

ANSYS Contact Technology Guide

ANSYS Release 9.0

002114
November 2004

ANSYS, Inc. is a
UL registered
ISO 9001: 2000
Company.

ANSYS Contact Technology Guide

ANSYS Release 9.0

ANSYS, Inc.
Southpointe
275 Technology Drive
Canonsburg, PA 15317
ansysinfo@ansys.com
<http://www.ansys.com>
(T) 724-746-3304
(F) 724-514-9494

Copyright and Trademark Information

Copyright © 2004 SAS IP, Inc. All rights reserved. Unauthorized use, distribution or duplication is prohibited.

ANSYS, DesignSpace, CFX, DesignModeler, DesignXplorer, ANSYS Workbench environment, AI*Environment, CADOE and any and all ANSYS, Inc. product names referenced on any media, manual or the like, are registered trademarks or trademarks of subsidiaries of ANSYS, Inc. located in the United States or other countries. ICEM CFD is a trademark licensed by ANSYS, Inc. All other trademarks and registered trademarks are property of their respective owners.

ANSYS, Inc. is a UL registered ISO 9001: 2000 Company.

ANSYS Inc. products may contain U.S. Patent No. 6,055,541.

Microsoft, Windows, Windows 2000 and Windows XP are registered trademarks of Microsoft Corporation.

Inventor and Mechanical Desktop are registered trademarks of Autodesk, Inc.

SolidWorks is a registered trademark of SolidWorks Corporation.

Pro/ENGINEER is a registered trademark of Parametric Technology Corporation.

Unigraphics, Solid Edge and Parasolid are registered trademarks of Electronic Data Systems Corporation (EDS).

ACIS and ACIS Geometric Modeler are registered trademarks of Spatial Technology, Inc.

FLEXIm License Manager is a trademark of Macrovision Corporation.

This ANSYS, Inc. software product and program documentation is ANSYS Confidential Information and are furnished by ANSYS, Inc. under an ANSYS software license agreement that contains provisions concerning non-disclosure, copying, length and nature of use, warranties, disclaimers and remedies, and other provisions. The Program and Documentation may be used or copied only in accordance with the terms of that license agreement.

See the ANSYS, Inc. online documentation or the ANSYS, Inc. documentation CD for the complete Legal Notice.

If this is a copy of a document published by and reproduced with the permission of ANSYS, Inc., it might not reflect the organization or physical appearance of the original. ANSYS, Inc. is not liable for any errors or omissions introduced by the copying process. Such errors are the responsibility of the party providing the copy.

Table of Contents

1. Contact Overview	1-1
1.1. General Contact Classification	1-1
1.2. ANSYS Contact Capabilities	1-1
1.2.1. Surface-to-Surface Contact Elements	1-2
1.2.2. Node-to-Surface Contact Elements	1-3
1.2.3. Node-to-Node Contact Elements	1-3
2. GUI Aids for Contact Analyses	2-1
2.1. The Contact Manager	2-1
2.2. The Contact Wizard	2-2
2.3. Managing Contact Pairs	2-3
3. Surface-to-Surface Contact	3-1
3.1. Using Surface-to-Surface Contact Elements	3-1
3.2. Steps in a Contact Analysis	3-1
3.3. Creating the Model Geometry and Mesh	3-2
3.4. Identifying Contact Pairs	3-2
3.5. Designating Contact and Target Surfaces	3-3
3.5.1. Asymmetric Contact vs. Symmetric Contact	3-4
3.5.1.1. Background	3-4
3.5.1.2. Using KEYOPT(8)	3-4
3.6. Defining the Target Surface	3-4
3.6.1. Pilot Nodes	3-5
3.6.2. Primitives	3-5
3.6.3. Element Types and Real Constants	3-5
3.6.3.1. Defining Target Element Geometry	3-5
3.6.4. Using Direct Generation to Create Rigid Target Elements	3-6
3.6.5. Using ANSYS Meshing Tools to Create Rigid Target Elements	3-7
3.6.5.1. Some Modeling and Meshing Tips	3-9
3.6.5.2. Verifying Nodal Number Ordering (Contact Direction) of Target Surface	3-10
3.7. Defining the Deformable Contact Surface	3-11
3.7.1. Element Type	3-11
3.7.2. Real Constants and Material Properties	3-12
3.7.3. Generating Contact Elements	3-13
3.8. Set the Real Constants and Element KEYOPTS	3-14
3.8.1. Real Constants	3-14
3.8.1.1. Positive and Negative Real Constant Values	3-16
3.8.2. Element KEYOPTS	3-17
3.8.3. Selecting a Contact Algorithm (KEYOPT(2))	3-18
3.8.3.1. Background	3-18
3.8.4. Determining Contact Stiffness and Allowable Penetration	3-19
3.8.4.1. Background	3-19
3.8.4.2. Using FKN and FTOLN	3-20
3.8.4.3. Using FKT and SLTO	3-20
3.8.4.4. Using KEYOPT(10)	3-21
3.8.4.5. Chattering Control Parameters	3-22
3.8.5. Choosing a Friction Model	3-22
3.8.5.1. Background	3-22
3.8.5.2. Using TAUMAX, FACT, DC, and COHE	3-23
3.8.5.3. Static and Dynamic Friction Coefficients	3-23
3.8.6. Selecting Location of Contact Detection	3-25
3.8.6.1. Background	3-25

3.8.6.2. Using KEYOPT(4) and TOLS	3–25
3.8.7. Adjusting Initial Contact Conditions	3–27
3.8.7.1. Background	3–27
3.8.7.2. Using PMIN, PMAX, CNOF, ICONT, KEYOPT(5), and KEYOPT(9)	3–27
3.8.8. Physically Moving Contact Nodes Towards the Target Surface	3–34
3.8.9. Determining Contact Status and the Pinball Region	3–35
3.8.9.1. Background	3–35
3.8.9.2. Using PINB	3–36
3.8.10. Avoiding Spurious Contact in Self Contact Problems	3–36
3.8.11. Selecting Surface Interaction Models	3–37
3.8.11.1. Background	3–37
3.8.11.2. Using KEYOPT(12) and FKOP	3–37
3.8.11.3. Bonded Contact for Shell-Shell Assemblies	3–38
3.8.12. Modeling Contact with Superelements	3–39
3.8.12.1. Background	3–39
3.8.12.2. Using KEYOPT(3)	3–39
3.8.13. Accounting for Thickness Effect	3–39
3.8.13.1. Background	3–39
3.8.13.2. Using KEYOPT(11)	3–40
3.8.14. Using Time Step Control	3–40
3.8.14.1. Background	3–40
3.8.14.2. Using KEYOPT(7)	3–40
3.8.15. Using the Birth and Death Option	3–40
3.9. Controlling the Motion of the Rigid Target Surface (Rigid-to-Flexible Contact)	3–40
3.10. Applying Necessary Boundary Conditions to the Deformable Elements	3–41
3.11. Defining Solution and Load Step Options	3–42
3.12. Solving the Problem	3–43
3.13. Reviewing the Results	3–43
3.13.1. Points to Remember	3–44
3.13.2. Reviewing Results in POST1	3–44
3.13.3. Reviewing Results in POST26	3–45
4. Node-to-Surface Contact	4–1
4.1. The Node-to-Surface Contact Element	4–1
4.2. Performing a Node-to-Surface Contact Analysis	4–2
4.2.1. CONTA175 KEYOPTS	4–2
4.2.1.1. KEYOPT(3)	4–2
4.2.1.2. KEYOPT(4)	4–3
4.2.2. CONTA175 Real Constants	4–3
4.3. Using CONTA175 for Multiphysics Contact	4–3
5. Multiphysics Contact	5–1
5.1. Modeling Thermal Contact	5–1
5.1.1. Thermal Contact Behavior vs. Contact Status	5–1
5.1.2. Free Thermal Surface	5–1
5.1.3. Temperature on Target Surface	5–2
5.1.4. Modeling Conduction	5–2
5.1.4.1. Using TCC	5–2
5.1.4.2. Using the Quasi Solver Option	5–3
5.1.5. Modeling Convection	5–3
5.1.6. Modeling Radiation	5–3
5.1.6.1. Background	5–3
5.1.6.2. Using SBCT and RDVF	5–4
5.1.7. Modeling Heat Generation Due to Friction	5–4

5.1.7.1. Background	5-4
5.1.7.2. Using FHTG and FWGT	5-4
5.1.8. Modeling External Heat Flux	5-5
5.2. Modeling Electric Contact	5-5
5.2.1. Modeling Surface Interaction	5-5
5.2.1.1. Background	5-5
5.2.1.2. Using ECC	5-6
5.2.2. Modeling Heat Generation Due to Electric Current	5-6
5.3. Modeling Magnetic Contact	5-7
5.3.1. Using MCC	5-7
5.3.2. Modeling Perfect Magnetic Contact	5-8
6. Node-to-Node Contact	6-1
6.1. Node-to-Node Contact Elements	6-1
6.2. Performing a Node-to-Node Contact Analysis	6-1
6.2.1. Creating Geometry and Meshing the Model	6-2
6.2.2. Generating Contact Elements	6-2
6.2.2.1. Generating Contact Elements Automatically at Coincident Nodes	6-2
6.2.2.2. Generating Contact Elements Automatically at Offset Nodes	6-2
6.2.2.3. Node Ordering	6-3
6.2.3. Defining the Contact Normal	6-3
6.2.4. Defining the Initial Interference or Gap	6-4
6.2.5. Selecting the Contact Algorithm	6-4
6.2.6. Applying Necessary Boundary Conditions	6-5
6.2.7. Defining the Solution Options	6-5
6.2.8. Solving the Problem	6-6
6.2.9. Reviewing the Results	6-7
7. Multipoint Constraints and Assemblies	7-1
7.1. Modeling Solid-Solid and Shell-Shell Assemblies	7-1
7.2. Modeling a Shell-Solid Assembly	7-3
7.3. Surface-Based Constraints	7-8
7.3.1. Defining Surface-Based Constraints	7-9
7.3.2. Modeling a Beam-Solid Assembly	7-10
7.4. Restrictions and Recommendations for Internal MPC	7-11
8. Spot Welds	8-1
8.1. Defining a Spot Weld Set	8-2
8.1.1. Creating a Basic Spot Weld Set with SWGEN	8-2
8.1.2. The Components of a Spot Weld	8-6
8.1.3. Adding Surfaces to a Basic Set	8-8
8.2. Listing and Deleting Spot Welds	8-11

List of Figures

2.1. Contact Manager Toolbar	2-1
2.2. Example of a Contact Wizard Dialog	2-3
3.1. Localized Contact Zones	3-3
3.2. ANSYS Geometric Entities and Their Corresponding Rigid Target Elements	3-7
3.3. A Single Circular Target Segment Created From Arc Line Segments	3-8
3.4. Meshing Patterns for Arbitrary Target Surfaces	3-9
3.5. Smoothing Convex Corner	3-10
3.6. Correct Node Ordering	3-10
3.7. Contact Element Types	3-12

3.8. Specification of the Contact Surface's Outward Normal	3-14
3.9. Depth of the Underlying Element	3-17
3.10. Sliding Contact Resistance	3-23
3.11. Friction Decay	3-24
3.12. Contact Detection Located at Gauss Point	3-25
3.13. Contact Detection Point Location at Nodal Point	3-26
3.14. Node Slippage Using Nodal Integration KEYOPT(4) = 1 or 2	3-26
3.15. Contact Surface Adjustment With ICONT	3-29
3.16. Contact Surface Adjustment (PMIN, PMAX)	3-30
3.17. A Scenario in Which Initial Adjustment Will Fail	3-31
3.18. Ignoring Initial Penetration, KEYOPT(9) = 1	3-32
3.19. Components of True Penetration	3-33
3.20. Ramping Initial Interference	3-34
3.21. Effect of Moving Contact Nodes	3-35
3.22. Auto Spurious Prevention	3-36
3.23. Penalty-Based Shell-Shell Assembly	3-38
4.1. Node-to-Surface Contact Elements	4-1
5.1. Target Temperature	5-2
6.1. Node-to-Node Contact Elements	6-1
6.2. Contact Between Two Concentric Pipes	6-3
6.3. Two Concentric Pipes, Normals Rotated Properly	6-4
6.4. Example of Overconstrained Contact Problem	6-5
7.1. Example of Shell-Solid Assembly	7-3
7.2. Shell-Solid Assembly (Original Mesh)	7-5
7.3. Shell-Solid Assembly with Solid-Solid Constraint Option	7-5
7.4. Shell-Solid Assembly with Shell-Shell Constraint Option	7-6
7.5. Shell-Solid Assembly with Shell-Solid Constraint Option	7-6
7.6. Shell-Solid Constraint, All Directions (KEYOPT(5) = 4)	7-7
7.7. Rigid Constraint Surface	7-8
7.8. Force-Distributed Surface	7-9
7.9. Beam-Solid Assembly Defined by Rigid Constraint Surface	7-11
7.10. Beam-Solid Assembly Defined by Force-distributed Surface	7-11
8.1. Example Spot Weld Configuration	8-1
8.2. Default Projection Direction for Node 1	8-3
8.3. Default Projection Direction for Nodes 1 and 2	8-4
8.4. User-specified Projection Direction, Node 1	8-4
8.5. User-specified Projection Direction, Nodes 1 and 2	8-5
8.6. Search Radius for Spot Weld	8-6
8.7. Nodes Included in Constraint Equations	8-7
8.8. Beam Element Created for Spot Weld	8-7
8.9. Surfaces Added to Basic Spot Weld Set	8-9
8.10. Node Reordering for Beam Elements	8-10

List of Tables

1.1. ANSYS Contact Capabilities	1-1
3.1. Summary of Real Constant Defaults in Different Environments	3-15
3.2. Summary of KEYOPT Defaults in Different Environments	3-18

Chapter 1: Contact Overview

Contact problems are highly nonlinear and require significant computer resources to solve. It is important that you understand the physics of the problem and take the time to set up your model to run as efficiently as possible.

Contact problems present two significant difficulties. First, you generally do not know the regions of contact until you've run the problem. Depending on the loads, material, boundary conditions, and other factors, surfaces can come into and go out of contact with each other in a largely unpredictable and abrupt manner. Second, most contact problems need to account for friction. There are several friction laws and models to choose from, and all are nonlinear. Frictional response can be chaotic, making solution convergence difficult.

In addition to these two difficulties, many contact problems must also address multi-field effects, such as the conductance of heat, electrical currents, and magnetic flux in the areas of contact.

If you do not need to account for friction in your model, and the interaction between the bodies is always bonded, you may be able to use the internal multipoint constraint (MPC) feature (available for certain contact elements) to model various types of contact assemblies and surface-based constraints (see Chapter 7, "Multipoint Constraints and Assemblies" for more information). Another alternative is to use constraint equations or coupled degrees of freedom instead of contact to model these situations (see Chapter 12, "Coupling and Constraint Equations" in the *ANSYS Modeling and Meshing Guide* for more information). Constraint equations are only available for small strain applications (**NLGEOM,off**).

In addition to the implicit contact capabilities discussed in this guide, ANSYS also offers explicit contact capabilities with the ANSYS LS-DYNA explicit dynamics product. Explicit capabilities are ideally suited for short-duration contact-impact problems. For more information on the ANSYS LS-DYNA product and its contact capabilities, see the *ANSYS LS-DYNA User's Guide*.

1.1. General Contact Classification

Contact problems fall into two general classes: rigid-to-flexible and flexible-to-flexible. In rigid-to-flexible contact problems, one or more of the contacting surfaces are treated as rigid (i.e., it has a much higher stiffness relative to the deformable body it contacts). In general, any time a soft material comes in contact with a hard material, the problem may be assumed to be rigid-to-flexible. Many metal forming problems fall into this category. The other class, flexible-to-flexible, is the more common type. In this case, both (or all) contacting bodies are deformable (i.e., have similar stiffnesses). An example of a flexible-to-flexible contact is bolted flanges.

1.2. ANSYS Contact Capabilities

ANSYS supports three contact models: node-to-node, node-to-surface, and surface-to-surface. Each type of model uses a different set of ANSYS contact elements and is appropriate for specific types of problems as shown in Table 1.1: "ANSYS Contact Capabilities".

Table 1.1 ANSYS Contact Capabilities

	Node-to-Node			Node-to-Surface	Surface-to-Surface	
	CONTAC 12	CONTAC 52	CONTA 178	CONTA 175, TARGET 169, 170	CONTA 171,172 TARGET 169	CONTA 173,174 TARGET 170
Point-to-Point	Y	Y	Y			
Point-to-Surface				Y		

	Node-to-Node			Node-to-Surface	Surface-to-Surface	
	CONTAC 12	CONTAC 52	CONTA 178	CONTA 175, TARGET 169, 170	CONTA 171,172 TARGET 169	CONTA 173,174 TARGET 170
Surface-to-Surface				Y	Y	Y
2-D	Y		Y	Y	Y	
3-D		Y	Y	Y		Y
Sliding	small	small	small	large	large	large
Cylindrical Gap	Y		Y			
Pure Lagrange Multiplier			Y	Y	Y	Y
Augmented Lagrange Multiplier			Y	Y	Y	Y
Lagrange Multiplier on Normal and Penalty on Tangent			Y	Y	Y	Y
Internal Multipoint Constraint (MPC)				Y	Y	Y
Contact Stiffness	user-defined	user-defined	semi-automatic	semi-automatic	semi-automatic	semi-automatic
Auto-meshing Tools	EINTF	EINTF	EINTF	ESURF	ESURF	ESURF
Lower-Order	Y	Y	Y	Y	Y	Y
Higher-Order				Y (2-D only)	Y	Y
Rigid-Flexible	Y	Y	Y	Y	Y	Y
Flexible-Flexible	Y	Y	Y	Y	Y	Y
Thermal Contact				Y	Y	Y
Electric Contact				Y	Y	Y
Magnetic Contact				Y	Y	Y

To model a contact problem, you first must identify the parts to be analyzed for their possible interaction. If one of the interactions is at a point, the corresponding component of your model is a node. If one of the interactions is at a surface, the corresponding component of your model is an element: either a beam, shell, or solid element. The finite element model recognizes possible contact pairs by the presence of specific contact elements. These contact elements are overlaid on the parts of the model that are being analyzed for interaction. The different contact elements that ANSYS uses, and procedures for using them, are described in the remaining chapters of this guide.

An overview of the ANSYS contact elements and their capabilities follows. For detailed information on any of these elements, refer to the *ANSYS Elements Reference* and the *ANSYS, Inc. Theory Reference*.

1.2.1. Surface-to-Surface Contact Elements

ANSYS supports both rigid-to-flexible and flexible-to-flexible surface-to-surface contact elements. These contact elements use a "target surface" and a "contact surface" to form a contact pair.

- The target surface is modeled with either TARGE169 or TARGE170 (for 2-D and 3-D, respectively).
- The contact surface is modeled with elements CONTA171, CONTA172, CONTA173, and CONTA174.

To create a contact pair, assign the same real constant number to both the target and contact elements. You can find more details on defining these elements and their shared real constant sets in Chapter 3, “Surface-to-Surface Contact”.

These surface-to-surface elements are well-suited for applications such as interference fit assembly contact or entry contact, forging, and deep-drawing problems. The surface-to-surface contact elements have several advantages over the node-to-node element CONTA175. These elements:

- Support lower and higher order elements on the contact and target surfaces (in other words, corner-noded or midside-noded elements).
- Provide better contact results needed for typical engineering purposes, such as normal pressure and friction stress contour plots.
- Have no restrictions on the shape of the target surface. Surface discontinuities can be physical or due to mesh discretization.

Using these elements for a rigid target surface, you can model straight and curved surfaces in 2-D and 3-D, often using simple geometric shapes such as circles, parabolas, spheres, cones, and cylinders. More complex rigid forms or general deformable forms can be modeled using special preprocessing techniques (see Section 3.6: Defining the Target Surface for more information).

Surface-to-surface contact elements are not well-suited for point-to-point, point-to-surface, or edge-to-surface contact applications, such as pipe whip or snap-fit assemblies. You should use the node-to-surface or node-to-node elements in these cases. You also can use surface-to-surface contact elements for most contact regions and use a few node-to-surface contact elements near contact corners.

The surface-to-surface contact elements only support general static and transient analyses, buckling, harmonic, modal or spectrum analyses, or substructure analyses.

1.2.2. Node-to-Surface Contact Elements

CONTA175 is a node-to-surface contact element. It supports large sliding, large deformation, and different meshes between the contacting components.

CONTA175 is typically used to model point-to-surface contact applications, such as two beams contacting each other (at a beam tip or sharp corner node), and the corners of snap-fit parts.

You can also use CONTA175 to model surface-to-surface contact, if the contacting surface is defined by a group of nodes and multiple elements are generated. The surfaces can be either rigid or deformable. An example of this type of contact problem is a wire inserted into a slot.

Unlike the node-to-node contact elements, you do not need to know the exact location of the contacting area beforehand, nor do the contacting components need to have a compatible mesh. Large deformation and large relative sliding are allowed, although this capability can also model small sliding.

CONTA175 does not support 3-D higher-order elements on the contact surface side. The element can fail if the target surface is severely discontinuous. No contour plots are available for contact results.

1.2.3. Node-to-Node Contact Elements

Node-to-node contact elements are typically used to model point-to-point contact applications. To use node-to-node contact elements, you need to know the location of contact beforehand. These types of contact problems usually involve small relative sliding between contacting surfaces (even in the case of geometric nonlinearities).

An example of a node-to-node contact application is the traditional pipe whip model, where the contact point is always located between the pipe tip and the restraint.

Node-to-node contact elements can also be used to solve a surface-to-surface problem if the nodes of the two surfaces line up, the relative sliding deformation is negligible, and deflections (rotations) of the two surfaces remain small. These are typically problems with faceted and simple geometry. An interference fit problem is an example of a surface-to-surface problem where the use of node-to-node contact may be sufficient.


Another use of node-to-node contact elements is in extremely precise analysis of surface stresses, such as in turbine blade analysis.

ANSYS element CONTA178 is the best choice for most node-to-node problems. It offers a wider range of options and solver types than the other elements. CONTAC12 and CONTAC52 are available largely for reasons of backward compatibility with existing models.

Chapter 2: GUI Aids for Contact Analyses

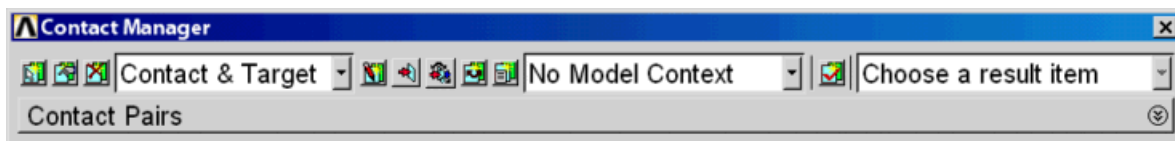
Several GUI aids are available to help you create and manage contact pairs. The Contact Manager allows you to define, view, and edit contact pairs. It provides a convenient way to manage all contact pairs for your entire model. The Contact Wizard, which is accessed from the Contact Manager, leads you through the process of creating contact pairs.

2.1. The Contact Manager

You can access the Contact Manager via the Contact Manager icon  in the ANSYS Standard Toolbar, or via the menu path **Preprocessor > Modeling > Create > Contact Pair**. The Contact Manager is available at the Begin level and in the following processors: preprocessor (PREP7), solution (SOLU), and general postprocessor (POST1).

The Contact Manager Toolbar provides an intuitive interface for the creation and management of contact pairs. The manager supports surface-to-surface contact analysis, node-to-surface contact analysis (using CONTA175), and the internal multipoint constraint (MPC) method of contact .

Figure 2.1 Contact Manager Toolbar



- **Contact Wizard** - Accesses the Contact Wizard GUI described in Section 2.2: The Contact Wizard. Allows you to manually define target and contact surfaces. Supports both 2-D and 3-D geometries as well as rigid-flexible (with optional pilot node) or flexible-flexible contact. (Primitives are not supported by contact wizard.) The wizard also supports surface-based constraint contact pairs.
- **Contact Properties** - Allows you to specify the properties of the contact pair(s) via real constants and KEYOPTs for the contact elements used.
- **Delete Contact Pairs** - Deletes the contact pairs selected in the contact pair list.
- **Contact Selection Options** - Specifies display of contact elements, target elements, or both.
- **Plot Elements/Results** - Displays the elements of selected contact pairs; or displays the contact results, if available (POST1 only), specified in the **Contact Results** field for selected contact pairs. Results are displayed if **Model Context** is specified as "Result-"; otherwise, elements are displayed. The display can be limited to contact, or target, or both as specified in the **Contact Selection Options** field. (Note that CONTA175 results will not display graphically, but can be listed with the **List Elements/Results** icon.)
- **Show Normals** - Specifies whether or not to display the normals on the elements when plotting contact pairs
- **Flip Normals** - Flips the element normals of the selected contact pair. This action is limited to the elements specified in the **Contact Selection Options** field.
- **Switch Contact and Target** - Inverts the target surface and the contact surface with each other. This is applicable only to flexible-to-flexible surface-to-surface contact pairs.
- **List Elements/Results** - Lists the elements of the selected contact pairs; or lists the contact results, if available (POST1 only), specified in the **Contact Results** field for selected contact pairs. Results are listed if **Model Context** is specified as "Result-"; otherwise, elements are listed. The listing is restricted to the elements specified in the **Contact Selection Options** field.

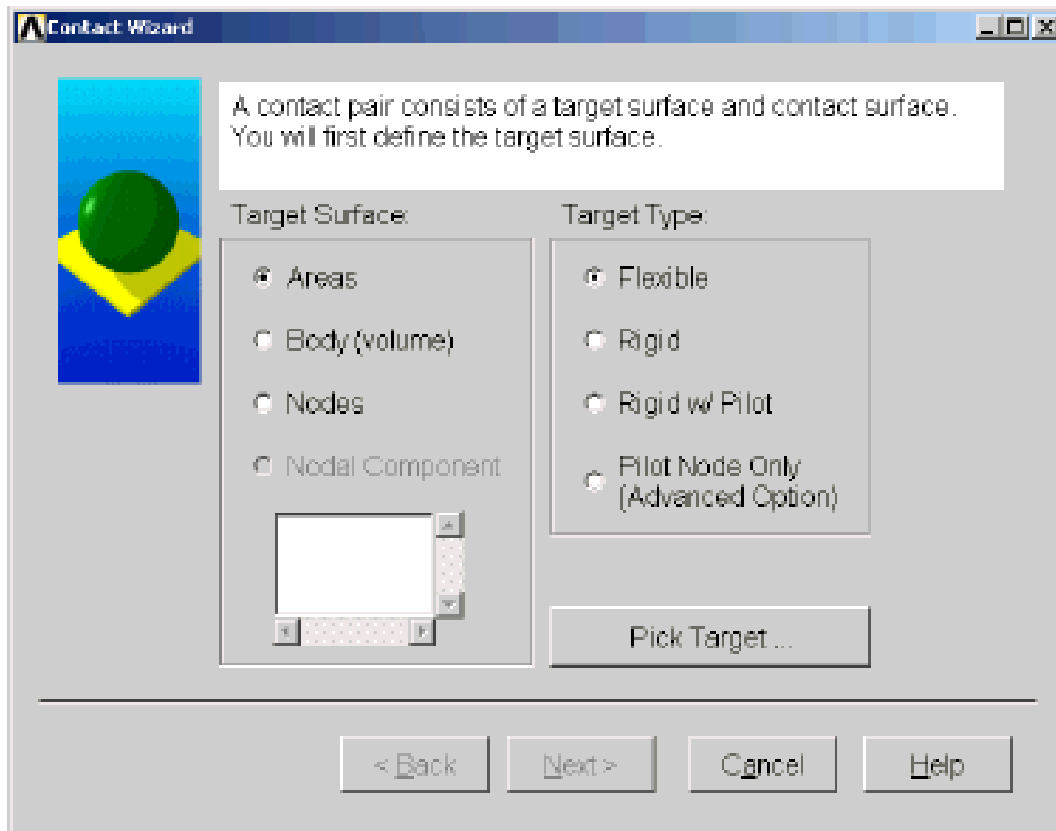
- **Model Context** - Displays the contact pairs in the context of the entire model using a translucent plot, or shows only the contact pairs. If set to "Result-" (POST1 only), controls display/listing of contact pair results.
- **Check Contact Status** - Provides contact status information for selected contact pairs. Several options are available (click and hold down the Check Contact Status button to access these options):
 - Display a detailed listing of status information for each contact pair.
 - Run a partial solution of the initial contact state (**CNCHECK**,POST command) that can be subsequently postprocessed from the contact manager. (See **CNCHECK** for more information.)
 - Physically move contact nodes to the target surface (**CNCHECK**,ADJUST command) in order to close a gap or reduce a penetration (see Section 3.8.8: Physically Moving Contact Nodes Towards the Target Surface.)
 - Reset target element and contact element key options and real constants to their default values (**CNCHECK**,RESET command).
- **Contact Results** - Shows the contact result items for subsequent viewing using the **Plot Elements/Results** and **List Elements/Results** icons.

The bar below the toolbar icons minimizes or maximizes the Contact Pair list box below it. The Contact Pair list box displays the defined contact pairs. You can select contact pairs from this list for displaying or editing purposes.

2.2. The Contact Wizard

This Contact Wizard will lead you through the process of manually creating contact pairs. The wizard supports rigid-flexible (with optional pilot node) and flexible-flexible contact, and supports both surface-to-surface and node-to-surface configurations. (Note that the wizard does not support rigid target primitives.) The Contact Wizard also supports surface-based constraint contact pairs.

To use the Contact Wizard, access the Contact Manager Toolbar and click on the **Contact Wizard** icon. The wizard steps you through setting up the contact analysis. Below is an example of the screens you will see.

Figure 2.2 Example of a Contact Wizard Dialog

The Contact Wizard remains unavailable (dimmed) if you haven't meshed any portion of your model. If you wish to create a rigid-flexible model, mesh only those parts of the model which will be used as flexible contact surfaces (do not mesh the rigid target surfaces) before launching the wizard. If you wish to create a flexible-flexible model, mesh all parts of the model which will be used as contact surfaces (including target surfaces) before launching the wizard.

You can specify target and contact surfaces using lines, areas, volumes, selected set of nodes, or node components. Note that the wizard allows you to choose more than one area for the target and contact surfaces, thus allowing multiple areas to form a single contact surface. If you specify a rigid target surface, you will then have the option to define a pilot node for that contact pair. (The pilot node step is mandatory if you define a surface-based constraint contact pair by picking the **Pilot Node Only** option.)

After you specify the target and contact surfaces, you can specify properties of the contact pair (real constants and KEYOPTs) before creating the contact pair. When you finish specifying all the required data, click the **Create** button to create the contact and target element types, a unique real constant set with the real constant values, and the elements that make up the contact and target surfaces.

2.3. Managing Contact Pairs

As discussed in Chapter 3, "Surface-to-Surface Contact" it is paramount that the contact elements be oriented correctly for proper contact detection. The contact manager provides tools that help you

1. Verify that the normals of the contact and target surfaces are in the correct direction
2. Reverse normals of elements that are not oriented correctly

You can choose to display one or more contact pairs on which to perform the above listed operations. Then you have an additional option of only displaying the contact surface or the target surface when verifying or reversing the element normals. In addition these elements can be displayed independently or in the context of your entire model. In the later case the contact elements are highlighted in a translucent plot of your model.


Another important function is to edit the properties of the contact pair(s) as needed. The properties include real constant values and key option values as discussed earlier. The **Contact Properties** button in the contact manager provides a simple to use interface that allows the properties of the selected contact pair(s) to be reviewed and modified if needed.

Note that when you have multiple contact pairs, it is possible to have conflicts in the real constant values or in the KEYOPT settings. When you have such conflicts, the properties dialog for those real constants or KEYOPT settings is left blank.

Finally, you can display or list specific contact result items (contact status, penetration, pressure, etc.) for the selected contact pairs. This option is only available in POST1, and only if a result set is available. The contact results can be displayed independently or in the context of your entire model. (For CONTA175, results can be listed but not displayed graphically.)

Note — Prior to displaying or listing result items associated with the initial contact configuration resulting from the **CNCHECK,POST** command, you need to issue the appropriate **FILE** and **SET,FIRST** commands to read the results from the proper results file (see **CNCHECK** for details).

Chapter 3: Surface-to-Surface Contact

You can use the surface-to-surface contact elements to model either rigid-flexible or flexible-flexible contact between surfaces. The Contact Manager provides an easy-to-use interface to help you construct and manage contact definitions. You can access the manager via the Contact Manager icon  in the ANSYS Standard Toolbar, or via the menu path **Main Menu > Preprocessor > Modeling > Create > Contact Pair**. See Chapter 2, "GUI Aids for Contact Analyses" for more information on using the Contact Manager.

This chapter explains how to set up a contact analysis using both the command and GUI approaches.

Note — The procedures described in this chapter also apply to node-to-surface contact analyses using CONTA175, unless otherwise noted.

3.1. Using Surface-to-Surface Contact Elements

In problems involving contact between two boundaries, one of the boundaries is conventionally established as the "target" surface, and the other as the "contact" surface. For rigid-flexible contact, the target surface is always the rigid surface, and the contact surface is the deformable surface. For flexible-to-flexible contact, both contact and target surfaces are associated with the deformable bodies. These two surfaces together comprise the "contact pair." Use TARGE169 with CONTA171, CONTA172, or CONTA175 to define a 2-D contact pair. For 3-D contact pairs, use TARGE170 with CONTA173, CONTA174, or CONTA175. Each contact pair is identified via the same real constant number.

3.2. Steps in a Contact Analysis

The basic steps for performing a typical surface-to-surface contact analysis are listed below. Each step is then explained in detail in the following sections.

1. Create the model geometry and mesh
2. Identify the contact pairs
3. Designate contact and target surfaces
4. Define the target surface
5. Define the contact surface
6. Set the element KEYOPTS and real constants
7. Define/control the motion of the target surface (rigid-to-flexible only)
8. Apply necessary boundary conditions
9. Define solution options and load steps
10. Solve the contact problem
11. Review the results

Each contact-specific step also has a corresponding GUI approach where you use functions and features on the Contact Toolbar.

3.3. Creating the Model Geometry and Mesh

First, create solid model entities that represent the geometry of the contacting bodies. Set element types, real constants, and material properties as you would for any ANSYS analysis. Mesh the contacting bodies by meshing the areas or volumes with the element type that you have chosen. For more information, see the *ANSYS Modeling and Meshing Guide*.

Command(s): AMESH, VMESH

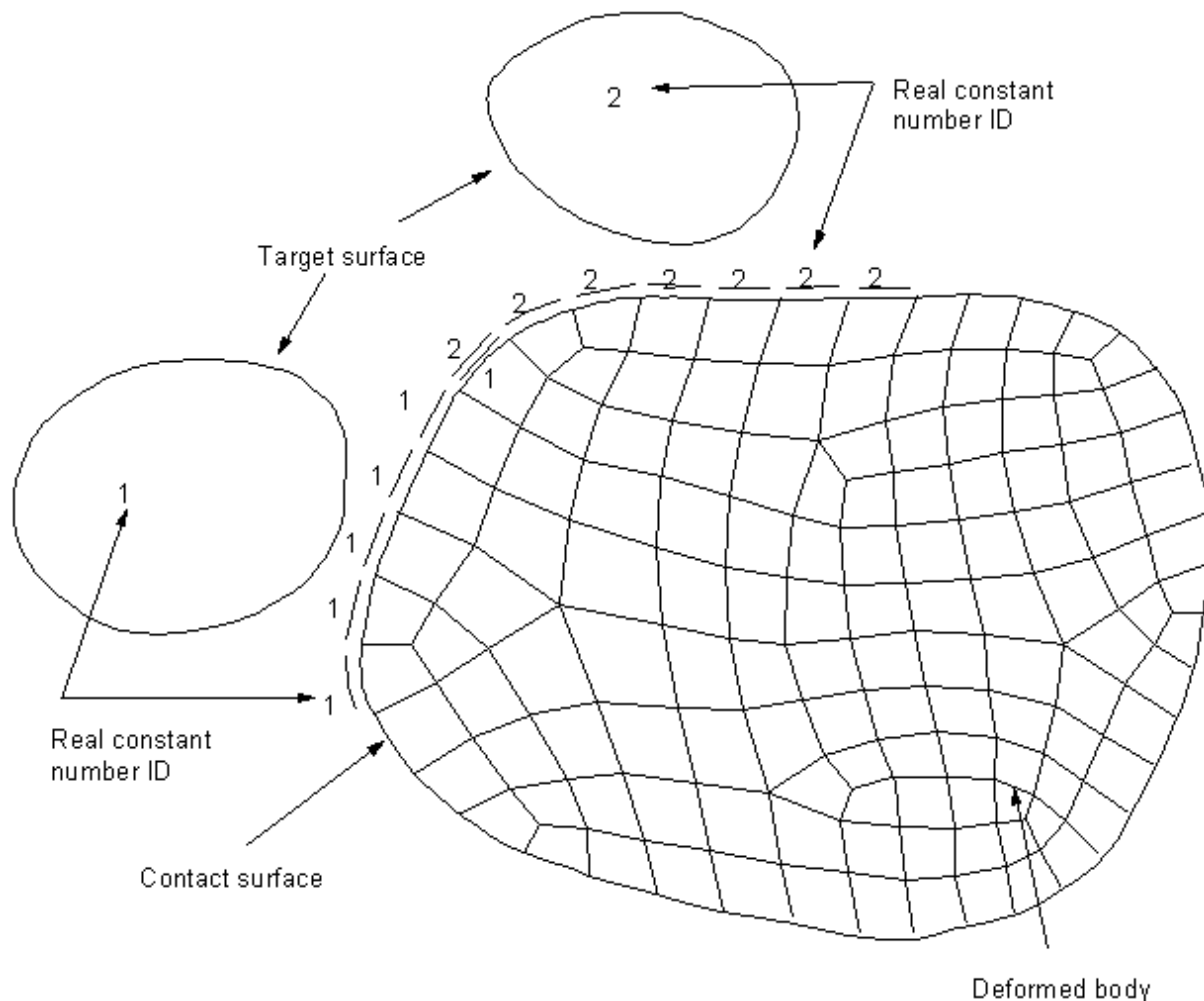
GUI: Main Menu > Preprocessor > Meshing > Mesh

Note — You should avoid midside-noded elements for 3-D contact surfaces when using node-to-surface element CONTA175.

3.4. Identifying Contact Pairs

You must identify where contact might occur during the deformation of your model. Once you've identified potential contact surfaces, you define them via target and contact elements, which will then track the kinematics of the deformation process. Target and contact elements that make up a contact pair are associated with each other via a shared real constant set.

The contact zone can be arbitrary; however, for the most efficient solution (primarily in CPU time), you may want to define smaller, localized contacting zones, but be sure your zones are adequate to capture all necessary contact. Different contact pairs must be defined by a different real constant set, even if the element real constant values do not change. There is no limit on the number of surfaces allowed.

Figure 3.1 Localized Contact Zones

Depending on the geometry of the model (and the potential deformation), multiple target surfaces could interact with the same zone of the contact surface. In such cases, you must define multiple contact pairs (using multiple overlapping contact elements), each with its own real constant number. See Figure 3.1: “Localized Contact Zones”.

3.5. Designating Contact and Target Surfaces

Contact elements are constrained against penetrating the target surface. However, target elements can penetrate through the contact surface. For rigid-to-flexible contact, the designation is obvious: the target surface is always the rigid surface and the contact surface is always the deformable surface. For flexible-to-flexible contact, the choice of which surface is designated contact or target can cause a different amount of penetration and thus affect the solution accuracy. Consider the following guidelines when designating the surfaces:

- If a convex surface is expected to come into contact with a flat or concave surface, the flat/concave surface should be the target surface.
- If one surface has a fine surface mesh and, in comparison, the other has a coarse mesh, the fine mesh should be the contact surface and the coarse mesh should be the target surface.
- If one surface is stiffer than the other, the softer surface should be the contact surface and the stiffer surface should be the target surface.
- If higher-order elements underly one of the external surfaces and lower-order elements underly the other surface, the surface with the underlying higher-order elements should be the contact surface and the

other surface should be the target. However, for 3-D node-to-surface contact, the lower-order elements should be the contact surface. The higher-order elements should be the target surface.

- If one surface is markedly larger than the other surface, such as in the instance where one surface surrounds the other surface, the larger surface should be the target surface.

These guidelines are true for asymmetric contact; however, asymmetric contact may not perform satisfactorily for your model. The following section details the difference between asymmetric and symmetric contact and outlines some of the situations that require symmetric contact.

3.5.1. Asymmetric Contact vs. Symmetric Contact

3.5.1.1. Background

Asymmetric contact is defined as having all contact elements on one surface and all target elements on the other surface. This is sometimes called "one-pass contact." This is usually the most efficient way to model surface-to-surface contact. However, under some circumstances asymmetric contact does not perform satisfactorily. In such cases, you can designate each surface to be both a target and a contact surface. You can then generate two sets of contact pairs between the contacting surfaces (or just one contact pair; for example, a self-contact case). This is known as symmetric contact (or "two-pass contact"). Obviously, symmetric contact is less efficient than asymmetric contact. However, many analyses will require its use (typically to reduce penetration). Specific situations that require symmetric contact include models where

- The distinction between the contact and target surfaces is not clear.
- Both surfaces have very coarse meshes. The symmetric contact algorithm enforces the contact constraint conditions at more surface locations than the asymmetric contact algorithm.

If the meshes on both surfaces are identical and sufficiently refined, the symmetric contact algorithm may not significantly improve performance and may, in fact, be more "expensive" in CPU time. In such circumstances, pick one surface to be the target and the other the contact surface.

For a symmetric contact definition, ANSYS may find one side of a contact surface as closed and the other side of the surface as closed. In this case, it can be difficult to interpret the results. The total contact pressure acting on both sides is the average of the contact pressures on each side of the surface.

3.5.1.2. Using KEYOPT(8)

When there are several contact pairs involved in the model, and the graphical picking of contact and target surfaces is difficult, you can just define the symmetric contact pairs and, by setting KEYOPT(8) = 2, ANSYS will internally select which asymmetric pair is to be used at the solution stage based on the guidelines mentioned above in Designating Contact and Target Surfaces.

Note — In any contact model, you can mix different types of contact pairs: rigid-to-flexible or flexible-to-flexible contact; symmetric contact or asymmetric contact. However, only one type can exist with a contact pair.

3.6. Defining the Target Surface

The target surface can be 2-D or 3-D and either rigid or deformable. For deformable target surfaces, you will normally use the **ESURF** command to generate the target elements along the boundaries of an existing mesh. You can follow the same method to generate the deformable contact surface (see Section 3.7: Defining the Deformable Contact Surface for details). You should not use the following rigid target segments for a deformable

target surface: ARC, CARC, CIRC, CYL1, CONE, SPHE, or PILO. For rigid target surfaces, the following provides general guidelines.

In 2-D cases, the shape of the target surface is described by a sequence of straight lines, circular arcs, and parabolas, all of which can be represented with the target segment element TARGE169. You can use any combination to define the complex target surface geometry. In 3-D cases, the shape of the target surfaces is described by a sequence of triangles, quadrilaterals, cylinders, cones, and spheres, which can be represented with TARGE170. You can use any combination of low/high-order triangles and quadrilaterals to model a target surface with a complex, arbitrary geometry.

3.6.1. Pilot Nodes

The rigid target surface can also be associated with a "pilot node," which is really an element with one node, whose motion governs the motion of the entire target surface. You can think of a pilot node as a handle for the rigid target surface. Forces/moments or rotations/displacements for the entire target surface can be prescribed on just the pilot node. The pilot node can be one of the nodes on the target element or a node at any arbitrary location. The location of the pilot node is important only when rotation or moment loading is required. If you define a pilot node, ANSYS checks for boundary conditions only on the pilot node and ignores any constraints on other nodes.

3.6.2. Primitives

You can use circle, cylinder, cone, and sphere primitives to model the target (which require real constants to define the radius). You can combine primitive segments with general segments (such as lines, parabolas, triangles, and quadrilaterals) to define a target surface. Primitives cannot be defined directly in the Contact Wizard.

3.6.3. Element Types and Real Constants

Before generating the target element, first define the element type (TARGE169 for 2-D or TARGE170 for 3-D):

Command(s): ET

GUI: Main Menu > Preprocessor > Element Type > Add/Edit/Delete

3.6.3.1. Defining Target Element Geometry

You define characteristics of the target element geometry through real constants R1 and R2 as follows:

- For CONTA171 and CONTA172:
 - R1 is the radius if the target shape (TARGE169) is a circle.
 - R2 is the element thickness if the underlying element is a superelement set as plane stress with thickness (KEYOPT(3) = 3). The default value is 1.
- For CONTA173 and CONTA174 (also applies to node-to-surface element CONTA175):
 - R1 is the radius if the target shape (TARGE170) is a cylinder, cone, or sphere.
 - R2 is the radius of a cone at the second node.

To set the real constant number for the target elements:

Command(s): REAL

GUI: Main Menu > Preprocessor > Real Constants

For TARGE169 and TARGE170, you need only set real constants R1 and R2 (if required). For a complete description of the target elements, element shapes, and real constants, see the description of TARGE169 and TARGE170 in the *ANSYS Elements Reference*.

Note — Specifying real constants (R1, R2) manually is necessary only if you use direct generation to create your target elements. You can also use the ANSYS meshing tools to create the elements, or use the Contact Manager Toolbar.

3.6.4. Using Direct Generation to Create Rigid Target Elements

To generate target elements directly, use the following command or GUI path:

Command(s): TSHAP

GUI: Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes

You then specify the element shape. Possible shapes are:

- Straight line (2-D)
- Parabola (2-D)
- Clockwise arc (2-D)
- Counterclockwise arc (2-D)
- Circle (2-D)
- Three-node triangle (3-D)
- Six-node triangle (3-D)
- Four-node quadrilateral (3-D)
- Eight-node quadrilateral (3-D)
- Cylinder (3-D)
- Cone (3-D)
- Sphere (3-D)
- Pilot node (both 2-D and 3-D)

Once you specify a target element shape, all subsequent elements will have that shape until you specify another shape.

Note — You cannot mix 2-D and 3-D target elements on the same target surface.

You cannot mix rigid target elements with deformable target elements on the same target surface. During solution, ANSYS assigns a deformable status to target elements with underlying elements and assigns a rigid status to target elements without underlying elements. If a portion of the underlying elements of a deformable surface are deleted, an error will occur in solution.

You can generate the nodes and elements using standard ANSYS direct generation techniques. For more information on direct generation modeling techniques, see Chapter 9, “Direct Generation” in the *ANSYS Modeling and Meshing Guide*.

Command(s): N, E

GUI: Main Menu > Preprocessor > Modeling > Create > Nodes

Main Menu > Preprocessor > Modeling > Create > Elements

You can then verify your element shapes by listing the elements.

Command(s): ELIST

GUI: Utility Menu> List> Elements> Nodes + Attributes

3.6.5. Using ANSYS Meshing Tools to Create Rigid Target Elements

You can also let ANSYS generate the elements automatically using the standard ANSYS meshing capabilities. ANSYS will recognize the proper target element shape based on the solid model and will ignore the **TSHAP** setting.

To generate a pilot node, use the following command or GUI path:

Command(s): KMESSH

GUI: Main Menu> Preprocessor> Meshing> Mesh> Keypoints

Note — **KMESSH** always creates pilot nodes.

To generate 2-D rigid target elements, use the following command or GUI path. ANSYS creates a single line over each line, parabolic segments over B-splines, and arc segments over each arc and line fillet (see Figure 3.2: “ANSYS Geometric Entities and Their Corresponding Rigid Target Elements”). If all the arcs form a closed circle, ANSYS creates a single circular segment (see Figure 3.3: “A Single Circular Target Segment Created From Arc Line Segments”). However, if the arcs that form a closed circle are created from imported or archived geometry (such as IGES), ANSYS might not create a single circular segment.

Command(s): LMESH

GUI: Main Menu> Preprocessor> Meshing> Mesh> Lines

Figure 3.2 ANSYS Geometric Entities and Their Corresponding Rigid Target Elements

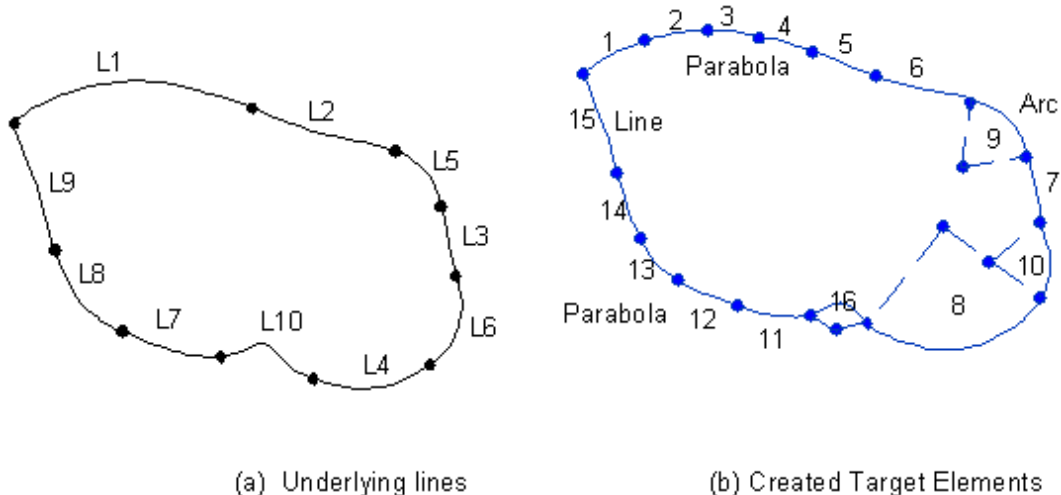
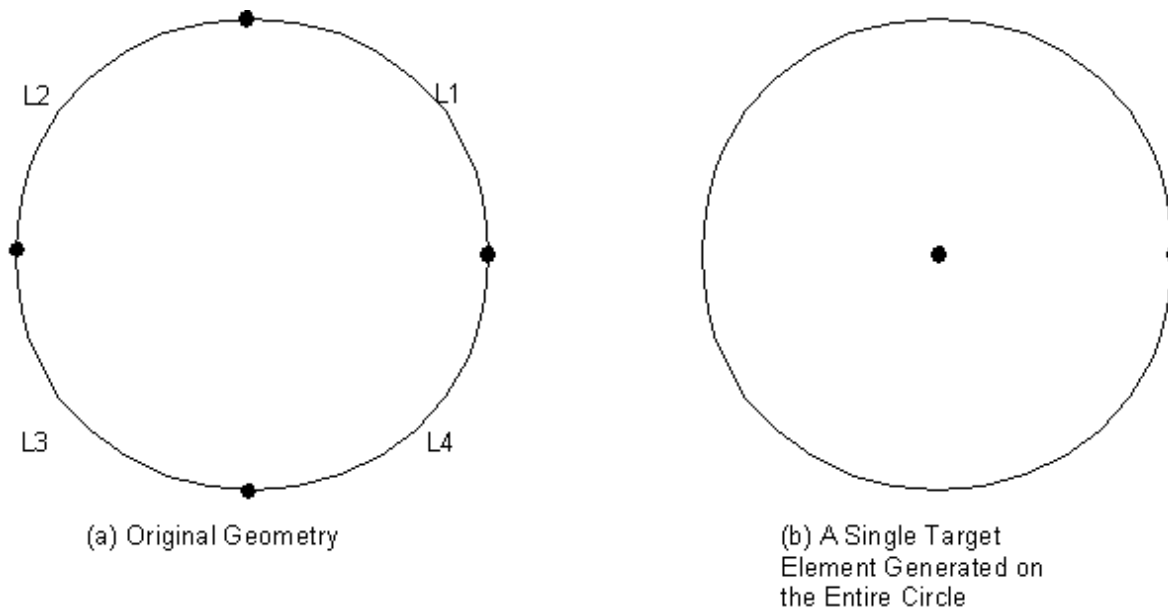


Figure 3.3 A Single Circular Target Segment Created From Arc Line Segments

To generate 3-D rigid target elements, use the following command or GUI path.

Command(s): AMESH

GUI: Main Menu > Preprocessor > Meshing > Mesh > Areas

If the surface segments on the solid model form a complete sphere, cylinder, or cone, then ANSYS automatically generates a single primitive 3-D target element through the **AMESH** command. By creating fewer elements, the analysis becomes more computationally efficient. For arbitrary surfaces, you should use **AMESH** to generate target elements. In these cases, the quality of the meshed target shape is not important. It is more important that the target elements represent the rigid surface geometry well.

We recommend using mapped meshing on all possible areas. If there is no curvature on the edges of the surface, assign one division on that edge. TARGE169 with a rigid specification will always mesh with one element division, per line, ignoring any **LESIZE** setting. The default target element shape is quadrilateral. If you want a triangular target element shape, use **MSHAPE,1**. Figure 3.4: "Meshing Patterns for Arbitrary Target Surfaces" shows the meshing patterns for arbitrary target surfaces. The following command or GUI path will generate a mapped mesh wherever possible (otherwise, if not possible, it will generate a free mesh).

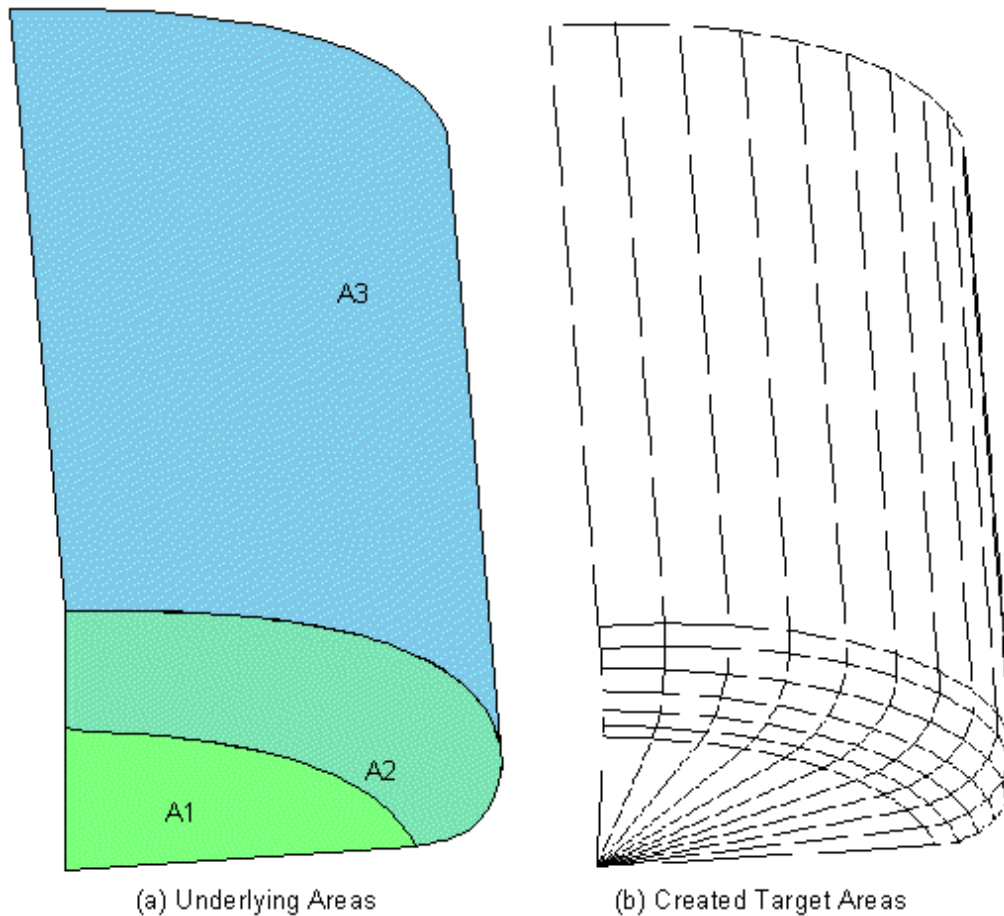
Command(s): MSHKEY,2

GUI: Main Menu > Preprocessor > Meshing > Mesh > Areas > Target Surf

If the target surface is flat (or nearly flat), you may select low-order target elements (3-node triangular or 4-node quadrilateral elements). If the target surface is curved you should select high-order target elements (6-node triangular or 8-node quadrilateral). By doing so, set **KEYOPT(1) = 1** in the target element definition.

Note — Low-order target elements result in "cheaper" CPU usage in getting penetration and gap; however, the meshed surface may not be smooth. Higher-order target elements are more "expensive" to use in getting the penetration and gap, but they need many fewer elements to discretize the whole curved target surface.

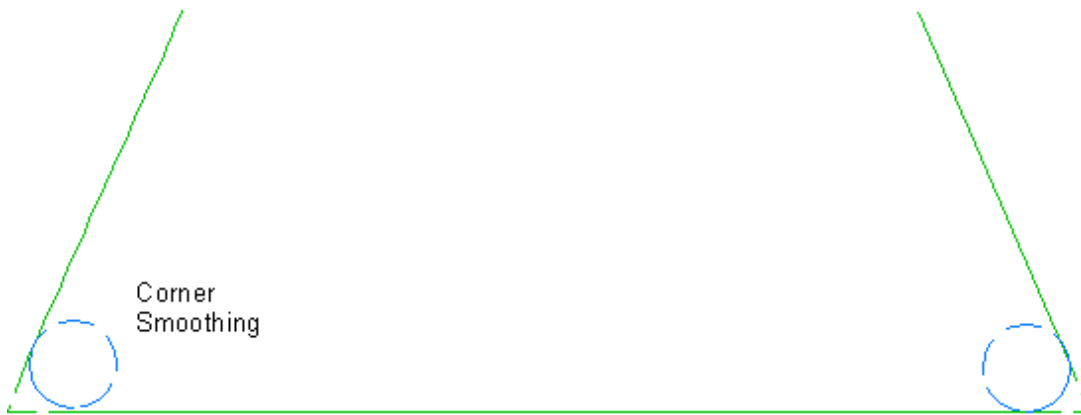
Note — If target elements are created via program meshing (through the **KMESH**, **LMESH**, or **ESURF** commands) the **TSHAP** command is ignored and ANSYS chooses the correct shape codes automatically.

Figure 3.4 Meshing Patterns for Arbitrary Target Surfaces

3.6.5.1. Some Modeling and Meshing Tips

A target surface can be made up of two or more disconnected regions. Where possible, you should localize the contact zone by defining multiple target surfaces (each with a different real constant number). There are no restrictions on the shape of the rigid surfaces. Smoothing is not required. However, you must ensure that the mesh discretization of the curved surfaces on the rigid target surface is adequate. Excessively coarse discretization can cause numerical convergence problems. It can be difficult to obtain a converged solution in a large sliding simulation if the target surface has sharp convex corners. To avoid such modeling problems, use line or area fillet functions on the solid model to smooth out the sharp corner, use a more refined mesh, or use high-order element in the region of abrupt curvature changes (see Figure 3.5: "Smoothing Convex Corner").

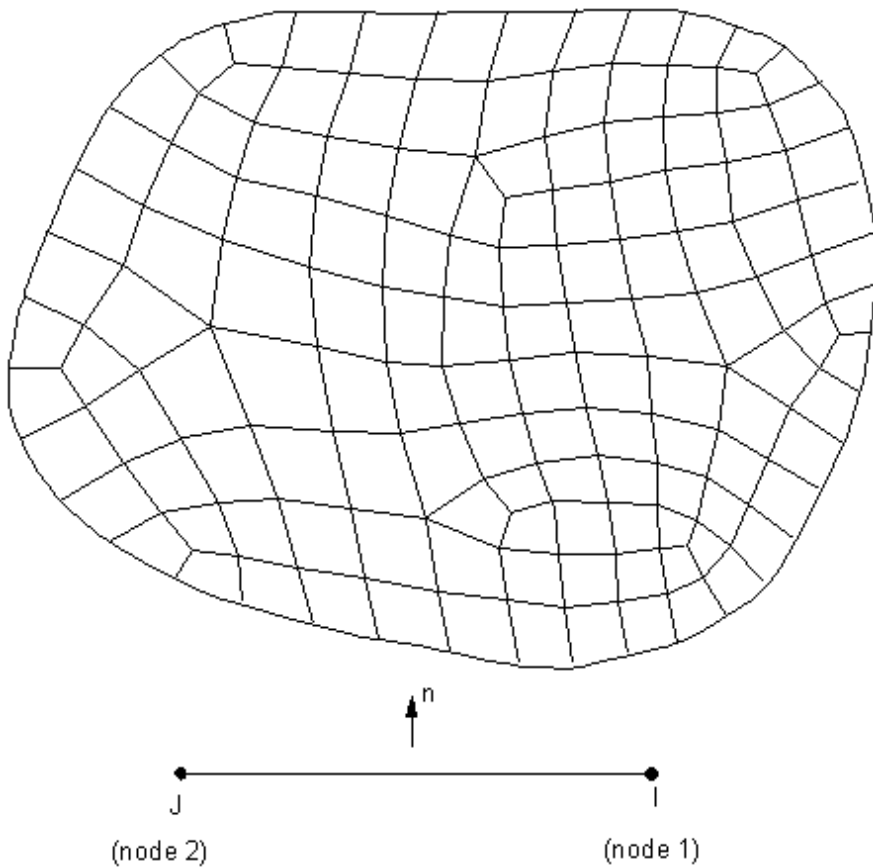
Figure 3.5 Smoothing Convex Corner



3.6.5.2. Verifying Nodal Number Ordering (Contact Direction) of Target Surface

The node order of the target surface elements is critical because it defines contact direction. For 2-D contact, the associated (deformable) contact elements must lie to the right of the target surface when moving from the first node to the second node along the target surface line (see Figure 3.6: “Correct Node Ordering”).

Figure 3.6 Correct Node Ordering



For 3-D contact, the target triangular element numbering should be such that the rigid surface's outward normal points toward the contact surface. The outward normal is determined by the right-hand rule.

To check the direction of the normals, turn on the element coordinate systems.

Command(s): /PSYMB,ESYS,1

GUI: Utility Menu> PlotCtrls> Symbols

If the element normals do not point toward the contact surface, select this element and reverse the direction of the surface normals.

Command(s): ESURF,,REVE

GUI: Main Menu> Preprocessor> Modeling> Create> Elements> Surf/Contact> Surf to Surf

or, reorient the element normals:

Command(s): ENORM

GUI: Main Menu> Preprocessor> Modeling> Move/Modify> Elements> Shell Normals

Note — Contact on target primitives (such as a complete circle, cylinder, cone, or sphere), can occur only on the outside surfaces of such target bodies.

3.7. Defining the Deformable Contact Surface

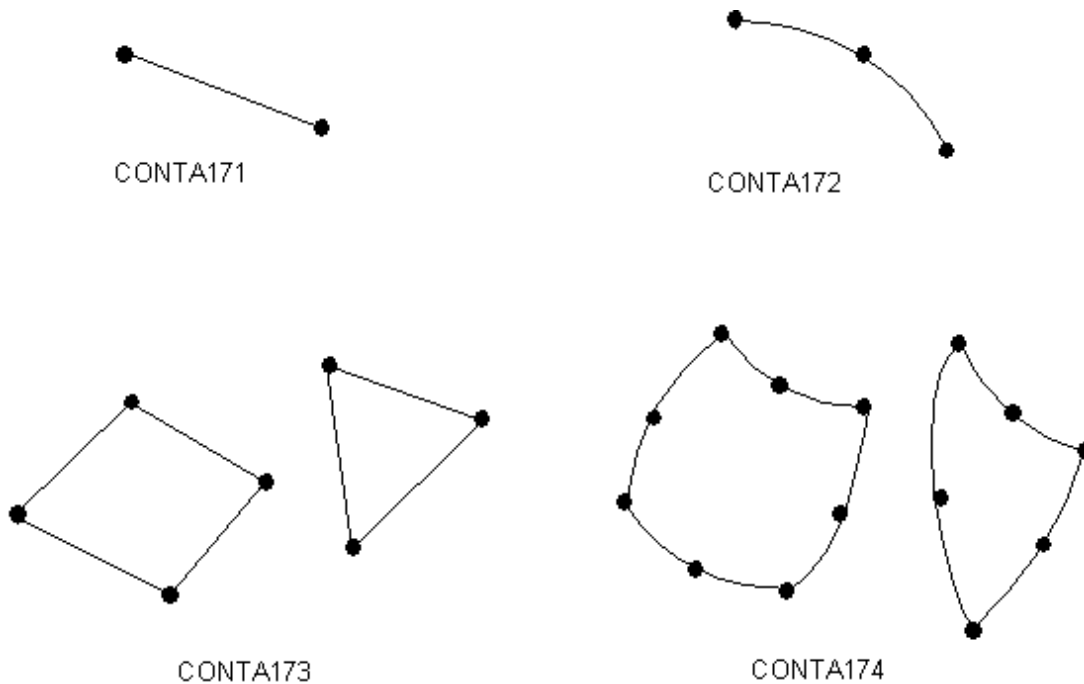
To create the deformable contact surface, you must define that surface using contact elements CONTA171 or CONTA172 (for 2-D), or CONTA173 or CONTA174 (for 3-D), or CONTA175 for (2- or 3-D).

The contact surface is defined by the set of contact elements that comprise the surface of the deformable body. These contact elements have the same geometric characteristics as the underlying elements of the deformable body. The contact surface elements are of the same order as the underlying elements (lower- or higher-order), with compatible nodes along the edges. The higher-order contact elements can match lower-order underlying elements by dropping the midside nodes. The underlying elements can be solid, shell, or 2-D beam elements. The contact surface can be on either side of the shell or beam elements. The underlying elements may also be a superelement. However, axisymmetric harmonic elements may not be used as underlying elements.

As with the target surface elements, you must define the contact surface element type, then select the correct real constant number (the real constant number must be the same as the one used for the target surface for each contact pair), and finally generate the elements.

3.7.1. Element Type

The four contact element types are listed below, along with a brief description. For complete information on these element types, see the *ANSYS Elements Reference*.

Figure 3.7 Contact Element Types

- CONTA171: This element is a 2-D, 2-node, lower-order line element that can be located on the surfaces of 2-D solid, shell, or beam elements (such as BEAM3, PLANE42, or SHELL51).
- CONTA172: This element is a 2-D, 3-node, higher-order parabolic element that can be located on the surfaces of 2-D solid or beam elements with midside nodes (such as PLANE82 or VISCO88).
- CONTA173: This element is a 3-D, 4-node, lower-order quadrilateral element that can be located on the surfaces of 3-D solid or shell elements (such as SOLID45 or SHELL181). It can be degenerated to a 3-node triangular element.
- CONTA174: This element is a 3-D, 8-node, higher-order quadrilateral element that can be located on the surfaces of 3-D solid or shell elements with midside nodes (such as SOLID92, SOLID95, or SHELL93). It can be degenerated to 3 - 7 node quadrilateral/triangular shapes.
- CONTA175: This element is a 2- or 3-D 1-node element that can be located on the surface of 2-D low order and higher order solid or beam elements or 3-D low order solid or shell elements.

Command(s): ET

GUI: Main Menu > Preprocessor > Element Type > Add/Edit/Delete

Note — Use CONTA175 for node-to-surface contact.

3.7.2. Real Constants and Material Properties

After defining the element type, you need to select the correct real constant set. The real constant set for each contact surface must be the same one used for the corresponding target surface for each contact pair. Each contact pair must reference its own real constant number.

ANSYS uses the material properties of the underlying elements to calculate an appropriate contact (or penalty) stiffness. In cases where the underlying element has **TB** plasticity material properties defined (whether active or not), the contact normal stiffness will be reduced by a factor of 100. ANSYS automatically defines a default value for tangent (sliding) contact stiffness that is proportional to MU and the normal stiffness. If the underlying element is a superelement, the material property set for the contact elements must be the same as that of the original structural elements used during the formation of the superelement.

3.7.3. Generating Contact Elements

You can generate contact elements either through direct generation or by generating the surface automatically from the exterior faces of the underlying elements. We recommend that you use automatic generation; this approach is simpler and more reliable.

To automatically generate contact elements, follow these steps:

1. Select the nodes on the meshed deformable body. For each surface, view the node list. If you are certain that particular nodes will never come into contact, you can omit those nodes and reduce CPU time. However, you should always include more nodes than you think you'll need so that you don't miss unexpected areas of contact.

Command(s): NSEL

GUI: Utility Menu > Select > Entities

2. Generate the contact elements.

Command(s): ESURF

GUI: Main Menu > Preprocessor > Modeling > Create > Elements > Surf/Contact > Surf to Surf

If the contact surface is attached to areas or volumes that are meshed with solid elements, ANSYS automatically determines the outward normal needed for contact calculations. If the underlying elements are beam or shell elements, you must indicate which surface (top or bottom) is the target surface.

Command(s): ESURF,,TOP or BOTTOM

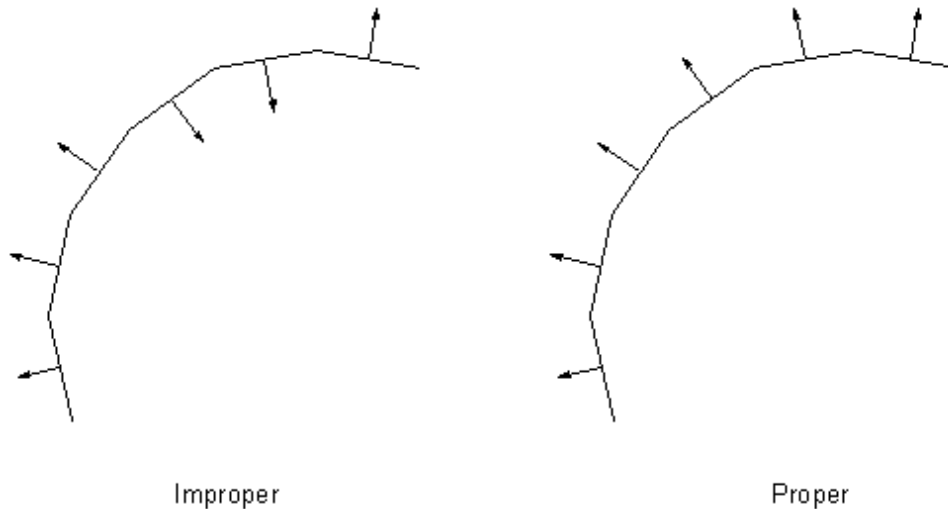
GUI: Main Menu > Preprocessor > Modeling > Create > Elements > Surf/Contact > Surf to Surf

Use the TOP setting (default) to generate contact elements with their outward normals the same as the beam or shell elements' normals. Use the BOTTOM setting to generate contact elements with their outward normals opposite the beam or shell elements' normals. You must make sure that all elements in the beam or shell element selection have their normals consistently oriented. If the underlying elements are solid elements, then the TOP or BOTTOM setting has no effect.

3. Check the direction of the outward normals for the contact elements. The direction of the contact surface's outward normal is critical for proper contact detection. For 3-D elements, the node numbering follows the right hand rule to define its outward normal. The contact surface's outward normal should point toward the target surface. Otherwise, ANSYS may detect over-penetration of the surfaces at the beginning of the analysis and have difficulty finding an initial solution. In most of these cases, the analysis will fail immediately. Figure 3.8: "Specification of the Contact Surface's Outward Normal" illustrates both proper and improper specification of the contact surface's outward normal.

Command(s): /PSYMB,ESYS

GUI: Utility Menu > PlotCtrls > Symbols

Figure 3.8 Specification of the Contact Surface's Outward Normal

If the surface normals are specified incorrectly, you must either change them by reversing the node number order of the selected elements.

Command(s): `ESURF,,REVE`

GUI: Main Menu > Preprocessor > Modeling > Create > Elements > Surf/Contact > Surf to Surf

or, reorient the element normals.

Command(s): `ENORM`

GUI: Main Menu > Preprocessor > Modeling > Move/Modify > Elements > Shell Normals

Note — `ESURF,,TOP` or `BOTTOM` and checking the direction of outward normals will not apply to node-to-surface contact (`CONTA175`) because it is a single-node element with no surface normal associated with it.

3.8. Set the Real Constants and Element KEYOPTS

ANSYS uses several real constants and KEYOPTs to control contact behavior using surface-to-surface contact elements. For more information in addition to what is presented here, refer to the individual contact element descriptions in the *ANSYS Elements Reference*.

If you decide the real constant and KEYOPT settings you have specified for a particular contact pair are not appropriate, you can use the `CNCHECK,RESET` command to reset all values back to their default settings. Some real constants and key options are not affected by this command; see `CNCHECK` for details.

3.8.1. Real Constants

Two real constants, R1 and R2, are used to define the geometry of the target surface elements. The remaining are used by the contact surface elements.

- R1 and R2 define the target element geometry.
- FKN defines a normal contact stiffness factor.
- FTOLN is a factor based on the thickness of the element which is used to calculate allowable penetration.
- ICONT defines an initial closure factor (or adjustment band).
- PINB defines a "pinball" region.

- PMIN and PMAX define an allowable penetration range for initial penetration.
- TAUMAX specifies the maximum contact friction.
- CNOF specifies the positive or negative offset value applied to the contact surface.
- FKOP specifies the stiffness factor applied when contact opens.
- FKT specifies the tangent contact stiffness.
- COHE specifies the cohesion sliding resistance.
- TCC specifies the thermal contact conductance coefficient.
- FHTG specifies the fraction of frictional dissipated energy converted into heat.
- SBCT specifies the Stefan-Boltzmann constant.
- RDVF specifies the radiation view factor.
- FWGT specifies the weight factor for the distribution of heat between the contact and target surfaces for thermal contact or for electric contact.
- ECC specifies the electric contact conductance or capacitance per unit area.
- FHEG specifies the fraction of electric dissipated energy converted into heat.
- FACT specifies the ratio of static to dynamic coefficients of friction.
- DC specifies the decay coefficient for static/dynamic friction.
- SLTO controls maximum sliding distance when MU is nonzero and the tangent contact stiffness (FKT) is updated at each iteration (KEYOPT(10) = 2).
- TNOP specifies the maximum allowable tensile contact pressure.
- TOLS adds a small tolerance that extends the edge of the target surface.
- MCC specifies the magnetic contact permeance (3-D only).

Real constant defaults can vary depending on the environment you are working in. The following table compares the default values between ANSYS and the ANSYS Workbench. See your ANSYS sales representative for more information about ANSYS Workbench.

Table 3.1 Summary of Real Constant Defaults in Different Environments

Real Constants		Description	ANSYS Default	ANSYS Workbench Default
No.	Name			
1	R1	Target circle radius	0	n/a
2	R2	Superelement thickness	1	n/a
3	FKN	Normal penalty stiffness factor	1	[1]
4	FTOLN	Penetration tolerance factor	0.1	0.1
5	ICONT	Initial contact closure	0	0
6	PINB	Pinball region	[2]	[2]
7	PMAX	Upper limit of initial penetration	0	0
8	PMIN	Lower limit of initial penetration	0	0
9	TAUMAX	Maximum friction stress	1.00E+20	1.00E+20
10	CNOF	Contact surface offset	0	0

Real Constants		Description	ANSYS Default	ANSYS Workbench Default
No.	Name			
11	FKOP	Contact opening stiffness	1	1
12	FKT	Tangent penalty stiffness	1	1
13	COHE	Contact cohesion	0	0
14	TCC	Thermal contact conductance	0	[3]
15	FHTG	Frictional heating factor	1	1
16	SBCT	Stefan-Boltzmann constant	0	n/a
17	RDVF	Radiation view factor	1	n/a
18	FWGT	Heat distribution weighing factor	0.5	0.5
19	ECC	Electric contact conductance	0	n/a
20	FHEG	Joule dissipation weighting factor	1	n/a
21	FACT	Static/dynamic ratio	1	1
22	DC	Exponential decay coefficient	0	0
23	SLTO	Allowable elastic slip	1%	1%
24	(blank)			
25	TOLS	Target edge extension factor	[4]	[4]
26	MCC	Magnetic contact permeance	0	n/a

1. FKN = 10 for bonded. For all other, FKN = 1.0, but if bonded and other contact behavior exists, FKN = 1 for all.
2. Depends on contact behavior (rigid vs. flex target), **NLGEOM,ON** or OFF, KEYOPT(9) setting, KEYOPT(12) setting, and the value of CNOF (see Section 3.8.9.2: Using PINB).
3. Calculated as a function of highest conductivity and overall model size.
4. 10% of target length for **NLGEOM,OFF**.
2% of target length for **NLGEOM,ON**.

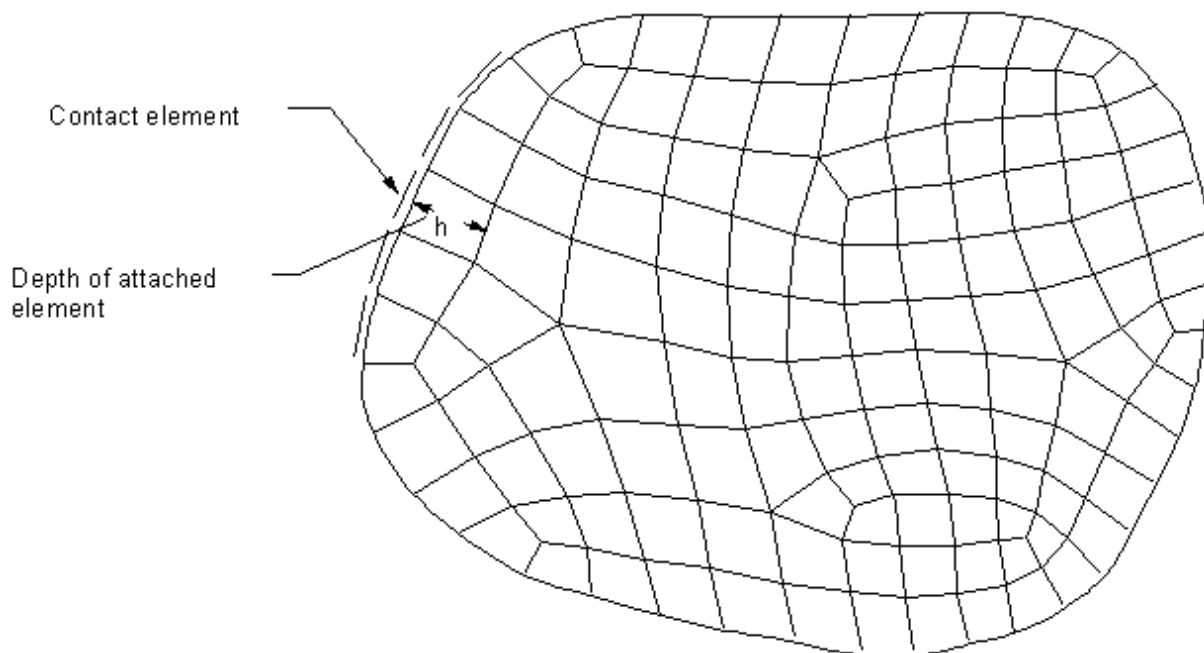
Command(s): R

GUI: Main Menu > Preprocessor > Real Constants

3.8.1.1. Positive and Negative Real Constant Values

For the real constants FKN, FTOLN, ICONT, PINB, PMAX, PMIN, FKOP, FKT, SLTO, and TNOP, you can specify either a positive or negative value. ANSYS interprets a positive value as a *scaling factor* and interprets a negative value as the *absolute value*. ANSYS uses the depth of the underlying element as the reference value to be used for ICONT, FTOLN, PINB, PMAX, and PMIN. For example, a positive value of 0.1 for ICONT indicates an initial closure factor of 0.1 x depth of the underlying element. However, a negative value of 0.1 indicates an actual adjustment band of 0.1 units. When KEYOPT(10) = 0, 1, or 2, all the contact related settings (ICONT, FTOLN, PINB, PMAX, PMIN, FKN, FKT, SLTO) are averaged across all contact elements in a contact pair. However, when KEYOPT(10) = 3, 4, or 5, the settings are based on each individual contact element (geometry and material behaviors).

Figure 3.9: “Depth of the Underlying Element” shows the depth of the underlying element for a solid element. If the underlying elements are shell or beam elements, the depth will usually be 4 times the element thickness.

Figure 3.9 Depth of the Underlying Element

When KEYOPT(10) = 0, 1, or 2, each contact pair has a pair-based depth which is obtained by averaging the depth of each contact element across all the contact elements in a contact pair. This can avoid the problem of very different element-based depths when there are meshes with large variations in element sizes.

Note — When the contact pair depth is too small (for example, 10^{-5}), the machine precision may not guarantee the accuracy of penetration to be calculated. You should scale the length unit in the model.

3.8.2. Element KEYOPTS

Each contact element includes several KEYOPTS. We recommend using the default settings, which are suitable for most contact problems. For some specific applications, you can override the defaults. The element KEYOPTS allow you to control several aspects of contact behavior.

- Degrees of freedom (KEYOPT(1))
- Contact algorithm (defaults to augmented Lagrangian) (KEYOPT(2))
- Stress state when superelements are present (KEYOPT(3)) for 2-D surface-to-surface contact. Contact model for node-to-surface contact.
- Location of contact detection point (KEYOPT(4))
- CNOF Automated adjustment (KEYOPT(5))
- Time step control (KEYOPT(7))
- Asymmetric contact selection (KEYOPT(8))
- Effect of initial penetration or gap (KEYOPT(9))
- Contact stiffness update (KEYOPT(10))
- Shell thickness effect (KEYOPT(11))
- Behavior of contact surface (rough, bonded, etc.) (KEYOPT(12))

KEYOPT defaults can vary depending on the environment you are working in. The following table compares the default values between ANSYS, the ANSYS Contact Wizard, and the ANSYS Workbench. See your ANSYS sales representative for more information about ANSYS Workbench.

Table 3.2 Summary of KEYOPT Defaults in Different Environments

KEYOPT	Description	ANSYS	ANSYS Contact Wizard	ANSYS Workbench Default Linear (bonded, no sep)	ANSYS Workbench, Default Nonlinear (standard, rough)
1	Selects DOF*	Manual	Auto	Auto	Auto
2	Contact Algorithm	Aug. Lagr.	Aug. Lagr.	Pure Penalty	Pure Penalty
3	Stress state when superelement is present	no super elem	no super elem	n/a	n/a
4	Location of contact detection point	gauss	gauss	gauss	gauss
5	CNOF/ICONT adjustment	No adjust	No adjust	No adjust	No adjust
6	(blank)				
7	Element level time increment control	No control	No control	No control	No control
8	Asymmetric contact selection	No action	No action	No action	No action
9	Effect of initial penetration or gap	Include all	Include all	Exclude all	Include all/ramped
10	Contact stiffness update	Between load steps	Between substeps	Between load steps	Between load steps
11	Beam/shell thickness effect	Exclude	Exclude	Exclude	Exclude
12	Behavior of contact surface	Standard	Standard	Bonded	n/a

*Manual: Requires user to define.

Auto: Selection is based on DOF of underlying element.

Command(s): KEYOPT, ET

GUI: Main Menu > Preprocessor > Element Type > Add/Edit/Delete

3.8.3. Selecting a Contact Algorithm (KEYOPT(2))

3.8.3.1. Background

For surface-to-surface contact elements, ANSYS offers several different contact algorithms:

- Penalty method (KEYOPT(2) = 1)
- Augmented Lagrangian (default) (KEYOPT(2) = 0)
- Lagrange multiplier on contact normal and penalty on tangent (KEYOPT(2) = 3)
- Pure Lagrange multiplier on contact normal and tangent (KEYOPT(2) = 4)
- Internal multipoint constraint (MPC) (KEYOPT(2) = 2)

The penalty method uses a contact “spring” to establish a relationship between the two contact surfaces. The spring stiffness is called the contact stiffness. This method uses the following real constants: FKN and FKS for all values of KEYOPT(10), plus FTOLN and SLTO if KEYOPT(10) = 1 or 2.

The augmented Lagrangian method (which is the default) is an iterative series of penalty methods. The contact tractions (pressure and frictional stresses) are augmented during equilibrium iterations so that the final penetration is smaller than the allowable tolerance (FTOLN). Compared to the penalty method, the augmented Lagrangian method usually leads to better conditioning and is less sensitive to the magnitude of the contact stiffness. However, in some analyses, the augmented Lagrangian method may require additional iterations, especially if the deformed mesh becomes too distorted.

The pure Lagrange multiplier method enforces zero penetration when contact is closed and “zero slip” when sticking contact occurs. The pure Lagrange multiplier method does not require contact stiffness, FKN and FKS. Instead it requires chattering control parameters, FTOLN and TNOP. This method adds contact traction to the model as additional degrees of freedom and requires additional iterations to stabilize contact conditions. It often increases the computational cost compared to the augmented Lagrangian method.

An alternative algorithm is the Lagrange multiplier method applied on the contact normal and the penalty method (tangential contact stiffness) on the frictional plane. This method enforces zero penetration and allows a small amount of slip for the sticking contact condition. It requires chattering control parameters, FTOLN and TNOP, as well as the maximum allowable elastic slip parameter SLTO.

Another method, the internal multipoint constraint (MPC) algorithm, is used in conjunction with bonded contact (KEYOPT(12) = 5 or 6) and no separation contact (KEYOPT(12) = 4) to model several types of contact assemblies and kinematic constraints. See Chapter 7, “Multipoint Constraints and Assemblies” for more information on how to use this feature.

Note — The Lagrange multiplier methods (KEYOPT(2) = 3, 4) and MPC approach (KEYOPT(2) = 2) do not support the Gauss point detection option (KEYOPT(4) = 0) for surface-to-surface contact. They support the nodal detection options for surface-to-surface contact and node-to-surface contact. When using these options, be careful not to overconstrain the model. The model is overconstrained when a contact node has prescribed boundary conditions, CE and CP equations. ANSYS usually detects and eliminates the overconstraints. However, there is no guarantee that the program will eliminate all the cases of overconstraint. You should always verify your model carefully to address this issue. The Lagrange multiplier also introduces more degrees of freedom which may result in spurious modes for modal and linear eigenvalue buckling analyses. The augmented Lagrangian method would be a better choice for these analysis types.

Note — The Lagrange multiplier methods (KEYOPT(2) = 3, 4) introduce zero diagonal terms in the stiffness matrix. Any iterative solver (PCG or AMG) will encounter a preconditioning matrix singularity with these methods. Therefore, you should switch to sparse solver.

Note — If overconstraint occurs in bonded shell-shell assemblies when using the MPC algorithm, you can switch to the penalty method or the augmented Lagrangian method. See Section 3.8.11.3: Bonded Contact for Shell-Shell Assemblies for more information.

3.8.4. Determining Contact Stiffness and Allowable Penetration

3.8.4.1. Background

For the augmented Lagrangian method and penalty method, normal and tangential contact stiffnesses are required. The amount of penetration between contact and target surfaces depends on the normal stiffness. The amount of slip in sticking contact depends on the tangential stiffness. Higher stiffness values decrease the amount of penetration/slip, but can lead to ill-conditioning of the global stiffness matrix and to convergence difficulties. Lower stiffness values can lead to a certain amount of penetration/slip and produce an inaccurate solution. Ideally, you want a high enough stiffness that the penetration/slip is acceptably small, but a low enough stiffness that the problem will be well-behaved in terms of convergence.

ANSYS provides default values for contact stiffnesses (FKN, FKT), allowable penetration (FTOLN), and allowable slip (SLTO). In most cases, you do not need to define the contact stiffness. In addition, we recommend that you use KEYOPT(10) = 1 to allow the program to update the contact stiffness automatically.

3.8.4.2. Using FKN and FTOLN

For certain contact problems, you may choose to use the real constant FKN to define a normal contact stiffness factor. The usual factor range is from 0.01-1.0, with a default of 1.0. The default value is appropriate for bulk deformation. If bending deformation dominates, we recommend using a smaller value (0.1). In cases where the underlying element has a **TB** plasticity material defined (whether plasticity is active or not), the contact normal stiffness will be reduced by a factor of 100.

Use real constant FTOLN in conjunction with the augmented Lagrangian method. FTOLN is a tolerance factor to be applied in the direction of the surface normal. The range for this factor is less than 1.0 (usually less than 0.2), with a default of 0.1, and is based on the depth of the underlying solid, shell, or beam element (see Figure 3.9: "Depth of the Underlying Element"). This factor is used to determine if penetration compatibility is satisfied. Contact compatibility is satisfied if penetration is within an allowable tolerance (FTOLN times the depth of underlying elements). The depth is defined by the average depth of each individual contact element in the pair. If ANSYS detects any penetration larger than this tolerance, the global solution is still considered unconverged, even though the residual forces and displacement increments have met convergence criteria. You can also define an absolute allowable penetration by specifying a negative value for FTOLN.

Note — When the contact stiffness is too large (for example, 10^{16}), the machine precision may not guarantee good conditioning of the global stiffness matrix. In this case, you should scale the force unit in the model if possible.

Note — FTOLN is also used in the Lagrange multiplier methods (KEYOPT(2) = 3, 4) as a chattering control parameter.

3.8.4.3. Using FKT and SLTO

ANSYS automatically defines a default tangential contact stiffness that is proportional to MU and the normal stiffness FKN. The default tangential stiffness corresponds to a default value of FKT = 1.0. A positive value for FKT is a factor; a negative value indicates an absolute value of tangential stiffness.

For KEYOPT(10) = 1 or 2, or when the Lagrange multiplier on normal and penalty on tangent option is used (KEYOPT(2) = 3), ANSYS updates tangential contact stiffness based on current contact normal pressure, PRES, and maximum allowable elastic slip, SLTO (FKT = MU* PRES/SLTO). The real constant SLTO is used to control maximum sliding distance when FKT is updated at each iteration. ANSYS provides default tolerance values which work well in most cases. You can override the default values for SLTO (1% of average contact length in pair) by defining a scaling factor (positive value when using command input) or an absolute value (negative value when using command input). A larger value will enhance convergence but compromise accuracy. Based on the tolerance, current normal pressure, and friction coefficient, the tangential contact stiffness FKT can be obtained automatically. In certain cases users can override FKT by defining a scaling factor (positive input value) or absolute value (negative input value) (see Positive and Negative Real Constants for more information).

Note — FKN, FTOLN, FKT, and SLTO can be modified from one load step to another. They can also be adjusted in a restart run. Determining a good stiffness value may require some experimentation on your part. To arrive at a good stiffness value, you can try the following procedure as a “trial run”:

1. Use a low value for the contact stiffness to start. In general, it is better to underestimate this value rather than overestimate it. Penetration problems resulting from a low stiffness are easier to fix than convergence difficulties that arise from a high stiffness.
2. Run the analysis up to a fraction of the final load (just enough to get the contact fully established).
3. Check the penetration and the number of equilibrium iterations used in each substep. If the global convergence difficulty is caused by too much penetration (rather than by residual forces and displacement increments), FKN may be underestimated or FTOLN may be too small. If the global convergence requires many equilibrium iterations for achieving convergence tolerances of residual forces and displacements rather than the resulting penetration, FKN or FKT may be overestimated.
4. Adjust FKN, FKT, FTOLN, or SLTO as necessary and run the full analysis. If the penetration control becomes dominant in the global equilibrium iterations (that is, if more iterations were used to converge the problem to within the penetration tolerance than to converge the residual forces), you may increase FTOLN to permit more allowable penetration or increase FKN.

Note — Generally, the contact stiffness FKN, FKT has units FORCE/LENGTH³. However, for contact force-based models (KEYOPT(3) = 0) in CONTA175, the contact stiffness has units FORCE/LENGTH.

3.8.4.4. Using KEYOPT(10)

The normal and tangential contact stiffness can be updated during the course of an analysis, either automatically (due to large strain effects that change the underlying element's stiffness) or explicitly (by user-specified FKN or FKT values). KEYOPT(10) governs how the normal and tangential contact stiffness is updated when the augmented Lagrangian or penalty method is used. In most cases we recommend that you use KEYOPT(10) = 1 to allow the program to update contact stiffnesses automatically. The possible settings for KEYOPT(10) are outlined below.

- KEYOPT(10) = 0, the contact stiffness will be updated at each load step if FKN or FKT is redefined by the user. Stiffness and other settings (ICONT, FTOLN, SLTO, PINB, PMAX, and PMIN) are averaged across contact elements in a contact pair. The default contact stiffness is determined by underlying element depth and material properties.
- KEYOPT(10) = 1 (covers KEYOPT(10) = 0), the normal contact stiffness will be updated at every substep based on the mean stress of the underlying elements from the previous substep and the allowable penetration, FTOLN, except in the first substep of the first load step. The default normal contact stiffness for the first substep of the first load step is the same as described for KEYOPT(10) = 0. If bisections occur in the beginning of the analysis, the normal contact stiffness will be reduced by a factor of 0.2 for each bisection. The tangential contact stiffness will be updated at each iteration based on the current contact pressure, MU, and allowable slip (SLTO).
- KEYOPT(10) = 2 (covers KEYOPT(10) = 1), the normal contact stiffness will be updated at each iteration based on the current mean stress of the underlying elements and the allowable penetration, FTOLN, except in the very first iteration. The default normal contact stiffness for the first iteration is the same as described for KEYOPT(10) = 0. If bisections occur in the beginning of the analysis, the normal contact stiffness will be reduced by a factor of 0.2 for each bisection. The tangential contact stiffness will be updated at each iteration based on the current contact pressure, MU, and allowable slip (SLTO).
- KEYOPT(10) = 3, same as KEYOPT(10) = 0, except stiffness and settings are not averaged across the contact elements in a contact pair. If bisections occur in the beginning of the analysis, the normal contact stiffness will be reduced by a factor of 0.2 for each bisection.

- KEYOPT(10) = 4, same as KEYOPT(10) = 1, except stiffness and settings are not averaged across the contact elements in a contact pair.
- KEYOPT(10) = 5, same as KEYOPT(10) = 2, except stiffness and settings are not averaged across the contact elements in a contact pair.

Note — When a Lagrange multiplier method (KEYOPT(2) = 3, 4) or MPC algorithm (KEYOPT(2) = 2) is used, KEYOPT(10) is ignored.

3.8.4.5. Chattering Control Parameters

The Lagrange multiplier methods (KEYOPT(2) = 3, 4) do not require contact stiffness, FKN and FKT. Instead they require chattering control parameters, FTOLN and TNOP, by which ANSYS assumes that the contact status remains unchanged. FTOLN is the maximum allowable penetration and TNOP is the maximum allowable tensile contact pressure.

Note — A negative contact pressure occurs when the contact status is closed. A tensile contact pressure (positive) refers to a separation between the contact surfaces, but not necessarily an open contact status.

The behavior can be described as follows:

- If the contact status from the previous iteration is open and the current calculated penetration is smaller than FTOLN, then contact remains open. Otherwise the contact status switches to closed and another iteration is processed.
- If the contact status from the previous iteration is closed and the current calculated contact pressure is positive but smaller than TNOP, then contact remains closed. If the tensile contact pressure is larger than TNOP, then the contact status changes from closed to open and ANSYS continues to the next iteration.

ANSYS will provide reasonable defaults for FTOLN and TNOP. FTOLN defaults to the displacement convergence tolerance. TNOP defaults to the force convergence tolerance divided by contact area at contact nodes.

Keep in mind the following when providing values for FTOLN and TNOP:

- A positive value is a scaling factor applied to the default values.
- A negative value is used as an absolute value (which overrides the default).

The objective of FTOLN and TNOP is to provide stability to models which exhibit contact chattering due to changing contact status. If the values you use for these tolerances are too small, the solution will require more iterations. However, if the values are too large, the accuracy of the solution will be affected since a certain amount of penetration or tensile contact force is allowed.

3.8.5. Choosing a Friction Model

3.8.5.1. Background

In the basic Coulomb friction model, two contacting surfaces can carry shear stresses up to a certain magnitude across their interface before they start sliding relative to each other. This state is known as sticking. The Coulomb friction model defines an equivalent shear stress τ , at which sliding on the surface begins as a fraction of the contact pressure p ($\tau = \mu p + \text{COHE}$, where μ is the friction coefficient and COHE specifies the cohesion sliding resistance). Once the shear stress is exceeded, the two surfaces will slide relative to each other. This state is known as sliding. The sticking/sliding calculations determine when a point transitions from sticking to sliding or vice versa.

For frictionless, rough, and bonded contact, the contact element stiffness matrices are symmetric. Contact problems involving friction produce unsymmetric stiffnesses. Using an unsymmetric solver is more computationally expensive than a symmetric solver for each iteration. For this reason, ANSYS uses a symmetrization algorithm by which most frictional contact problems can be solved using solvers for symmetric systems. If frictional stresses have a substantial influence on the overall displacement field and the magnitude of the frictional stresses is highly solution dependent, the symmetric approximation to the stiffness matrix may provide a low rate of convergence. In such cases, choose the unsymmetric solution option (**NROPT,UNSYM**) to improve convergence.

3.8.5.2. Using TAUMAX, FACT, DC, and COHE

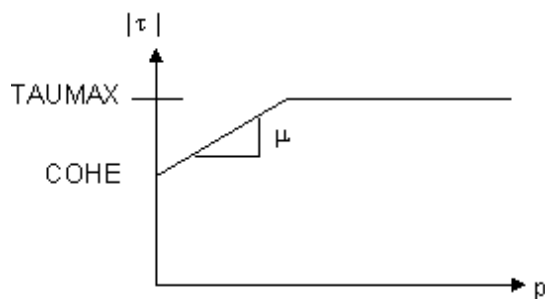
One of the material properties used for contact elements is input via the interface coefficient of friction, MU, for the Coulomb friction model. Use MU = 0 for frictionless contact. For rough or bonded contact (KEYOPT(12) = 1, 3, 5, or 6, see Selecting Surface Interaction Models), ANSYS assumes infinite frictional resistance regardless of specified value of μ . MU can be specified as a function of temperature. If the underlying element is a superelement (MATRIX50), the material property set must be the same as the one used for the original elements that were assembled into the superelement.

ANSYS provides one extension of classical Coulomb friction: TAUMAX is maximum contact friction with units of stress. This maximum contact friction stress can be introduced so that, regardless of the magnitude of normal contact pressure, sliding will occur if the friction stress reaches this value. You typically use TAUMAX when the contact pressure becomes very large (such as in bulk metal forming processes). TAUMAX defaults to 1.0e20.

Empirical data is often the best source for TAUMAX. Its value may be close to $\frac{\sigma_y}{\sqrt{3}}$, where σ_y is the yield stress of the material being deformed.

Another real constant used for the friction law is the cohesion, COHE (default COHE = 0), which has units of stress. It provides sliding resistance, even with zero normal pressure (see Figure 3.10: "Sliding Contact Resistance").

Figure 3.10 Sliding Contact Resistance



Two other real constants, FACT and DC are involved in specifying static and dynamic friction coefficients, as described in the next section.

3.8.5.3. Static and Dynamic Friction Coefficients

The coefficient of friction can depend on the relative velocity of the surfaces in contact. Typically, the static coefficient of friction is higher than the dynamic coefficient of friction.

ANSYS provides the following exponential decay friction model:

$$\mu = MU \times (1 + (\text{FACT} - 1) \exp(-DC \times V_{\text{rel}}))$$

where:

μ = coefficient of friction.

MU = dynamic coefficient of friction input using the **MP** command.

FACT = ratio of static to dynamic coefficients of friction. It defaults to the minimum value of 1.0

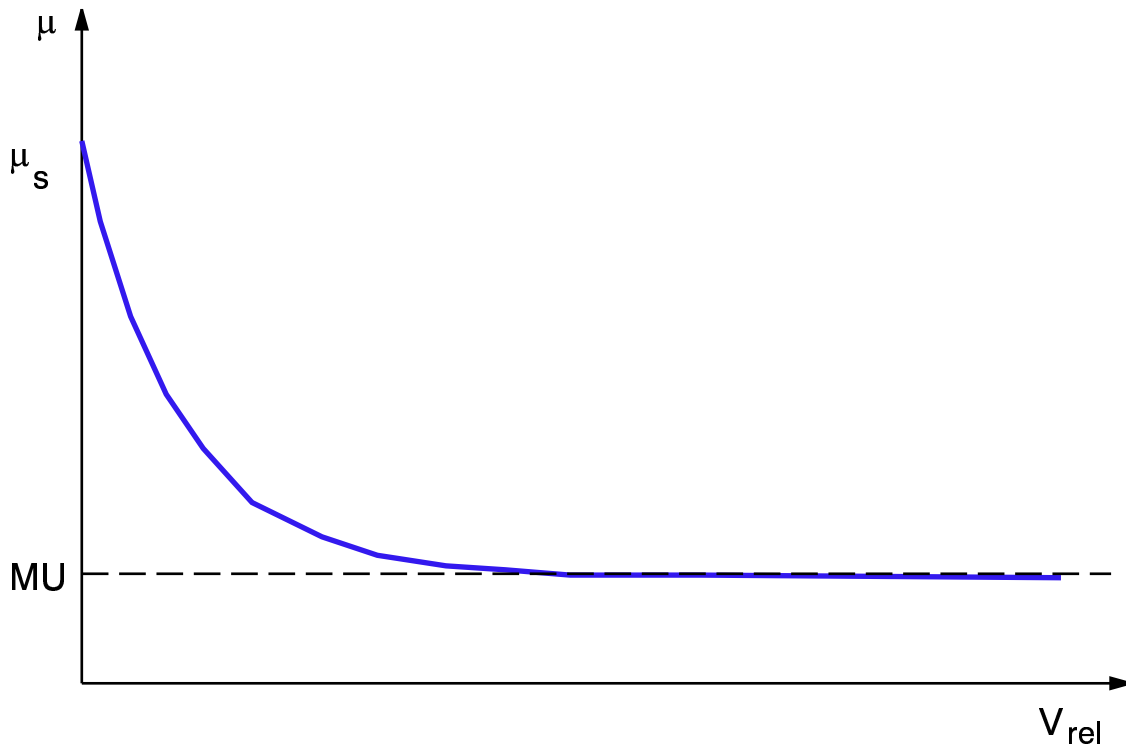
DC = decay coefficient. It defaults to 0.0 and it has units of time/length. Therefore, time has some meaning in a static analysis.

Vrel = slip rate calculated by ANSYS.

Figure 3.11: "Friction Decay" shows the exponential decay curve where the static coefficient of friction is given by:

$$\mu_s = \text{FACT} \times \text{MU}$$

Figure 3.11 Friction Decay



You can determine the decay coefficient if you know the static and dynamic coefficients of friction and at least one data point ($\mu_1 ; V_{rel_1}$). The equation for friction decay can be rearranged to give:

$$\text{DC} = -\frac{1}{V_{rel_1}} \times \ln \left(\frac{\mu_1 - \text{MU}}{(\text{FACT} - 1) \times \text{MU}} \right)$$

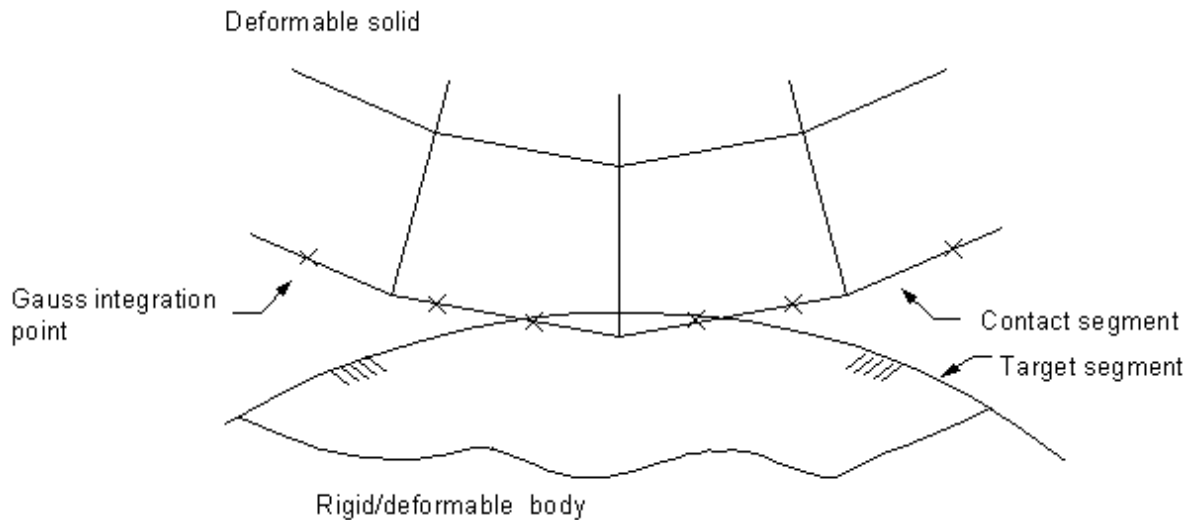
If you do not specify a decay coefficient and FACT is greater than 1.0, the coefficient of friction will change suddenly from the static to the dynamic value as soon as contact reaches the sliding state. This behavior is not recommended because the discontinuity may lead to convergence difficulties.

3.8.6. Selecting Location of Contact Detection

3.8.6.1. Background

Contact detection points are located at the integration points of the contact elements which are interior to the element surface. The contact element is constrained against penetration into the target surface at its integration points. However, the target surface can, in principle, penetrate through into the contact surface, see Figure 3.12: “Contact Detection Located at Gauss Point”.

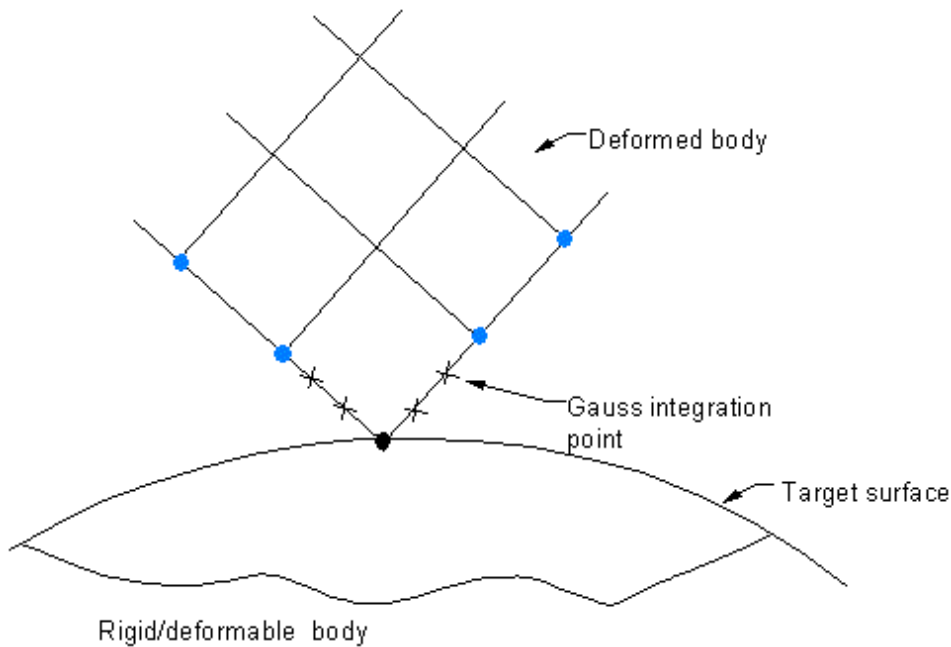
Figure 3.12 Contact Detection Located at Gauss Point



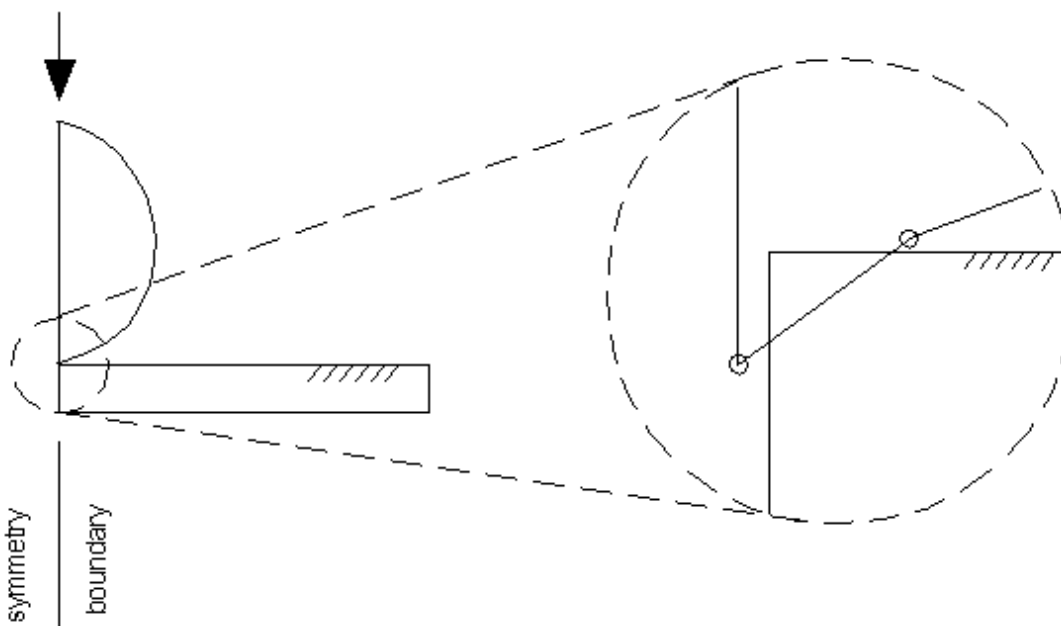
ANSYS surface-to-surface contact elements use Gauss integration points as a default, which generally provide more accurate results than the nodal detection scheme, which uses the nodes themselves as the integration points. The node-to-surface contact element CONTA175 always uses the nodal detection scheme.

3.8.6.2. Using KEYOPT(4) and TOLS

The nodal detection algorithms require the smoothing of the contact surface ($\text{KEYOPT}(4) = 1$) or the smoothing of the target surface ($\text{KEYOPT}(4) = 2$), which is quite time consuming. You should use this option only to deal with corner, point-surface, or edge-surface contact (see Figure 3.13: “Contact Detection Point Location at Nodal Point”). $\text{KEYOPT}(4) = 1$ specifies that the contact normal be perpendicular to the contact surface. $\text{KEYOPT}(4) = 2$ specifies that the contact normal be perpendicular to the target surface. Use this option ($\text{KEYOPT}(4) = 2$) when the target surface is smoother than the contact surface.

Figure 3.13 Contact Detection Point Location at Nodal Point

Be aware, however, that using nodes as the contact detection points can lead to other convergence difficulties, such as "node slippage," where the node slips off the edge of the target surface, see Figure 3.14: "Node Slippage Using Nodal Integration KEYOPT(4) = 1 or 2". In order to prevent node slippage, you can use real constant TOLS to extend the target surface when the default setting still cannot avoid the problem. For most point-to-surface contact problems, we recommend using CONTA175; see Chapter 4, "Node-to-Surface Contact" later in this guide.

Figure 3.14 Node Slippage Using Nodal Integration KEYOPT(4) = 1 or 2

Smoothing is required for nodal detection algorithms, and it is performed by averaging surface normals connected to the node. As a result, the variation of the surface normal is continuous over the surface, which leads to a better calculation of friction behavior and a better convergence.

Real constant TOLS is used to add a small tolerance that will internally extend the edge of the target surface when you define the contact detection at the nodal point (KEYOPT(4) = 1 or 2). TOLS is useful for problems where contact nodes are likely to lie on the edge of targets (as at symmetry planes or for models generated in a node-to-node contact pattern). In these situations, the contact node may repeatedly "slip" off the target surface and go completely out of contact, resulting in convergence difficulties from oscillations. Units for TOLS are percent (1.0 implies a 1.0% increase in the target edge length). A small value of TOLS will usually prevent this situation from occurring. The default value is 10 for small deflection and 2 for large deflection (**NLGEOM, ON**).

Note — The definition of KEYOPT(4) in node-to-surface contact element CONTA175 is different. KEYOPT(4) = 1 for surface-to-surface contact is equivalent to KEYOPT(4) = 1 for node-to-surface contact. However, KEYOPT(4) = 2 for surface-to-surface contact is equivalent to KEYOPT(4) = 0 for node-to surface contact. See Section 4.2.1.2: KEYOPT(4).

3.8.7. Adjusting Initial Contact Conditions

3.8.7.1. Background

Rigid body motion is usually not a problem in dynamic analyses. However, in static analyses, rigid body motion occurs when a body is not sufficiently restrained. "Zero or negative pivot" warning messages and impractical, excessively large displacements indicate unconstrained motion in a static analysis.

In simulations where rigid body motions are constrained only by the presence of contact, you must ensure that the contact pairs are in contact in the initial geometry. In other words, you want to build your model so that the contact pairs are "just touching." However, you can encounter various problems in doing so:

- Rigid body profiles are often complicated, making it difficult to determine where the first point of contact might occur.
- Small gaps between element meshes on both sides of the element pair can be introduced by numerical round-off, even if the solid model is built in an initially-contacting state.
- Small gaps can exist between the integration points of the contact elements and target surface elements.

For the same reasons, too much initial penetration between target and contact surfaces can occur. In such cases, the contact elements may overestimate the contact forces, resulting in nonconvergence or in breaking-away of the components in contact.

The definition of initial contact is perhaps the most important aspect of building a contact analysis model. Therefore, you should always issue the **CNCHECK** command before starting the solution to verify the initial contact status. You may find that you need to adjust the initial contact conditions. ANSYS offers several ways to adjust the initial contact conditions of a contact pair.

3.8.7.2. Using PMIN, PMAX, CNOF, ICONT, KEYOPT(5), and KEYOPT(9)

The following techniques can be performed independently or in combinations of one or more at the beginning of the analysis. They are intended to eliminate small gaps or penetrations caused by numerical round-off due to mesh generation. They are not intended to correct gross errors in either the mesh or in the geometric data.

1. Use real constant CNOF to specify a contact surface offset.

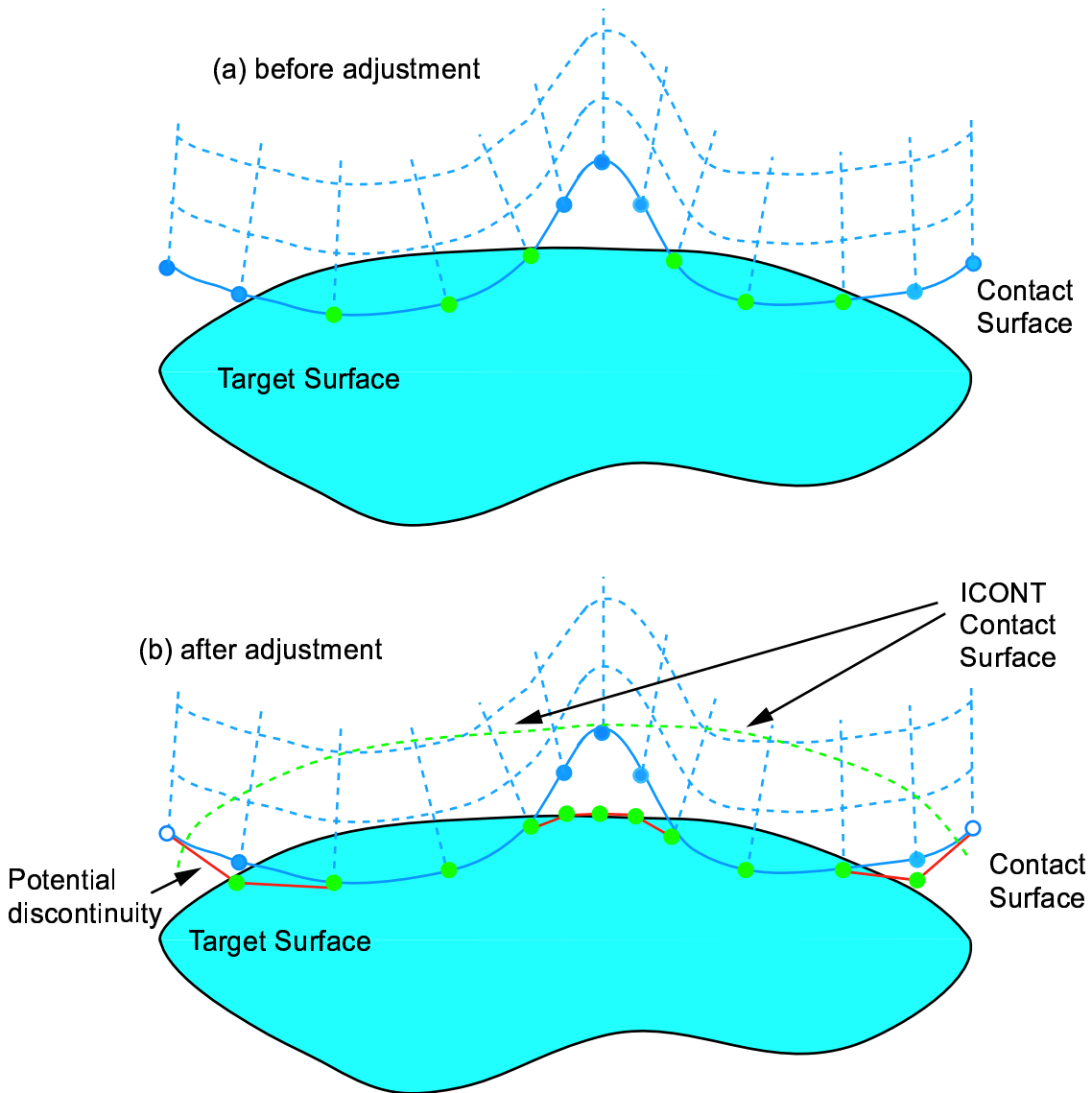
Specify a positive value to offset the entire contact surface towards the target surface. Use a negative value to offset the contact surface away from the target surface.

Note — If user-defined values are input for both CNOF and PINB, you must ensure that PINB is greater than CNOF. Otherwise, CNOF will be ignored. However, if a user-defined CNOF is input and the PINB value is left at its default value, the PINB value will be adjusted so that it is larger than the CNOF value, as described in Using PINB.

ANSYS can automatically provide the CNOF value to either just close the gap or reduce initial penetration. Set KEYOPT(5) as follows:

- 1:** Closes the gap
 - 2:** Reduces initial penetration
 - 3:** Either closes the gap or reduces initial penetration
2. Use the real constant ICONT to specify a small initial contact closure. This is the depth of an "adjustment band" around the target surface. A positive value for ICONT indicates a scaling factor relative to the depth of the underlying elements. A negative value indicates an absolute contact closure value. The value of ICONT defaults to zero if KEYOPT(5) = 0, 1, 2, or 3. (The ICONT default is different when KEYOPT(12) = 6 for bonded-initial contact; see Section 3.8.11: Selecting Surface Interaction Models for more information). If KEYOPT(5) = 4, ANSYS provides a small (but meaningful) value for ICONT according to the geometric dimensions, and prints a warning message stating what value was assigned. Any contact detection points that fall within this adjustment band are internally shifted to be on the target surface (see Figure 3.15: "Contact Surface Adjustment With ICONT"(a)). Only a very small correction is suggested; otherwise, severe discontinuity may occur (see Figure (b)).

The difference between CNOF and ICONT is that the former shifts the entire contact surface with the distance value CNOF, the latter moves all initially open contact points which are inside of adjustment band ICONT onto the target surface.

Figure 3.15 Contact Surface Adjustment With ICONT

- Use real constants PMIN and PMAX to specify an initial allowable penetration range. When either PMAX or PMIN is specified, ANSYS brings the target surface into a state of initial contact at the beginning of the analysis (see Figure 3.16: "Contact Surface Adjustment (PMIN, PMAX)"). If the initial penetration is larger than PMAX, ANSYS adjusts the target surface to reduce penetration. If the initial penetration is smaller than PMIN (and within the pinball region), ANSYS adjusts the target surface to ensure initial contact. Initial adjustment for contact status is performed only in translational modes.

Such adjustment of initial contact status will be performed for a rigid target surface that has either prescribed loads or displacements. Similarly, a target surface that has no boundary conditions specified may also be adjusted for initial contact.

When all the target surface nodes have a prescribed value of zero, the initial adjustment using PMAX and PMIN will not be performed.

Note that ANSYS treats applicable degrees of freedom for target surface nodes independently. For example, if you specify the UX degree of freedom to be "zero," then no initial adjustment is possible along the X direction. However, the PMAX and PMIN options will still be activated in the Y and Z directions.

The initial status adjustment is an iterative process. ANSYS uses a maximum of 20 iterations. If the target surface cannot be brought into an acceptable penetration range (i.e., in the range of PMIN to PMAX), the analysis proceeds with the original geometry. ANSYS issues a warning message in such circumstances, and you may need to manually adjust your initial geometry.

Figure 3.17: "A Scenario in Which Initial Adjustment Will Fail" illustrates a problem in which initial contact adjustment iteration will fail. The UY degree of freedom for the target has been restrained. Therefore, the only possible adjustment for initial contact is in the X direction. However, in this problem, any movement of the rigid target surface in the X direction will not establish initial contact.

For flexible-to-flexible contact, this technique not only moves the entire target surface but also moves the whole deformable body which attaches to the target surface. Make sure there is no other contact surface or target surface connecting with the deformable body.

Figure 3.16 Contact Surface Adjustment (PMIN, PMAX)

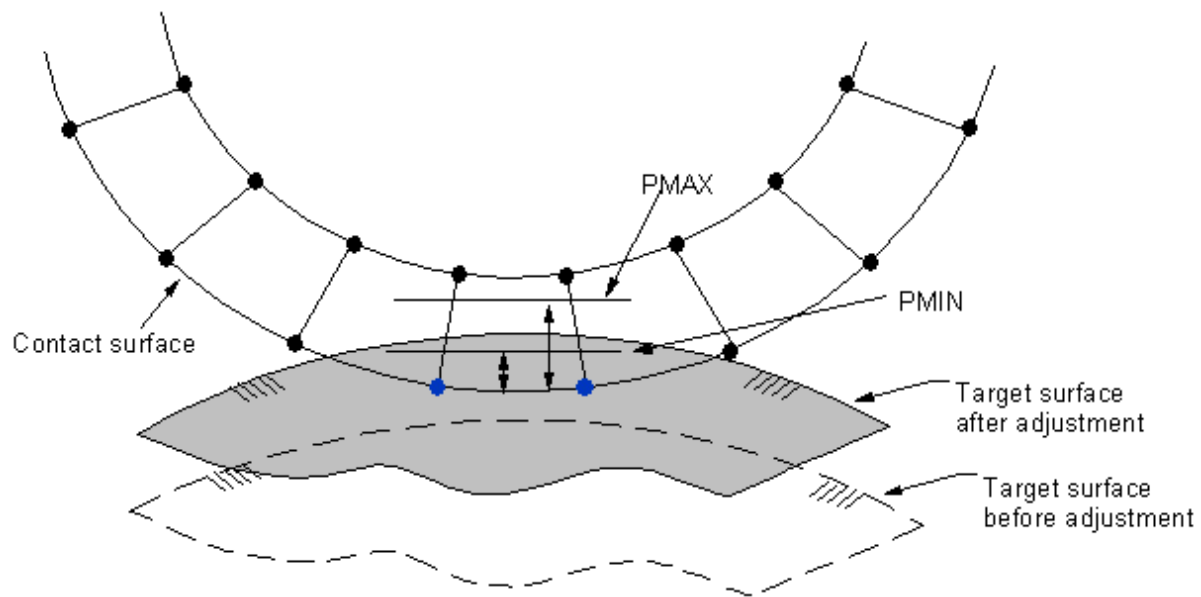
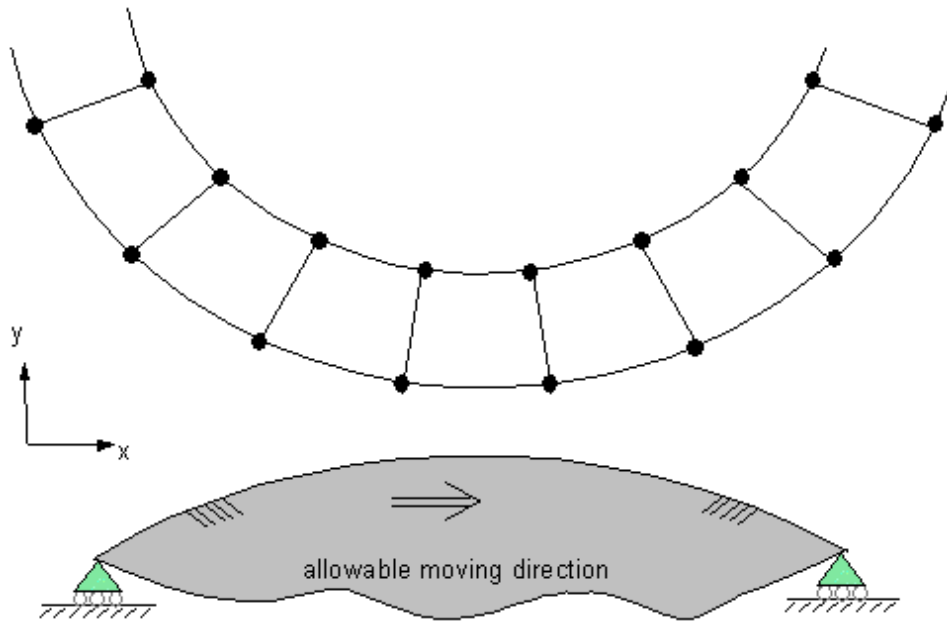


Figure 3.17 A Scenario in Which Initial Adjustment Will Fail

4. Set KEYOPT(9) to adjust initial penetration or gap; see Figure 3.18: "Ignoring Initial Penetration, KEYOPT(9) = 1".

True initial penetration includes two parts:

- Penetration or gap due to geometry
- Penetration or gap due to user-defined contact surface offset (CNOF).

See Figure 3.19: "Components of True Penetration".

KEYOPT(9) provides the following capabilities:

- To include initial penetration from both geometry and contact surface offset, set KEYOPT(9) = 0. This is the default.
- To ignore initial penetration from both effects, set KEYOPT(9) = 1. When KEYOPT(12) = 4 or 5, this setting for KEYOPT(9) will also ignore the initial force in open-gap springs, thus creating an initially "perfect" contacting surface having no initial forces acting across the contact interface.
- To include the defined contact surface offset (CNOF) but ignore the initial penetration due to geometry, set KEYOPT(9) = 3. When KEYOPT(12) = 4 or 5, this setting for KEYOPT(9) will also ignore the initial force in open-gap springs, thus creating an initially "perfect" contacting surface having no initial forces acting across the contact interface.

For problems such as an interference fit, over-penetration is expected. These problems often have convergence difficulties if the initial penetration is step-applied in the first load step. You may overcome convergence difficulties by ramping the initial penetration over the first load step, see Figure 3.20: "Ramping Initial Interference". The following KEYOPT(9) settings provide ramped capabilities:

- To ramp the total initial penetration (CNOF + the offset due to geometry), set KEYOPT(9) = 2.
- To ramp the defined contact surface penetration, but ignore the penetration due to geometry, set KEYOPT(9) = 4.

For both of the above KEYOPT(9) settings, you should also set **KBC,0** and not specify any external loads in the first load step. Also, be sure that the pinball region is big enough to capture the initial interference.

You can use the above techniques in conjunction with each other. For example, you may wish to set a very precise initial penetration or gap but the initial coordinates of the finite element nodes may not be able to provide sufficient precision. To accomplish this, you could:

1. Use **ICONT** to move the initial open contact points just onto the target surface.
2. Use **CNOF** to specify a penetration (positive value) or gap (negative value).
3. Use **KEYOPT(9) = 3** to resolve the initial penetration in the first substep (or **KEYOPT(9) = 4** to gradually resolve the initial penetration).

Figure 3.18 Ignoring Initial Penetration, KEYOPT(9) = 1

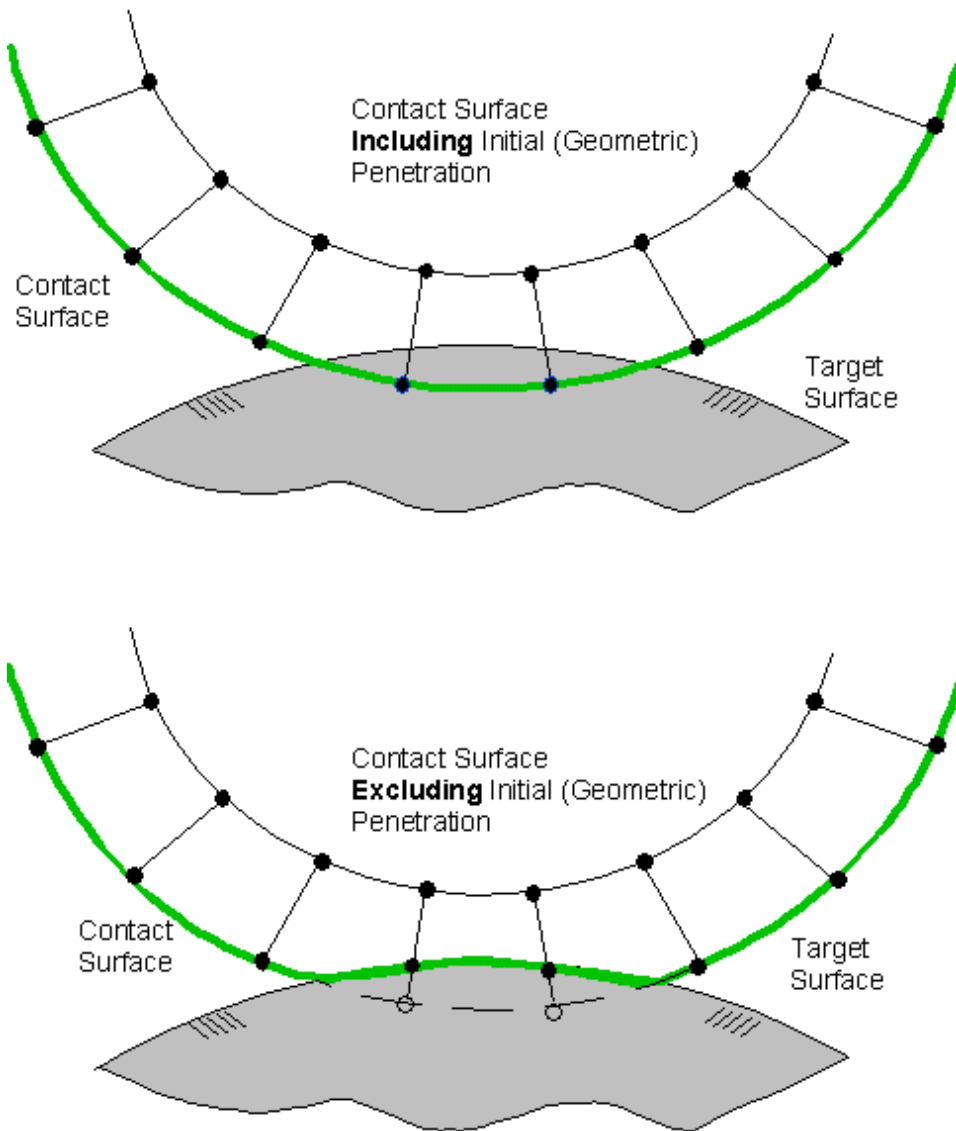


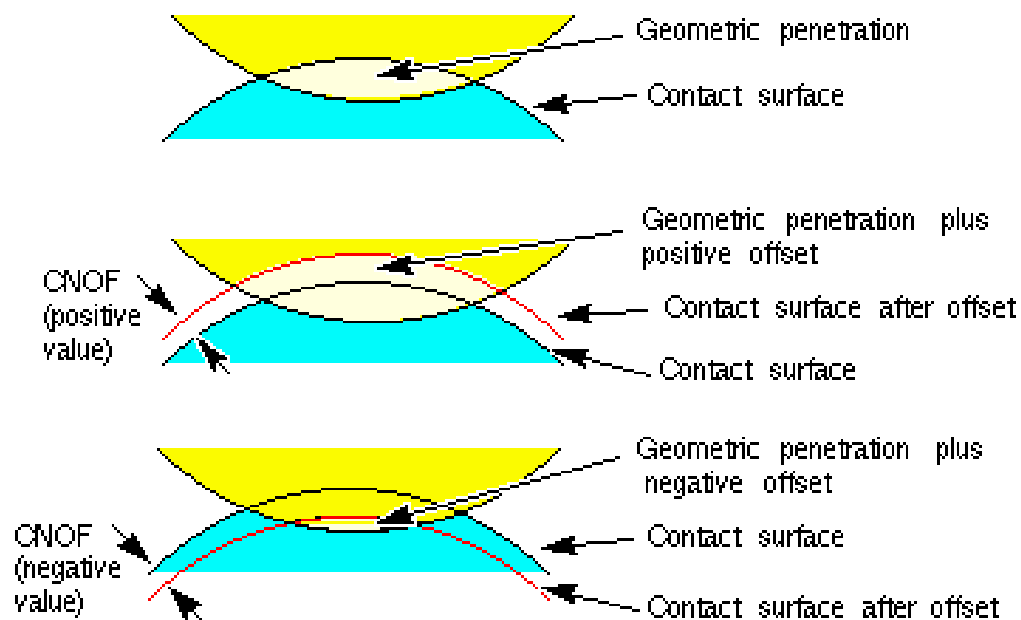
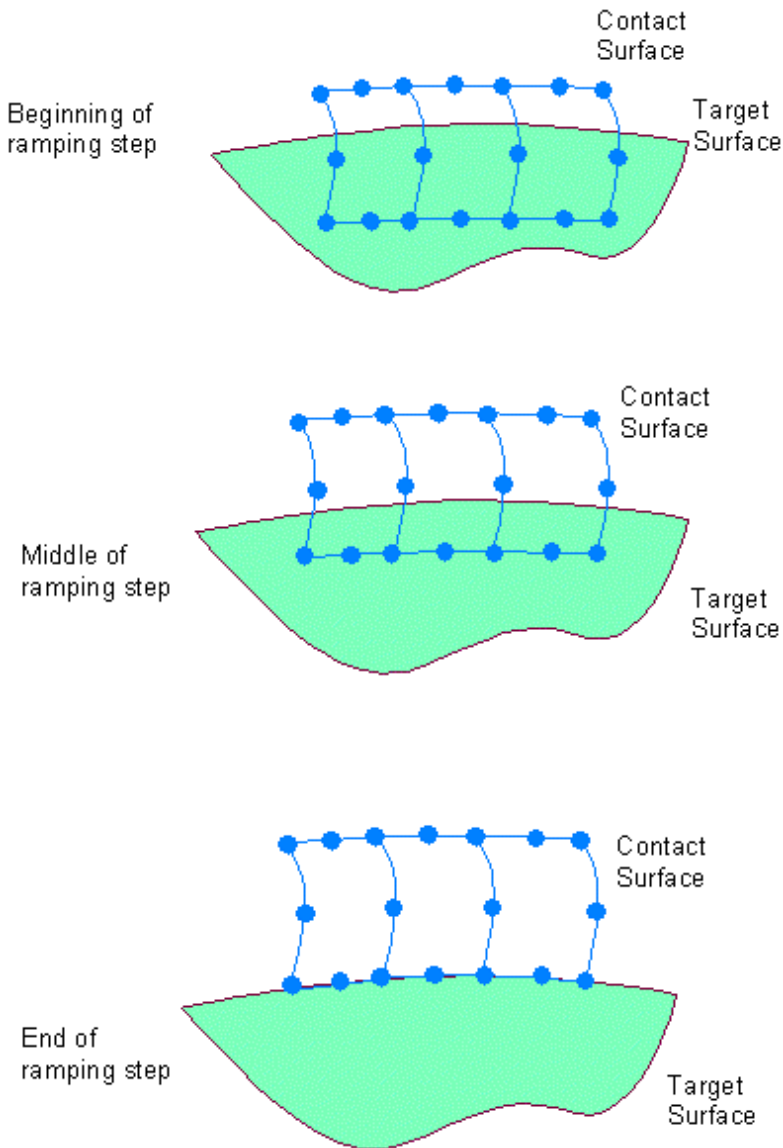
Figure 3.19 Components of True Penetration

Figure 3.20 Ramping Initial Interference

ANSYS provides a printout (in the output window or file or via the **CNCHECK**) of the model's initial contact state for each target surface at the beginning of the analysis. This information is helpful for determining the maximum penetration or minimum gap for each target surface.

If no contact is detected for a specific target surface, ANSYS issues a warning. This occurs when the target surface is far from contact (i.e., outside of the pinball region), or when the contact/target elements have been killed.

See Positive and Negative Real Constants for more information on these real constants.

3.8.8. Physically Moving Contact Nodes Towards the Target Surface

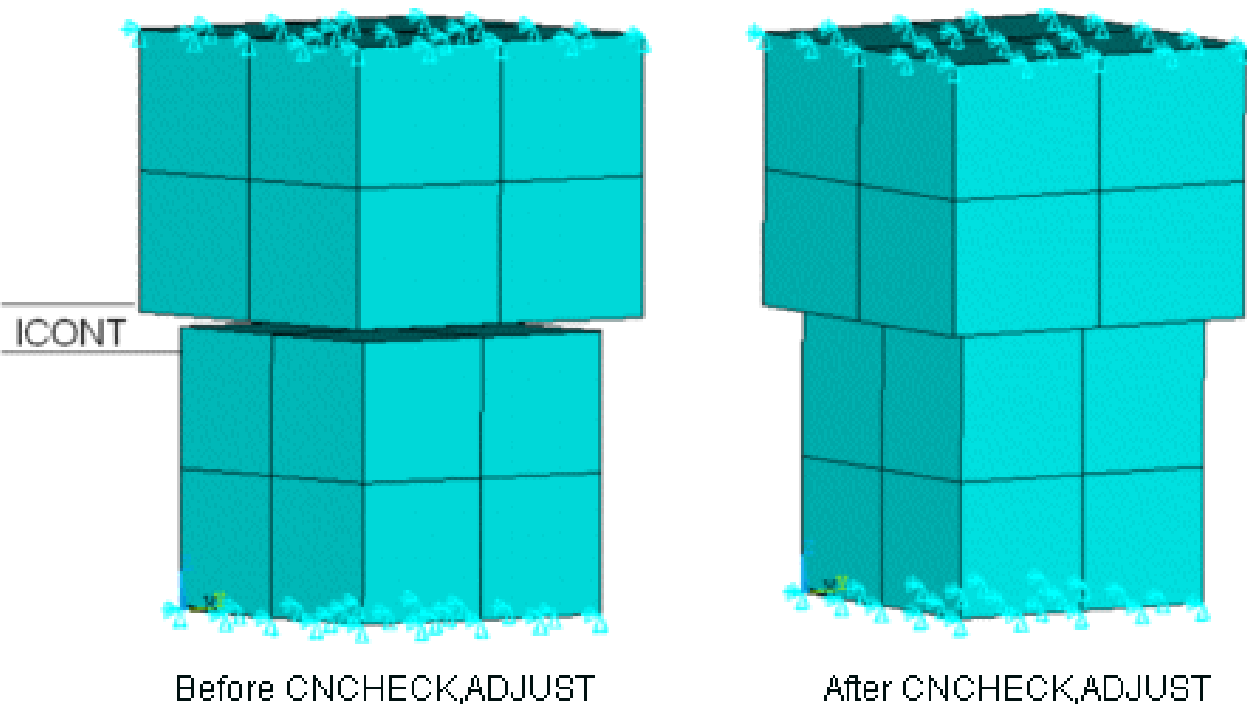
The initial contact status can be adjusted to close the gap by defining real constant **ICONT** or by ignoring the penetration (setting **KEYOPT(9) = 1**). However the initial contact adjustment is kept during the entire analysis as a rigid zone. The initial contact adjustment can cause a certain amount of residual force if a large rotation appears at the contact surface. This problem can be alleviated by issuing the **CNCHECK,ADJUST** command, which physically moves contact nodes towards the target surface under the following circumstances:

- Only when using the nodal detection option (KEYOPT(4) = 1 or 2), or when using CONTA175.
- Initially open contact nodes inside the ICONT zone.
- Initially penetrated nodes with KEYOPT(9) = 1.

After issuing the **CNCHECK,ADJUST** command, the coordinates of the nodes that have been moved are modified as shown in Figure 3.21: “Effect of Moving Contact Nodes”. You can change other contact related settings in PREP7 (for example, set KEYOPT(4) = 0 to use the Gauss detection option) and save the db file. Issuing the **SAVE** command before issuing the **CNCHECK,ADJUST** command is recommended in order to resume the .DB file with the original contact configuration.

For those contact pairs whose contact nodes you do not wish to physically move towards target surface, do not define KEYOPT(4) = 1 or 2.

Figure 3.21 Effect of Moving Contact Nodes



3.8.9. Determining Contact Status and the Pinball Region

3.8.9.1. Background

The position and motion of a contact element relative to its associated target surface determines the contact element status. ANSYS monitors each contact element and assigns a status:

- STAT = 0 Open far-field contact
- STAT = 1 Open near-field contact
- STAT = 2 Sliding contact
- STAT = 3 Sticking contact

A contact element is considered to be in near-field contact when its integration points (Gauss points or nodal points) are within a code-calculated (or user-defined) distance to the corresponding target surface. This distance

is referred to as the pinball region. The pinball region is a circle (in 2-D) or a sphere (in 3-D) centered about the Gauss point.

3.8.9.2. Using PINB

Use real constant PINB to specify a scaling factor (positive value for PINB when using command input) or absolute value (negative value for PINB when using command input) for the pinball region. You can specify PINB to have any value. By default, and assuming that large deflection effects apply (**NLGEOM,ON**), ANSYS defines the pinball region as a circle for 2-D or a sphere for 3-D of radius $4 \times \text{depth}$ (if rigid-to-flexible contact) or $2 \times \text{depth}$ (if flexible-to-flexible contact) of the underlying element. (See the discussion of element depth in Section 3.8.1.1: Positive and Negative Real Constant Values.) If you include no large-deflection effects (**NLGEOM,OFF**), the default pinball region is half that of the large-deflection case. (For the no-separation (**KEYOPT(12) = 4**) and bonded-always (**KEYOPT(12) = 5**) options, the PINB default is different than described here. See Section 3.8.11: Selecting Surface Interaction Models for more information.)

Note — If you input a value for real constant CNOF (contact surface offset) and the default PINB value (as described above) is less than the absolute value of CNOF, the default for PINB will be set to the absolute value of $(1.1 \times \text{CNOF})$.

The computational cost of searching for contact depends on the size of the pinball region. Far-field contact element calculations are simple and add little computational demands. The near-field calculations (for contact elements that are nearly or actually in contact) are slower and more complex. The most complex calculations occur once the elements are in actual contact.

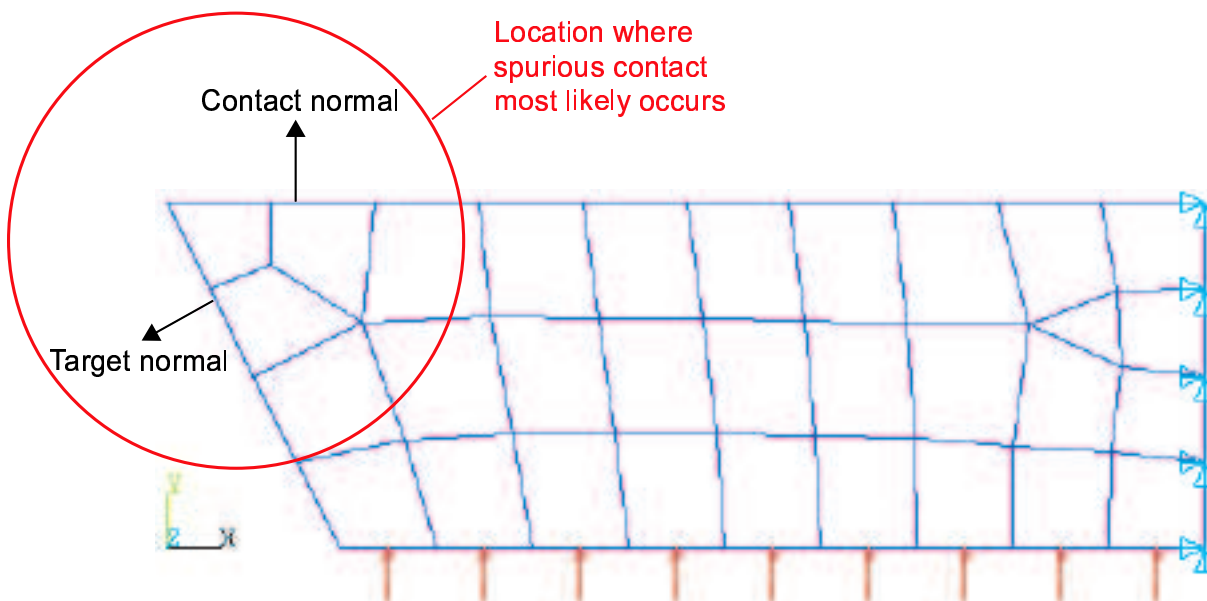
Setting a proper pinball region is useful to overcome spurious contact definitions if the target surface has several convex regions. However, the default setting should be appropriate for most contact problems.

See Positive and Negative Real Constants for more information on this real constant.

3.8.10. Avoiding Spurious Contact in Self Contact Problems

In some cases of self contact, ANSYS may erroneously assume contact between a contact and target surface that are in very close geometrical position as shown below.

Figure 3.22 Auto Spurious Prevention



ANSYS will alert you when it first detects spurious contact in each load step. If ANSYS encounters such contact on the first load step, you'll see the following error message:

```
Contact element x has too much penetration related to target element y. We
assume it (may be more elements) is spurious contact.
```

If ANSYS encounters an abrupt change in contact that it classifies as spurious contact, you'll see the following message:

```
Contact element x status changed abruptly with target element y. We assume
it (may be more elements) is spurious contact.
```

ANSYS issues such messages only once per load step. It does not notify you of additional cases of spurious contact that were ignored during the load step.

3.8.11. Selecting Surface Interaction Models

3.8.11.1. Background

The surface-to-surface contact elements support normal unilateral contact models as well as other mechanical surface interaction models.

3.8.11.2. Using KEYOPT(12) and FKOP

Use KEYOPT(12) to model different contact surface behaviors.

- KEYOPT(12) = 0 models standard unilateral contact; that is, normal pressure equals zero if separation occurs.
- KEYOPT(12) = 1 models perfectly rough frictional contact where there is no sliding. This case corresponds to an infinite friction coefficient and ignores the material property MU.
- KEYOPT(12) = 2 models no separation contact, in which the target and contact surfaces are tied (although sliding is permitted) for the remainder of the analysis once contact is established.
- KEYOPT(12) = 3 models "bonded" contact, in which the target and contact surfaces are bonded in all directions (once contact is established) for the remainder of the analysis.
- KEYOPT(12) = 4 models no separation contact, in which contact detection points that are either initially inside the pinball region or that once involve contact always attach to the target surface along the normal direction to the contact surface (sliding is permitted).
- KEYOPT(12) = 5 models bonded contact, in which contact detection points that are either initially inside the pinball region or that once involve contact always attach to the target surface along the normal and tangent directions to the contact surface (fully bonded).
- KEYOPT(12) = 6 models bonded contact, in which the contact detection points that are initially in a closed state will remain attached to the target surface and the contact detection points that are initially in an open state will remain open throughout the analysis.

For modeling either no-separation or bonded contact, you may need to set a value for the FKOP real constant. This provides a stiffness factor applied when contact opens. If FKOP is a scaling factor (positive value for command input), the true contact opening stiffness equals FKOP times the contact stiffness applied when contact closes. If FKOP is an absolute value (negative value for command input), the value is applied as an absolute contact opening stiffness. The default FKOP value is 1.

No separation and bonded contact generate a "pull-back" force when contact opening occurs, and that force may not completely prevent separation. To reduce separation, define a larger value for FKOP. Also, in some cases separation is expected while connection between the contacting surfaces is required to prevent rigid body

motion. In such instances, you can specify a small value for FKOP to maintain the connection between the contact surfaces (this is a "weak spring" effect).

For the no-separation option (KEYOPT(12) = 4) and the bonded-always option (KEYOPT(12) = 5), a relatively small PINB value (pinball region) may be used to prevent any false contact. For these KEYOPT(12) settings, the default for PINB is 0.25 (25% of the contact depth) for small deformation analysis (**NLGEOM,OFF**) and 0.5 (50% of the contact depth) for large deformation analysis (**NLGEOM,ON**). (The default PINB value may differ from what is described here if CNOF is input. See Section 3.8.9.2: Using PINB for more information.)

For the bonded-initial option (KEYOPT(12) = 6), a relatively large ICONT value (initial contact closure) may be used to capture the contact. For this KEYOPT(12) setting, the default for ICONT is 0.05 (5% of the contact depth) when KEYOPT(5) = 0 or 4.

See Positive and Negative Real Constants for more information on these real constants.

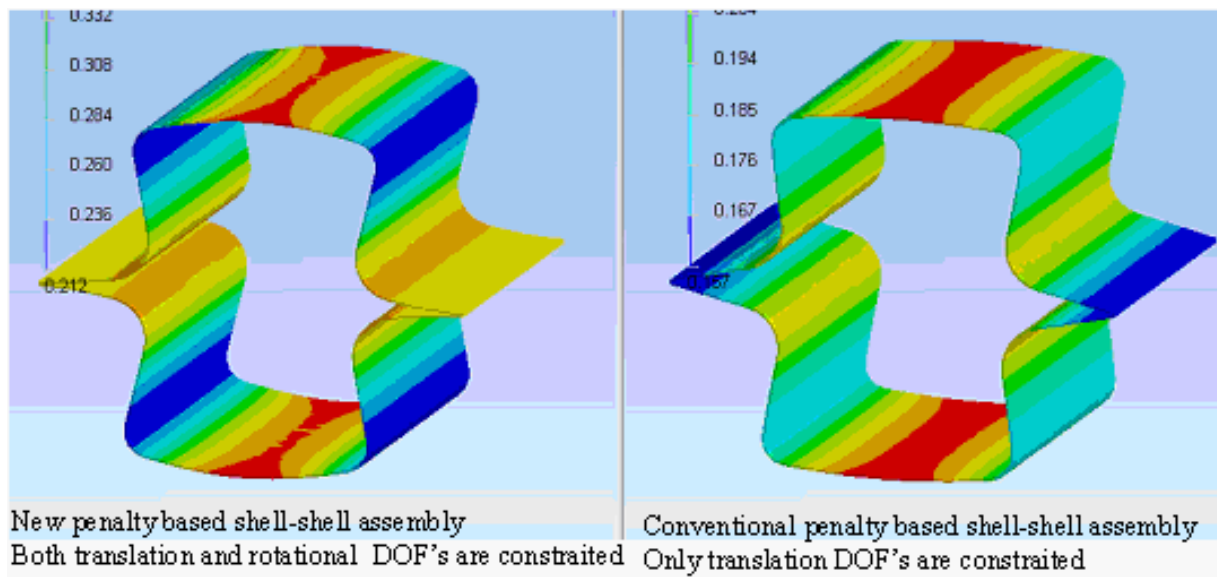
3.8.11.3. Bonded Contact for Shell-Shell Assemblies

The bonded contact options (KEYOPT(12) = 5 or 6) can be used with the MPC approach (KEYOPT(2) = 2) to model various types of assemblies (see Chapter 7, "Multipoint Constraints and Assemblies"). When this method is used to model shell-shell assemblies, there may be cases where the MPC approach causes the model to be overconstrained. To alleviate this problem, you can use a penalty based method for shell-shell assemblies. Using the penalty based method constrains rotational DOFs in addition to translational DOFs. This capability is available for contact elements CONTA173, CONTA174, and CONTA175 in conjunction with TARGE170.

To use this method, first set KEYOPT(2) = 0 or 1 (augmented Lagrangian or penalty function) and KEYOPT(12) = 5 or 6 (bonded always or bonded initial contact) in the contact elements. Setting KEYOPT(5) = 2 (shell-shell constraint) for the target elements will cause this penalty based method to be used.

The penalty stiffness used for rotational DOFs is equal to (contact stiffness used for translational DOFs) * (contact length). The contact stiffness for translational DOFs is input by real constant FKN, or defaults to an internal value. The contact length is always calculated internally and it is printed in the output file. The figure below shows the difference in using conventional penalty based shell-shell assembly and this method.

Figure 3.23 Penalty-Based Shell-Shell Assembly



3.8.12. Modeling Contact with Superelements

3.8.12.1. Background

The surface-to-surface contact elements can model a rigid body (or one linear elastic body) contacting another linear elastic body undergoing small motions. These elastic bodies can be modeled using superelements, which greatly reduces the number of degrees of freedom involved in the contact iteration. Remember that any contact or target nodes must be either all master nodes of the superelements or all slave nodes of the superelements. When the contact pair is built in original elements used to generate superelements, the contact status will not change from its initial status.

Because the superelement consists only of a group of retained nodal degrees of freedom, it has no surface geometry on which ANSYS can define a contact and target surface. Therefore, the contact and target surface must be defined on the surface of the original elements before they are assembled into a superelement. Information taken from the superelement includes nodal connection and assembly stiffness, but no material property or stress states (whether axisymmetric, plane stress, or plane strain). One restriction is that the material property set used for the contact elements must be the same as the one used for the original elements before they were assembled into superelements.

3.8.12.2. Using KEYOPT(3)

Use KEYOPT(3) to provide information for the 2-D analysis with superelements. In elements CONTA171 and CONTA172, the options are as follows:

- No superelement used (KEYOPT(3) = 0)
- Axisymmetric, use with superelements only (KEYOPT(3) = 1)
- Plane strain or plane stress with unit thickness, use with superelements only (KEYOPT(3) = 2)
- Plane stress with thickness input use with superelements only (KEYOPT(3) = 3). Note that for this case, use real constant R2 to specify the thickness.

In 3-D contact analysis, KEYOPT(3) in elements CONTA173 and CONTA174 is ignored. ANSYS will automatically detect whether the underlying element is a superelement.

Note — KEYOPT(3) in node-to surface contact element CONTA175 has a different meaning. In CONTA175, KEYOPT(3) = 1 defines the contact traction-based model. In this case, all of the real constant inputs and contact result quantities have the same units as the surface-to-surface contact elements. KEYOPT(3) = 0 (default) defines the contact force model. In this model, certain real constants and contact result quantities can have different units (a factor of AREA (Length²) difference). See Section 4.2.1.1: KEYOPT(3).

3.8.13. Accounting for Thickness Effect

3.8.13.1. Background

You can account for the thickness of shells (2-D and 3-D) and beams (2-D) using KEYOPT(11). For rigid-to-flexible contact, ANSYS will automatically shift the contact surface to the bottom or top of the shell/beam surface. For flexible-to-flexible contact, ANSYS will automatically shift both the contact and target surfaces which are attached to shell/beam elements. By default, ANSYS does not account for the element thickness, and beams and shells are discretized at their mid-surface in which penetration distance is calculated from the mid-surface.

3.8.13.2. Using KEYOPT(11)

When you set KEYOPT(11) = 1 to account for beam or shell thickness, the contact distance is calculated from either the top or the bottom surface as specified previously in Section 3.2: Steps in a Contact Analysis.

Note — Only use KEYOPT(11) = 1 to account for thickness when you have shell or beam elements with nodes located at the middle surface (for example, KEYOPT(11) = 0 for SHELL91).

When building your model geometry, if you are going to account for thickness, remember the offsets which may come from either the contact surface or target surface or from both. When you specify a contact offset (CNOF) along with setting KEYOPT(11) = 1, it is defined from the top or bottom of the shell/beam, not the mid-surface. When used with SHELL181, SHELL208, or SHELL209, changes in thickness during deformation are also taken into account.

3.8.14. Using Time Step Control

3.8.14.1. Background

Time step control is an automatic time stepping feature that predicts when the status of a contact element will change and cuts the current time step back.

3.8.14.2. Using KEYOPT(7)

Use KEYOPT(7) to take one of four actions to control time stepping, where KEYOPT(7) = 0 provides no control (the default), and KEYOPT(7) = 3 provides the most control.

- KEYOPT(7) = 0: No control. The time step size is unaffected by the prediction. This setting is appropriate for most analyses when automatic time stepping is activated and a small time step size is allowed.
- KEYOPT(7) = 1: Time step size is bisected if too much penetration occurs during an iteration, or if the contact status changes dramatically.
- KEYOPT(7) = 2: Predict a reasonable increment for the next substep.
- KEYOPT(7) = 3: Predict a minimal time increment for the next substep.

3.8.15. Using the Birth and Death Option

The surface-to-surface contact and target elements allow birth and death and also follow the birth and death status of their underlying elements. The elements can be removed for part of an analysis and then reactivated for a later stage. This feature is useful for modeling complex metal forming processes where multiple rigid target surfaces need to interact with the contact surface at different stages of the analysis. Springback modeling often requires removing the rigid tools at the end of the forming processes. This option cannot be used with "no separation" or bonded contact.

3.9. Controlling the Motion of the Rigid Target Surface (Rigid-to-Flexible Contact)

Rigid target surfaces are defined in their original configuration, and the motion of the entire surface is then defined by the imposed displacements on the pilot node (or the different nodes of the target surface if no pilot node was defined).

You must use a pilot node in any of the following situations to control the boundary conditions (and motion) of the entire target surface:

- The target surface is subjected to applied forces.
- The target surface is subjected to rotations.
- The target surface is connected to other elements (e.g., structural mass element MASS21).
- The motion of the target surface is adjusted by the equilibrium condition.

The degrees of freedom of the pilot node represent the motion of the entire rigid surface, including two translational and one rotational degree of freedom in 2-D, and three translational and three rotational degrees of freedom in 3-D. You can apply boundary conditions (displacement, initial velocity), concentrated loads, rotations, etc. to the pilot node. To account for a rigid body's mass, define a mass element (MASS21) on the pilot node.

Keep in mind the following restrictions on the target surface when using a pilot node:

- Each target surface can have only one pilot node.
- ANSYS ignores all boundary conditions on all nodes other than the pilot node.
- Only the pilot node can connect to other elements. If you need to attach the rigid surface to another element, you must use the pilot node to do so.
- You cannot use constraint equations (**CE**) or node coupling (**CP**) to control the degree of freedom of the target surface when a pilot node has been defined. If you want to apply any loads or constraints on the rigid target surface, you must define a pilot node and apply the loads to that pilot node. If you do not use a pilot node, you can have rigid body motions only.

Note — The pilot node can be one of the nodes on the target elements or a node at any arbitrary location. However, it should not be the node on the contact element. The location of the pilot node becomes important only when rotations or moments are to be applied. For each pilot node, ANSYS will automatically define an internal node and an internal constraint equation. The rotational DOF of the pilot node is connected to the translational DOF of the internal node by the internal constraint equation.

By default, KEYOPT(2) = 0 for the target element, ANSYS checks the boundary conditions for each target surface. If all of the following conditions are met, then ANSYS treats the target nodes along the respective degree of freedom as fixed:

- There are no explicit boundary conditions or prescribed forces for target surface nodes.
- Target surface nodes are not connected to other elements.
- Neither constraint equations nor node coupling have been used to constrain such nodes.

At the end of each load step, ANSYS releases the constraint conditions that were set internally.

The constraint conditions stored in the results file (**Jobname.RST**) and the database file (**Jobname.DB**) may be updated due to this change. You should carefully verify whether the current constraint conditions are expected before restarting an analysis or resolving the problem in interactive mode.

If you wish, you can control the constraint conditions of target nodes by setting KEYOPT(2) = 1 in the target element definition.

3.10. Applying Necessary Boundary Conditions to the Deformable Elements

You can now apply any necessary boundary conditions as you would for any ANSYS analysis. For more information on applying boundary conditions, see the appropriate analysis descriptions in the *ANSYS Structural Analysis Guide*.

3.11. Defining Solution and Load Step Options

Convergence behavior for contact problems depends strongly on the particular problem. The options listed below are either typical or recommended for most surface-to-surface contact analyses. Please see the *ANSYS Commands Reference* for further details.

The time step size must be small enough to capture the proper contact zone. The smooth transfer of contact forces is disrupted if the time step size is too large. The time step size is specified by a number of substeps or the time step size itself. The following commands adjust these parameters.

Command(s): NSUBST

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Freq and Substps

Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time and Substps

Main Menu> Solution> Analysis Type> Sol'n Control (: Basic Tab)

Main Menu> Solution> Unabridged Menu> Load Step Opts> Time/Frequenc> Freq and Substps

Main Menu> Solution> Unabridged Menu> Load Step Opts> Time/Frequenc> Time and Substps

Command(s): DELTIM

GUI: Main Menu> Preprocessor> Loads> Load Step Opts> Time/Frequenc> Time-Time Step

Main Menu> Solution> Analysis Type> Sol'n Control (: Basic Tab)

Main Menu> Solution> Unabridged Menu> Load Step Opts> Time/Frequenc> Time-Time Step

Note — A reliable way to set an accurate time step size is to turn automatic time stepping on.

The following options are automatically invoked, but may override them if needed.

Command(s): AUTOTS,ON

GUI: Main Menu> Solution> Analysis Type> Sol'n Control (: Basic Tab)

Main Menu> Solution> Unabridged Menu> Load Step Opts> Time/Frequenc> Time-Time Step or Time and Substps

If the contact status changes during the iteration process, discontinuity can occur. To avoid a slow convergence rate and use an updated stiffness matrix, set the Newton-Raphson option to FULL.

Command(s): NROPT,FULL,,OFF

GUI: Main Menu> Solution> Unabridged Menu> Analysis Type> Analysis Options

Also, do not use adaptive descent. Adaptive descent usually does not provide any help for surface-to-surface contact applications, and we recommend turning it off.

In cases where frictional sliding dominates, set the unsymmetric solver option (**NROPT, UNSYM,,OFF**) to avoid slow convergence or divergence.

Set the number of equilibrium equations to a number that is appropriate for a reasonable time step size. This command defaults to between 15 and 26 iterations, depending upon the physics of the problem.

Command(s): NEQIT

GUI: Main Menu> Solution> Analysis Type> Sol'n Control (: Nonlinear Tab)

Main Menu> Solution> Unabridged Menu> Load Step Opts> Nonlinear> Equilibrium Iter

Because the iterations tend to become unstable for large increments, use the line search option to stabilize the calculations.

Command(s): LNSRCH

GUI: Main Menu> Solution> Analysis Type> Sol'n Control (: Nonlinear Tab)

Main Menu> Solution> Unabridged Menu> Load Step Opts> Nonlinear> Line Search

Turn the predictor-corrector option on, except for large rotations or dynamic analyses.

Command(s): PRED

**GUI: Main Menu> Solution> Analysis Type> Sol'n Control (: Nonlinear Tab)
Main Menu> Solution> Unabridged Menu> Load Step Opts> Nonlinear> Predictor**

Many convergence failures in contact analyses are the result of using too large a value for contact stiffness (real constant FKN). Be sure to follow the recommendations given earlier in this chapter for estimating contact stiffness. If such estimated values lead to a convergence failure, reduce the contact stiffness and restart.

Conversely, if overpenetration occurs in your contact analysis, you probably need a larger value of FKN. In this case, gradually increase the contact stiffness value to an appropriate level by redefining it using **RMODIF** commands over several load steps in a restart.

Note — For most small strains, small displacements, and small sliding problems, set **NLGEOM**, OFF. This setting will speed up the searching time; however, if the contact problem involves large sliding, set **NLGEOM**, ON.

3.12. Solving the Problem

You can now solve the analysis the same as you would for any nonlinear analysis. Keep the following points in mind:

- Always check the real constant sets which are related to contact pairs and check the constraint conditions on the target surfaces. Any previous "trial runs" could have changed the settings.
- Always check the target surface contact status in the beginning of the analysis. If you detect any unexpected gap (or no contact) or overestimated penetration, abort the analysis and then check your geometric model. You can issue the **CNCHECK** command to verify the initial contact status.
- If your model is experiencing convergence difficulties due to contact, use the **NLHIST** command as a debugging tool to monitor contact information during the solution. Before starting the solution, issue **NLHIST** to specify the pair-based contact items (such as contact penetration or gap, contact normal stiffness, etc.) to be tracked. The resulting data are written to a file named **Jobname.nlh**.

NLDIAG,CONT can also be used to monitor contact information. When turned on, this feature will track the same contact information available through **NLHIST** for all defined contact pairs according to the specified writing frequency (each iteration, substep, or load step). The resulting output is stored in a text file called **Jobname.cnd**.

- Always check your results carefully using standard engineering guidelines.

See Section 2.3.5: Solve the Analysis in the *ANSYS Structural Analysis Guide*.

If you are restarting a contact analysis, follow the normal restart procedures as discussed in Section 3.10: Restarting an Analysis in the *ANSYS Basic Analysis Guide*. However, be aware that the constraint conditions of target surfaces may have been set internally. Verify the constraints carefully before restarting an analysis. Only the real constants FKN, FTOLN, PINB, and FKOP can be changed, and they can only be changed at the point of restart or at the beginning of a new load step.

Note — Singleframe restart does not support surface-to-surface or node-to-surface contact. Use multiframe restart if your model contains any of the following contact elements: CONTA171, CONTA172, CONTA173, CONTA174, CONTA175.

3.13. Reviewing the Results

Results from a contact analysis consist mainly of displacements, stresses, strains, reaction forces, and the contact information (e.g., contact pressure, sliding, etc.). You can review these results in POST1, the general postprocessor,

or in POST26, the time-history postprocessor. For contact-related results, you can select CONT as a plotting or list item. While in POST1, you can also review the results from within the Contact Manager (via the Contact Manager icon in the ANSYS Standard Toolbar). See the "Output Data" section of the element descriptions (the *ANSYS Elements Reference*) for the available output components.

Remember that in POST1, only one substep can be read in at a time, and that the results from that substep should have been written to **Jobname.RST** (or **Jobname.RCN** for the initial contact configuration calculated by **CNCHECK,POST**). (The load step option command **OUTRES** controls which substep results are stored on **Jobname.RST**.) A typical POST1 postprocessing sequence is described below.

3.13.1. Points to Remember

See Section 2.3.6.2: Points to Remember in the *ANSYS Structural Analysis Guide*.

3.13.2. Reviewing Results in POST1

The steps for reviewing results in POST1 are the same as those for a typical nonlinear analysis (see Section 8.5.6.2: Reviewing Results in POST1 in the *ANSYS Structural Analysis Guide*) with the following exceptions:

For step 4, the following shows the various CONT options for the **PLNSOL** and **PLESOL** commands.

CONT	STAT[1]	Contact status:
		3-closed and sticking
		2-closed and sliding
		1-open but near contact
		0-open and not near contact
"	PENE	Contact penetration
"	PRES	Contact pressure
"	SFRIC	Contact friction stress
"	STOT	Contact total stress (pressure plus friction)
"	SLIDE	Contact sliding distance
"	GAP	Contact gap distance
"	FLUX	Heat flux at contact surface
"	CNOS	Total number of contact status changes during substep

1. Status notation is for all contact elements except CONTACT12 and CONTACT52. If you are using these elements, check the output listing in the corresponding element documentation.

Note — You can set these options through the **Main Menu > General Postproc > Plot Results > Contour Plot > Nodal Solu** or **Element Solu** menu item. Choose Contact for Item, Comp and you'll see a list of the options detailed above.

For rigid-to-flexible contact or asymmetric flexible-to-flexible contact, the contact element provides the true pressure and friction stress. However, for symmetric flexible-to-flexible contact, the true pressure and friction stress is the average of the pressures and friction stresses from both sides of the contact elements.

Note — For the contact force-based model ($KEYOPT(3) = 0$) in CONTACT175, PRES, SFRIC, and STOT are the contact normal force, contact friction force, and total contact force, respectively.

Note that the contact-specific information (CONT) plots as follows. For 2-D contact analyses, the model will plot in gray and the requested item will be contoured as an area (trapezoid) along the edge of the model where the

contact elements are located. Use the **FACT** item to scale 2-D contour size. For 3-D contact analyses, the model will plot in gray and the requested item will be contoured as a 2-D surface where the contact elements overlay the model.

For tabular listings, you may also list contact-specific information by using the **CONT** label and its arguments with the **PRNSOL** or **PRESOL** commands or their related menu items

You can also animate contact results over time. See the **ANTIME** command for details.

Command(s): ANTIME

GUI: Utility Menu > PlotCtrls > Animate > Over Time

If you used **CNCHECK,POST** to evaluate the initial contact state, you can view the contact results items for the initial contact configuration as you would for any other load step. To do so, you must explicitly read the results of the first load step from the results file **Jobname.RCN** by issuing the **FILE** and **SET,FIRST** commands before postprocessing. Otherwise, the result file may be read improperly.

3.13.3. Reviewing Results in POST26

The steps for reviewing results in POST26 are the same as those for a typical nonlinear analysis See Section 8.5.6.3: Reviewing Results in POST26 in the *ANSYS Structural Analysis Guide*.

Chapter 4: Node-to-Surface Contact

You can use the node-to-surface contact element CONTA175 to model flexible-flexible or rigid-flexible contact between a surface and a node. Additionally, you can use these elements to represent contact between two surfaces by specifying one surface as a group of nodes.

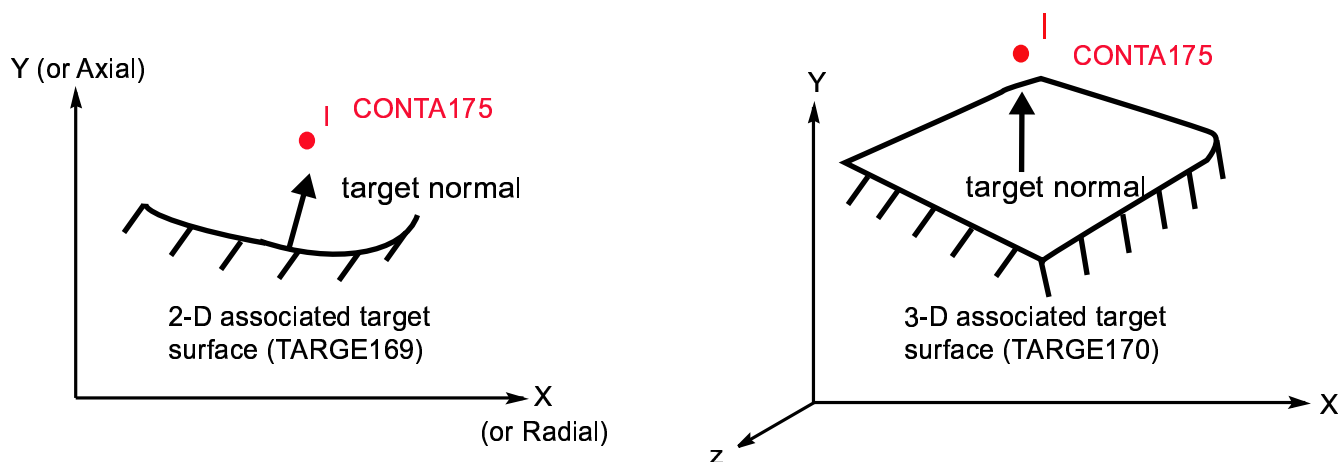
Node-to-surface contact is a phenomenon that occurs in most engineering applications: fasteners (nuts, bolts, rivets, pins), metal forming, rolling operations, dynamic pipe whip, etc. Engineers are interested in the stresses, deflections, forces, and temperature changes that occur due to contact between structural parts.

4.1. The Node-to-Surface Contact Element

Node-to-surface contact is represented in the ANSYS program by following the positions of points on one surface (the *contact* surface, modeled by CONTA175) relative to lines or areas of another surface (the *target* surface, modeled by TARGE169 or TARGE170).

CONTA175, in two or three dimensions, is depicted in Figure 4.1: “Node-to-Surface Contact Elements”.

Figure 4.1 Node-to-Surface Contact Elements



Presented below are characteristics of CONTA175:

- Has one node and its target surface is defined by TARGE169 or TARGE170. CONTA175 uses the same real constant set as the surface-to-surface contact elements. See Real Constants for more information.
- Supports 2-D/3-D rigid-flexible and flexible-flexible contact.
- Does not support 3-D contact surfaces with midside nodes, but can support 2-D/3-D target surfaces with midside nodes or 2-D contact surfaces with midside nodes.
- Generates elements using the **ESURF** command, as do the surface-to-surface contact elements.
- Supports structural, thermal, electric, and magnetic analyses.

Note — CONTA175 is recommended for point-to-surface or edge-to-surface problems. It can also be used to supplement a surface-to-surface pair at strategic locations where edge contact exists. For general surface-to-surface contact, CONTA171 through CONTA174 are recommended.

4.2. Performing a Node-to-Surface Contact Analysis

CONTA175 follows the contact pair concept used by surface-to-surface elements CONTA171 through CONTA174. CONTA175 is paired off with target elements TARGE169 and TARGE170. See Identifying Contact Pairs for more information. CONTA175 uses most of the same element KEYOPTS and real constants as the surface-to-surface contact elements. These are described below.

You should avoid midside-noded underlying elements of the contact surface, especially in 3-D. The “effective stiffness” at the contact surface nodes is very nonuniform. For instance, for the 20-node bricks SOLID95 or HYPER86, the corner nodes have a negative stiffness associated with them. However, the node-to-surface contact algorithm assumes that the stiffness is uniformly distributed across all the surface nodes when contact is made. This condition can lead to convergence difficulties when using midside-noded elements in contact. The midside-noded elements can only be used when bonded or no-separation contact is defined. You can still use midside nodes on 2-D contact surfaces or on 2-D/3-D target surfaces.

The basic steps for performing a node-to-surface contact analysis using CONTA175 are the same as those used for a typical surface-to-surface analysis using CONTA171 through CONTA174. See Steps in a Contact Analysis for details.

The Contact Manager provides an easy-to-use interface to help you construct and manage contact definitions.

You can access the manager via the Contact Manager icon  in the ANSYS Standard Toolbar, or via the menu path **Main Menu > Preprocessor > Modeling > Create > Contact Pair**. See Chapter 2, “GUI Aids for Contact Analyses” for more information on using the Contact Manager.

As mentioned, CONTA175 uses the **ESURF** command to generate elements between corresponding contact pairs, similar to the surface-to-surface contact elements. The GUI path is:

Main Menu > Preprocessor > Modeling > Create > Elements > Surf/Contact > Node to Surf

Since CONTA175 is a one node element, you are not able to plot the contact results. However, you can list the results using the **PLESOL**, **CONT** or **PRETAB** commands.

4.2.1. CONTA175 KEYOPTS

CONTA175 uses most of the same KEYOPTS that are used by the surface-to-surface contact elements CONTA171 through CONTA174. KEYOPT(3) and KEYOPT(4) are used but have different meanings when used with CONTA175. See Section 3.8.2: Element KEYOPTS for a listing of the remaining KEYOPTS.

4.2.1.1. KEYOPT(3)

KEYOPT(3) in CONTA175 allows you to choose between a contact force-based model (KEYOPT(3) = 0, default), and a contact traction-based model (KEYOPT(3) = 1). For the contact traction-based model, ANSYS can determine the area associated with the contact node. For the single point contact case, a unit area will be used which is equivalent to the contact force-based model.

When the traction-based model is defined, the real constants FKN, FKT, TCC, ECC, and MCC have the same units used in surface-to-surface contact elements (CONTA171 through CONTA174), as do postprocessing items PRES, TAUR, and TAUS.

When the force-based model is defined, the units of these quantities have a factor of AREA with respect to those used in the traction-based model. For instance, contact stiffness FKN has units FORCE/LENGTH for the force-

based model, but $\text{FORCE}/\text{LENGTH}^3$ for the traction-based model. PRES is the contact normal force in the force-based model, but contact pressure in the traction-based model.

4.2.1.2. KEYOPT(4)

KEYOPT(4) in CONTA175 allows you to choose the contact normal direction. The contact normal can be either perpendicular to the target surface (KEYOPT(4) = 0, default or KEYOPT(4) = 3), or perpendicular to the contact surface (KEYOPT(4) = 1, 2). When contact occurs on the bottom surface of a shell or beam, and shell thickness effect is included (KEYOPT(11) = 1), or CNOF is defined, KEYOPT(4) = 2 or 3 should be used in order to capture the contact.

Real constant TOLS is used to add a small tolerance that will internally extend the edge of the target surface. TOLS is useful for problems where contact nodes are likely to lie on the edge of the target (as at symmetry planes or for models generated in a node-to-node contact pattern). In these situations, the contact node may repeatedly “slip” off the target surface and so completely out of contact, resulting in convergence difficulties from oscillations.

Units for TOLS are percent (1.0 implies a 1.0% increase in the target edge length). A small value of TOLS will usually prevent this situation from occurring. The default value is 10 for small deflection and 2 for large deflection (NLGEOM,ON).

4.2.2. CONTA175 Real Constants

CONTA175 uses the same real constants used by the surface-to-surface contact elements CONTA171 through CONTA174), except the units of TCC, ECC, and MCC for the contact force-based model. See a listing of the real constants in Section 3.8.1: Real Constants.

4.3. Using CONTA175 for Multiphysics Contact

You can use node-to-surface contact element CONTA175 to model thermal contact, electric contact, and magnetic contact as you would use the surface-to-surface contact elements. For multiphysics contact, we recommend that you use the contact traction-based model (KEYOPT(3) = 1), which allows you to use real constants TCC, ECC, and MCC consistently with the surface-to-surface contact elements.

If you use the contact force-based model (KEYOPT(3) = 0), you should adjust the values of TCC, ECC, and MCC up or down as you make the element mesh coarser or finer.

Chapter 5: Multiphysics Contact

Multiphysics applications (involving two or more fields) often require the inclusion of contact interaction between two or more surfaces or between a surface and its surroundings. These contact interactions can be modeled by using the thermal, electric, and magnetic capabilities of the surface-to-surface and node-to-surface contact elements (CONTA171, CONTA172, CONTA173, CONTA174, and CONTA175).

5.1. Modeling Thermal Contact

You can use surface-to-surface contact elements and the node-to-surface contact element, in combination with thermal-structural coupled field solid elements or thermal elements, to model heat transfer that occurs in the contact surface. To activate both the structural and thermal DOFs, set KEYOPT(1) = 1. To activate the thermal DOF only, set KEYOPT(1) = 2.

The following thermal contact features are supported.

- Thermal contact conduction between two contacting surfaces.
- Heat convection from a “free surface” to the environment or between two open surfaces separated by small gap (“near field” convection).
- Heat radiation from a “free surface” to the environment or between two open surfaces separated by a small gap (“near field” radiation).
- Heat generation due to frictional dissipation.
- Heat flux input.

Note — When KEYOPT(1) is set to 2, ANSYS ignores heat generation due to friction.

5.1.1. Thermal Contact Behavior vs. Contact Status

Each contact pair can cover one or more thermal contact features. Which feature is active depends on the contact status:

Closed Contact: Thermal contact conduction transfers heat between two contacting surfaces.

Frictional Sliding: Frictional dissipated energy generates the heat to both the contact and target surfaces.

Near-Field Contact: Both heat convection and radiation between the contact and target surfaces are taken into account. The external flux value contributes to the contact surface.

Free-Surface Contact: Heat convection and radiation between the contact surface and the environment are taken into account. The external flux value only contributes to both contact and target surfaces.

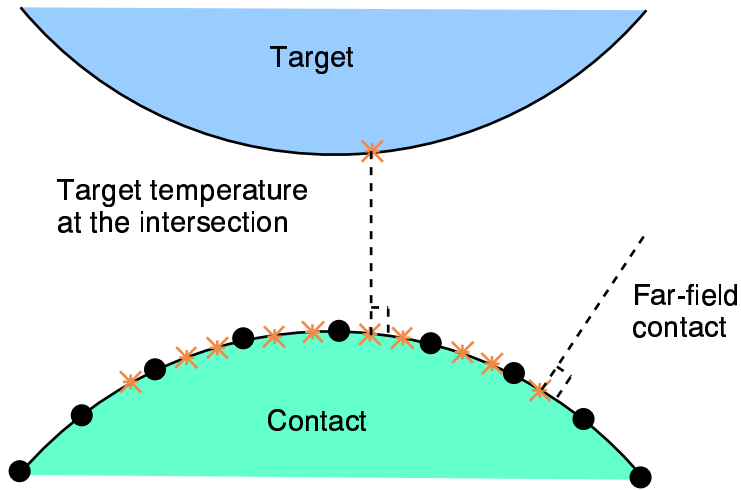
5.1.2. Free Thermal Surface

If you wish to model free surface convection, free surface radiation, or a surface with a supplied heat flux value, you can define a “free” thermal surface. A free thermal surface can be a contact surface with no associated target (that is, the contact pair lacks target elements). You can also set KEYOPT(3) = 1 of the target element type definition to define a free thermal surface. When this KEYOPT is set, both free surface radiation and convection are considered as long as open contact is detected. In this case, there is no convective and radiative heat transfer between the contact and target surfaces.

5.1.3. Temperature on Target Surface

For interface heat conduction, near field convection, or near field radiation, a temperature for both the contact and target surfaces is required. The temperature at the intersection between the target surface and the normal from the contact detection point represents the target temperature. The temperature on the pilot node represents the entire rigid target surface temperature if the pilot node exists.

Figure 5.1 Target Temperature



5.1.4. Modeling Conduction

To take into account the conductive heat transfer between contact and target surfaces, you need to specify the thermal contact conductance coefficient TCC through a real constant table.

5.1.4.1. Using TCC

The conductive heat transfer between two contacting surfaces is defined by

$$q = TCC \times (T_t - T_c)$$

where:

q: is the heat flux per area.

TCC: is the thermal contact conductance coefficient, having units of HEAT/(TIME * TEMPERATURE * AREA) for force-based node-to-surface contact, or units of HEAT/(TIME * TEMPERATURE) for the traction-based model.

T_t and T_c: are the temperatures of the contact points on the target and contact surfaces.

The TCC value is input through a real constant, which can be made a function of temperature $[(T_c + T_t)/2]$, pressure, time, and location by using the %TABLE% option. TCC has units of heat/(time x area x temp). If contact occurs, a small value of TCC yields a measure amount of imperfect contact and a temperature discontinuity across the interface. For large values of TCC, the resulting temperature discontinuity tends to vanish and perfect thermal contact is approached. When not in contact, however, it is assumed that no heat is transferred across the interface. To model contact conduction between two surfaces where a small gap exists, use KEYOPT(12) = 4 or 5 to define either the "bonded contact" or "no-separation contact" options (see Selecting Surface Interaction Models).

5.1.4.2. Using the Quasi Solver Option

You can take advantage of the fast thermal transient solver option (**THOPT,QUASI**) in the contact analysis. (See Nonlinear Options in the *ANSYS Thermal Analysis Guide* for more information on this solver option.) To do so, you must use the following contact element key options:

```
KEYOPT(1) = 2 - Temperature DOF only
KEYOPT(12) = 5 or 6 - Bonded always or bonded initial
```

The following solver options must also be set:

```
ANTYPE,TRANS
THOPT,QUASI
EQSLV,JCG/ICCG
```

The following two cases are supported:

- Thermal conductivity at contact. The only real constant used is TCC, which can be a function of time and temperature.
- Perfect thermal contact which supports dissimilar meshes on both sides of the contacting interface (TCC = infinity). This case requires the internal MPC approach (KEYOPT(2) = 2) and contact nodal detections (KEYOPT(4) = 1 or 2) or CONTA175.

5.1.5. Modeling Convection

To model convective heat transfer, you must specify the heat convection coefficient CONV using the **SFE** command. CONV can be a constant value (only uniform is allowed) or a function of temperature, time, and location as specified through tabular input. For free surface convection, you must specify bulk temperature through the **SFE** command. You can access this command through the following GUI paths:

```
Main Menu> Preprocessor> Loads> Define Loads> Apply> Thermal> Convection> On Elements> Uniform
Main Menu> Solution> Define Loads> Apply> Thermal> Convection> On Elements> Uniform
```

5.1.6. Modeling Radiation

5.1.6.1. Background

To model radiative heat transfer, you need to specify one of the following:

- Emissivity value EMIS, specified through the material property definition.
- Stefan-Boltzmann constant SBCT through a real constant.
- Offset temperature TOFFST. If you define your data in terms of degrees Fahrenheit or degrees Celsius, you must specify a temperature offset using the **TOFFST** command. You can access this command through the following GUI paths:

```
Main Menu> Preprocessor> Loads> Analysis Type> Analysis Options
Main Menu> Preprocessor> Material Props
```

- Radiation view factor RDVF, specified through a real constant.
- Environment (ambient) temperature. It is only used for free radiation and input on the **SFE** command with KVAL = 2 and CONV as a table parameter (this is the same as the bulk temperature in free surface convection modeling).

5.1.6.2. Using SBCT and RDVF

When contact is open, heat transfer through radiation can occur. The equation is defined by

$$q = \text{RDVF} \times \text{EMIS} \times \text{SBCT} [(T_e + \text{TOFFST})^4 - (T_c + \text{TOFFST})^4]$$

Where

TOFFST: The temperature offset from absolute zero to zero (defined through the **TOFFST** command.)

EMIS: The surface emissivity (input as material property).

SBCT: The Stefan-Boltzmann constant (input as a real constant). There is no default for SBCT, and if this is not defined the radiation effect is excluded.

RDVF: The radiation view factor input as a real constant (defaults to 1). RDVF can be defined as a function of temperature, gap distance, time, and location by using the %TABLE% option. It is only used for near field radiation. For far field radiation, RDVF is set to 1.0 and a user-assigned value is ignored. Other free-surface conditions recognize user-specified RDVF.

For “near field” radiation, when an intersection from a contact detection point to the target surface (in the direction of normal to the contact point) is detected, and the gap distance is smaller than the pinball radius, T_e is the target temperature at the intersection. The radiation modeling here assumes that the radiative heat transfer occurs in the direction of the normal between two surfaces with a small gap. By defining RDVF as a function of gap, you can account for geometry effects. Use the Radiosity Solver method for more generalized radiation problems (see the *ANSYS Thermal Analysis Guide* for more information).

For “free surface” radiation, T_e becomes the “ambient” temperature defined by “bulk temperature” input from the **SFE** command (using $\text{KVAL} = 2$ and **CONV** as the table).

5.1.7. Modeling Heat Generation Due to Friction

5.1.7.1. Background

In order to model heat generation due to frictional dissipated energy, you should perform a coupled transient thermal-structural analysis. If you wish you can turn off transient effects on structural DOFs by using **TIMINT,STRUC,OFF**. However, you must include transient effects on the thermal DOF. Two real constants are required:

- FHTG is the frictional dissipated energy converted into heat.
- FWGT is the weight factor for the distribution of heat between contact and target surfaces.

5.1.7.2. Using FHTG and FWGT

In the coupled thermal-structural contact modeling, the rate of frictional dissipation is given by

$$q = \text{FHTG} \times \tau \times V$$

Where

τ : The equivalent frictional stress.

V: The sliding rate.

FHTG: The fraction of frictional dissipated energy converted into heat. This value defaults to 1 and can be input as a real constant. For an input of true 0, you must enter a very small value (for example, 1E-8). If you enter 0, ANSYS interprets this as an input of the default value.

The amount of frictional dissipation on contact and target surfaces is defined by

$$q_c = \text{FWGT} \times \text{FHTG} \times \tau \times V$$

and

$$q_T = (1 - \text{FWGT}) \times \text{FHTG} \times \tau \times V$$

Where q_c is the contact side and q_T is the target side, and FWGT is the weight factor for the distribution of heat between the contact and target surfaces (input as a real constant). By default, FWGT = 0.5. For an input of true 0, you must enter a very small value (for example, 1E-8). If you enter 0, ANSYS interprets this as an input of the default value.

5.1.8. Modeling External Heat Flux

You can apply heat flux on the contact elements through the **SFE** command. Only uniform flux can be applied. Heat flux cannot be applied on target elements. However, for near field contact, the external flux is applied on contact and will contribute to target elements.

For a free thermal surface, if KEYOPT(3) of the target element is set to 1, the external flux is only applied on the contact side. On a given contact element either CONV or HFLUX (but not both) may be specified. However, you can define two different contact pairs: one models convection and the other models heat flux.

5.2. Modeling Electric Contact

You can use surface-to-surface contact elements or the node-to-surface contact element, in combination with thermal-electric elements and solid coupled field elements to model electric current conduction. You can also use surface-to-surface contact elements, in combination with piezoelectric and electrostatic elements, to model electric charge across a contacting interface.

KEYOPT(1) provides degree of freedom options for modeling electric contact. For combined structural/thermal/electric contact, set KEYOPT(1) = 3 to activate the structural, thermal, and electric current DOFs. For pure thermal/electric contact, set KEYOPT(1) = 4 to activate the thermal and electric current DOFs. For piezoelectric contact, set KEYOPT(1) = 5 to activate the structural and piezoelectric DOFs. For electrostatic contact, set KEYOPT(1) = 6 to activate the electrostatic DOF.

The electric contact features are:

- Electric conduction between two contacting surfaces.
- Heat generation due to electric dissipation.
- Electric charge across the contacting interface.

5.2.1. Modeling Surface Interaction

5.2.1.1. Background

To take into account the surface interaction for electric contact, you need to specify the electric contact *conductance* per unit area if you are using the electric current degree of freedom, or the electric contact *capacitance* per unit area if you are using the piezoelectric or electrostatic degrees of freedom. For either case, this parameter is ECC. You specify ECC through a real constant table. You can use a tabular input to define ECC as a function of contact pressure (pressure as a table), average temperature on contact detection point (temperature as a table), and time. If the “bonded contact” or “no-separation contact” option is set, contact interaction can occur between two surfaces separated by a narrow gap.

5.2.1.2. Using ECC

The interaction between two contacting surfaces is defined by

$$J = \text{ECC} \times (V_t - V_c)$$

where:

J = current density for the electric potential (VOLT) degree of freedom (KEYOPT(1) = 3 or 4), or the electric charge density (KEYOPT(1) = 5, or 6).

ECC = electric contact conductance for the electric potential (VOLT) degree of freedom (KEYOPT(1) = 3 or 4), or the electric contact capacitance per unit area for (KEYOPT(1) = 5, or 6).

V_t and V_c = voltages at the contact points on the target and contact surfaces.

The ECC value is input through a real constant, which can be a function of temperature $[(T_t + T_c)/2]$, pressure, and time, by using the %TABLE% option. For the current conduction option, the electric contact conductance ECC has units of electric conductivity/length. For the piezoelectric and electrostatic options, the electric contact capacitance ECC has units of capacitance per unit area. To model surface interaction between two surfaces where a small gap exists, use KEYOPT(12) = 4 or 5 to define either the “bonded contact” or “no-separation contact” options (see Selecting Surface Interaction Models).

Note — For force-based node-to-surface contact, ECC has units of (electric conductivity)(LENGTH) or the capacitance.

5.2.2. Modeling Heat Generation Due to Electric Current

For electric current field analyses (KEYOPT(1) = 3 or 4), the heat generation due to electric current is given by

$$q = \text{FHEG} \times J \times (V_t - V_c)$$

Where

FHEG: The fraction of electric dissipated energy converted into heat (Joule heating). This value defaults to 1 and can be input as a real constant. For an input of true 0, you must enter a very small value (for example, 1E-8). If you enter 0, ANSYS interprets this as an input of the default value.

J: The current density.

V_t and V_c : The voltages at the contact points on the target and contact surfaces.

The amount of electric heat dissipation on contact and target surfaces is defined by

$$q_c = \text{FWGT} \times q$$

and

$$q_T = (1 - \text{FWGT}) \times q$$

Where q_c is the contact side and q_T is the target side, and FWGT is the weight factor for the contact heat dissipation between the contact and target surfaces (input as a real constant). FWGT is the same real constant used for frictional heat generation. By default, FWGT = 0.5. For an input of true 0, you must enter a very small value (for example, 1E-8). If you enter 0, ANSYS interprets this as an input of the default value.

5.3. Modeling Magnetic Contact

You can use surface-to-surface contact elements or the node-to-surface contact element to model magnetic flux across two contacted bodies. The following situations are possible.

- Non-perfect contact to account for the effects of a small air gap between mating components. This typically occurs at the interface between adjoining bodies. In this situation, there is a gap permeance effect where an MMF drop occurs. You can model this effect by inputting the gap permeance real constant, MCC. This option works best if the magnetic flux is normal to the gap interface.
- Perfect contact across dissimilar meshes. This is typically used to model the air gap in a machine, for example, where the rotor and stator meshes meet.

For both types of magnetic contact, you must set KEYOPT(1) = 7 to select the degree-of-freedom option. For the 2-D case, the magnetic potential degree of freedom, AZ, is active. For the 3-D case, only the scalar potential degree of freedom, MAG, is active, and scalar potential formulations (reduced (RSP), difference (DSP), or general (GSP)) are available (see **MAGOPT**).

Note — 3-D magnetic contact is not supported for the MVP formulation (AX, AY, AZ), and the edge-based formulation (AZ).

Note — Non-perfect magnetic contact is only available for the 3-D contact elements, CONTA173 and CONTA174.

For more information on which element types should be used for a particular analysis, see the element discussions in the appropriate chapter of the *ANSYS Low-Frequency Electromagnetic Analysis Guide*. For information on the use of the AZ degree of freedom, see Section 2.3.1.3: Specifying Element Types and Options. For more information on the use of the MAG DOF, see Section 5.3.5: Building the Model.

For details on how to set up a contact analysis, see Steps in a Contact Analysis.

For an example input listing showing a 2-D static magnetic contact analysis, see Section 2.6: Doing an Example 2-D Static Magnetic Contact Analysis (Command Method).

5.3.1. Using MCC

The magnetic flux across the contacting interface is defined by:

$$\text{MFLUX} = \text{MCC} \times (M_t - M_c)$$

where:

MFLUX = magnetic flux density

M_t, M_c = magnetic potential at the contact points on the target and contact surfaces

MCC = contact permeance coefficient (Henries/meters² in MKS units)

The MCC value is input through a real constant, which can be a function of temperature $[(T_t + T_c)/2]$, pressure, and time, by using the %TABLE% option. MCC values can be approximated as μ/t , where μ is the gap permeability and t is the gap width.

If the “no-separation contact” or “bonded contact” option is set (KEYOPT(12) = 4 or 5), contact interaction can occur between two surfaces separated by a narrow gap.

5.3.2. Modeling Perfect Magnetic Contact

Perfect magnetic contact supports dissimilar meshes on both sides of the contacting interface (MCC = infinity). You must use the internal MPC approach by setting KEYOPT(2) = 2. You must also set KEYOPT(4) = 1 or 2 for contact nodal detection, and KEYOPT(12) = 5, 6 for bonded contact.

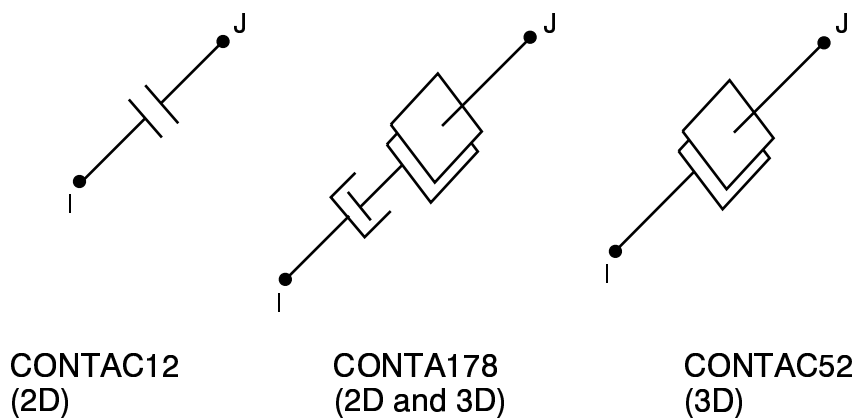
Chapter 6: Node-to-Node Contact

You can use node-to-node contact elements to model point-to-point contact (flexible-flexible or rigid-flexible). Additionally, you can use these elements to represent contact between two surfaces by specifying individual node-to-node contact between the opposing nodes of each surface. This use requires that the nodes of the two opposing surfaces match up geometrically and that the relative sliding between the two surfaces will be negligible. In addition, the deflections (rotations) of the two surfaces must remain small.

6.1. Node-to-Node Contact Elements

The most commonly used node-to-node elements are shown in the following figure.

Figure 6.1 Node-to-Node Contact Elements



In terms of solution CPU time, the elements shown are the simplest and least expensive of all contact element types. When modeling conditions warrant their use, they can be an effective tool for modeling a variety of contact situations. CONTA178 offers more features and flexibility than CONTAC12 and CONTAC52. The following lists the features available with CONTA178:

- More contact algorithms, including the Lagrange method (KEYOPT(2))
- Semiautomatic contact stiffness (real constants FKN-FKS)
- More flexibility for defining contact normal
- More contact behaviors (KEYOPT(10))
- Cylindrical gap with friction (KEYOPT(4) = 4)
- Damper (real constants CV1, CV2)

Note — CONTA178 supports only elastic Coulomb friction behavior. For rigid Coulomb friction models, use CONTAC12 and CONTAC52

6.2. Performing a Node-to-Node Contact Analysis

The procedure for performing a node-to-node contact analysis is similar to that described for the node-to-surface contact elements in Chapter 4, “Node-to-Surface Contact”. The following describes the typical steps in a node-to-node contact analysis. Each of these steps is explained in detail.

1. Create the geometry and mesh the model.
2. Generate the contact elements.
3. Define the contact normal.
4. Define the initial interference or gap.
5. Select the contact algorithm.
6. Apply the necessary boundary conditions.
7. Define the solution options.
8. Solve the problem.
9. Review the results.

6.2.1. Creating Geometry and Meshing the Model

Node-to-node contact elements transmit forces at the nodes (compared to surface-to-surface contact elements which transmit pressures at Gauss points). This feature limits their use to low order elements only.

You must identify where contact might occur during the deformation of your model. The nodes on the two surfaces of potential contact must line up. Once you've identified potential contact surfaces and created an adequate mesh you can create the contact elements.

6.2.2. Generating Contact Elements

You can generate node-to-node contact element in one of two ways:

1. Use the direct generation method:
Command(s): E
GUI: Main Menu > Preprocessor > Modeling > Create > Elements > User Numbered > Thru Nodes
2. Use the **EINTF** command to generate the contact elements automatically at coincident nodes or offset nodes. This is discussed in detail in the following subsections.

6.2.2.1. Generating Contact Elements Automatically at Coincident Nodes

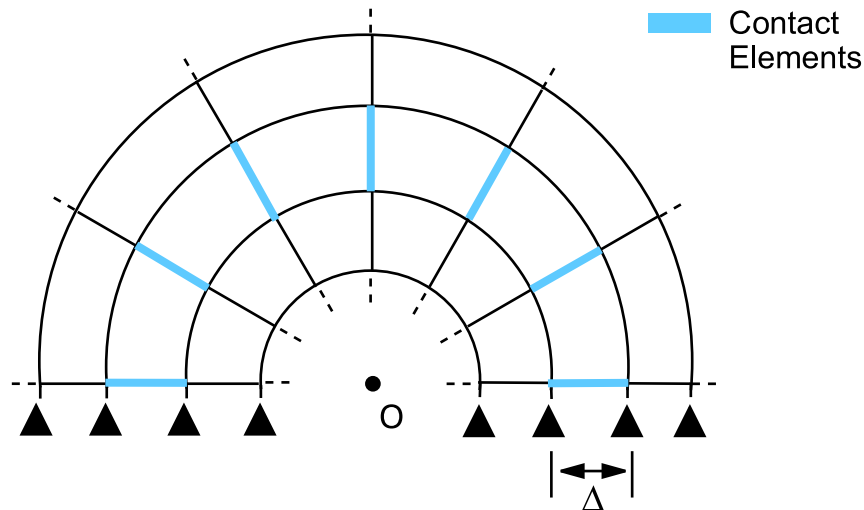
If the two bodies are in the “just touching” position, you can use the **EINTF** command to automatically generate the contact elements. In this case, the menu path is **Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > At Coincident Nodes**. Only nodes within the tolerance value set in the first argument (*TOLER*) are considered as coincident. If you wish to check a subset of nodes for coincidence, first select all the nodes you wish to check using the **NSEL** command.

6.2.2.2. Generating Contact Elements Automatically at Offset Nodes

If the two bodies are separated by a gap, the **EINTF** command can be used to create elements between offset nodes by specifying the node location increments (DX, DY, DZ) in the coordinate system KCN. The GUI path is **Main Menu > Preprocessor > Modeling > Create > Elements > Auto Numbered > Offset Nodes**.

If KOPT is set to 1, the nodes belonging to the contact elements created are rotated into the coordinate system KCN.

For example, the following figure shows contact between two concentric pipes separated by a gap. In this example, KCN would be a cylindrical coordinate system centered in O, and DX would be set in Δ ($\Delta \pm TOLER$).

Figure 6.2 Contact Between Two Concentric Pipes

6.2.2.3. Node Ordering

Node ordering can be critical in determining the contact normal. You can use **EINTF,,,LOW** or **EINTF,,,HIGH** to control node ordering. When using the LOW argument, the 2-node elements are generated from the lowest numbered node to the highest numbered node. When using HIGH, 2-node elements are generated from the highest to the lowest. To check the contact normal, use **/PSYMB,ESYS**. If you discover that the ordering is incorrect, you can reverse it using **EINTF,,,REVE**. To determine which side of the interface contains the I nodes, use the following commands:

```
ESEL,,ENAME,,178
NSLE,,POS,1
ESLN
NSLE
EPL0T
```

6.2.3. Defining the Contact Normal

The contact normal direction is of primary importance in a contact analysis with CONTA178. By default [KEYOPT(5) = 0 and NX, NY, NZ = 0], ANSYS will calculate the contact normal direction based on the initial positions of the I and J nodes, such that a positive displacement (in the element coordinate system) of node J relative to node I opens the gap. However, you must specify the contact normal direction for any of the following conditions:

- If nodes I and J have the same initial coordinates.
- If the model has an initial interference condition in which the underlying elements' geometry overlaps.
- If the initial open gap distance is very small.

In the above cases, the ordering of nodes I and J is critical. The correct contact normal usually points from node I toward node J unless contact is initially overlapped.

You can specify the contact normal by means of real constants NX, NY, NZ (direction cosines related to the global Cartesian system) or element KEYOPT(5). The following lists the various options for KEYOPT(5):

KEYOPT(5) = 0

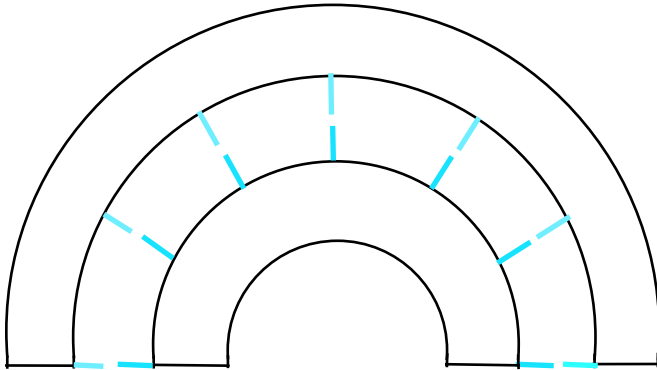
The contact normal is either based on the real constant values of NX, NY, NZ or on node locations when NX, NY, NZ are not defined. For two dimensional contact, NZ = 0.

KEYOPT(5) = 1 (2,3)

The contact normal points in a direction which averages the direction cosines of the X (Y, Z) axis of the nodal coordinates on both nodes I and J. The direction cosines on nodes I and J should be very close. This option may be supported by the NORA and NORL commands, which rotate the X axis of the nodal coordinate system to point to the surface normal of solid models. The GUI paths for these commands are, respectively:

Main Menu> Preprocessor> Modeling> Move/Modify> Rotate Node CS> To Surf Normal> On Areas
Main Menu> Preprocessor> Modeling> Move/Modify> Rotate Node CS> To Surf Normal> On Lines
Main Menu> Preprocessor> Modeling> Move/Modify> Rotate Node CS> To Surf Normal> With Area

Figure 6.3 Two Concentric Pipes, Normals Rotated Properly



For a pipe-within-a-pipe, this portrays the normals rotated properly.

KEYOPT(5) = 4 (5,6)

The contact normal points to X (Y, Z) of the element coordinate system issued by the ESYS command. If you use this option, make sure that the element coordinate system specified by ESYS is the Cartesian system.

Note — For CONTACT12 you can define the orientation of the contact surface with real constant THETA. For CONTACT52, you can use real constants NX, NY, NZ.

6.2.4. Defining the Initial Interference or Gap

With CONTA178, the gap size can be automatically calculated from the GAP real constant plus the node locations (projection of vector points from node I to J on the contact normal). This is the default (KEYOPT(4) = 0) This means that if you want the initial gap to be determined by the node locations only, set KEYOPT(4) = 0 and the real constant GAP to 0.

If KEYOPT(4) = 1, the initial gap size is only based on the real constant GAP (it ignores the node location). A negative value for the gap size can be used to model initial interference.

You can ramp initial interference using KEYOPT(9) = 1. This option isn't supported by CONTACT12 and CONTACT52. Note that CONTACT52's real constant GAP is the opposite of CONTACT12's real constant INTF, which defines an interference. KEYOPT(4) for CONTACT12 and CONTACT52 is also different from CONTA178.

6.2.5. Selecting the Contact Algorithm

You can choose among the following four different contact algorithms for CONTA178:

- Pure Lagrange Multiplier

- Lagrange multiplier on the contact normal, penalty on the frictional (tangential) direction.
- Augmented Lagrange method
- Pure penalty method.

CONTAC12 and CONTAC52 offer only the pure penalty method, in which you must specify the contact stiffness. The normal stiffness KN should be based upon the stiffness of the surface in contact. However, if you choose to use CONTA178 with the pure penalty method or the augmented Lagrange method, a “semiautomatic” setting is provided for the contact normal and tangential stiffnesses.

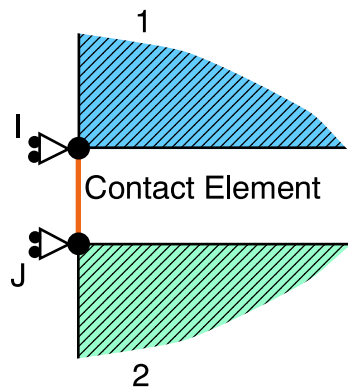
ANSYS provides a default normal contact stiffness FKN, which is based on Young's modulus E and the size of the underlying elements. FKN and FKS are factors. If you want to import absolute values for FKN and FKS, use *negative* values.

6.2.6. Applying Necessary Boundary Conditions

You can now apply necessary boundary conditions as you would do for any ANSYS analysis. For more information on applying boundary conditions, see the appropriate analysis description in the *ANSYS Structural Analysis Guide*.

When using the Lagrange multiplier method, be careful not to overconstrain the model. The model is overconstrained when a contact constraint at a node conflicts with a prescribed boundary condition on that degree of freedom at the same node. The following figure illustrates this issue:

Figure 6.4 Example of Overconstrained Contact Problem



Both nodes I and J are fixed in the X direction. The model is overconstrained only when the two bodies are in contact and bonded contact or rough friction has been defined. It can also occur if the contact status is “sticking.” In these cases, the constraint in the X-direction is duplicated which causes an overconstrained model.

“Zero Pivot” and “Numerical Singularity” warning messages indicate overconstraints in a model. Overconstraints can lead to convergence difficulties and/or inaccurate results. They can be easily avoided by changing the contact definition or the boundary conditions.

6.2.7. Defining the Solution Options

Convergence behavior for contact problems depends strongly on the particular problem. The options listed below are either typical or recommended for most node-to-node contact analyses.

- Set the appropriate auto time step behavior using KEYOPT(7). The **SOLCONTROL** command turns this **off** by default.

- The time step size must be small enough to capture the proper contact zone. The smooth transfer of contact forces is disrupted if the time step size is too large. A reliable way to set an accurate time step size is to turn automatic time stepping on. The **SOLCONTROL** command turns this **on** by default.

Command(s): AUTOTS,ON

GUI: Main Menu> Solution> Analysis Type> Sol'n Control (: Basic Tab)

Main Menu> Solution> Unabridged Menu> Load Step Opts> Time/Frequenc> Time-Time Step

Main Menu> Solution> Unabridged Menu> Load Step Opts> Time/Frequenc> Time and Substps

- Set the number of equilibrium equations to a number that is appropriate for a reasonable time step size. With **SOLCONTROL,ON**, this command defaults to between 15 and 26 iterations, depending upon the physics of the problem.

Command(s): NEQIT

GUI: Main Menu> Solution> Analysis Type> Sol'n Control (: Nonlinear Tab)

Main Menu> Solution> Unabridged Menu> Load Step Opts> Nonlinear> Equilibrium Iter

- Turn the predictor-corrector option on, unless you expect large rotations.

Command(s): PRED

GUI: Main Menu> Solution> Analysis Type> Sol'n Control (: Nonlinear Tab)

Main Menu> Solution> Unabridged Menu> Load Step Opts> Nonlinear> Predictor

- Set the Newton-Raphson option to FULL, with adaptive descent on.

Command(s): NROPT,FULL,,ON

GUI: Main Menu> Solution> Unabridged Menu> Analysis Type> Analysis Options

- For analysis involving friction, using **NROPT, UNSYM** is useful (and sometimes required if the coefficient of friction μ is > 0.2) for problems where the normal and tangential (sliding) motions are strongly coupled.
- NLGEOM, ON** is supported but the contact normal is not updated during the analysis. Make sure that only small rotations occur along the contact surfaces (except for the cylindrical gap option).
- Many convergence failures in contact analysis are the result of using too large a value for contact stiffness (real constant KN). Be sure to follow the recommendations given previously in this section for an initial estimate for contact stiffness. If such estimated values lead to a convergence failure, reduce the contact stiffness and restart. (You must also explicitly define the tangent stiffness - see the Note below.)
- Conversely, if overpenetration occurred in your contact analysis, you probably used a value of KN that was too small. In this case, gradually increase the contact stiffness value to an appropriate level by redefining it using new **R** commands over several load steps in a restart. (You must also explicitly define the tangent stiffness - see the **Note** statement below.)

Note — Although you can change the contact stiffness value (real constant KN), you *cannot* change any other real constants between load steps. Therefore, if you plan to change KN in a restart (or from one load step to the next), you cannot allow the value of the tangent (or "sticking") contact stiffness (real constant KT) to be defined by default, because the program would then attempt to redefine the tangent stiffness as the contact stiffness changed. You must explicitly define the tangent stiffness whenever you change the contact stiffness to maintain a consistent value for the tangent stiffness throughout all load steps.

6.2.8. Solving the Problem

You can now solve the analysis the same as you would for any linear or nonlinear analysis. See Section 2.3.5: Solve the Analysis in the *ANSYS Structural Analysis Guide*.

6.2.9. Reviewing the Results

Postprocessing of contact result items for these elements requires the use of **ETABLE** commands.

The steps for reviewing results in POST26 are the same as those for a typical nonlinear analysis. See Section 8.5.6.3: Reviewing Results in POST26 in the *ANSYS Structural Analysis Guide*.

For general information on post processing, refer to the *ANSYS Basic Analysis Guide*.

Chapter 7: Multipoint Constraints and Assemblies

You can use the internal multipoint constraint (MPC) approach ($\text{KEYOPT}(2) = 2$), in conjunction with certain bonded and no separation contact definitions ($\text{KEYOPT}(12) = 4, 5, \text{ or } 6$), to define various contact assemblies and kinematic constraints. This capability is available for contact elements CONTA171, CONTA172, CONTA173, CONTA174, and CONTA175. By this method, the program builds MPC equations internally based on the contact kinematics. You can use this method to model the following contact assemblies and surface-based constraints:

- Solid-solid assembly - both contact and target surfaces paste onto solid element faces
- Shell-shell assembly - both contact and target surfaces paste onto shell element faces
- Shell-solid assembly - the contact surface pastes onto shell element faces and the target surfaces paste onto solid element faces
- Rigid constraint surface - the contact nodes are constrained to the rigid body motion defined by the pilot node (similar to the **CERIG** command)
- Force-distributed surface - the forces or displacements applied to the pilot node are distributed to contact nodes, in an average sense, through shape functions (similar to the **RBE3** command)
- Beam-solid assembly - one beam end-node is the pilot node which connects to the solid or shell surface (use the rigid constraint surface or force-distributed surface type of MPC)

The internal MPC approach can overcome the drawbacks of the traditional contact algorithms and other multipoint constraint tools available in ANSYS. For example:

- Degrees of freedom of the contact surface nodes are eliminated, reducing the wave front size of the system equation solver.
- No contact stiffness is required as input. For a small deformation problem, this represents true linear contact behavior; no iterations are needed to solve the system of equations. For large deformation problems, the MPC equations are updated during each iteration, overcoming the small strain restriction in conventional constraint equations.
- Both translational and rotational degrees of freedom can be constrained.
- Generation of internal MPC is simple because it uses contact pair definitions.
- Shape functions are taken into account automatically; no weight factor is needed for a force-distributed multipoint constraint (which is similar to the **RBE3** command) if you use higher order elements or axisymmetric elements. In addition to forces, displacements can be applied on the pilot node for this type of MPC.

The following sections describe how to use the internal MPC approach to define assemblies and surface-based constraints. Before using the internal MPC feature, be sure to review Section 7.4: Restrictions and Recommendations for Internal MPC.

7.1. Modeling Solid-Solid and Shell-Shell Assemblies

To define solid-solid or shell-shell assemblies using the internal MPC approach, you must set the following key options on the contact elements:

$\text{KEYOPT}(2) = 2$ - MPC based approach

$\text{KEYOPT}(12) = 4, 5, \text{ or } 6$ - No separation (see note below), bonded always, or bonded initial

KEYOPT(4) = 1 or 2 - Nodal detection for CONTA171, CONTA172, CONTA173, CONTA174

KEYOPT(4) = 0 or 1 - Contact normal direction for CONTA175

Note — When used with the MPC approach, the no separation option (KEYOPT(12) = 4) is only valid for modeling solid-solid assemblies that represent a slider line or slider plane.

The following key options are ignored: KEYOPT(8) (it is always set to 2 internally), KEYOPT(10).

The following real constants are used: R1, R2, ICONT, PINB, CNOF, PMAX, PMIN, TOLS. All other real constants are ignored.

This function works similarly to the **CEINTF** command for small deformation analysis (**NLGEOM,OFF**). Comparing it to the **CEINTF** command, the contact surface acts as “region A nodes,” and the target surface acts as “region B elements.”

The contact surface must be defined on the deformable bodies, and the target surface must be defined on either deformable or rigid bodies in the contact pair. In order to prevent over-constraint, only asymmetric contact is supported. If symmetric pairs are defined, ANSYS will automatically pick one pair and ignore the other pair (acting as KEYOPT(8) = 2). The self-contact pair definition is not supported.

If the temperature degree of freedom is active in the model (KEYOPT(1) = 1 or 2), ANSYS will build MPC equations not only for structural degrees of freedom, but also for the temperature DOF. In this case, the real constant TCC is ignored. If only the temperature DOF is set (KEYOPT(1) = 2) and other solution options are defined (**AN-TYPE,,TRANS; THOPT,QUASI; EQSLV,JCG/ICCG**), the internal MPC approach can support fast thermal transient analysis (see Section 3.5.4: Nonlinear Options in the *ANSYS Thermal Analysis Guide*). Internal MPC equations for temperature DOF are built to support heat transfer between the two bonded surfaces.

For the bonded always option (KEYOPT(12) = 5) and no separation option (KEYOPT(12) = 4), any contact node that lies inside the pinball region (PINB) can be the constrained node in the MPC definition if an intersection with the target surface is detected in the contact normal direction. This holds true at the beginning of deformation, as well as during the deformation process. A relatively small PINB may be used to prevent any false contact. When KEYOPT(12) is set to the bonded always or no separation option, PINB defaults to 0.25 (25% of the contact depth) for small deformation analysis (**NLGEOM,OFF**), and 0.5 (50% of the contact depth) for large deformation analysis (**NLGEOM,ON**). (The default PINB value may differ from what is described here if CNOF is input. See Section 3.8.9.2: Using PINB for more information.)

For the “bonded initial” option (KEYOPT(12) = 6), only those contact nodes that are initially in contact or have a very small gap but lie inside the adjustment zone (ICONT) are always constrained via internal MPC. Those contact nodes that are initially open will never be constrained, even though they may later penetrate into the target surface during the deformation process. In order to capture contact, you should specify proper ICONT or CNOF values. Use **CNCHECK** in conjunction with ICONT to move all the contact nodes that are inside the ICONT zone onto the target surface in the initial configuration, without causing any strain. When the “bonded initial” option is set and KEYOPT(5) = 0 or 4, ICONT defaults to 0.05. See Section 3.8.7: Adjusting Initial Contact Conditions for more information on using KEYOPT(5), CNOF, and ICONT.

In most cases, ANSYS will automatically constrain the translational degrees of freedom for a solid-solid assembly and constrain both translational and rotational degrees of freedom for a shell-shell assembly. However, you can use KEYOPT(5) of the target element (TARGE170) to explicitly define the type of constraint:

KEYOPT(5) = 0 - Auto constraint type detection

KEYOPT(5) = 1 - Solid-solid constraint (no rotational DOFs are constrained)

KEYOPT(5) = 2 - Shell-shell constraint (both translational and rotational DOFs are constrained)

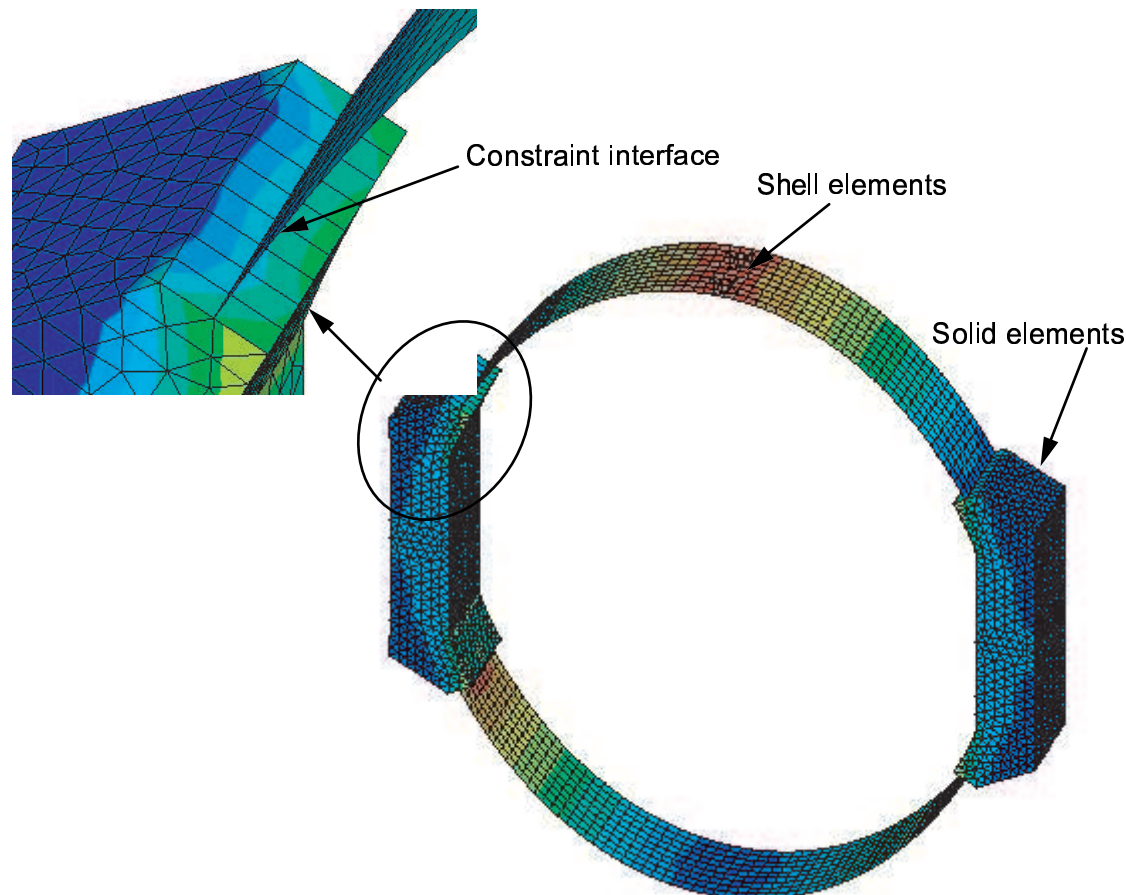
Note — When the no separation option ($\text{KEYOPT}(12) = 4$ on the contact element) is used with the MPC approach to model solid-solid assemblies, only the $\text{KEYOPT}(5) = 0$ and 1 options (auto detection or solid-solid constraint) described above are valid. If the auto detection option is set and the program finds a shell-shell or shell-solid constraint in this situation, the solution will terminate.

Note — If overconstraint occurs in bonded shell-shell assemblies when using the MPC algorithm, you can switch to the penalty method or the augmented Lagrangian method. See Section 3.8.11.3: Bonded Contact for Shell-Shell Assemblies for more information.

7.2. Modeling a Shell-Solid Assembly

The 3-D shell-solid assembly provides a transition from a shell element region to a solid element region. This approach is useful when local modeling requires a full three-dimensional model with a relatively fine mesh, but other parts of the structure can be represented by shell elements (see Figure 7.1: “Example of Shell-Solid Assembly”). No alignment is required between the solid element mesh and the shell element mesh. The contact surface or edge must be built on the shell element side. The target surface must be built on the solid elements side.

Figure 7.1 Example of Shell-Solid Assembly



To define a shell-solid assembly using the internal MPC approach, you must set the following key options on the contact elements:

$\text{KEYOPT}(2) = 2$ - MPC-based approach

$\text{KEYOPT}(12) = 5$ or 6 - Bonded always or bonded initial

$\text{KEYOPT}(4) = 1$ or 2 - Nodal detection for CONTA171, CONTA172, CONTA173, CONTA174

KEYOPT(4) = 0 or 1 - Contact normal direction for CONTA175

The following real constants are also used: ICONT, FTOL, PINB, CNOF, PMAX, PMIN, TOLS.

The following key options are ignored: KEYOPT(8), KEYOPT(10), KEYOPT(1) > 0.

In most cases, ANSYS will automatically constrain both translational and rotational degrees of freedom for a shell-solid assembly (see Figure 7.2: "Shell-Solid Assembly (Original Mesh)"). However, you can use KEYOPT(5) of the target element (TARGE170) to explicitly define the type of constraint:

KEYOPT(5) = 0 - Auto constraint type detection (default)

KEYOPT(5) = 1 - Solid-solid constraint; no rotational DOFs are constrained (see Figure 7.3: "Shell-Solid Assembly with Solid-Solid Constraint Option")

KEYOPT(5) = 2 - Shell-shell constraint; both translational and rotational DOFs are constrained (see Figure 7.4: "Shell-Solid Assembly with Shell-Shell Constraint Option")

KEYOPT(5) = 3 - Shell-solid constraint; both translational and rotational DOFs are constrained on shell nodes; only translational DOFs are constrained on solid nodes (see Figure 7.5: "Shell-Solid Assembly with Shell-Solid Constraint Option")

KEYOPT(5) = 4 - Shell-solid constraint, all directions. This option acts the same as KEYOPT(5) = 3 if an intersection is found from the contact normal to the target surface. Otherwise, constraint equations are still built as long as contact nodes and target segments are inside the pinball region. (See Figure 7.6: "Shell-Solid Constraint, All Directions (KEYOPT(5) = 4)" later in this section.)

The solid-solid and shell-shell constraint types (KEYOPT(5) = 1 or 2) may require additional shell elements at the interface. These shell elements can be defined by typical modeling methods, or you can use the **SHSD** command to generate these elements automatically.

SHSD is a meshing tool used to build solid-solid and shell-shell constraint types. This command can be used only when the contact pair consists of CONTA175 and TARGE170 elements. Additional shell elements (SHELL181) and/or contact elements (CONTA175) are created through this command (see the **SHSD** command description for more information). For the shell-solid constraint type, no special mesh tool is required.

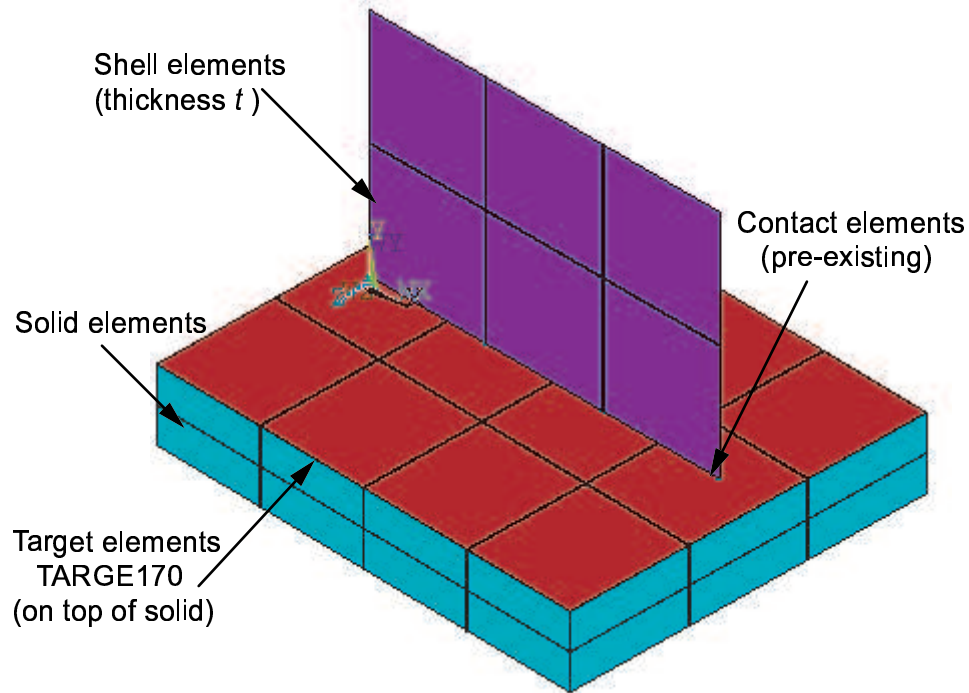
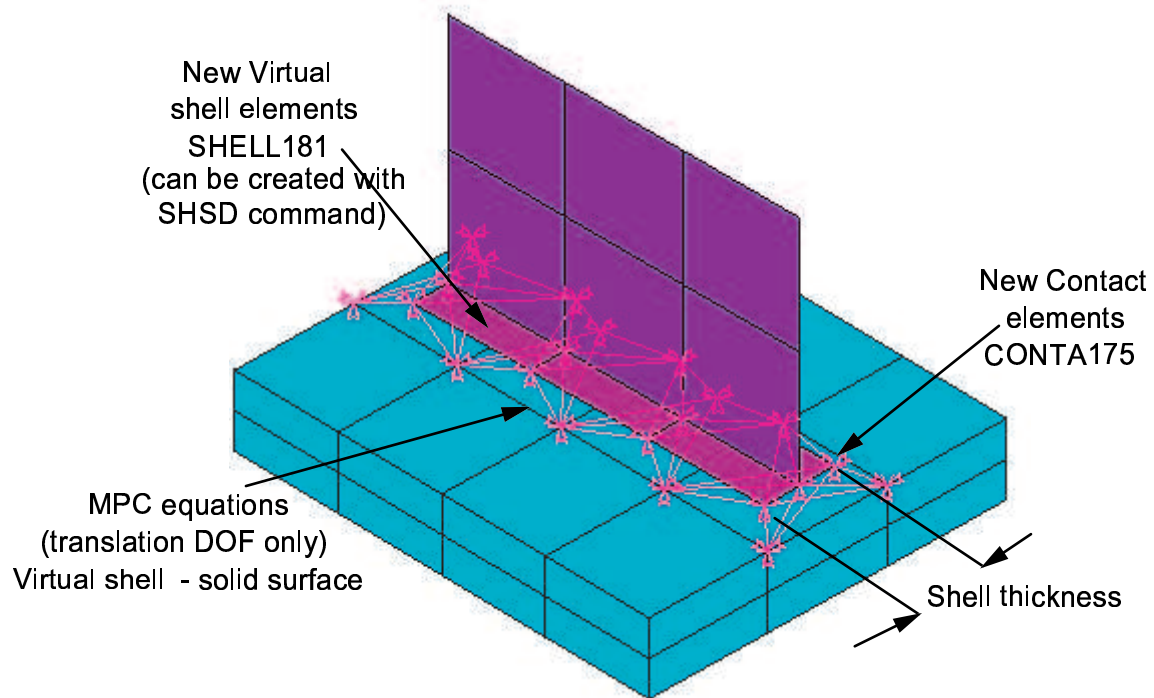
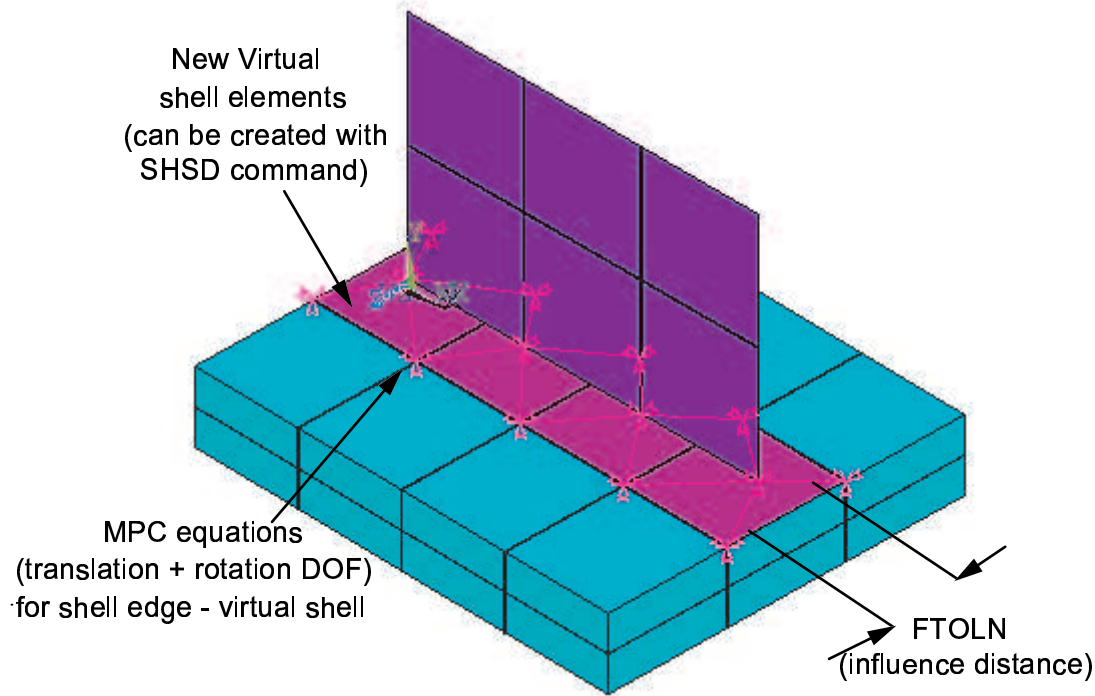
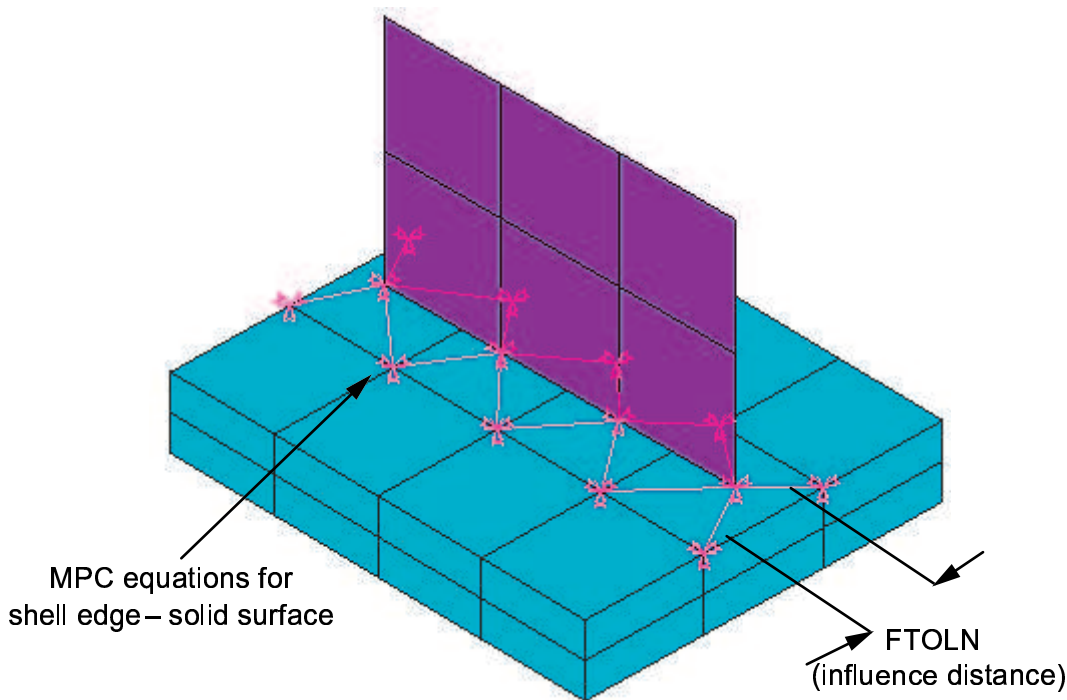
Figure 7.2 Shell-Solid Assembly (Original Mesh)**Figure 7.3 Shell-Solid Assembly with Solid-Solid Constraint Option**

Figure 7.4 Shell-Solid Assembly with Shell-Shell Constraint Option**Figure 7.5 Shell-Solid Assembly with Shell-Solid Constraint Option**

For the shell-solid constraint option (`KEYOPT(5) = 3` on `TARGE170`), ANSYS automatically creates an internal set of force-distributed constraints (similar to the **RBE3** command) between nodes on the shell edges and nodes on the solid surface. The program uses the pinball region (`PINB`), initial adjustment zone (`ICONT`), and influence distance (`FTOLN`) to determine which nodes on the shell edge will be constrained with which nodes on the solid surface. Each shell node acts as the master node, and associated solid nodes act as slave nodes.

For the bonded always option ($\text{KEYOPT}(12) = 5$), any shell node that lies inside the pinball region (PINB) will be included in the constraint if an intersection with the target surface is detected in the contact normal direction. This holds true at the beginning of deformation, as well as during the deformation process. A relatively small PINB value may be used to prevent any false contact. The default for PINB is 0.25 (25% of the contact depth) for small deformation analysis (**NLGEOM,OFF**), and 0.5 (50% of the contact depth) for large deformation analysis (**NLGEOM,ON**). (The default PINB value may differ from what is described here if CNOF is input. See Section 3.8.9.2: Using PINB for more information.)

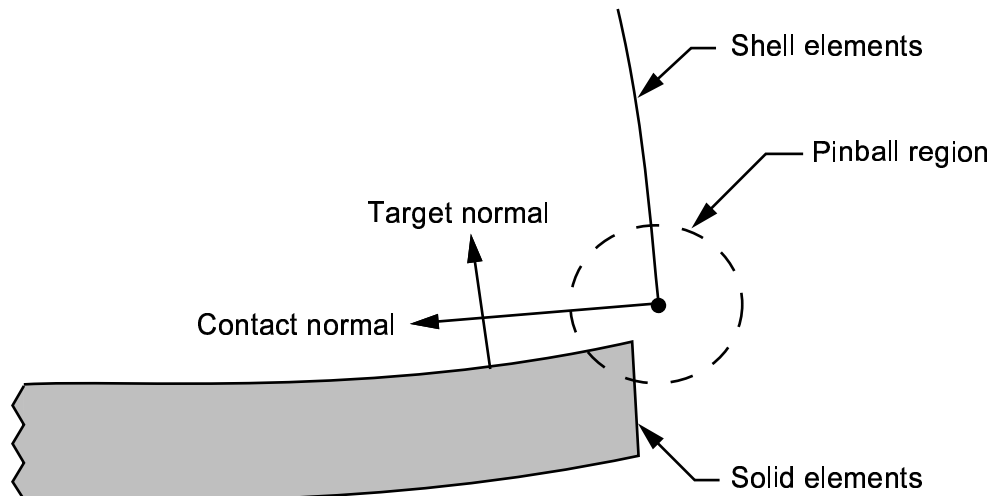
For the bonded initial option ($\text{KEYOPT}(12) = 6$), only the shell nodes that initially lie inside the adjustment zone (ICONT) are included in the constraint sets. Shell nodes that lie outside ICONT are not constrained with the solid nodes. The default for ICONT is 0.05 (5% of the contact depth).

The influence distance (FTOLN) is used for the shell-solid constraint option ($\text{KEYOPT}(5) = 3$ on TARGE170). Each solid node is included in the constraint set if the perpendicular distance from the solid node to any shell edge is smaller than the influence distance. FTOLN defaults to half the thickness of the shell. A positive FTOLN value represents a scaling factor on the shell half-thickness, and a negative value represents an absolute distance value.

A shell-solid assembly can be used in a substructure analysis. However, if a superelement is defined to represent the shell elements, shell thickness is unknown in the use pass. In this case, you must overwrite the default setting of FTOLN (originally a factor of the shell thickness) to account for the zero thickness of the superelement. Input an absolute value for FTOLN (that is, input a negative value) to capture all constrained nodes when the constraint equations are built.

Among the four constraint types, the shell-shell constraint approach ($\text{KEYOPT}(5) = 2$ on TARGE170) often provides a better solution. The solid-solid constraint ($\text{KEYOPT}(5) = 1$) can result in higher local stresses near the interface between the virtual shell surface and solid surface. The shell-solid constraint ($\text{KEYOPT}(5) = 3$ or 4) is not always capable of transmitting all components of the moment at the shell nodes due to transverse shear locking of shell elements. A typical example is the moment component that parallels to the normal of the solid surface. The $\text{KEYOPT}(5) = 4$ option (shell-solid constraint, all directions) is recommended for detecting contact in bodies that do not overlap or intersect (see figure below).

Figure 7.6 Shell-Solid Constraint, All Directions ($\text{KEYOPT}(5) = 4$)



For the solid-solid and shell-shell constraint types ($\text{KEYOPT}(5) = 1$ or 2 on TARGE170), any initial penetration or gap is closed if $\text{KEYOPT}(9) = 0$; the initial penetration or gap remains if $\text{KEYOPT}(9) = 1$. However, for the shell-solid constraint type ($\text{KEYOPT}(5) = 3$ or 4), $\text{KEYOPT}(9)$ is ignored, and initial penetration or gap always remains constant. In order to close the initial penetration or gap, issue the command **CNCHECK,ADJUST** in the beginning of the analysis.

7.3. Surface-Based Constraints

A surface-based constraint can be used to couple the motion of nodes on the contact surface to a single pilot node on the target surface. The multipoint constraint (MPC) capability of the contact elements (KEYOPT(2) = 2) allows you to define two types of surface-based constraints:

- Rigid constraint surface - In this type of constraint, the contact nodes are constrained to the rigid body motion defined by the pilot node (see Figure 7.7: "Rigid Constraint Surface"), similar to a constraint defined by the **CERIG** command.
- Force-distributed constraint surface - In this type of constraint, forces or displacements applied on the pilot node are distributed to contact nodes (in an average sense) through shape functions (see Figure 7.8: "Force-Distributed Surface"), similar to a constraint defined by the **RBE3** command.

Figure 7.7 Rigid Constraint Surface

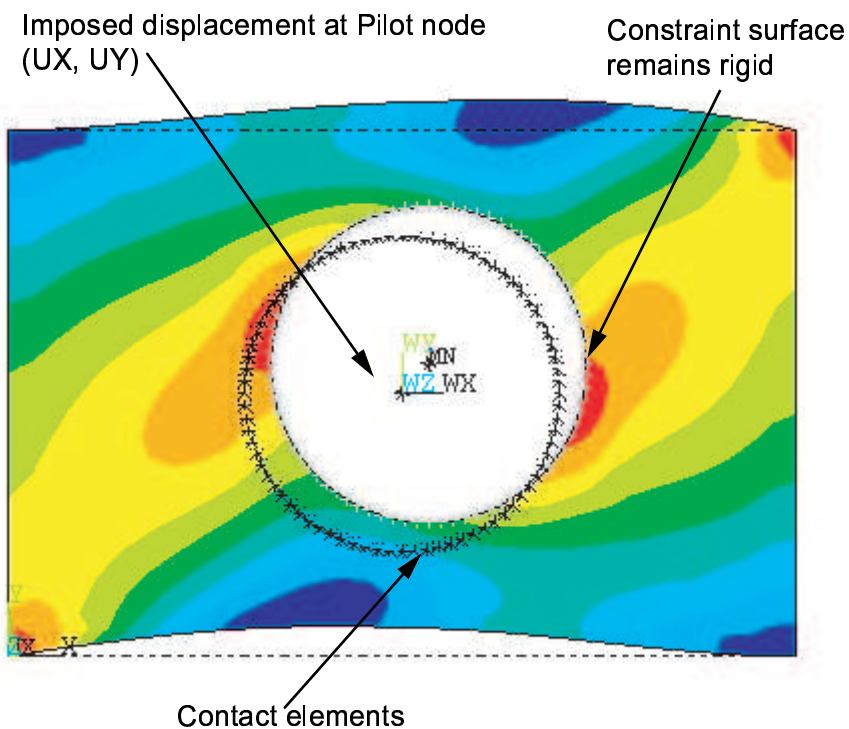
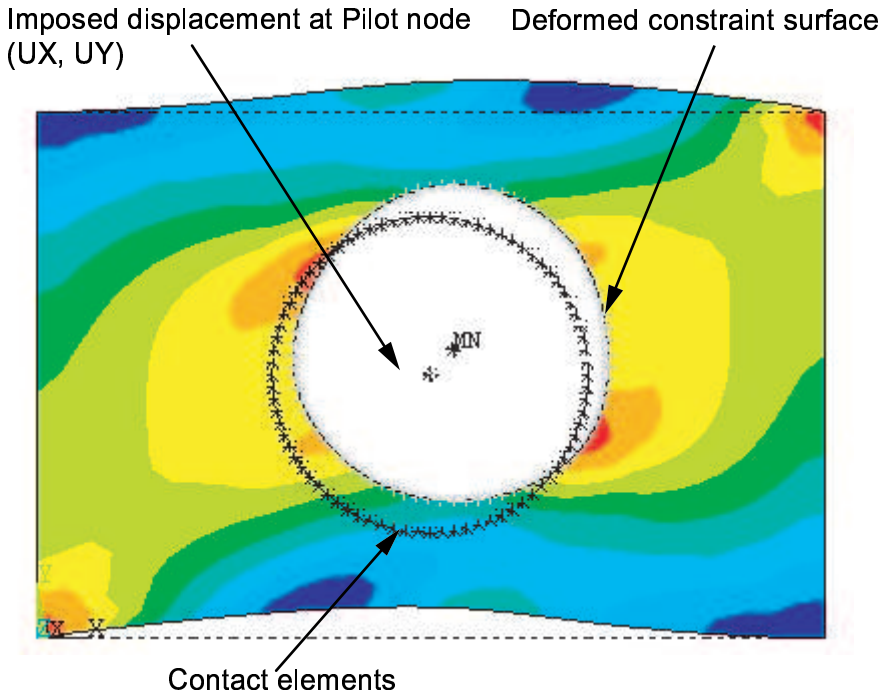


Figure 7.8 Force-Distributed Surface

These surface-based constraints can be used in the following applications:

- To apply loads and boundary conditions to the pilot node (such as torque load or drill rotation). Example: a bolt head submitted to a torque force using a force-distributed surface constraint.
- To model rigid end conditions. Example: using a rigid constraint surface to model a rigid end plate or rigid plane section of 3-D solid elements.
- To define transitions between solid and structure elements. Example: a beam element connected to a solid element face.

7.3.1. Defining Surface-Based Constraints

The contact surface can be generated via the **ESURF** command. The contact nodes on the contact surface are the slave nodes of the MPC equations. The pilot node is the only target segment on the target surface side. It is the master node of the MPC equations. Forces and displacements can be applied on the pilot node.

For a force-distributed surface constraint, use the following contact element key options:

KEYOPT(2) = 2 - MPC based approach

KEYOPT(12) = 5 or 6 - Bonded always or bonded initial (the two cases are the same)

KEYOPT(4) = 1 - Nodal detection for CONTA171, CONTA172, CONTA173, CONTA174; Contact normal direction for CONTA175

For a rigid constraint surface, use the following contact element key options:

KEYOPT(2) = 2 - MPC based approach

KEYOPT(12) = 5 or 6 - Bonded always or bonded initial (the two cases are the same)

KEYOPT(4) = 2 - Nodal detection for CONTA171, CONTA172, CONTA173, CONTA174

KEYOPT(4) = 0 - Contact normal direction for CONTA175

The following key options are ignored for surface-based constraints: KEYOPT(8), KEYOPT(5), KEYOPT(7), KEYOPT(10).

None of the standard contact real constants are used for surface-based constraints using internal MPC.

KEYOPT(1) > 0 is ignored for a force-distributed surface constraint since only structural degree-of-freedom constraints are included. However, for a rigid constraint surface, you can use KEYOPT(1) to include other field degrees of freedom (in addition to the structural DOFs) in the constraint sets.

You can specify the surface-based constraint in a local coordinate system. For the rigid constraint surface, rotate the contact nodes into a local coordinate system. For the force-distributed constraint, rotate the pilot node into a local coordinate system.

You can apply the surface-based constraint to a user-defined DOF set with respect to the global or the local coordinate system by using KEYOPT(4) of the target element (TARGE169 or TARGE170). For example, for the 3-D case (TARGE170), you might specify that only UX, UY, and ROTZ be used in the constraint. You can do this by entering a six digit value for KEYOPT(4). The first to sixth digits represent ROTZ, ROTY, ROTX, UZ, UY, UX, respectively. The number 1 (one) indicates the DOF is active, and the number 0 (zero) indicates the DOF is not active. Therefore, to specify that UX, UY, and ROTZ be used in the constraint, you would enter 100011 as the KEYOPT(4) value.

For a force-distributed surface constraint, these additional guidelines apply:

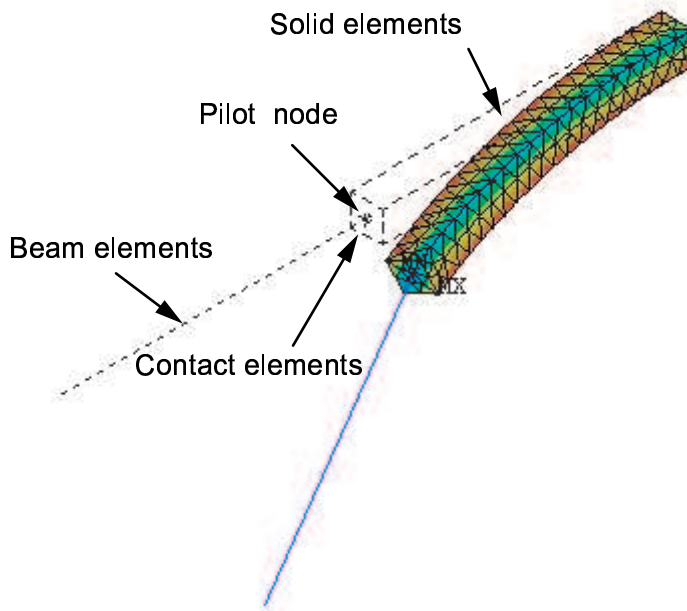
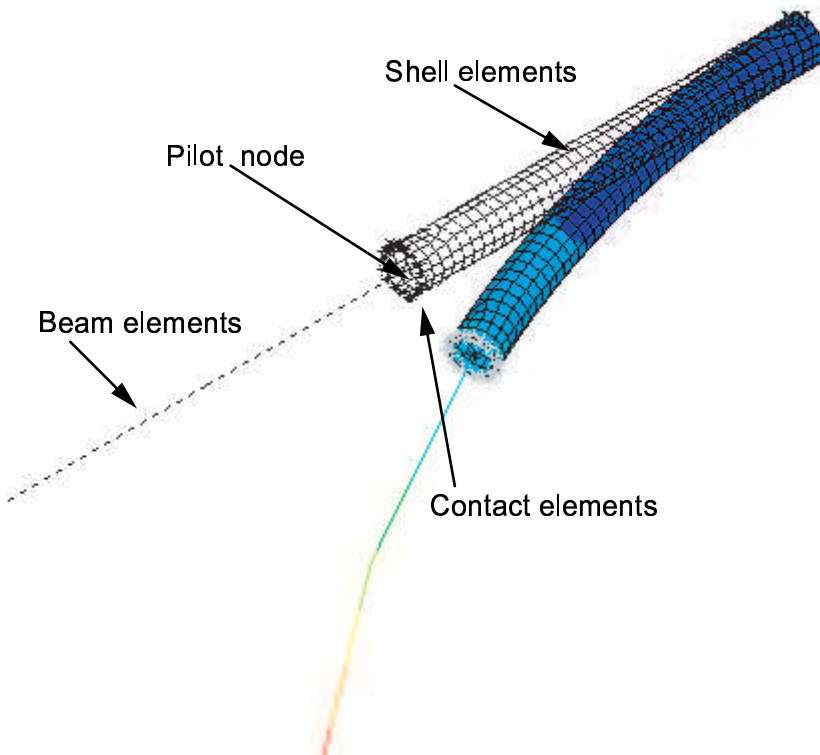
- The pilot node is a dependent node (meaning the degrees of freedom for this node are removed). The contact nodes are independent nodes (the degrees of freedom are retained). If the pilot node has constraints applied to it, internally-generated MPCs are rewritten so that the degrees of freedom of the pilot node are no longer dependent DOF.
- The pilot node has up to 3 degrees of freedom for 2-D or up to 6 degrees of freedom for 3-D.
- KEYOPT(4) of TARGE169 and TARGE170 controls the number of degrees of freedom of the pilot node.
- The number of internally-generated MPCs is equal to the number of degrees of freedom defined by KEYOPT(4) of TARGE169 and TARGE170.

For a rigid constraint surface, these additional guidelines apply:

- The pilot node is an independent (retained) node in the constraint equation. The contact nodes are the dependent (removed) nodes. You should not apply any displacement constraints, coupling (**CP** command), or constraint equations (**CE** command) on the contact nodes.
- The pilot node has 6 degrees of freedom for 3-D or 3 degrees of freedom for 2-D.
- KEYOPT(4) of TARGE169 and TARGE170 controls the DOF set (the number of DOF) of the contact (dependent) nodes used in the internally-generated MPCs.
- The number of internally-generated MPCs is equal to the number of contact nodes x the number of DOF.

7.3.2. Modeling a Beam-Solid Assembly

The surface-based constraint technique can be used to apply transitions between solid and structure elements; for example, a beam element connected to the solid or shell element face. One beam end node must be the pilot node and the solid/shell nodes must be the contact nodes. The rigid constraint surface is generally well-suited for the solid beam to solid surface case (see Figure 7.9: “Beam-Solid Assembly Defined by Rigid Constraint Surface”), and the force-distributed constraint surface is well-suited for the flexible beam (such as a thin wall beam) to solid/shell surface case (see Figure 7.10: “Beam-Solid Assembly Defined by Force-distributed Surface”).

Figure 7.9 Beam-Solid Assembly Defined by Rigid Constraint Surface**Figure 7.10 Beam-Solid Assembly Defined by Force-distributed Surface**

7.4. Restrictions and Recommendations for Internal MPC

The following restrictions apply when using the internal MPC approach.

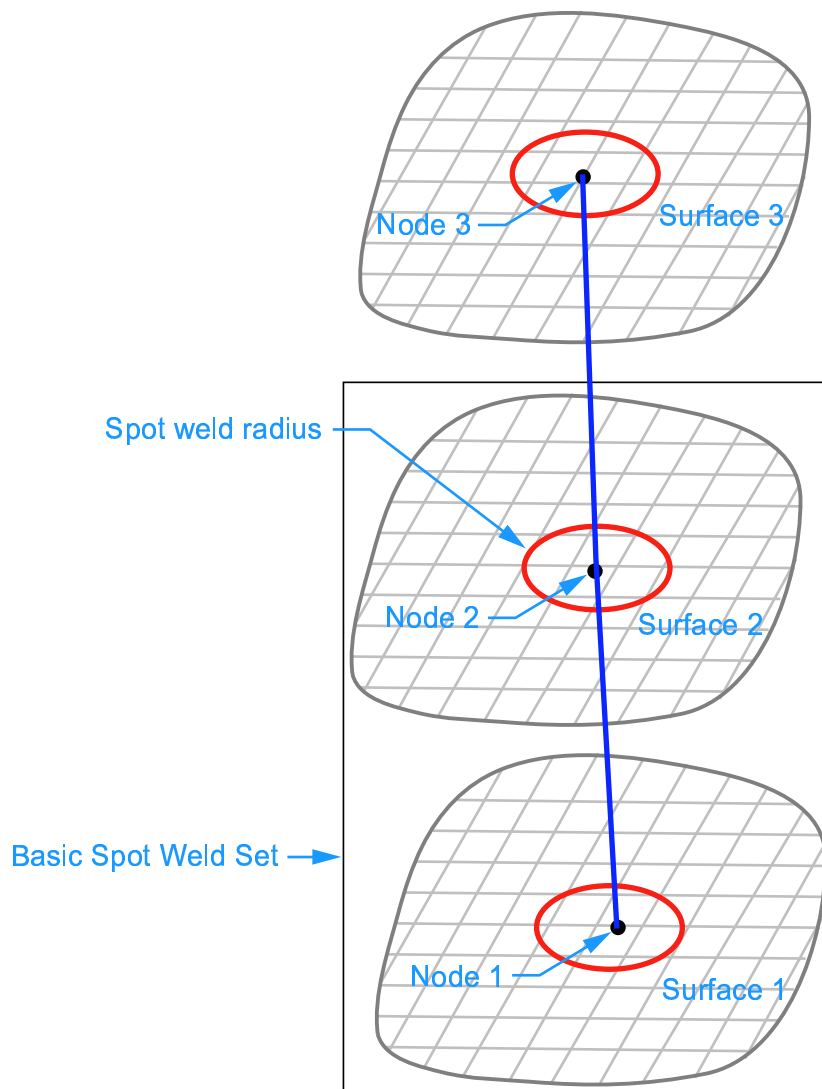
- Displacement boundary conditions and other constraint equations or coupling equations typically should not be applied on the contact nodes for solid-solid, shell-shell, and shell-solid assemblies, or for the rigid constraint surface MPC. Otherwise, over-constraint can occur because the degrees of freedom of contact nodes are eliminated through internal MPC equations.
- Sparse, PCG, and AMG solvers are recommended to solve models with constraint equations. In certain cases, the frontal solver may not solve problems containing constraint equations.
- The MPC-based algorithm must be used with nodal detection (`KEYOPT(4) = 1` or `2`) for `CONTA171`, `CONTA172`, `CONTA173`, and `CONTA174`. If nodal detection is not set, ANSYS will issue a warning message and automatically set `KEYOPT(4) = 2`.
- The internal MPC option does not support rigid-flexible contact when rigid surfaces are modeled by any primitive segments (such as circle, cylinder, cone, or sphere).
- The shell-solid assembly is usually used for the case where the solid mesh is fine with respect to the shell thickness. The shell-solid interface should be located in a region of the structure where shell theory is valid for an approximation. The accuracy of local stresses near the shell-solid interface (at least within the shell thickness range) is not guaranteed. We recommend you include at least two solid elements along the layer of the shell-solid interface.
- Using the force-distributed constraint type of MPC with a large number of contact nodes can result in a large wavefront for the global stiffness matrix. This may significantly increase the peak memory required during of element stiffness assembly. Consider reducing the number of contact nodes if real memory or virtual memory is limited.
- The force-distributed surface constraint creates internal constraint equations such that the motion of the pilot node is the average of the contact nodes. For the rotations, a least-squares approach is used to define the “average rotation” at the pilot node from the translations of the contact nodes. If the contact nodes are colinear, then one of the rotations on the pilot node that is parallel to the colinear direction can not be determined in terms of the translations of the contact nodes. Therefore, the associated moment component on the pilot node in that direction can not be transmitted. When this case occurs, a warning message is issued.
- The contact related postprocessing items (**ETABLE** items, **PRESOL**, **CONT**, etc.) are not supported for the internal MPC option.

Chapter 8: Spot Welds

The traditional way of modeling spot welds involves matching the meshes of different parts at the spot weld connection points, which introduces meshing difficulties and often requires parts to be meshed manually. In addition, this approach does not take into account the effects of the spot weld radius, and it underestimates the strength of the spot weld connection when the radius of the spot weld is the same or lesser order of magnitude than the mesh size.

The ANSYS spot weld feature, based on the internal multipoint constraint (MPC) approach, offers a robust alternative to the traditional method. This feature allows you to easily model thin sheet components that are connected with spot welds, rivets, or fasteners. The spot weld can be located anywhere between the parts that are to be connected, independent of the mesh and the node locations. The figure below shows an example of the spot weld configuration. Each spot weld set connects two or more surfaces.

Figure 8.1 Example Spot Weld Configuration



There are several advantages to using this mesh independent spot weld capability:

- Parts can be meshed independently.

- A basic spot weld set can be easily defined by specifying two surfaces to be connected, and a single node called the spot weld node near the surfaces. The spot weld node determines the location of the spot weld.
- This approach takes into account effects of the spot weld radius input by the user. ANSYS will create multipoint constraint equations (MPCs) internally through two contact pair definitions, one on each spot weld surface. The internal constraint equations (force-distributed constraints) couple the motion of the surface nodes to the motion of the spot weld node in an average sense.
- The spot weld can be rigid (default) or deformable. To specify a deformable spot weld, you simply define a deformable beam element type prior to creating the spot weld.

8.1. Defining a Spot Weld Set

The general procedure for defining a spot weld set consists of the following steps:

1. Define the basic spot weld set (**SWGEN** command)
2. Define any additional surfaces for the existing spot weld set (**SWADD** command)

In addition, ANSYS supplies the **SWLIST** and **SWDEL** commands for listing and deleting spot weld sets.

8.1.1. Creating a Basic Spot Weld Set with SWGEN

Use the **SWGEN** command to define a basic spot weld set (in the GUI, pick **Main Menu > Preprocessor > Modeling > Create > Element > SpotWeld > Create New Set**). The required arguments are:

- Name of the spot weld set
- Spot weld radius
- Spot weld surfaces
- First spot weld node

You can optionally specify the following:

- Second spot weld node
- Spot weld projection direction
- Search radius
- Contact and target element types

All of these inputs are discussed in detail below.

Spot Weld Set Name (*EComp*):

Use the *EComp* argument to assign a name to a new basic spot weld set. The spot weld set is a component composed of new beam, target, and contact elements. The name entered for the *EComp* argument will be used to list, delete, output, or add new surfaces to the spot weld set.

Spot Weld Surfaces (*NCM1*, *NCM2*):

Each basic spot weld set contains two surfaces, surface 1 and surface 2, to be connected together. Use arguments *NCM1* and *NCM2* to specify these surfaces. You can input the name of a predefined node component or a meshed area number. If no nodal component names matching *NCM1* and *NCM2* are found, they are assumed to be meshed area numbers.

Spot Weld Nodes (*SND1*, *SND2*):

You must specify the first spot weld node, *SND1*. The location of each spot weld depends on *SND1*. If you specify the second spot weld node (*SND2*), it is used to determine the spot weld projection direction. The

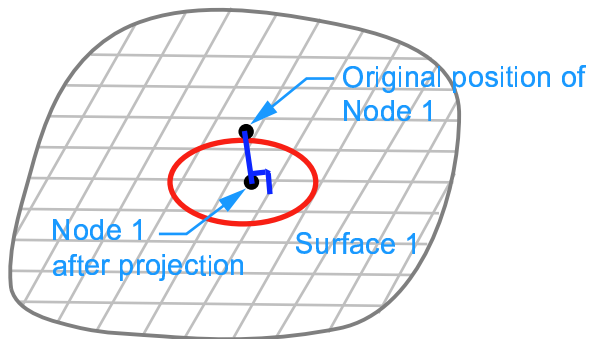
second node will be generated automatically if it is not specified. You should locate *SND1* and *SND2* (referred to as node 1 and node 2 in figures that follow) as close to the corresponding surfaces as possible. They can be nodes on the corresponding surfaces or independent nodes in space. If they are not on the corresponding surfaces, node 1 will be moved to surface 1 and node 2 will be moved to surface 2.

ANSYS will determine the actual location for the spot weld nodes by first projecting node 1 onto surface 1 along the spot weld projection direction. Next, ANSYS projects node 2 to surface 2 along the spot weld projection direction. The spot weld nodes are moved to the surfaces in the beginning of the analysis.

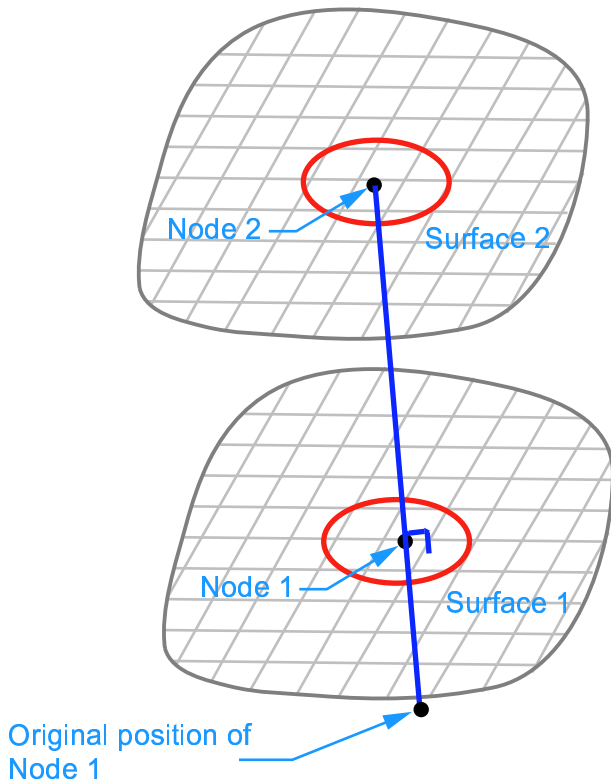
Spot Weld Projection Direction:

By default, the projection direction is a surface normal vector from node 1 to surface 1 (see figure below). If you specify node 2 (*SND2*), the projection direction is in line with the two spot weld nodes. If node 2 is not specified, you can still explicitly define the projection direction with the *DIRX*, *DIRY*, and *DIRZ* arguments.

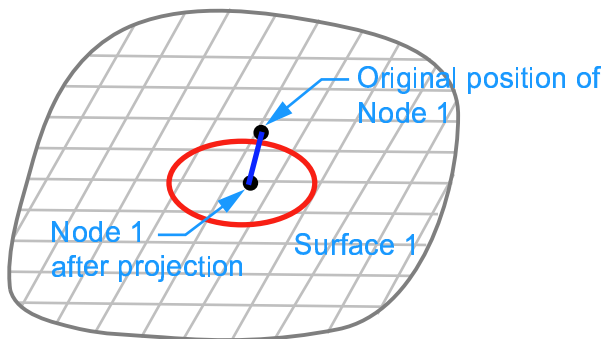
Figure 8.2 Default Projection Direction for Node 1



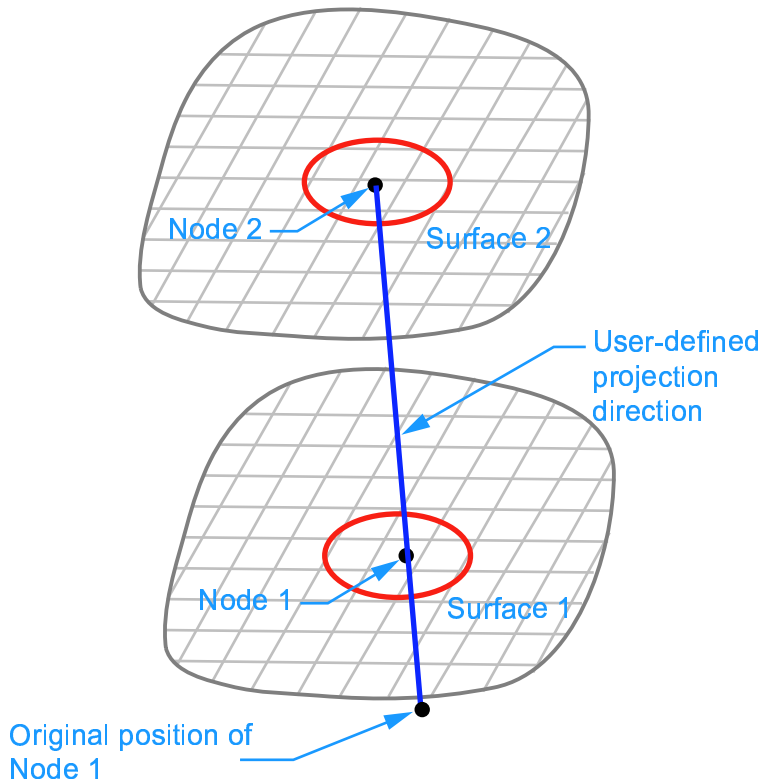
Once the actual location of node 1 has been determined, ANSYS will project node 2 to surface 2 along the spot weld projection direction that was used for node 1. The figure below shows the projection locations of nodes 1 and 2 given a default projection normal to surface 1.

Figure 8.3 Default Projection Direction for Nodes 1 and 2

The figure below shows spot weld node 1 projected as specified by the user (either through the input of *DIRX*, *DIRY*, and *DIRZ*, or by the specification of a second spot weld node).

Figure 8.4 User-specified Projection Direction, Node 1

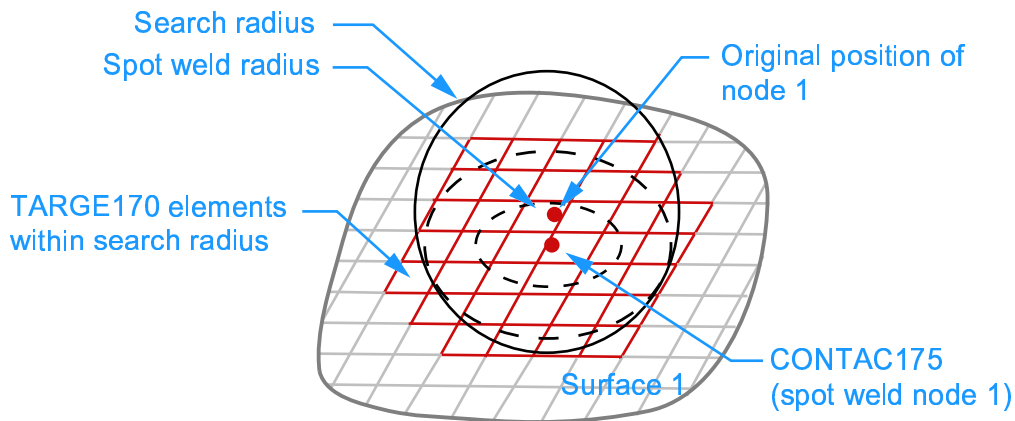
The figure below is another example showing the projection location of spot weld nodes 1 and 2 given a user-specified projection direction.

Figure 8.5 User-specified Projection Direction, Nodes 1 and 2**Spot Weld Radius (*SWRD*):**

Each spot weld is assumed to have a circular projection onto the spot weld surfaces. The projection is defined by the required spot weld radius argument *SWRD*. For each spot weld surface ANSYS will build 6 force-distributed constraint equations through a contact pair. The spot weld radius determines the influence range of the constraint equations. Each spot weld node couples with a group of nodes on the corresponding surface within the radius. (See Figure 8.7: “Nodes Included in Constraint Equations” later in this section.)

Search Radius (*SHRD*):

ANSYS will create a contact pair for each spot weld surface. The contact pair contains one contact element defined by the spot weld node, and target elements overlaid fully or partially on the corresponding spot weld surface. Only the surface nodes that fall within a sphere of radius, called the search radius, with its center at the original position of the corresponding spot weld node are used to form target elements (see figure below). The search radius must be larger than the spot weld radius. If the search radius argument (*SHRD*) is not defined, ANSYS uses a default search radius that is four times the spot weld radius.

Figure 8.6 Search Radius for Spot Weld

Target and Contact Element Types (*ITTY*, *ICTY*):

By default, the program automatically creates the target and contact element types required for the spot weld. However, you may use the *ITTY* and *ICTY* arguments to specify the target and contact element types, respectively. You must also set certain key options if you specify the element types to be used. See Section 8.1.2: The Components of a Spot Weld for more information.

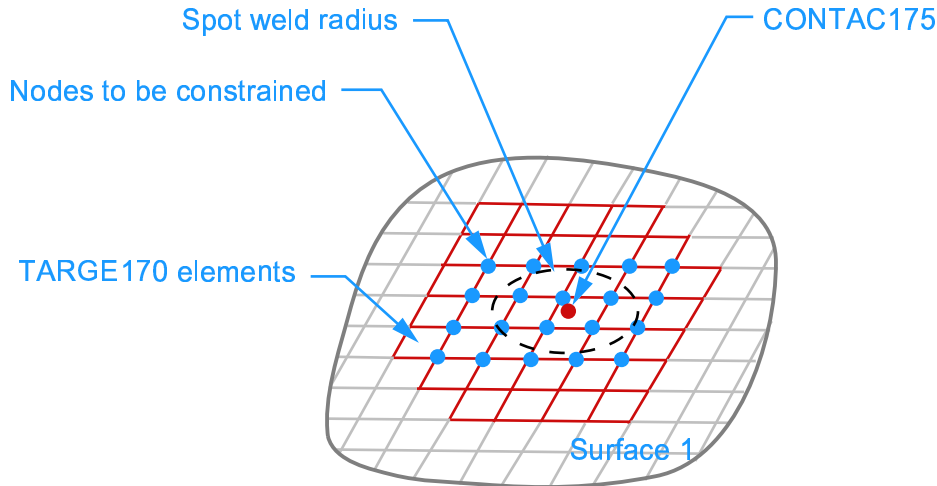
8.1.2. The Components of a Spot Weld

Each new spot weld set created by the **SWGEN** command consists of a beam element and two node-to-surface MPC contact pairs (see Section 7.2: Modeling a Shell-Solid Assembly for more information). Each node-to-surface pair will generate 6 constraint equations. You can choose either a rigid MPC184 element (the default) or a deformable BEAM188 element to link the spot weld surfaces. Below are detailed explanations of how these items are generated:

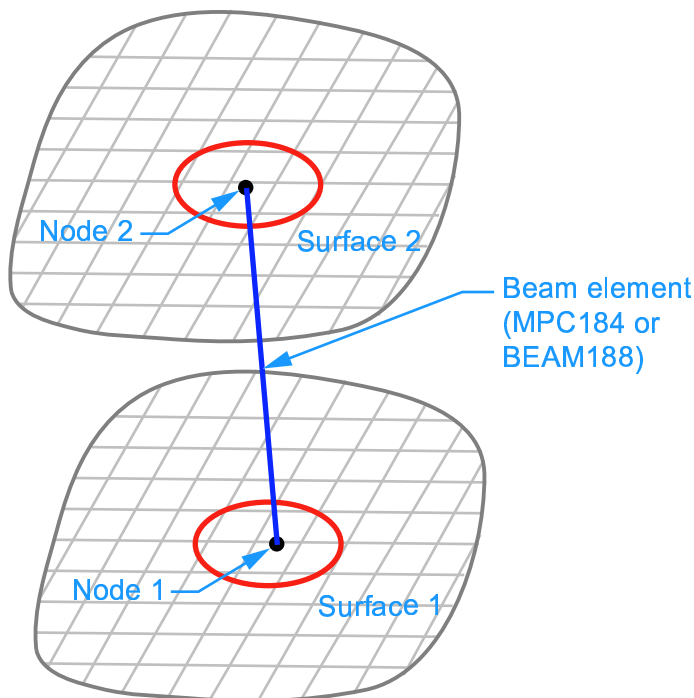
1. **Contact Pairs:** Two node-to-surface contact pairs are created for each new spot weld set. One pair is for spot weld surface 1 and the other is for surface 2. Each contact pair has only one contact element, CONTA175, which is defined by the associated spot weld node. The target elements (TARGE170) are formed by a group of surface nodes lying within the region of the search radius. ANSYS will create an independent real constant ID for each contact pair, and proper contact and target element type IDs.

If you prefer, you can specify the contact or target element type IDs to be used. To specify a contact element type, use the *ICTY* argument on **SWGEN** and set the following contact element options for that element type: KEYOPT(2) = 2 (MPC approach), KEYOPT(12) = 5 (bonded always). To specify a target element type, use the *ITTY* argument on **SWGEN** and set the target element option KEYOPT(5) = 4 (shell-solid constraint) for that element type.

2. **Force-distributed constraint equations:** For each contact pair, ANSYS will internally form force-distributed constraint equations that distribute the internal forces of the spot weld node (that is, the contact node) to the surface nodes (that is, the target nodes) lying within the region of the spot weld radius. This includes all elements that fall entirely or partially within the radius (see the figure below). In other words, it couples the motion of certain surface nodes to the motion of the spot weld node (in an average sense). There are six constraint equations generated for each spot weld surface (that is, each contact pair).

Figure 8.7 Nodes Included in Constraint Equations

3. **Beam element:** A beam element is created to connect spot weld surface 1 and surface 2. The two beam end-nodes are the spot weld nodes, node 1 and node 2 (see Figure below). The spot weld set can be rigid or deformable. By default ANSYS will create a rigid beam element (MPC184 with KEYOPT(1)=1) to represent a rigid spot weld. However, if the currently defined element type is BEAM188 with a solid circular cross-section, ANSYS assumes the spot weld set is deformable and a BEAM188 element is created. In this case, you must input the material properties with the **MP** command, and define the geometry of the beam section using the **SECTYPE** and **SECDATA** commands (see the example listing below). Generally, BEAM188 is capable of handling a short beam situation.

Figure 8.8 Beam Element Created for Spot Weld

The following example input shows typical commands used to define a spot weld with a deformable BEAM188 element having a circular cross section.

```
MP,EX,3,200000000000. ! spot weld material properties
MP,NUXY,3,0.3
SECTYPE,3,beam,csolid ! define a circular solid beam section
SECDATA,2.75e-002      ! beam circular radius
ET,3,188              ! element type for a deformable spot weld
TYPE,3
MAT,3
SECNUM,3
*SET,NODE1,9000       ! define parameter for node number
N,NODE1,0.1,0.5,10.2 ! define spot weld node
SWGEN,SWELD1,2.75e-2,2,3,NODE1 ! Spot weld name = SWELD1
                                ! Radius = 2.75e-2
                                ! Spot weld surfaces = areas 2 and 3
```

Note — All limitations that are documented for the MPC184 link/beam element in the Element Reference also apply when the element is used in a rigid spot weld definition. Please read the MPC184 description carefully, especially MPC184 Assumptions and Restrictions.

8.1.3. Adding Surfaces to a Basic Set

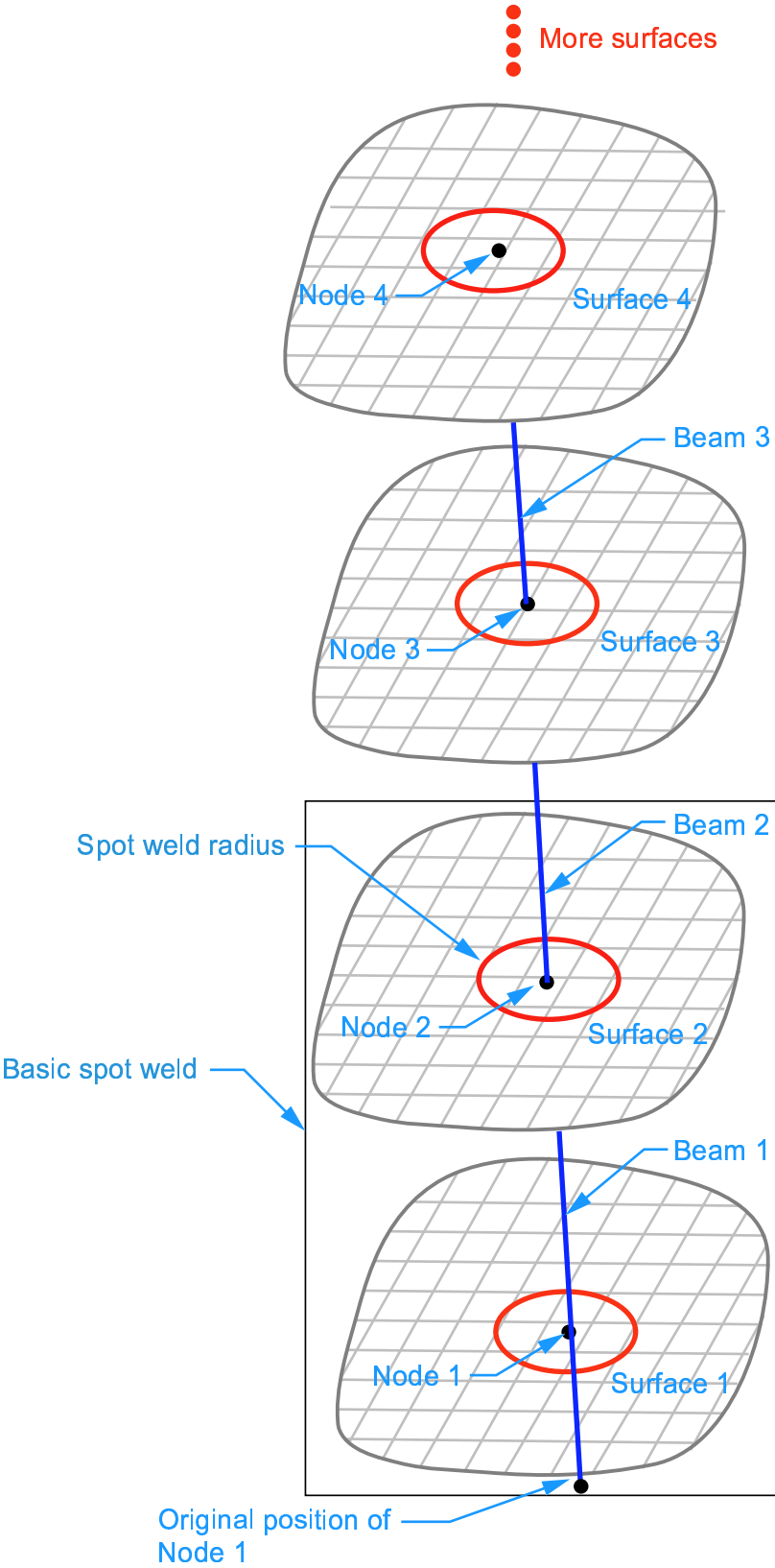
Once a basic spot weld set has been created, additional surfaces can be added to the existing spot weld set. Use the **SWADD** command to define the additional surfaces (in the GUI, pick **Main Menu > Preprocessor > Modeling > Create > Element > SpotWeld > Add More Surfaces**). Each added surface (input as *NCM1* through *NCM9*) can be specified by an existing nodal component name or by a meshed area number.

In the example command below, four surfaces are added to an existing spot weld set named SWELD1. The four surfaces include area number 3, area number 4, nodal component FAC5, and nodal component FAC6. The search radius is 0.8.

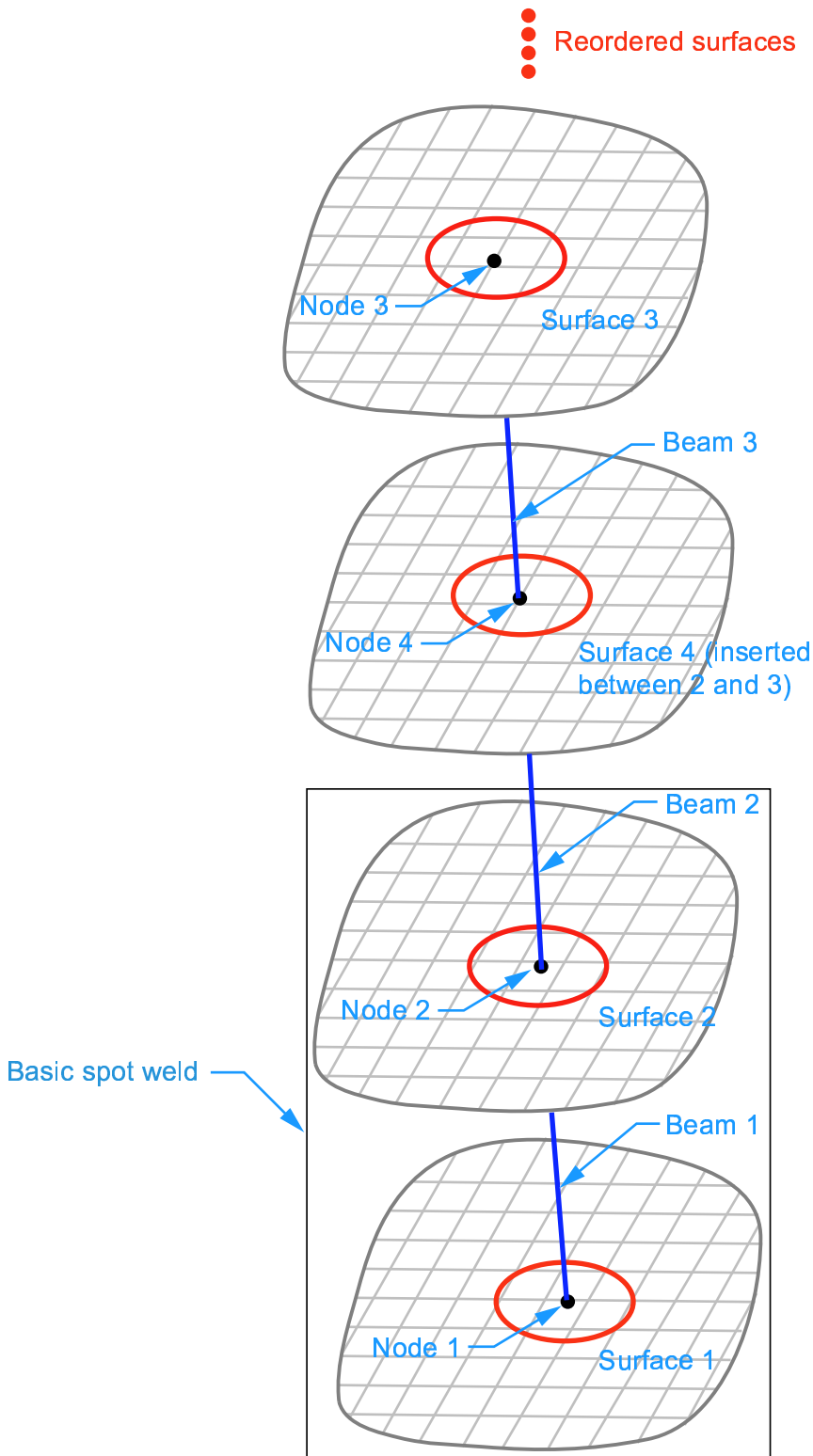
```
SWADD,SWELD1,0.8,3,4,FAC5,FAC6 ! Adds 4 surfaces to spot weld SWELD1.
```

For each new surface, ANSYS will create a new spot weld node, a new contact pair, and a new beam element (see figure below). The location of the new node is determined by projection of one of the closest existing spot weld nodes onto the newly added surface along the spot weld projection direction. The new node-to-surface contact pair contains the new spot weld node as a single contact node in the pair, and surface nodes within the range of the search radius (*SHRD*) specified on the **SWADD** command. The search radius defaults to 4 times the spot weld radius (*SWRD*) defined for the basic spot weld set. Each new contact pair will create 6 force-distributed constraint equations internally. A new beam is created to link the new surface to the closest existing surface through the spot weld nodes.

Figure 8.9 Surfaces Added to Basic Spot Weld Set



After all spot weld nodes are created, the nodes of the beam elements may be reordered to link every two adjacent spot weld nodes (see Figure below). The ordering is important for the output, which is calculated for each beam in a spot weld set.

Figure 8.10 Node Reordering for Beam Elements

Assuming the basic spot weld set is named SWELD, below is an example of commands that could have produced the above figure.

```
SWADD,SWELD, ,3 ! Add Area 3 to spot weld set SWELD
SWADD,SWELD, ,4 ! Add Area 4 (between surfaces 2 and 3) to SWELD
```


Initially, when spot weld surface 3 was added, a beam was created between surfaces 2 and 3. When surface 4 was added between surfaces 2 and 3, the beam that went between surfaces 2 and 3 was redefined to go between surfaces 2 and 4, and a new beam was created to link surfaces 4 and 3.

You can add more surfaces by repeating the **SWADD** command. However, the maximum number of allowable surfaces for each spot weld set (including the two surfaces used for the basic set) is 11.

8.2. Listing and Deleting Spot Welds

Use the **SWLIST** command to list the spot weld sets for a specific spot weld or for all spot weld sets (in the GUI, pick **Main Menu > Preprocessor > Modeling > Create > Element > SpotWeld > List**). The listing will include nodes, beams, and contact pair information. If the **SWLIST** command is issued in POST1 (in the GUI, pick **Main Menu > General Postproc > List Results > SpotWeld Solution**), the beam forces and moments, are also output in the beam local coordinate system. The output includes axial force, shear forces, torque, and bending moments. For a deformed spot weld set, axial stress and bending stress are also reported.

Use the **SWDEL** command to delete a specified spot weld set or all spot weld sets. In the GUI, pick **Main Menu > Preprocessor > Modeling > Create > Element > SpotWeld > Delete**.