFA	SURFATYP	SF	DF	CF	INTORD	
Type	I	I	I	F	F	F	I	
Default	none	none	none	1.	20.	0.5	0	

VARIABLE	DESCRIPTION
DID	Dummy ID, see *COMPONENT_GEBOD_*OPTION.*
SURFA	Tracked surface set ID, see *SET_NODE_*OPTION*, *PART, or *SET_PART.
SURFATYP	SURFA set type:

SURFATYP set type continued:

    EQ.0: Node set

    EQ.1: Part

    EQ.2: Part set

SF	Penalty scale factor. Useful to scale maximized penalty.
DF	Damping option, see description for *CONTACT_*OPTION*:

    EQ.0.0: No damping,

    GT.0.0: Viscous damping in percent of critical, e.g., 20 for 20% damping,

    LT.0.0: DF must be an integer. -DF is the load curve ID giving the damping force as a function of relative normal velocity (see Remark 1 below).

CF	Coulomb friction coefficient (see Remark 2 below). Assumed to be constant.
INTORD	Integration order (tracked materials only).

    EQ.0: Check nodes only,

    EQ.1: 1 point integration over segments,

    EQ.2: 2×2 integration,

    EQ.3: 3×3 integration,

    EQ.4: 4×4 integration,

    EQ.5: 5×5 integration.

This option allows a check of the penetration of the dummy segment into the deformable (tracked) material. Then virtual nodes at the location of the integration points are checked.

Card 2	1	2	3	4	5	6	7	8
Variable	BT	DT	SO					
Type	F	F	I					
Default	0.	10^{20}	0					

VARIABLE	DESCRIPTION

BT   Birth time

DT   Death time

SO   Flag to use penalty stiffness as in surface-to-surface contact:

        EQ.0: Contact entity stiffness formulation,

        EQ.1: Surface to surface contact method,

        LT.0: In this case SO must be an integer. |SO| references the load curve ID which defines the force as a function of the normal penetration. See Remark 1.

Remarks:

1. **Load Curves.** The optional load curves that are defined for damping as a function relative normal velocity and for force as a function of normal penetration should be defined in the positive quadrant. The sign for the damping force depends on the direction of the relative velocity and the treatment is symmetric if the damping curve is in the positive quadrant. If the damping force is defined in the negative and positive quadrants, the sign of the relative velocity is used in the table look-up.

2. **Friction and Contact.** Insofar as these ellipsoidal contact surfaces are continuous and smooth it may be necessary to specify Coulomb friction values larger than those typically used with faceted contact surfaces.

CONTACT_GUIDED_CABLE_*{OPTION1}_{OPTION2}

Purpose: Define a sliding contact that guides 1D elements, such as springs, trusses, and beams, along a path defined by a set of nodes. Only one 1D element can be in contact with any given node in the node set at a given time. If for some reason, a node is in contact with multiple 1D elements, one guided contact definition must be used for each contact. The ordering of the nodal points and 1D elements in the input is arbitrary.

OPTION1 specifies that a part set ID is given with the single option:

    <BLANK>

    SET

If not used, a part ID is assumed.

OPTION2 specifies that the first card read defines the heading and ID number of the contact interface and takes the single option:

    ID

Title Card. Additional card for ID keyword option.

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

VARIABLE	DESCRIPTION
CID	Contact interface ID. This must be a unique number.
HEADING	Interface descriptor. We suggest using unique descriptions.

Card 1	1	2	3	4	5	6	7	8
Variable	NSID	PID/PSID	SOFT	SSFAC	FRIC	ENDTOL		
Type	I	I	I	F	F	F		
Default	none	none	0	1.0	none	↓		

VARIABLE	DESCRIPTION
NSID	Node set ID that guides the 1D elements
PID/PSID	Part ID or part set ID if SET is included in the keyword line
SOFT	Flag for soft constraint option. Set to 1 for soft constraint.
SSFAC	Stiffness scale factor for penalty stiffness value. The default value is unity. This applies to SOFT set to 0 and 1.
FRIC	Contact friction
ENDTOL	Tolerance, in length units, applied at the ends of the cable elements beyond which contact will pass to the next cable element. The default is 0.002 times the element length.

*CONTACT_INTERIOR

Purpose: Define interior contact for solid elements. Frequently, when soft materials are compressed under high pressure, the solid elements used to discretize these materials may invert leading to negative volumes and error terminations. To keep these elements from inverting, it is possible to consider interior contacts between layers of interior surfaces made up of the faces of the solid elements. Since these interior surfaces are generated automatically, the part (material) ID's for the materials of interest are defined here, prior to the interface definitions.

Define as many cards as necessary. Input ends at the next keyword ("*") card. Multiple instances of this keyword may appear in the input.

Card 1	1	2	3	4	5	6	7	8
Variable	PSID1	PSID2	PSID3	PSID4	PSID5	PSID6	PSID7	PSID8
Type	I	I	I	I	I	I	I	I
Default	none	none	none	none	none	none	none	none

VARIABLE	DESCRIPTION
PSID*	Part set ID for which interior contact is desired.

Four attributes should be defined for each part set:

    Attribute 1:   PSF, penalty scale factor (Default = 1.00).

    Attribute 2:   Activation factor, F_a (Default = 0.10). When the crushing of the element reaches F_a times the initial thickness, the contact algorithm begins to act.

    Attribute 3:   ED, Optional modulus for interior contact stiffness.

    Attribute 4:   TYPE, Formulation for interior contact.

        EQ.1.0:   Default, recommended for uniform compression
        EQ.2.0:   Designed to control the combined modes of shear and compression. Works for type 1 brick formulation and type 10 tetrahedron formulation.

Define each part set with the *SET_PART_COLUMN option to specify independent attribute values for each part in the part set.

Remarks:

The interior penalty is determined by the formula:

$$K = \frac{\text{SLSFAC} \times \text{PSF} \times \text{Volume}^{2/3} \times \text{E}}{\text{Min. Thickness}}$$

where SLSFAC is the value specified on the *CONTROL_CONTACT card, volume is the volume of the brick element, E is a constitutive modulus, and min. thickness is approximately the thickness of the solid element through its thinnest dimension. If ED is defined above, the interior penalty is then given instead by:

$$K = \frac{\text{Volume}^{2/3} \times \text{ED}}{\text{Min. Thickness}}$$

where the scaling factors are ignored. Generally, ED should be taken as the locking modulus specified for the foam constitutive model.

Caution should be observed when using this option since if the time step size is too large an instability may result. The time step size is not affected by the use of interior contact.

*CONTACT_RIGID_SURFACE

Purpose: Define rigid surface contact. The purpose of rigid surface contact is to model large rigid surfaces, such as road surfaces, with nodal points and segments that require little storage and are written out at the beginning of the binary databases. The rigid surface motion, which can be optionally prescribed, is defined by a displacement vector which is written with each output state. The nodal points defining the rigid surface must be defined with *NODE_RIGID_SURFACE. These rigid nodal points do not contribute degrees-of-freedom.

Card 1	1	2	3	4	5	6	7	8
Variable	CID	PSID	BOXID	SSID	FS	FD	DC	VC
Type	I	I	I	I	F	F	F	F
Default	none	none	0	none	0.	0.	0.	0.

Card 2	1	2	3	4	5	6	7	8
Variable	LCIDX	LCIDY	LCIDZ	FSLCID	FDLCID			
Type	I	I	I	I	I			
Default	0	0	0	0	0			

Card 3	1	2	3	4	5	6	7	8
Variable	SFS	STTHK	SFTHK	XPENE	BSORT	CTYPE		
Type	F	F	F	F	I	I		
Default	1.0	0.0	1.0	4.0	10	0		

VARIABLE	DESCRIPTION
CID	Contact interface ID. This must be a unique number.

VARIABLE	DESCRIPTION
PSID	Part set ID of all parts that may contact the rigid surface. See *SET_PART.
BOXID	Include only nodes of the part set that are within the specified box, see *DEFINE_BOX, in contact. If BOXID is zero, all nodes from the part set, PSID, will be included in the contact.
SSID	Segment set ID defining the rigid surface. See *SET_SEGMENT.
FS	Static coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity $v_{\rm rel}$ of the surfaces in contact, $$\mu_c = {\rm FD} + ({\rm FS} - {\rm FD})e^{-{\rm DC}|v_{\rm rel}|}.$$ If FSLCID is defined, see below, then FS is overwritten by the value from the load curve.
FD	Dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity $v_{\rm rel}$ of the surfaces in contact, $$\mu_c = {\rm FD} + ({\rm FS} - {\rm FD})e^{-{\rm DC}|v_{\rm rel}|}.$$ If FDLCID is defined, see below, then FD is overwritten by the value from the load curve.
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity $v_{\rm rel}$ of the surfaces in contact $$\mu_c = {\rm FD} + ({\rm FS} - {\rm FD})e^{-{\rm DC}|v_{\rm rel}|}.$$
VC	Coefficient for viscous friction. This is necessary to limit the friction force to a maximum. A limiting force is computed, $$F_{\rm lim} = {\rm VC} \times A_{\rm cont}\ .$$ $A_{\rm cont}$ being the area of the segment contacted by the node in contact. The suggested value for VC is to use the yield stress in shear ${\rm VC} = \sigma_0/\sqrt{3}$ where σ_0 is the yield stress of the contacted material.
LCIDX	Load curve ID defining x-direction motion. If zero, there is no motion in the x-coordinate system.
LCIDY	Load curve ID defining y-direction motion. If zero, there is no motion in the y-coordinate system.
LCIDZ	Load curve ID defining z-direction motion. If zero, there is no motion in the z-coordinate system.

VARIABLE	DESCRIPTION
FSLCID	Load curve ID defining the static coefficient of friction as a function of interface pressure. This option applies to shell segments only.
FDLCID	Load curve ID defining the dynamic coefficient of friction as a function of interface pressure. This option applies to shell segments only.
SFS	Scale factor on default tracked surface penalty stiffness, see also *CONTROL_CONTACT.
STTHK	Optional thickness for tracked surface (overrides true thickness). This option applies to contact with shell, solid, and beam elements. True thickness is the element thickness of the shell elements. Thickness offsets are not used for solid element unless this option is specified.
SFTHK	Scale factor for tracked surface thickness (scales true thickness). This option applies only to contact with shell elements. True thickness is the element thickness of the shell elements.
XPENE	Contact surface maximum penetration check multiplier. If the penetration of a node through the rigid surface exceeds the product of XPENE and the tracked node thickness, the node is set free.
	EQ.0.0: default is set to 4.0.
BSORT	Number of cycles between bucket sorts. The default value is set to 10 but can be much larger, e.g., 50-100, for fully connected surfaces.
CTYPE	Contact formulation:
	EQ.0: Equivalent to the ONE_WAY_SURFACE_TO_SURFACE formulation (default)
	EQ.1: Penalty formulation. If the tracked surface belongs to a rigid body, this formulation must be used.

Remarks:

Thickness offsets do not apply to the rigid surface. There is no orientation requirement for the segments in the rigid surface, and the surface may be assembled from disjoint, but contiguous, arbitrarily oriented meshes. With disjoint meshes, the global searches must be done frequently, about every 10 cycles, to ensure a smooth movement of a tracked node between mesh patches. For fully connected meshes this frequency interval can be safely set to 50-200 steps between searches.

The modified binary database, d3plot, contains the road surface information prior to the state data. This information includes:

NPDS = Total number of rigid surface points in problem.

NRSC = Total number of rigid surface contact segments summed over all definitions.

NSID = Number of rigid surface definitions.

NVELQ = Number of words at the end of each binary output state defining the rigid surface motion. This equals 6 × NSID if any rigid surface moves or zero if all rigid surfaces are stationary.

PIDS = An array equal in length to NPDS. This array defines the ID for each point in the road surface.

XC = An array equal in length to 3 × NPDS. This array defines the global x, y, and z coordinates of each point.

For each road surface define the following NSID sets of data:

ID = Rigid surface ID.

NS = Number of segments in rigid surface.

IXRS = An array equal in length to 4 × NS. This is the connectivity of the rigid surface in the internal numbering system.

At the end of each state, 6 × NVELQ words of information are written. For each road surface the x, y, and z displacements and velocities are written. If the road surface is fixed, a null vector should be output. Skip this section if NVELQ = 0. LS-PrePost currently displays rigid surfaces and animates their motion.

*CONTACT_SPG

Purpose: Define contact between SPG particles from different SPG parts or self-contact of SPG particles from the same SPG part. This keyword was developed for high-speed deformations, such as projectile impact penetration problems.

Card 1	1	2	3	4	5	6	7	8
Variable	PID1	PID2	PID3	PID4	PID5	PID6	PID7	PID8
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

Card 2	1	2	3	4	5	6	7	8
Variable	ISELF1	ISELF2	ISELF3	ISELF4	ISELF5	ISELF6	ISELF7	ISELF8
Type	I	I	I	I	I	I	I	I
Default	0	0	0	0	0	0	0	0

Card 3	1	2	3	4	5	6	7	8
Variable	PFAC1	PFAC2	PFAC3	PFAC4	PFAC5	PFAC6	PFAC7	PFAC8
Type	F	F	F	F	F	F	F	F
Default	0.0	0.0	0.0	0.0	0.0	0.0	0.0	0.0

Card 4	1	2	3	4	5	6	7	8
Variable	FS	FD	DC	NFREQ				
Type	F	F	F	F				
Default	0.0	0.0	0.0	0.0				

VARIABLE	DESCRIPTION
PID*i*	Part ID of the SPG parts in particle contact.
ISELF*i*	Self-contact flag for PID*i*:
	EQ.0: No self-contact allowed for PIDi
	EQ.1: Self-contact allowed for PIDi
PFAC*i*	Penalty factor *i*
FS	Static coefficient of friction
FD	Dynamic coefficient of friction
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity, v_{rel}, of the surfaces in contact: $$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-\text{DC}|v_{\text{rel}}|}.$$
NFREQ	Contact searching frequency

*CONTACT_1D

Purpose: Define one-dimensional slide lines for rebar in concrete.

Card 1	1	2	3	4	5	6	7	8
Variable	NSIDR	NSIDC	ERR	SIGC	GB	SMAX	EXP	
Type	I	I	F	F	F	F	F	
Default	none	none	0.	0.	0.	0.	0.	

VARIABLE	DESCRIPTION
NSIDR	Nodal set ID for the rebar nodes that slide along the concrete; see *SET_NODE.
NSIDC	Nodal set ID for the concrete nodes that the rebar nodes may slide along; see *SET_NODE.
ERR	External radius of rebar
SIGC	Unconfined compressive strength of concrete, f_c
GB	Bond shear modulus
SMAX	Maximum shear strain
EXP	Exponent in damage curve

Remarks:

With this option the concrete is defined with solid elements and the rebar with truss elements, each with their own unique set of nodal points. A string of spatially <u>consecutive</u> nodes related to the truss elements (NSIDR) may slide along another string of spatially <u>consecutive</u> nodes related to the solid elements (NSIDC). The sliding commences after the rebar debonds.

The bond between the rebar and concrete is assumed to be elastic perfectly plastic. The maximum allowable slip strain is given as:

$$u_{\max} = \text{SMAX} \times e^{-\text{EXP} \times D}$$

where D is the damage parameter $D_{n+1} = D_n + \Delta u$. The shear force, acting on area A_S, at time $n + 1$ is given as:

$$f_{n+1} = \min[f_n - \text{GB} \times A_s \times \Delta u, \text{GB} \times A_s \times u_{\max}]$$

*CONTACT_2D_OPTION1_{OPTION2}_{OPTION3}_

Purpose: Define a two-dimensional contact interface or slide line. This option is to be used with two-dimensional solid and shell elements using the plane stress, plane strain or axisymmetric formulations; see *SECTION_SHELL and *SECTION_BEAM.

All the two-dimensional contacts are supported in SMP. Only *CONTACT_2D_AUTO-MATIC_SINGLE_SURFACE and *CONTACT_2D_AUTOMATIC_SURFACE_TO_SUR-FACE are supported for MPP, with and without the option MORTAR.

OPTION1 specifies the contact type. The following options activate kinematic constraints and should be used with deformable materials only but may be used with rigid bodies if the rigid body is the reference surface, and all rigid body motions are prescribed. Kinematic constraints are recommended for high pressure hydrodynamic applications.

    SLIDING_ONLY

    TIED_SLIDING

    SLIDING_VOIDS

    AUTOMATIC_TIED_ONE_WAY

The following option uses both kinematic constraints and penalty constraints.

    AUTOMATIC_TIED

The following options are penalty-based. These methods have no rigid-material limitations. We recommend them for lower pressure solid mechanics applications.

    PENALTY_FRICTION

    PENALTY

    AUTOMATIC_SINGLE_SURFACE

    AUTOMATIC_SINGLE_SURFACE_MORTAR

    AUTOMATIC_SURFACE_TO_SURFACE

    AUTOMATIC_SURFACE_TO_SURFACE_MORTAR

    AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE

    AUTOMATIC_SURFACE_IN_CONTINUUM

The following two contacts are used for SPH particles in contact with two-dimensional continuum elements (shell formulations 13, 14, 15). These contacts do not apply to two-dimensional shell elements (beam formulations 7, 8).

       NODE_TO_SOLID

       NODE_TO_SOLID_TIED

The following option is used to measure contact forces that are reported as RCFORC output.

       FORCE_TRANSDUCER

OPTION2 specifies a thermal contact and takes the single option:

       THERMAL

Only the AUTOMATIC contact options: SINGLE_SURFACE, SURFACE_TO_SURFACE, and ONE_WAY_SURFACE_TO_SURFACE including the Mortar contact versions may be used with the THERMAL option.

OPTION3 specifies that the first card to read defines the title and ID number of contact interface and takes the single option:

       TITLE

Title Card. Additional card for the TITLE keyword potion.

Card Title	1	2	3	4	5	6	7	8
Variable	CID	TITLE						
Type	I	A70						

Two-dimensional contact may be divided into 3 groups, each with a unique input format.

1.   The first group were adopted from LS-DYNA2D and originated in the public domain version of DYNA2D from the Lawrence Livermore National Laboratory. Contact surfaces are specified as ordered sets of nodes. These sets define either contact surfaces or slide lines. The keyword options for the first group are:

          SLIDING_ONLY

          TIED_SLIDING

          SLIDING_VOIDS

          PENALTY_FRICTION

PENALTY

> **NOTE:** We do not recommend TIED_SLIDING, PENALTY_-
> FRICTION and PENALTY since easier to use auto-
> matic options with same functionality exist in the
> second group.

2. The second group contains the automatic contacts. These contact surfaces may
    be defined using part sets or unordered node sets. Segment orientations are de-
    termined automatically. The keywords for these are:

    AUTOMATIC_SINGLE_SURFACE

    AUTOMATIC_SINGLE_SURFACE_MORTAR

    AUTOMATIC_SURFACE_TO_SURFACE

    AUTOMATIC_SURFACE_TO_SURFACE_MORTAR

    AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE

    AUTOMATIC_SURFACE_IN_CONTINUUM

    AUTOMATIC_TIED

    AUTOMATIC_TIED_ONE_WAY

    FORCE_TRANSDUCER

3. The third group is for contact between SPH particles and continuum elements:

    NODE_TO_SOLID

    NODE_TO_SOLID_TIED

Each of the 3 groups has a section below with a description of input and additional re-
marks.

*CONTACT_2D_[SLIDING, TIED, & PENALTY]_OPTION

This section documents the *CONTACT_2D variations derived from DYNA2D:

      SLIDING_ONLY

      TIED_SLIDING

      SLIDING_VOIDS

      PENALTY_FRICTION

      PENALTY

Card 1	1	2	3	4	5	6	7	8
Variable	SURFA	SURFB	TBIRTH	TDEATH				
Type	I	I	F	F				
Default	none	none	0.	10^{20}				

Card 2	1	2	3	4	5	6	7	8
Variable	EXT_PAS	THETA1	THETA2	TOL_IG	PEN	TOLOFF	FRCSCL	ONEWAY
Type	I	F	F	F	F	F	F	F
Default	none	none	none	0.001	0.1	0.025	0.010	0.0

Friction Card. Additional card for the PENALTY_FRICTION keyword option.

Card 3	1	2	3	4	5	6	7	8
Variable	FRIC	FRIC_L	FRIC_H	FRIC_S				
Type	F	F	F	F				

VARIABLE	DESCRIPTION
SURFA	Nodal set ID for the SURFA nodes, see *SET_NODE. The surface specified with SURFA must be to the left of the surface specified with SURFB. For nonsymmetric contact, this surface is the tracked surface (all contacts in this section except PENALTY and PENALTY_FRICTION).
SURFB	Nodal set ID for the SURFB nodes, see *SET_NODE. For nonsymmetric contact, this surface is the reference surface (all contacts in this section except PENALTY and PENALTY_FRICTION).
TBIRTH	Birth time for contact
TDEATH	Death time for contact
EXT_PAS	Slide line extension bypass option. EQ.0: Extensions are used. EQ.1: Extensions are not used.
THETA1	Angle in degrees of slide line extension at first SURFB node. EQ.0: Extension remains tangent to first SURFB segment.
THETA2	Angle in degrees of slide line extension at last SURFB node. EQ.0: Extension remains tangent to last SURFB segment.
TOL_IG	Tolerance for determining initial gaps. EQ.0.0: default set to 0.001
PEN	Scale factor or penalty. EQ.0.0: default set to 0.10
TOLOFF	Tolerance for stiffness insertion for implicit solution only. The contact stiffness is inserted when a node approaches a segment a distance equal to the segment length multiplied by TOLOFF. The stiffness is increased as the node moves closer with the full stiffness being used when the nodal point finally makes contact. EQ.0.0: default set to 0.025.
FRCSCL	Scale factor for the interface friction. EQ.0.0: Default set to 0.010

VARIABLE	DESCRIPTION
ONEWAY	Flag for one way treatment. If set to 1.0, the nodal points on SURFA are constrained to SURFB. This option is generally recommended if SURFB is rigid. EQ.1.0: Activate one way treatment.
FRIC	Coefficient of friction
FRIC_L	Coefficient of friction at low velocity
FRIC_H	Coefficient of friction at high velocity
FRIC_S	Friction factor for shear

Remarks:

1. **Option descriptions.** The following are descriptions of the DYNA2D variations:

   a) The SLIDING_ONLY option is a two-surface method based on a kinematic formulation. The two surfaces can slide arbitrarily large distances without friction but are not permitted to separate or interpenetrate. Surfaces should be initially in contact. This option performs well when extremely high interface pressures are present. The more coarsely meshed surface should be chosen as the reference surface (SURFB) for best performance.

   b) The TIED_SLIDING option joins two parts of a mesh with differing mesh refinement. It is a kinematic formulation, so the more coarsely meshed surface should be chosen as the reference surface (SURFB).

   c) The SLIDING_VOIDS option is a kinematic formulation without friction which permits two surfaces to separate if tensile forces develop across the interface. The surfaces may be initially in contact or initially separated.

   d) The PENALTY_FRICTION and PENALTY options are penalty formulations, so the designation of SURFA and SURFB is not important. The two bodies may be initially separate or in contact. A rate-dependent Coulomb friction model is available for PENALTY_FRICTION.

2. **Slide line.** Consider two slide line surfaces in contact. It is necessary to designate one as a tracked surface and the other as a reference surface. Nodal points defining the tracked surface are called tracked nodes, and similarly, nodes defining the reference surface are called reference nodes. Each tracked-reference surface combination is referred to as a slide line.
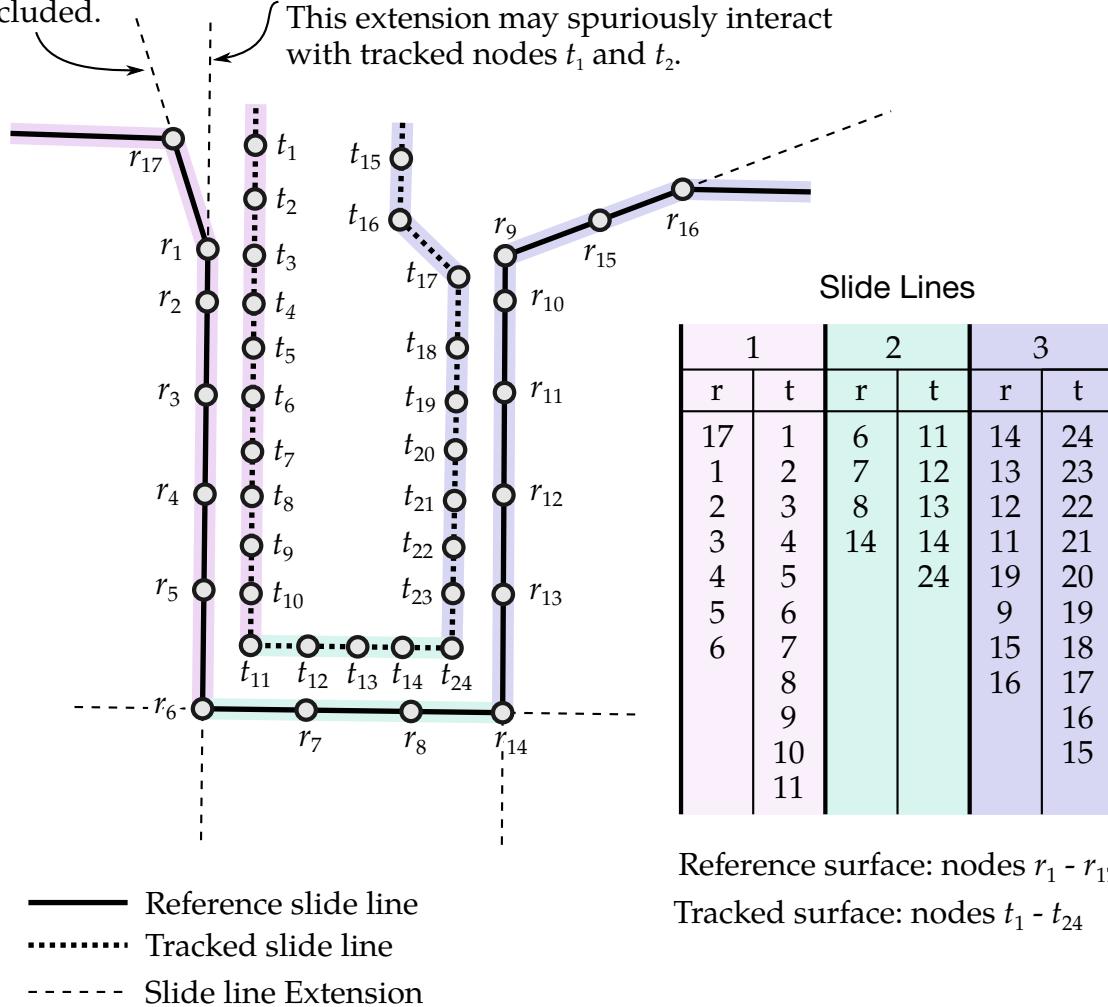
Figure 11-36. *Slide line Example.* Note: (1) as recommended, for 90° angles each facet is assigned a distinct slide line; (2) the reference slide line is more coarsely meshed; (3) the tracked slide line is to the left of the reference (following the node ordering, see inset table); (4) as shown for tracked nodes 1 and 2 it is important the slide line extension does not spuriously come into contact.

3. **Modeling precautions.** Many potential problems with the options can be avoided by observing the following precautions:

   a) Metallic materials should contain the reference surface along high explosive-metal interfaces.

   b) SLIDING_ONLY type slide lines are appropriate along high explosive-metal interfaces. The penalty formulation is not recommended along such interfaces.

c) If one surface is more finely zoned, it should be used as the tracked surface (SURFA). If penalty slide lines, PENALTY and PENALTY_FRICTION, are used, then the tracked-reference distinction is irrelevant.

d) A tracked node may have more than one reference segment and may be included as a member of a reference segment if a slide line intersection is defined.

e) Angles in the reference side of a slide line that approach 90° must be avoided.

   Whenever such angles exist in a reference surface, two or more slide lines should be defined. This procedure is illustrated in Figure 11-36. An exception for the foregoing rule arises if the surfaces are tied. In this case, only one slide line is needed.

f) Whenever two surfaces are in contact, the smaller of the two surfaces should be used as the tracked surface. For example, in modeling a missile impacting a wall, the contact surface on the missile should be used as the tracked surface.

g) Care should be used when defining a reference surface to prevent the extension from interacting with the solution. In Figures 11-36 and 11-37, slide line extensions are shown.
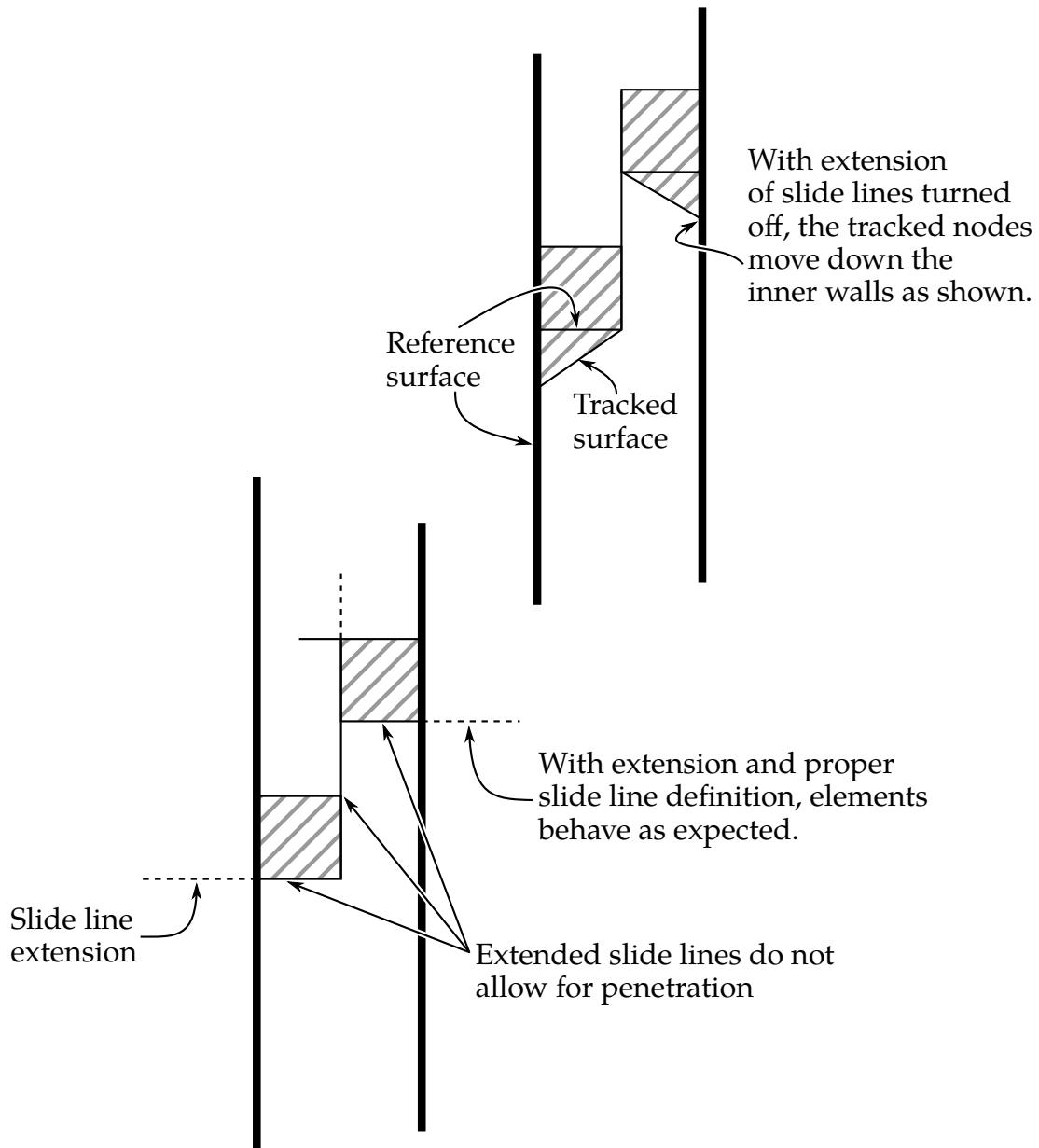
Figure 11-37. *With and without extension.* Extensions may be turned off by setting EXT_PAS (card 2), but, when turned off, tracked nodes may "leak" out as shown in the upper version of the figure.

*CONTACT

CONTACT_2D_[AUTOMATIC & FORCE_TRANSDUCER]_*OPTION

This section documents the following variations of *CONTACT_2D:

       AUTOMATIC_SINGLE_SURFACE

       AUTOMATIC_SINGLE_SURFACE_MORTAR (see Remark 12)

       AUTOMATIC_SURFACE_TO_SURFACE

       AUTOMATIC_SURFACE_TO_SURFACE_MORTAR (see Remark 12)

       AUTOMATIC_ONE_WAY_SURFACE_TO_SURFACE

       AUTOMATIC_SURFACE_IN_CONTINUUM (see Remark 11)

       AUTOMATIC_TIED

       AUTOMATIC_TIED_ONE_WAY

       FORCE_TRANSDUCER (see Remark 2)

Card 1	1	2	3	4	5	6	7	8
Variable	SURFA	SURFB	SFACT	FREQ	FS	FD	DC	
Type	I	I	F	I	F	F	F	
Default	none	↓	1.0	50	0.	0.	0.	
Remarks	1, 2	1, 2						

Card 2	1	2	3	4	5	6	7	8
Variable	TBIRTH	TDEATH	SOA	SOB	NDA	NDB	COF	INIT
Type	F	F	F	F	I	I	I	I
Default	0.	10^{20}	1.0	1.0	0	0	0	0
Remarks			4	4	3	3, 11		6

Automatic Thermal Card. Additional card for keywords with both the AUTOMATIC and THERMAL options. For example, *CONTACT_2D_AUTOMATIC_..._THERMAL_..... . See Remark 7.

Card 3	1	2	3	4	5	6	7	8
Variable	K	RAD	H	LMIN	LMAX	CHLM	BC_FLAG	
Type	F	F	F	F	F	F	I	
Default	none	none	none	none	none	1.0	0	

Automatic Optional Card 1. Optional card for the AUTOMATIC keyword option.

Card 4	1	2	3	4	5	6	7	8
Variable	VC	VDC	IPF	SLIDE	ISTIFF	TIEDGAP	IGAPCL	TIETYP
Type	F	F	I	I	I	R	I	I
Default	0.	10.0	0	0	0		0	0
Remarks				8	9	10		10

Automatic Optional Card 2. Optional card for the AUTOMATIC keyword option. This card can only be used if Card 4 is included, but Card 4 can be left blank.

Card 5	1	2	3	4	5	6	7	8
Variable	SLDSOA	SLDSOB	TDPEN					
Type	F	F	F					
Default	0.	0.	0.					
Remarks	13	13						

VARIABLE	DESCRIPTION
SURFA	Set ID for SURFA. If SURFA > 0, a part set is assumed; see *SET_-PART. If SURFA < 0, a node set with ID equal to the absolute value of SURFA is assumed; see *SET_NODE. For nonsymmetric contact, this surface is the tracked surface.
SURFB	Set ID to define the SURFB surface. If SURFB > 0, a part set is assumed; see *SET_PART. If SURFB < 0, a node set with ID equal to the absolute value of SURFB is assumed; see *SET_NODE. Do not define for single surface contact. For nonsymmetric contact, this surface is the reference surface.
SFACT	Scale factor for the penalty force stiffness
FREQ	Search frequency. The number of timesteps between bucket sorts. For implicit contact this parameter is ignored, and the search frequency is 1. EQ.0: Default set to 50.
FS	Static coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact according to the relation given by: $$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-\text{DC}|v_{\text{rel}}|}.$$
FD	Dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-\text{DC}|v_{\text{rel}}|}.$$ This parameter does not apply to mortar contact.
DC	Exponential decay coefficient. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-\text{DC}|v_{\text{rel}}|}.$$ This parameter does not apply to mortar contact.
TBIRTH	Birth time (contact surface becomes active at this time): LT.0.0: Birth time is set to |TBIRTH|. When negative, birth time is followed during the dynamic relaxation phase of the calculation. After dynamic relaxation has completed, contact is activated regardless of the value of TBIRTH.

VARIABLE	DESCRIPTION
	EQ.0.0: Birth time is inactive, meaning contact is always active
	GT.0.0: If TDEATH = -9999, TBIRTH is interpreted as the curve defining multiple pairs of birth-time/death-time. Otherwise, if TDEATH > 0, birth time applies both during and after dynamic relaxation.
TDEATH	Death time (contact surface is deactivated at this time):
	LT.0.0: If TDEATH = -9999, TBIRTH is interpreted as the curve ID defining multiple pairs of birth-time/death-time. Otherwise, negative TDEATH indicates that contact is inactive during dynamic relaxation. After dynamic relaxation the birth and death times are followed and set to |TBIRTH| and |TDEATH|, respectively.
	EQ.0.0: TDEATH defaults to 10^{20}.
	GT.0.0: TDEATH sets the time at which the contact is deactivated.
SOA	Surface offset from midline for two-dimensional shells of SURFA surface:
	EQ.0.0: Default to 1.0.
	GT.0.0: Scale factor applied to actual thickness
	LT.0.0: Absolute value is used as the offset
SOB	Surface offset from midline for 2D shells of SURFB surface:
	EQ.0.0: Default to 1.0.
	GT.0.0: Scale factor applied to actual thickness
	LT.0.0: Absolute value is used as the offset
NDA	Normal direction flag for two-dimensional shells of SURFA surface:
	EQ.0: Normal direction is determined automatically.
	EQ.1: Normal direction is in the positive direction.
	EQ.-1: Normal direction is in the negative direction.
NDB	Normal direction flag for two-dimensional shells of SURFB surface:
	EQ.0: Normal direction is determined automatically.

VARIABLE	DESCRIPTION
	EQ.1: Normal direction is in the positive direction.
	EQ.-1: Normal direction is in the negative direction.
COF	Closing/opening flag for implicit contact:
	EQ.0: Recommended for most problem where gaps are only closing.
	EQ.1: Recommended when gaps are opening to avoid sticking.
	This parameter does not apply to mortar contact.
INIT	Special processing during initialization:
	EQ.0: No special processing
	EQ.1: Forming option
K	Thermal conductivity (k) of fluid between the slide surfaces. If a gap with a thickness l_{gap} exists between the slide surfaces, then the conductance due to thermal conductivity between the slide surfaces is $$h_{\mathrm{cond}} = \frac{k}{l_{\mathrm{gap}}}$$ Note that LS-DYNA calculates l_{gap} based on deformation.
RAD	Radiation factor, f, between the slide surfaces. A radiant-heat-transfer coefficient (h_{rad}) is calculated (see *BOUNDARY_RADIATION). If a gap exists between the slide surfaces, then the contact conductance is calculated by $$h = h_{\mathrm{cond}} + h_{\mathrm{rad}}$$
H	Heat transfer conductance (h_{cont}) for closed gaps. Use this heat transfer conductance for gaps in the range: $$0 \le l_{\mathrm{gap}} \le l_{\mathrm{min}} \ ,$$ where l_{min} is GCRIT defined below.
LMIN	Critical gap (l_{min}), use the heat transfer conductance defined (HTC) for gap thicknesses less than this value.
LMAX	No thermal contact if gap is greater than this value (l_{max}).

VARIABLE	DESCRIPTION
CHLM	Multiplier used on the element characteristic distance for the search routine. The characteristic length is the largest interface surface element diagonal.
	EQ.0: Default set to 1.0
BC_FLAG	Thermal boundary condition flag:
	EQ.0: Thermal boundary conditions are on when parts are in contact.
	EQ.1: Thermal boundary conditions are off when parts are in contact.
VC	Coefficient for viscous friction. This is used to limit the friction force to a maximum. A limiting force is computed

$$F_{\lim} = VC \times A_{\text{cont}}.$$

A_{cont} being the area of contacted between segments. The suggested value for VC is to use the yield stress in shear:

$$VC = \frac{\sigma_o}{\sqrt{3}} \ ,$$

where σ_o is the yield stress of the contacted material.

VDC	Viscous damping coefficient in percent of critical for explicit contact. This parameter does not apply to Mortar contact.
IPF	Initial penetration flag (see Remark 14):
	EQ.0: Allow initial penetrations to remain.
	GE.1: Push apart initially penetrated surfaces.
SLIDE	Sliding option:
	EQ.0: Off
	EQ.1: On
ISTIFF	Stiffness scaling option:
	EQ.0: Use default option.
	EQ.1: Scale stiffness using segment masses and explicit time step (default for explicit contact).
	EQ.2: Scale stiffness using segment stiffness and dimensions (default for implicit contact).

VARIABLE	DESCRIPTION

TIEDGAP   Search gap for tied contacts.

    EQ.0: Default, use 1% of the SURFB segment length

    GT.0: Use the input value

    LT.0: Use –TIEDGAP % of the SURFB segment length.

IGAPCL   Flag to close gaps in tied contact:

    EQ.0: Default, allow gaps to remain

    EQ.1: Move SURFA nodes to SURFB segments to close gaps

TIETYP   Flag to control constraint type of tied contact:

    EQ.0: Default, use kinematic constraints when possible

    EQ.1: Use only penalty type constraints

SLDSOA   Solid surface offset for the SURFA surface

SLDSOB   Solid surface offset for the SURFB surface

TDPEN   Time span of penetration removal for 2D Mortar contacts. Each initial penetration will be gradually reduced linearly in time, so that it is removed by time TDPEN. For instance, if a given penetration has an initial distance D, then the penetration distance over time, $d(t)$ will be $d(t) = \frac{D}{\text{TDPEN}}(\text{TDPEN} - t)$. This is the interference option analogue to MPAR1 for IGNORE = 3 in 3D automatic Mortar contacts.

Remarks:

1. **Penalty force methods.** The SINGLE_SURFACE, SURFACE_TO_SURFACE, and ONE_WAY_SURFACE_TO_SURFACE options use penalty forces to prevent penetration between two-dimensional shell elements and external faces of two-dimensional continuum elements. Contact surfaces are defined using SURFA and SURFB to reference either part sets or node sets. If part sets are used, all elements and continuum faces of the parts in the set are included in contact. If node sets are used, elements or continuum faces that have both nodes in the set are included in the contact surface. The SINGLE_SURFACE option uses only the SURFA set and checks for contact between all elements and continuum faces in the set. If SURFA is blank or zero, contact will be checked for all elements and continuum faces in the model. With the other options, both SURFA and SURFB are required.

2.   **Force transducer.** The FORCE_TRANSDUCER option should be used in conjunction with at least one AUTOMATIC contact option. It does nothing to prevent penetration but measures the forces generated by other contact definitions. The FORCE_TRANSDUCER option requires only SURFA. SURFB is optional. If only SURFA is defined, the force transducer measures the resultant contact force on all the elements and continuum faces in the SURFA surface. If both SURFA and SURFB are defined, then the force transducer measures contact forces between the elements and continuum faces in the SURFA and SURFB surfaces. The measured forces are included in the rcforc output. In the case of an axisymmetric analysis, values output to rcforc and ncforc are in units of force per radian (this includes both shell types 14 and 15).

3.   **Normal direction.** By default, the normal direction of 2D shell elements is evaluated automatically for SINGLE_SURFACE, SURFACE_TO_SURFACE and ONE_WAY_SURFACE_TO_SURFACE contact. You can override the automatic algorithm using NDA or NDB, and contact will occur with the positive or negative face of the element.

4.   **Shell thickness.** By default, the true thickness of two-dimensional shell elements is taken into account for the SURFACE_TO_SURFACE, SINGLE_SURFACE, and ONE_WAY_SURFACE_TO_SURFACE options. You can override the true thickness by using SOA and SOB. If the surface offset is reduced to a small value, the automatic normal direction algorithm may fail, so it is best to specify the normal direction using NDA or NDB.

5.   **AUTOMATIC contact and rrosion.** For all AUTOMATIC contact options, eroding materials are treated by default. At present, subcycling is not possible.

6.   **Thin Solid Parts and Contact Interface.** The INIT parameter activates a forming option that is intended for implicit solutions of thin solid parts when backside segments may interfere with the solution. It automatically removes backside segments during initialization. Alternatively, the user can input INIT = 0 and use node set input to limit the contact interface to just the front of a thin part.

7.   **THERMAL option.** For the THERMAL option:

$h = h_{\text{cont}}$, if the gap thickness is $0 \leq l_{\text{gap}} \leq l_{\text{min}}$

$h = h_{\text{cond}} + h_{\text{rad}}$, if the gap thickness is $l_{\text{min}} \leq l_{\text{gap}} \leq l_{\text{max}}$

$h = 0$, if the gap thickness is $l_{\text{gap}} > l_{\text{max}}$

8.   **Sliding with kinks and vorners.** The SLIDE parameter activates a sliding option which uses additional logic to improve sliding when surfaces in contact have kinks or corners. This option is off by default.

9. **Penalty stiffness control.** The ISTIFF option allows control of the equation used in calculating the penalty stiffness. For backward compatibility, the default values are different for implicit and explicit solutions. When ISTIFF = 1 is used, the explicit time step appears in the stiffness equation, regardless of whether the calculation is implicit or explicit.

10. **Kinematic constraints.** The TIED_ONE_WAY contact creates two degree of freedom translational kinematic constraints to nodes on the SURFA surface that are initially located on or near SURFB segments. The TIED option creates kinematic constraints between SURFA nodes and SURFB segments and creates penalty constraints between SURFB nodes and SURFA segments. With either contact option, a kinematic constraint may be switched to penalty if there is a conflict with another constraint. The TIEDGAP parameter determines the maximum normal distance from a segment to a node for a constraint to be formed. Nodes will not be moved to eliminate an initial gap, and the initial gap will be maintained throughout the calculation. If TIETYP = 1, then only penalty constraints will be used.

11. **SURFACE_IN_CONTINUUM option.** Note that the SURFACE_IN_CONTINUUM option has been deprecated in favor of the *CONSTRAINED_LAGRANGE_IN_SOLID keyword, which allows coupling between fluids and structures. However, this option is maintained to provide backward compatibility for existing data.

    For the SURFACE_IN_CONTINUUM option, penalty forces prevent the flow of tracked element material (the continuum) through the reference surfaces. Flow of the continuum tangent to the surface is permitted. Only two-dimensional solid parts are permitted in the tracked part set. Both 2D solid and 2D shell parts are permitted in the reference part set. Flow through two-dimensional shell elements is prevented in both directions by default. If NDB is set to ±1, flow in the direction of the normal is permitted. The thickness of two-dimensional shell elements is ignored.

    When using the SURFACE_IN_CONTINUUM option, you do not need to mesh the continuum around the structure because contact is not with continuum nodes but with material in the interior of the continuum elements. The algorithm works well for Eulerian or ALE elements since the structure does not interfere with remeshing. However, a structure will usually not penetrate the surface of an ALE continuum since the nodes are Lagrangian normal to the surface. Therefore, if using an ALE fluid, the structure should be initially immersed in the fluid and remain immersed throughout the calculation. Penetrating the surface of an Eulerian continuum is not a problem.

12. **Mortar contact.** Mortar contact (MORTAR) is available for implicit and explicit calculations in SMP and MPP. The apparent behavior compared to non-Mortar

contact is very similar; the difference lies in details concerning the constitutive relation (contact stress as a function of the relative motion of contact surfaces) and the kinematics (the relative motion of contact surfaces as a function of nodal coordinates). Mortar contact is designed for continuity and smoothness, which is beneficial for an implicit solution scheme. It is intended to enhance robustness for implicit. For details regarding the two-dimensional mortar contact, see the LS-DYNA Theory Manual.

13. **Segment thickness.** By default, segments on the surface of two-dimensional continuum elements have zero thickness, so the contact surface is the line between the nodes. The contact surface can be optionally offset from this line by using the SLDSOA and SLDSOB parameters. If set to a positive number, SLDSOA is the offset for two-dimensional continuum element segments on the SURFA surface, and SLDSOB is the offset for two-dimensional continuum element segments on the SURFB surface. The SOA and SOB parameters provide similar options for segments on two-dimensional shell elements.

14. **IPF.** If a segment pair is detected to be in contact, and the penetration depth exceeds the penetration that has occurred since the last solution cycle, then the excess initial penetration will not be penalized for as long as the segment pair remains in contact. This is done to prevent spikes in contact forces and possible shooting nodes. If IPF > 0, then the excess penetration will be penalized, but the penalty force is ramped up from zero. The value of IPF is the additional percentage of excess initial penetration will be penalized each cycle. For example, IPF = 1 penalizes 1% and IPF = 2 penalizes 2%. We do not recommend larger values. The IPF flag is available for the SURFACE_TO_SURFACE and SINGLE_SURFACE type contact options when used in explicit dynamic solutions. For Mortar contacts we recommend using TDPEN instead of IPF.

CONTACT_2D_NODE_TO_SOLID_*OPTION

This section documents the following variations of *CONTACT_2D:

      NODE_TO_SOLID

      NODE_TO_SOLID_TIED

Card 1	1	2	3	4	5	6	7	8
Variable	SPH	SOLID	TBIRTH	TDEATH				
Type	I	I	F	F				
Default	none	None	0.	10^{20}				

Card 2	1	2	3	4	5	6	7	8
Variable	SOFT	MAXPAR	VC	OFFD	PEN	FS	FD	DC
Type	I	F	F	F	F	F	F	F
Default	0	1.05	0.0	0.0	↓	0.0	0.0	0.0

VARIABLE	DESCRIPTION

    SPH   Nodal set ID or part set ID for the SPH nodes, If SPH > 0, a nodal set ID is assumed. If SPH < 0, a part set ID is assumed.

    SOLID   Solid part set ID. SOLID < 0 since only part set is allowed.

    TBIRTH   Birth time for contact

    TDEATH   Death time for contact

    SOFT   Soft constraint option:

            EQ.0: Penalty formulation

            EQ.1: Soft constraint formulation

VARIABLE	DESCRIPTION
	The soft constraint may be necessary if the material constants of the parts in contact have a wide variation in the elastic bulk moduli. In the soft constraint option, the interface stiffness is based on the nodal mass and the global time step size. The soft constraint option is also recommended for axisymmetric simulations.
MAXPAR	Maximum parametric coordinate in segment search (values between 1.025 and 1.20 are recommended). If zero, the default is set to 1.05. This factor allows an increase in the size of the segments which may be useful at sharp corners.
VC	Coefficient for viscous friction. This is used to limit the friction force to a maximum. A limiting force is computed $$F_{\lim} = \text{VC} \times A_{\text{cont}} \ .$$ A_{cont} being the area of contacted between segments. The suggested value for VC is to use the yield stress in shear: $$\text{VC} = \frac{\sigma_o}{\sqrt{3}} \ ,$$ where σ_o is the yield stress of the contacted material.
OFFD	Contact offset distance for SPH nodes. It does not currently apply to tied contacts. Recommended to be half of the original particle spacing in contact direction.
PEN	Scale factor for penalty. EQ.0.0: default set to 1.0 for penalty formulation, or 0.1 for soft constraint formulation.
FS	Static coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact according to the relationship given by: $$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-\text{DC}|v_{\text{rel}}|}.$$
FD	Dynamic coefficient of friction. The frictional coefficient is assumed to be dependent on the relative velocity v_{rel} of the surfaces in contact $$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-\text{DC}|v_{\text{rel}}|}.$$
DC	Exponential decay coefficient. The frictional coefficient is assumed

VARIABLE	DESCRIPTION

to be dependent on the relative velocity v_{rel} of the surfaces in contact

$$\mu_c = \text{FD} + (\text{FS} - \text{FD})e^{-\text{DC}|v_{\text{rel}}|}.$$

Remarks:

NODE_TO_SOLID contact is a penalty-based contact type used only for SPH particles with solid elements using the plane stress, plane strain or axisymmetric formulation. NODE_TO_SOLID_TIED contact is used only for SPH particles tied with solid elements, an offset of distance h (smooth length) is adopted for each SPH particle.