ANSYS Mechanical Getting Started

Module 05 Student Reference Guide: Connections

Release 2023 R1

Please note:
- These training materials were developed and tested in Ansys Release 2023 R1. Although they are expected to behave similarly in later releases, this has not been tested and is not guaranteed.
- The screen images included with these training materials may vary from the visual appearance of a local software session.

Module 05: Learning Objectives

Upon successful completion of this module, the student should be able to:

- Explain the purpose of connections in ANSYS Mechanical
- Select, review, and modify individual contact regions
- Explain the purpose and limitations of Auto Detection
- Understand the importance of verifying each contact region in the model
- Describe the physical meaning of the Bonded contact type

Module 05: Learning Objectives

Upon successful completion of this module, the student should be able to:

- Understand the basic geometry definition of a contact region: red and blue surfaces, contact and target surfaces
- Describe the behavior of these ease-of-use features:
  - Body Views
  - RMB—Go To…
  - Contact Tool
- Understand the purpose of the Pinball Region

Module 05: Graphics

Module 05: Reference Material

- Setting Connections https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Contact.html

Setting Connections

Supported connection features consist of Contact, Joint, Spring, Beam Connection, End Release, Spot Weld and Body Interaction (Explicit Dynamics only). Each of these connections can be created manually in the application. Only Contact and Joint can also be generated automatically.

This section describes **Connections** folder, **Connection Group** folder, Automatic Generated Connections, as well as each connection type as outlined below.

- Connections Folder
- Connections Worksheet
- Connection Group
- Connection Features and Operations
- Contact
- Joints
- Springs
- Beam Connections
- Spot Welds
- End Releases
- Bearings

« TRANSFERRING COORDINATE SYSTEMS TO THE MECHANICAL APDL APPLICATION

CONNECTIONS FOLDER »

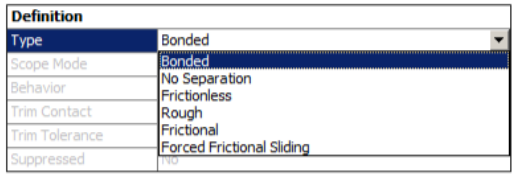
Module 05: Reference Material

- Contact Type https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Contact_Definition.html

Type

Choosing the appropriate contact type depends on the type of problem you are trying to solve. If modeling the ability of bodies to separate or open slightly is important and/or obtaining the stresses very near a contact interface is important, consider using one of the nonlinear contact types (**Frictionless**, **Rough**, **Frictional**), which can model gaps and more accurately model the true area of contact. However, using these contact types usually results in longer solution times and can have possible convergence problems due to the contact nonlinearity. If convergence problems arise or if determining the exact area of contact is critical, consider using a finer mesh (using the Sizing control) on the contact faces or edges.

The available contact types are listed below. Most of the types apply to Contact Regions made up of faces only.



- **Bonded**: This is the default configuration and applies to all contact regions (surfaces, solids, lines, faces, edges). If contact regions are bonded, then no sliding or separation between faces or edges is allowed. Think of the region as *glued*. This type of contact allows for a linear solution since the contact length/area will not change during the application of the load. If contact is determined on the mathematical model, any gaps will be closed and any initial penetration will be ignored. [Not supported for Rigid Dynamics. Fixed joint can be used instead.]
- **No Separation**: This contact setting is similar to the **Bonded** case. It only applies to regions of faces (for 3D solids) or edges (for 2D plates). Separation of the geometries in contact is not allowed.
- **Frictionless**: This setting models standard unilateral contact; that is, normal pressure equals zero if separation occurs. Therefore, gaps can form in the model between bodies depending on the loading. This solution is nonlinear because the area of contact may change as the load is applied. A zero coefficient of friction is assumed and allows free sliding. The model should be well constrained when using this contact setting. Weak springs are added to the assembly to help stabilize the model in order to achieve a reasonable solution.
- **Rough**: Similar to the frictionless setting, this setting models perfectly rough frictional contact where there is no sliding. It only applies to regions of faces (for 3D solids) or edges (for 2D plates). By default, no automatic closing of gaps is performed. This case corresponds to an infinite friction coefficient between the contacting bodies. [Not supported for Explicit Dynamics analyses.]

Module 05: Reference Material

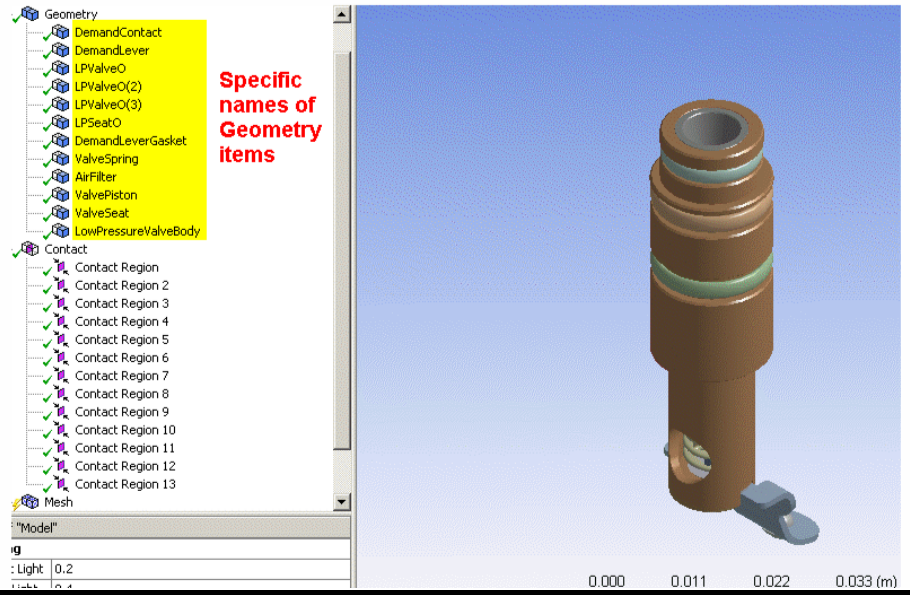
- Renaming Contact Regions Based on Geometry Names
  https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Renaming_Using_Geometry.html

Renaming Contact Regions Based on Geometry Names

You can change the name of any contact region using the following choices available in the context menu that appears when you click the right mouse button on a particular contact region:

- **Rename**: Enables you to change the contact region name to a name that you type (similar to renaming a file in Windows Explorer).
- **Rename Based on Definition**: Enables you to change the contact region name to include the corresponding names of the items in the **Geometry** branch of the tree that make up the contact region. The items are separated by the word "To" in the new contact region name. You can change all the contact region names at once by clicking the right mouse button on the **Connections** branch, then choosing **Rename Based on Definition** from that context menu. A demonstration of this feature follows.

The following demo is presented as an animated GIF. View online if you are reading the PDF version of the help. Interface names and other components shown in the demo may differ from those in the released product.

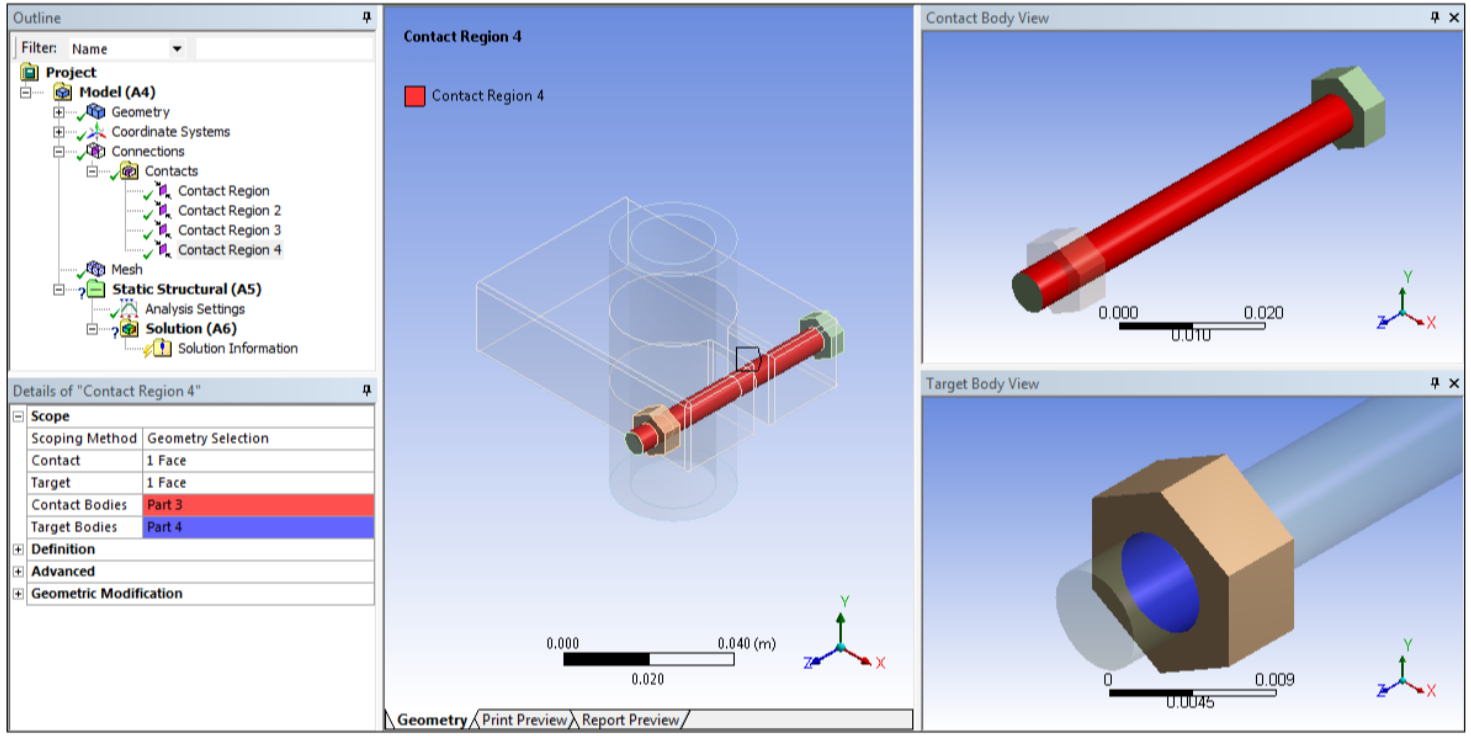


Module 05: Reference Material

- Displaying Contact Bodies in Separate Windows
  https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_contact_body_views.html

Displaying Contact Bodies in Separate Windows

By default, the **Body Views** option on the Connections Context Tab, is active to display parts and connections in separate auxiliary windows. As illustrated below, the different contact bodies (**Contact** and **Target**) have colors codes associated with them, in the **Details** view as well as the graphic windows. The Body Views feature is available for joints, contacts, springs, and beam connections. This display not only highlights the geometric entities in contact, but displays the bodies of the corresponding part(s).



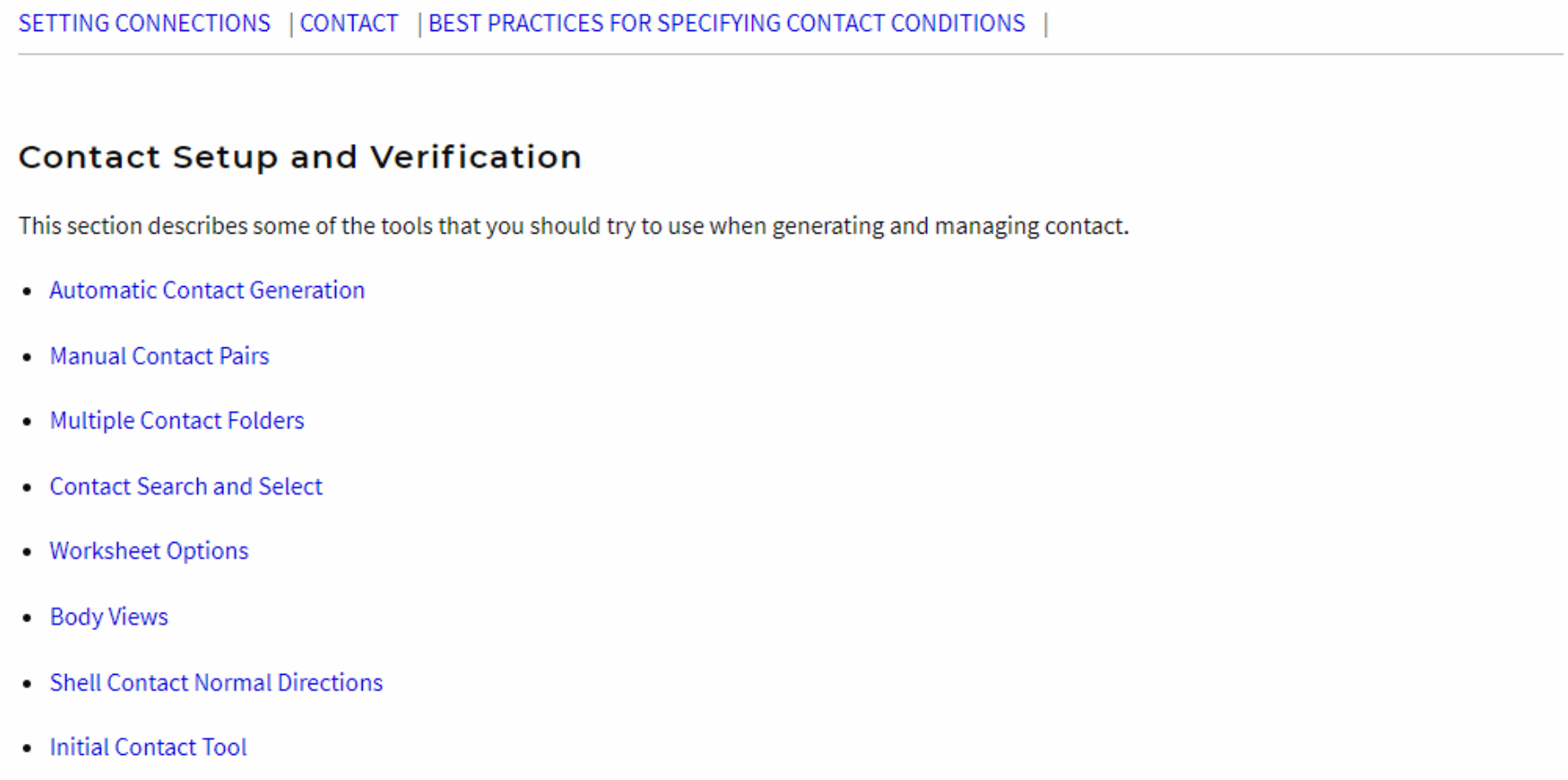
Module 05: Reference Material

- Correlating Tree Outline Objects with Model Characteristics https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_tree_go_to.html

Go To Option	Description / Application	Required Conditions for Option to Appear
Corresponding Bodies in Tree	Identifies body objects in the tree that correspond to selections in the **Geometry** window.	At least one vertex, edge, face, or body is selected.
Hidden Bodies in Tree	Identifies body objects in the tree that correspond to hidden bodies in the **Geometry** window.	At least one body is hidden.
Suppressed Bodies in Tree	Identifies body objects in the tree that correspond to suppressed bodies in the **Geometry** window.	At least one body is suppressed.
Bodies Without Contacts in Tree	Identifies bodies that are not in contact with any other bodies. When you are working with complex assemblies of more than one body, it is helpful to find bodies that are not designated to be in contact with any other bodies, as they generally cause problems for a solution because they are prone to rigid body movements.	More than one body in an assembly.
Parts Without Contacts in Tree	Identifies parts that are not in contact with any other parts. When you are working with complex assemblies of more than one multibody part, it is helpful to find parts that are not designated to be in contact with any other parts. For example, this is useful when dealing with shell models which can have parts that include many bodies each. Using this feature is preferred over using the **Bodies Without Contact in Tree** option when working with multibody parts mainly because contact is not a typical requirement for bodies within a part. Such bodies are usually connected by shared nodes at the time of meshing.	More than one part in an assembly.

Module 05: Reference Material

- Initial Contact Tool
  https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_contact_best_prac_setup.html



SETTING CONNECTIONS | CONTACT | BEST PRACTICES FOR SPECIFYING CONTACT CONDITIONS |

Contact Setup and Verification

This section describes some of the tools that you should try to use when generating and managing contact.

- Automatic Contact Generation
- Manual Contact Pairs
- Multiple Contact Folders
- Contact Search and Select
- Worksheet Options
- Body Views
- Shell Contact Normal Directions
- Initial Contact Tool

Module 05: Reference Material

- Pinball Radius https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v231/en/wb_sim/ds_Contact_Advanced.html

Rigid Dynamics Solver

For the Rigid Dynamics solver, the **Pinball Region** property is used to control the touching tolerance. By default, the Rigid Dynamics solver automatically computes the touching tolerance using the sizes of the surfaces in the contact region. These default values are sufficient in most of cases, but inadequate touching tolerance may arise in cases where contact surfaces are especially large or small (small fillet for instance). In such cases, the value of the touching tolerance can be directly specified using the following properties:

Option	***Description***
Program Controlled (default)	The touching tolerance is automatically computed by the Rigid Body Dynamics solver from the sizes of the contact surfaces.
Radius	The value of the touching tolerance is directly given by user.

Pinball Radius

The numerical value for the **Pinball Radius**. This property is displayed only if **Pinball Region** is set to **Radius**.

End of presentation