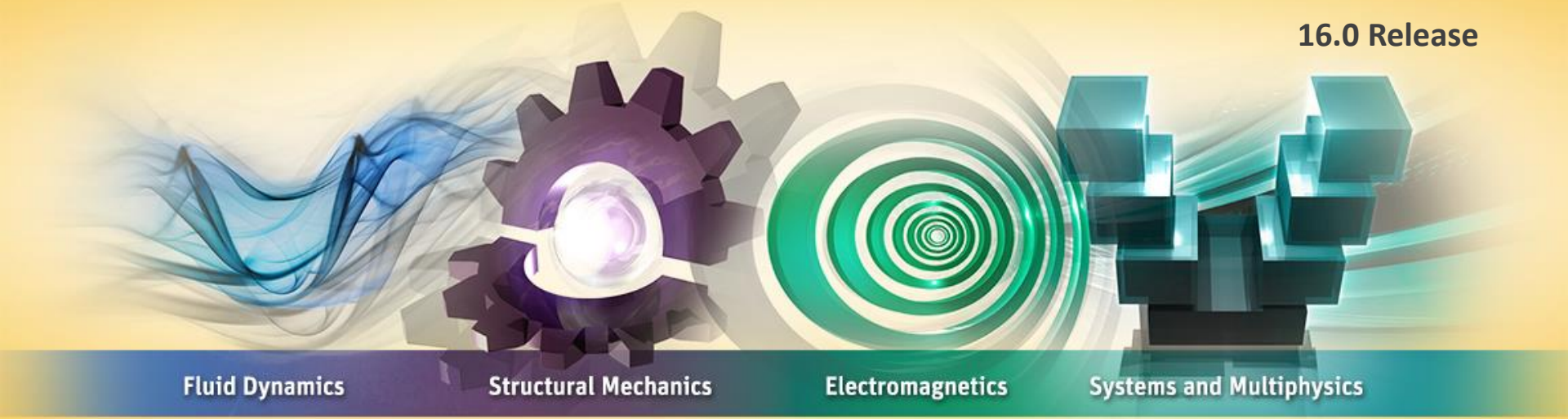
Lecture 5
Modeling Connections

16.0 Release



Introduction to ANSYS Mechanical

Realize Your Product Promise®

Chapter Overview

In this chapter, we will extend the discussion of contact control begun previously in this course. We also introduce the mesh connection capability for use with surface models:

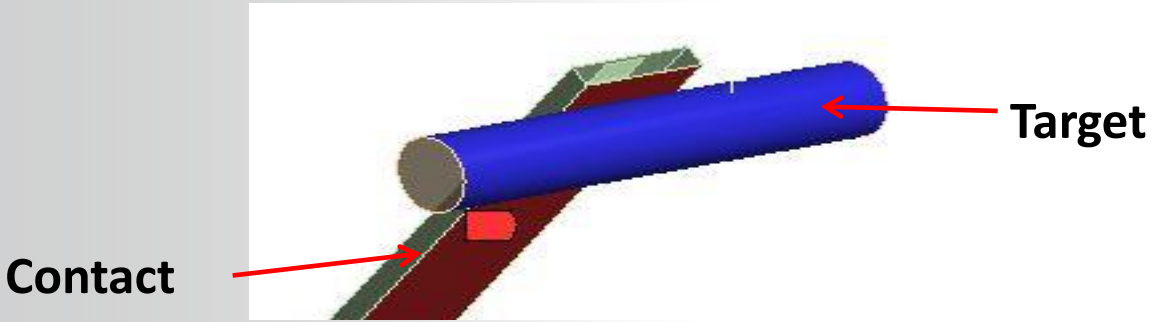
A. Contact
B. Contact Controls
C. Contact Results
D. Spot Welds
E. Mesh Connections
F. Connections Worksheet
G. Workshop 5.1 - Contact Offset Control
H. Joint Definitions
I. Springs and Beams
J. Workshop 5.2 - Using Joints
K. Appendix

© 2015 ANSYS, Inc.

A. Contact

In this section we will touch on some of the concepts relating to contact analysis. Keep in mind, however, contact could be a highly nonlinear feature (status changing) and is covered in its entirety in the Mechanical Nonlinearities training course.

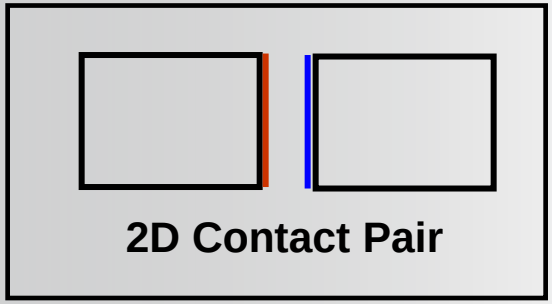
- **Contact elements can be thought of as a "skin" covering the surfaces that are expected to interact with one another.**
- **One side of a contact pair is referred to as the "contact" and its mate as the "target".**
- **Mechanical uses a color coding system to differentiate the contact and target surfaces.**



 © 2015 ANSYS, Inc.

. . . Contact

- One side of a contact pair is referred to as a contact surface, the other side is referred to as a target surface.
  - Contact and target scoping does not need to be equal. For example, a contact can be scoped to 2 faces while its target is scoped to 5 faces.
- Contact pairs are color coded in the details and on the geometry.



Details of "Bonded - TopPlate To BottomPlate"

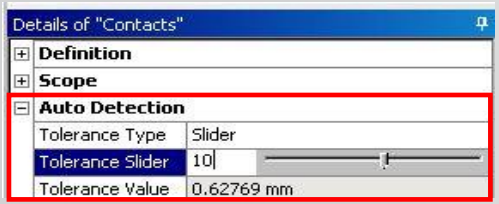
Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	4 Faces
Contact Bodies	TopPlate
Target Bodies	BottomPlate
Definition	
Type	Bonded
Scope Mode	Manual
Behavior	Program Controlled
Trim Contact	Program Controlled
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Program Controlled
Geometric Modification	
Contact Geometry Correction	None
Target Geometry Correction	None

© 2015 ANSYS, Inc.

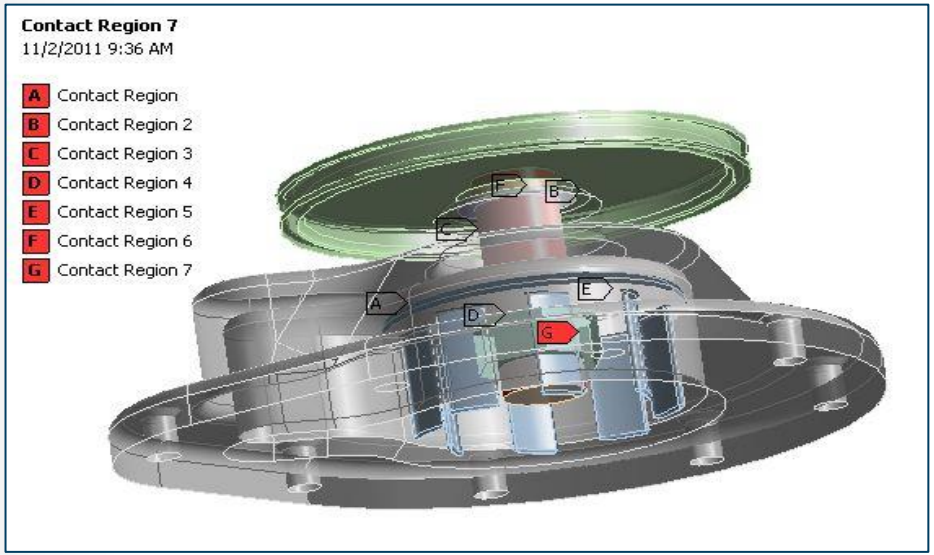
. . . Contact

Contact regions are automatically created between parts during assembly import.

- **Contacts are contained in the Connections branch and can be grouped in multiple “Contacts” folders.**
  - **Contact detection tolerance controls are available (low = loose tolerance; high = tight tolerance).**



Loose **Tight**

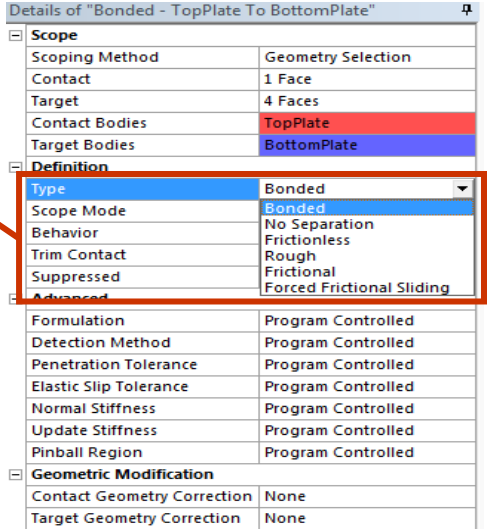


© 2015 ANSYS, Inc.

. . . Contact

Five contact behaviors are available:

Contact Type	Iterations	Normal Behavior (Separation)	Tangential Behavior (Sliding)
Bonded	1	No Gaps	No Sliding
No Separation	1	No Gaps	Sliding Allowed
Frictionless	Multiple	Gaps Allowed	Sliding Allowed
Rough	Multiple	Gaps Allowed	No Sliding
Frictional	Multiple	Gaps Allowed	Sliding Allowed



- ***Bonded* and *No Separation* contact are linear and require only 1 iteration.**
  - **Bonded: surfaces are fixed to one another so no gaps can open and no sliding takes place.**
  - **No Separation: no gaps can open however small sliding can take place.**

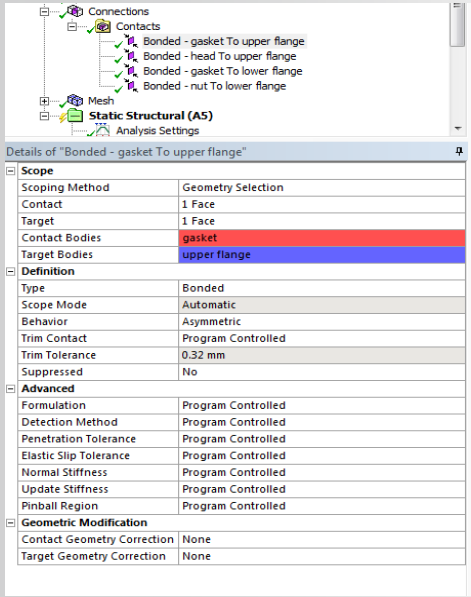
- ***Frictionless, Rough and Frictional* contact are nonlinear and require multiple iterations. *These contact types will be introduced here but detailed fully in the ANSYS Mechanical Structural Nonlinearities training course.***

© 2015 ANSYS, Inc.

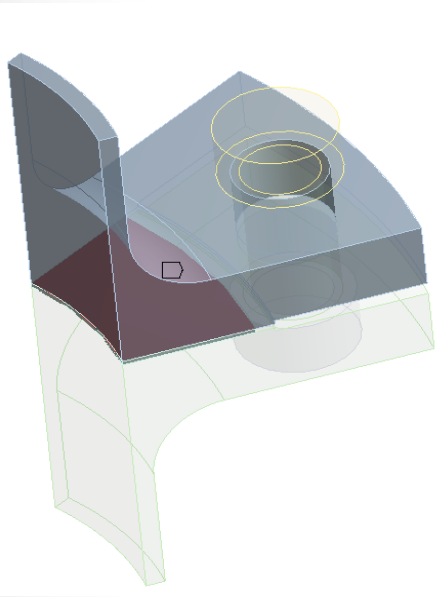
. . . Contact

When a contact region is highlighted in the connections branch, parts are made translucent for easier viewing.

- **Contact surfaces are color coded for easy identification.**



Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	gasket
Target Bodies	upper flange
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Asymmetric
Trim Contact	Program Controlled
Trim Tolerance	0.32 mm
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Program Controlled
Geometric Modification	
Contact Geometry Correction	None
Target Geometry Correction	None

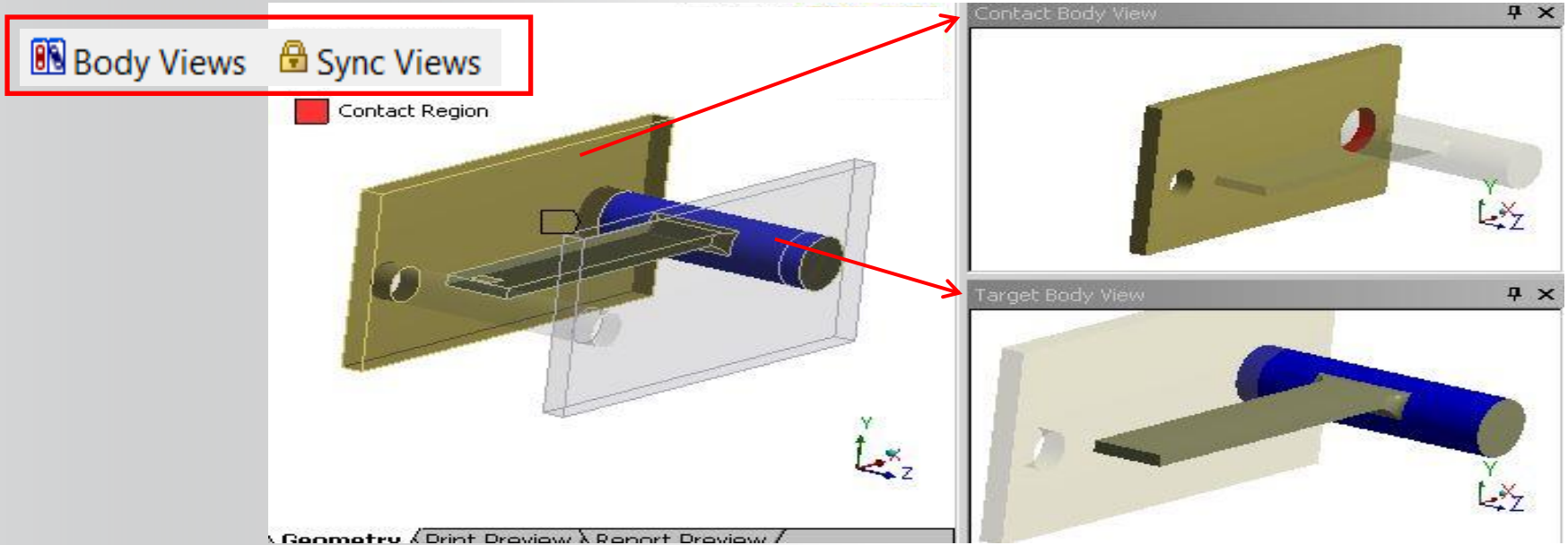


© 2015 ANSYS, Inc.

. . . Contact

For ease of viewing or selecting, “Body Views” can be activated:

- **Separate windows display the full model, contact body and target body.**
- **Views can be “synched” (all windows move together).**
- **Selecting (for contact scoping) can be done in any window.**

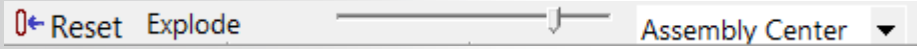


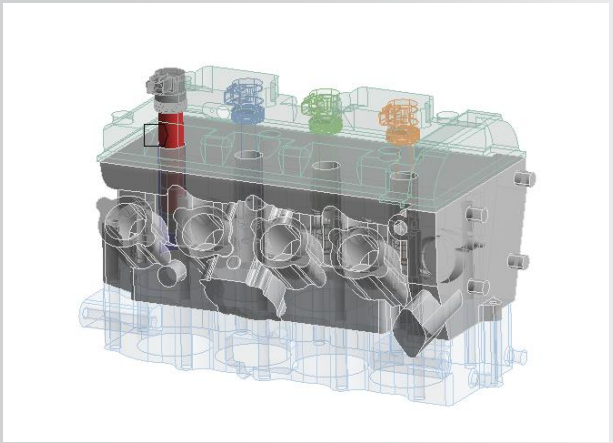
© 2015 ANSYS, Inc.

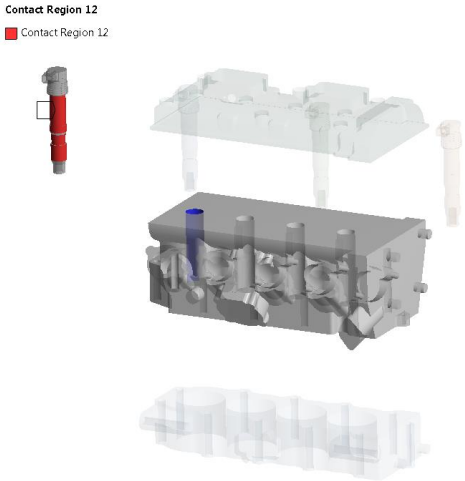
. . . Contact

For ease of viewing, user can « explode » assembly model

- **Use slider to modify how are seperate different parts**
- **User can choose the coordinate system**

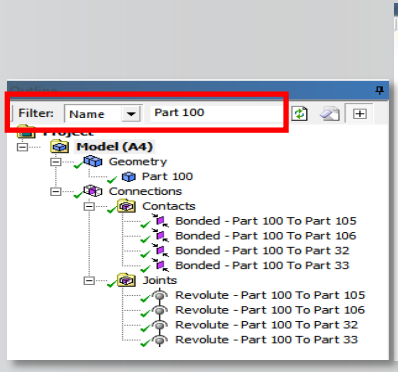


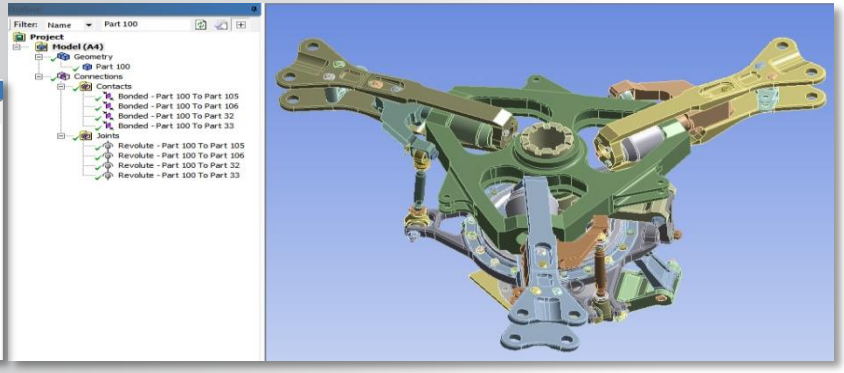


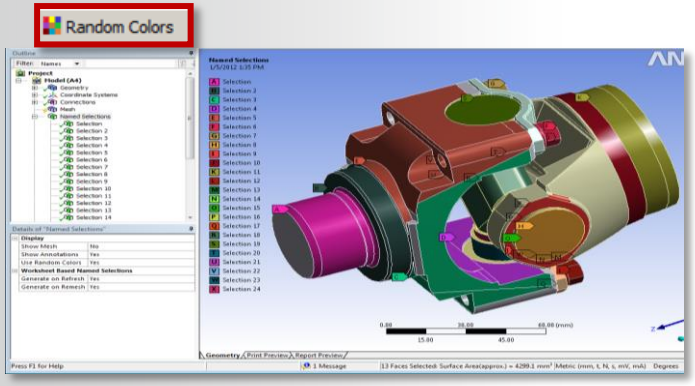


© 2015 ANSYS, Inc.

. . . Contact







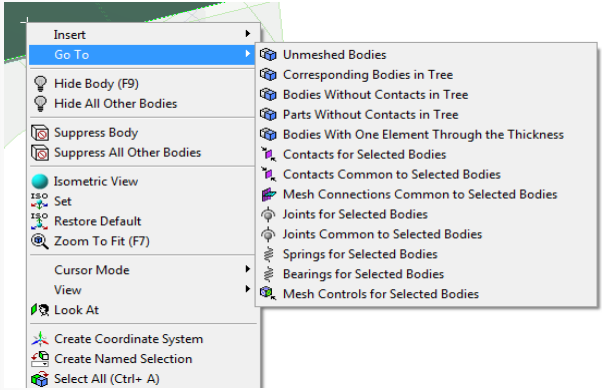
To improve the readability of models, filters based on name are available as well as unique random colors for the display of numerous loads, boundary conditions or named selections.

© 2015 ANSYS, Inc.

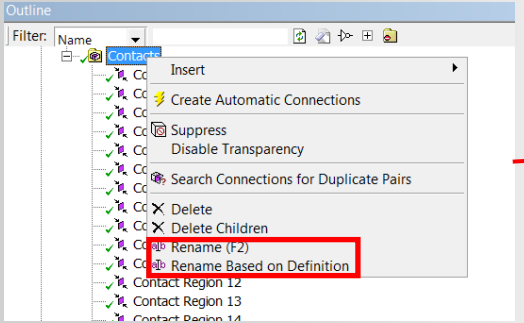
. . . Contact

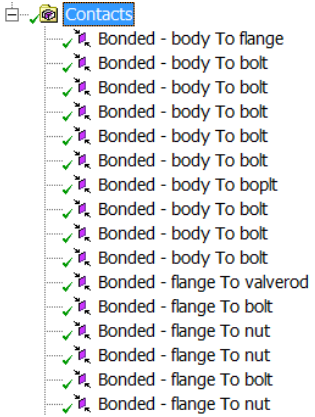
"Go To" utilities provide a simple way of verifying contact definitions:

- **Bodies without contact**
- **Parts without contact**
- **Contact regions for selected bodies**
- **Contacts common to selected bodies**
- **Corresponding bodies in tree**



- **Contacts can be quickly renamed to match part names**





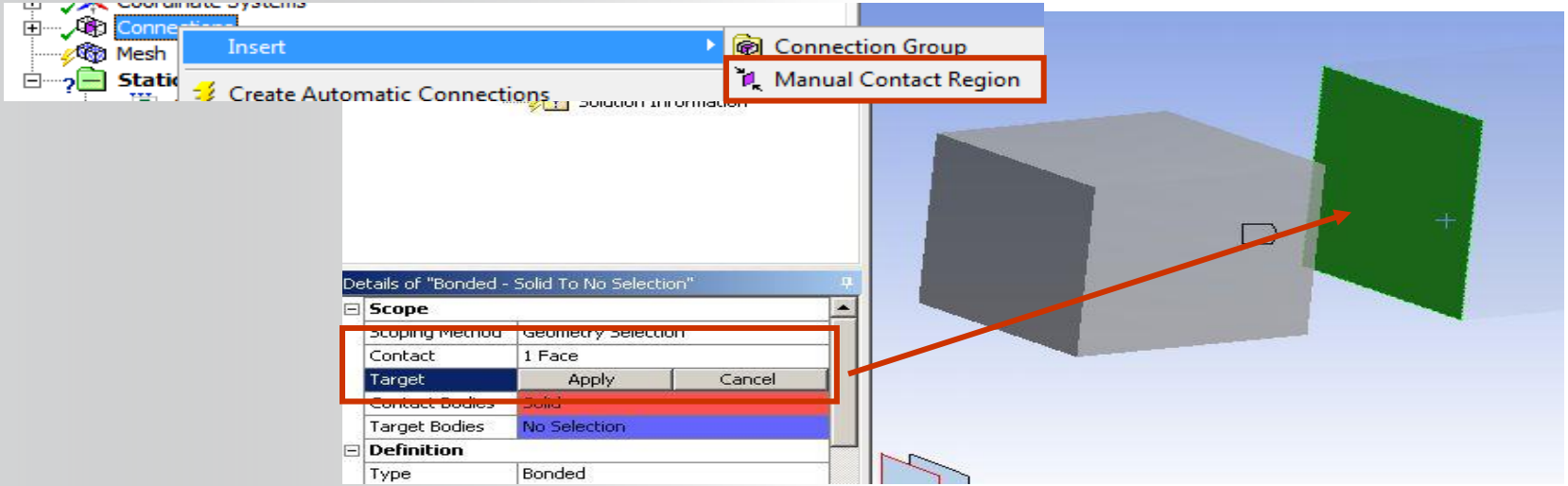
 © 2015 ANSYS, Inc.

. . . Contact

Where surfaces are not automatically detected a manual contact pair can be defined.

- **Insert a manual contact region and select the “contact” and “target” surfaces.**

RMB



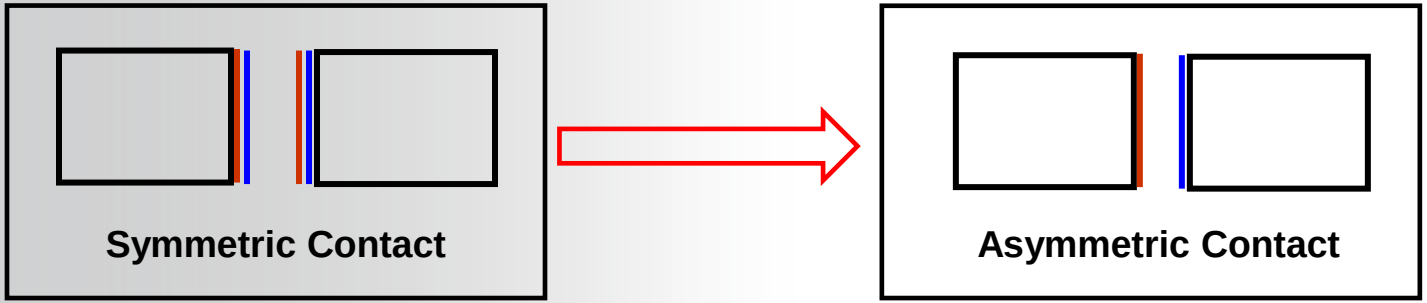
 © 2015 ANSYS, Inc.

B. Contact Controls

In Mechanical one side of a contact pair is referred to as the *contact* while the other is referred to as the *target.*

- **By default (Program Chosen) Mechanical uses what is called *auto-asymmetric* contact.**
- **In some situations there is a "preferred" for which side is designated the contact versus the target. Since the solver does not determine this preference ahead of time, initially a contact pair is duplicated (*symmetric contact).* When the solver detects the preferred arrangement one of the contact pairs is removed. This is called *asymmetric contact*.**

Details of "Frictionless - PumpHousing To Impeller"	
Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	PumpHousing
Target Bodies	Impeller
Definition	
Type	Frictionless
Scope Mode	Automatic
Behavior	Program Controlled
Trim Contact	Program Controlled
Trim Tolerance	0.79243 mm
Suppressed	No
Advanced	

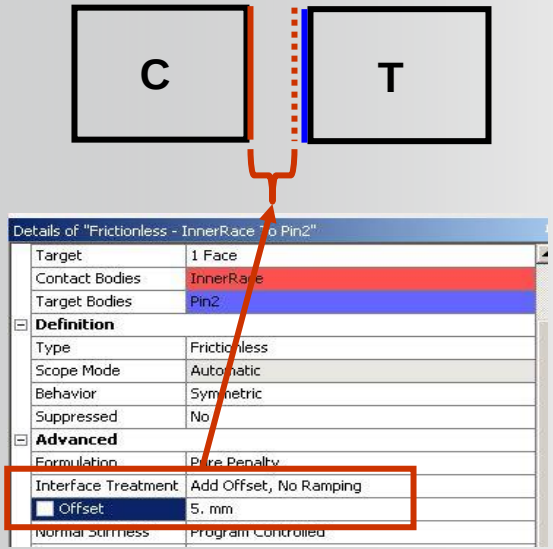


 © 2015 ANSYS, Inc.

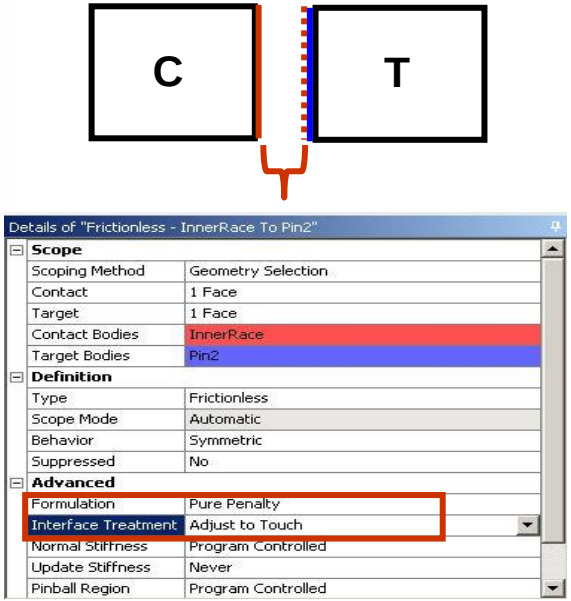
. . . Contact Controls

Nonlinear contact types allow an “interface treatment” option:

- **“Add Offset”: input zero or non-zero value for initial adjustment.**
- **“Adjusted to Touch”: ANSYS closes any gap to a just touching position**



Add offset: positive or negative can be ramped on.



Adjusted to touch

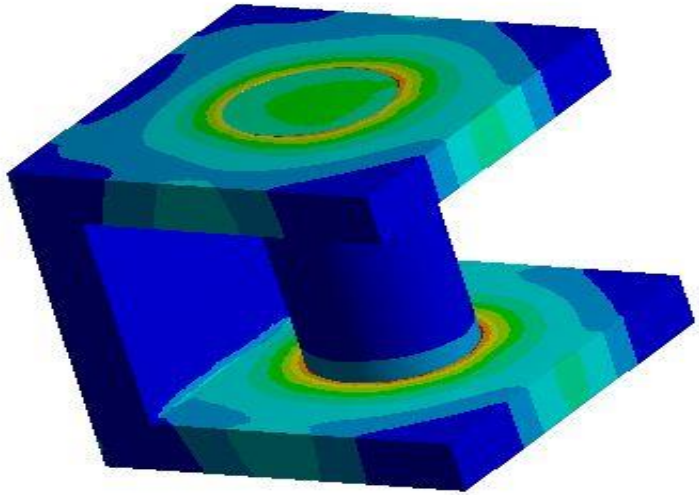
© 2015 ANSYS, Inc.

. . . Contact Controls

Interface treatment example modeling a press fit:

The geometry model contains a pin in a hole which are of the same diameter. Contact offset is used to simulate a press fit of an over sized pin. The resulting stress plot is shown.

Details of "Frictionless - Solid To Solid"	
⊞ **Scope**	
⊞ **Definition**	
⊟ **Advanced**	
Formulation	Program Controlled
Detection Method	Program Controlled
Interface Treatment	Add Offset, No Ramping
Offset	0.5 mm



 © 2015 ANSYS, Inc.

. . . Contact Controls

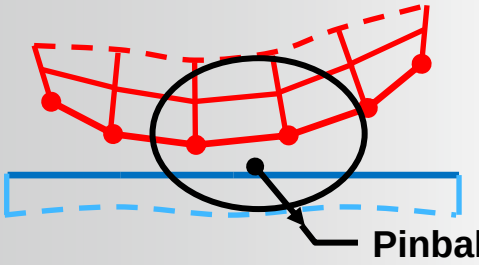
The Pinball Region is a zone that designates far field or near field open contact status (inside or outside the radius). It can be thought of as a zone bordering each contact region.

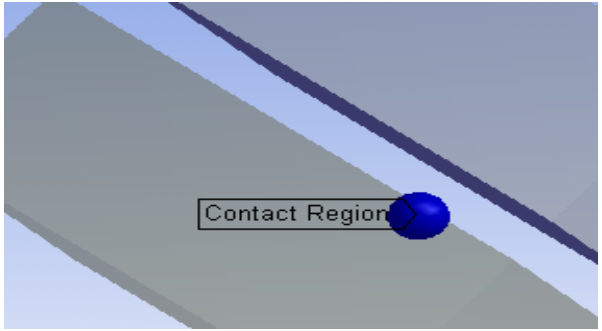
The pinball's main purpose is to provide efficiency when solving contacts which are "far" apart. For most applications simply use the program controlled setting.

During this course we'll point out several situations where it may be useful to manually control the pinball radius.

Details of "Bonded - gasket To upper flange"

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	1 Face
Contact Bodies	gasket
Target Bodies	upper flange
Definition	
Type	Bonded
Scope Mode	Automatic
Behavior	Asymmetric
Trim Contact	Program Controlled
Trim Tolerance	0.32 mm
Suppressed	No
Advanced	
Formulation	Program Controlled
Detection Method	Program Controlled
Penetration Tolerance	Program Controlled
Elastic Slip Tolerance	Program Controlled
Normal Stiffness	Program Controlled
Update Stiffness	Program Controlled
Pinball Region	Radius
Pinball Radius	0. mm





 © 2015 ANSYS, Inc.

. . . Contact Controls

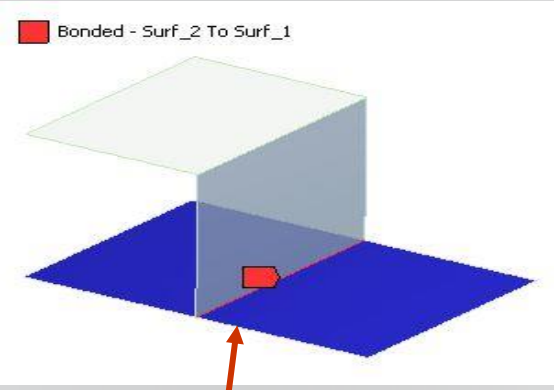
Shell contact includes face-to-face, edge-to-face or edge-to-edge contact:

- **Automatic shell contact is not turned on by default but can be set to detect face-to-edge or edge-to-edge contact.**
- **Priority can be set to prevent multiple contact regions in a given region.**

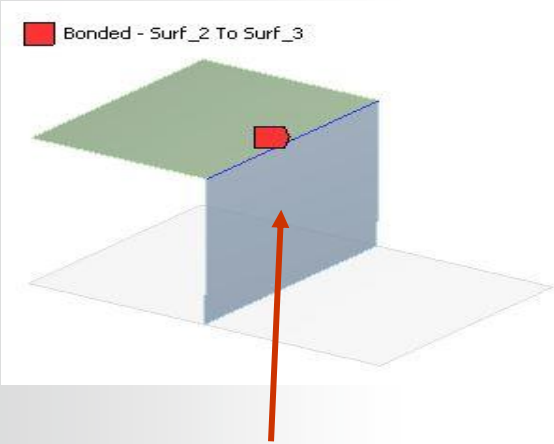
Contacts

Details of "Contacts"

Definition	
Connection Type	Contact
Scope	
Scoping Method	Geometry Selection
Geometry	All Bodies
Auto Detection	
Tolerance Type	Slider
Tolerance Slider	0.
Tolerance Value	1.9866 mm
Use Range	No
Face/Face	Yes
Face/Edge	No
Edge/Edge	No
Priority	Include All
Group By	Bodies
Search Across	Bodies
Statistics	
Connections	46
Active Connections	10



Edge to Surface



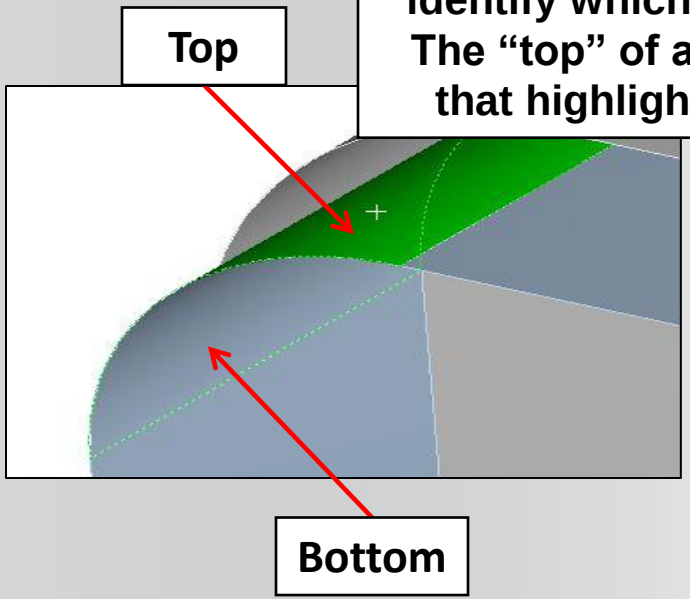
Edge to Edge

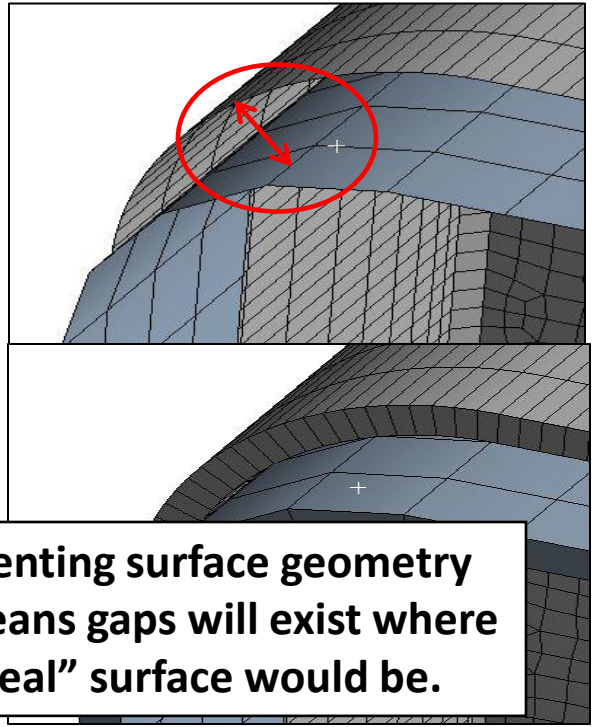
 © 2015 ANSYS, Inc.

. . . Contact Controls

Several unique aspects of surface geometry must be addressed when using contact:

With a single face it's important to identify which side is to be used. The "top" of a surface is the one that highlights when selected.





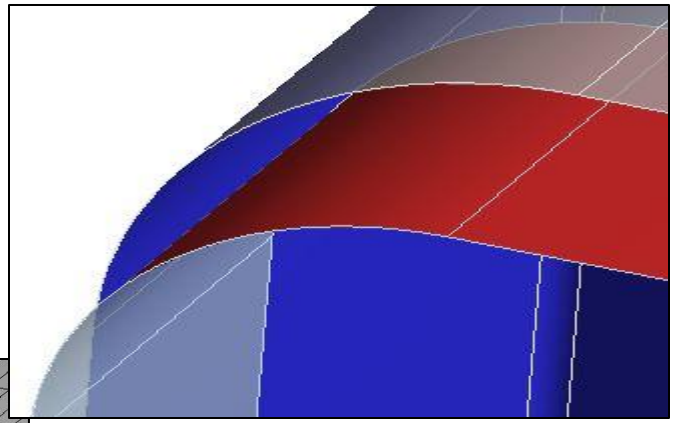
Representing surface geometry often means gaps will exist where the "real" surface would be.

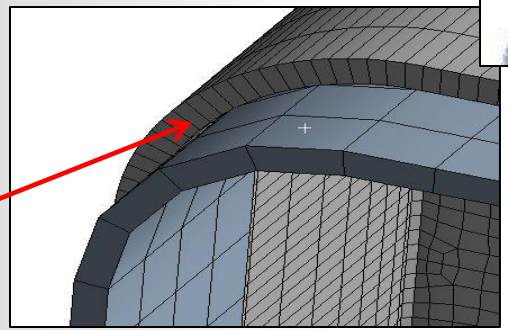
© 2015 ANSYS, Inc.

. . . Contact Controls

The details for surface contact contain controls for assigning contact to the top or bottom of a shell as well as including the effect of the thickness of the shell. Including the thickness effect here means the gap will be ignored and the surfaces will behave as if they were in contact.

Details of "Bonded - 1 To 1"	
Scope	
Scoping Method	Geometry Selection
Contact	8 Faces
Target	17 Faces
Contact Bodies	1
Target Bodies	1
Contact Shell Face	Top
Target Shell Face	Bottom
Shell Thickness Effect	Yes





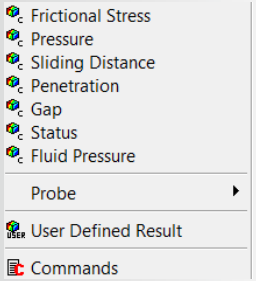
In this example by activating the shell thickness effect we are assuming the initial configuration is as shown here. Just touching

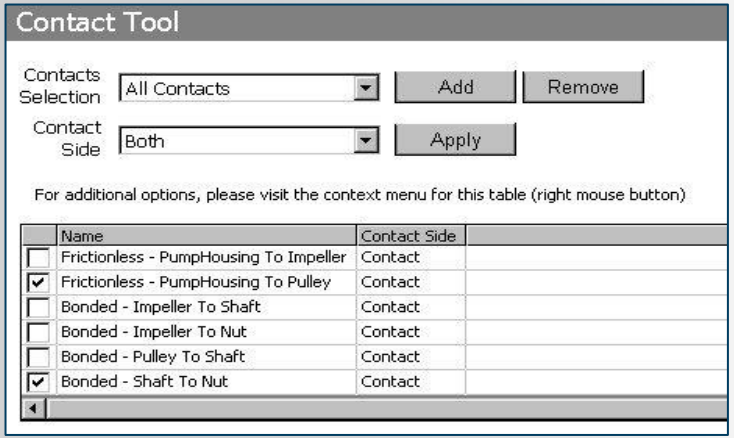
 © 2015 ANSYS, Inc.

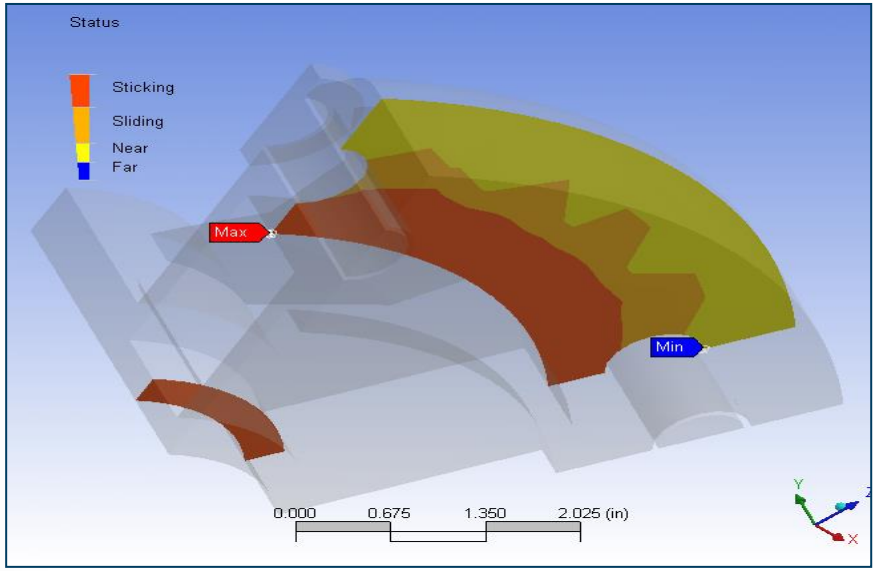
C. Contact Results

Contact specific results are requested via a “Contact Tool”.

- **Geometry selection or a worksheet (shown below) can be used to choose the contacts of interest.**







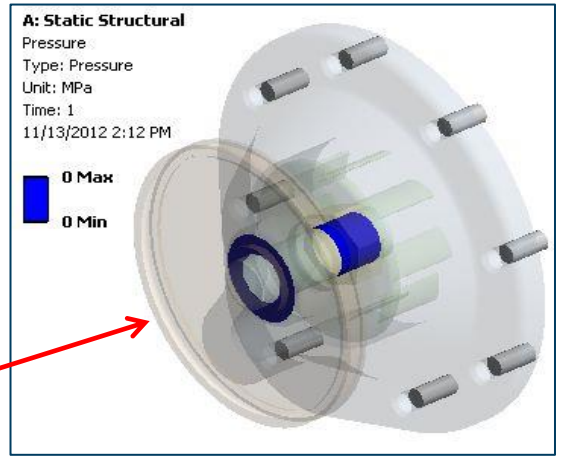
ANSYS License	Availability
Design Space	
Professional	x
Structural	x
Mechanical/Multiphysics	x

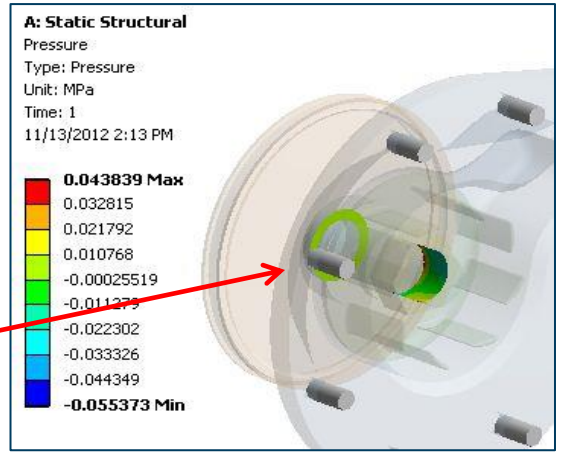
© 2015 ANSYS, Inc.

. . . Contact Results

Contact results are displayed on the contact side only. With auto-asymmetric contact, since the solver chooses which side is designated contact it may not be obvious at the outset which side will display the results.

In this example the original contact/target designation was flipped by the solver.
A zero result will be displayed on the target side of a contact pair as shown in the top figure.



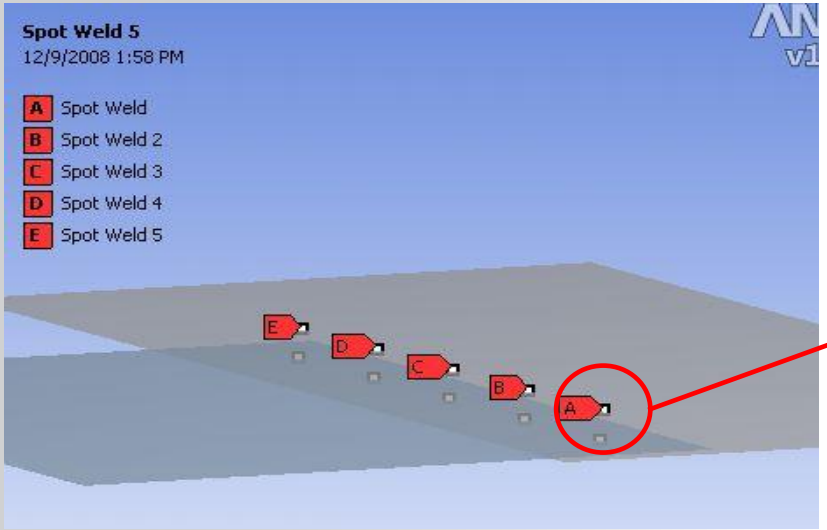


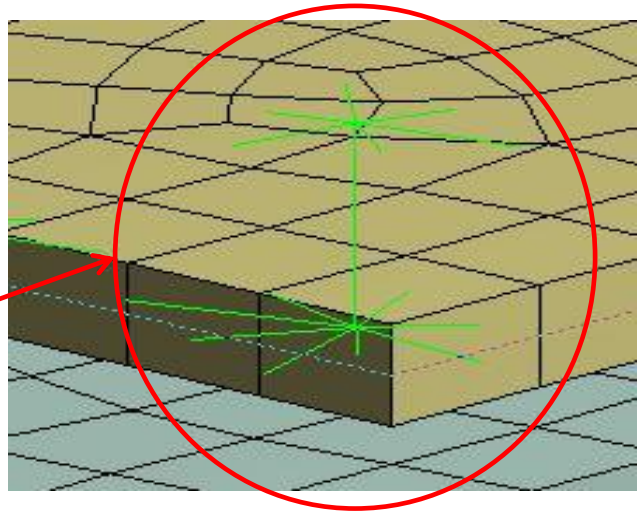
 © 2015 ANSYS, Inc.

D. Spot Welds

Spot welds provide a means of connecting shell assemblies at discrete points:

- **Spotwelds are defined on the geometry as vertex point pairs. Currently, only DesignModeler and NX can be used to create automatic spot welds (note, spot welds can be defined manually if vertices exist in the proper locations).**
- **The spot weld connection is accomplished using a beam connection between the points.**
- **A "spider web" of beams is radiated from each point to distribute the load.**



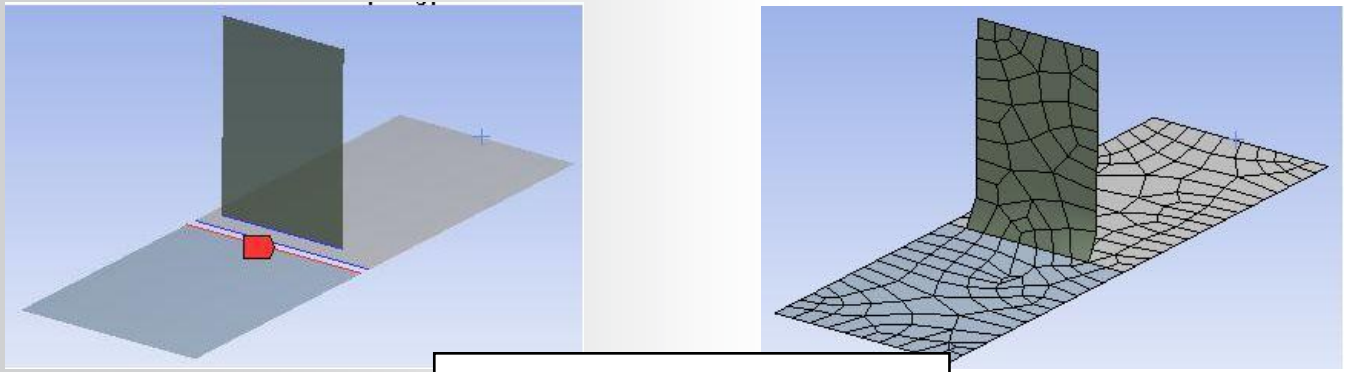


 © 2015 ANSYS, Inc.

E. Mesh Connections

The mesh connection feature allows you to join the meshes of topologically disconnected surface bodies:

- **Previously connections such as this required a geometry application to repair gaps (e.g. DesignModeler or CAD).**
- **Mesh connections are made at the mesh level using either edge to edge or edge to face configurations.**
- **Unlike geometry solutions, a multibody part is not required.**



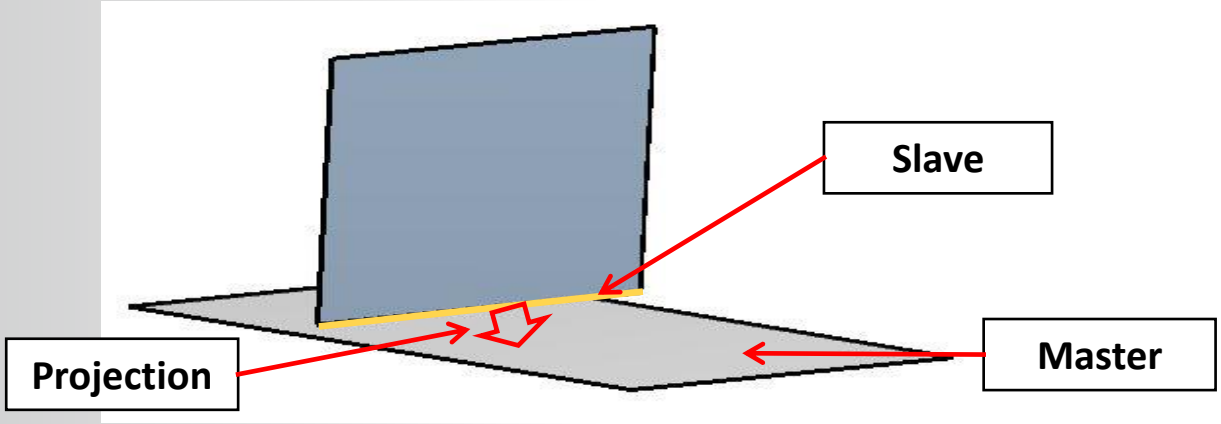
Mesh Connection Example

© 2015 ANSYS, Inc.

. . . Mesh Connections

Mesh connections use the concept of master and slave geometry to control how the connection is made:

- **Master: indicates the geometry/topology onto which other geometry is projected.**
- **Slave: indicates the geometry that will be projected onto the master geometry.**
- **Master geometry can be faces or edges whereas slave geometry can only be edges.**

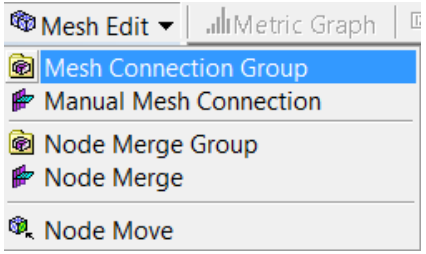


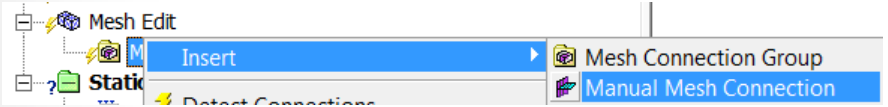
© 2015 ANSYS, Inc.

. . . Mesh Connections

Mesh connections are displayed under the "Mesh Edit" branch from Mesh in the tree using "Mesh Connection Group":

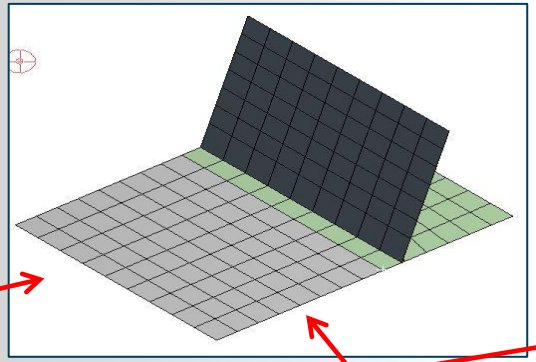
- **As with other connection types mesh connections can be manually or automatically created.**
- **Manual mesh connections offer some additional controls for configuration of the connection.**

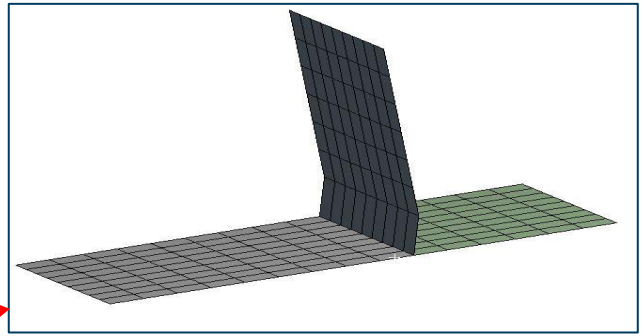




Details of "Mesh Connection"

Scope	
Scoping Method	Geometry Selection
Master Geometry	1 Face
Slave Geometry	1 Edge
Master Bodies	Surface Body
Slave Bodies	Surface Body
Definition	
Scope Mode	Manual
Tolerance Type	Value
Tolerance Value	2.25 mm
Suppressed	No
Snap to Boundary	No





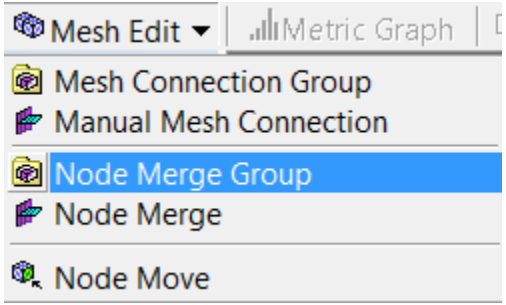
Use "Snap to Boundary=Yes" to avoid sliver region by snapping to the boundary (pictures exaggerated).

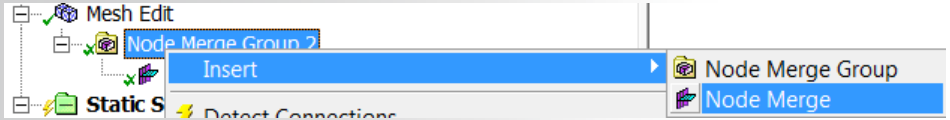
© 2015 ANSYS, Inc.

. . . Mesh Connections

Node Merge are displayed under the “Mesh Edit” branch from Mesh in the tree using “Node Merge group”

- **This object defines nodes that have been merged on a generated mesh**
- **Multiple Node Merge Group options can be added as child objects to a Mesh Edit Object**
- **Requires mesh generation**





Details of "Node Merge"

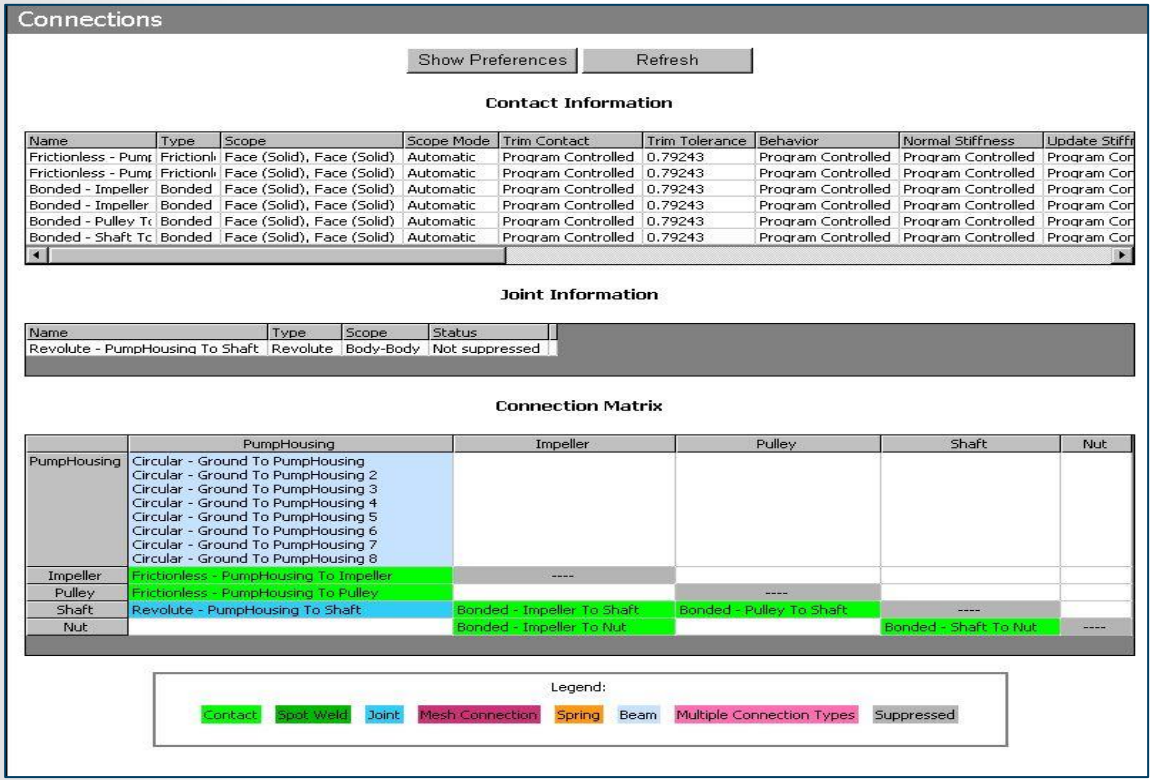
Scope	
Scoping Method	Geometry Selection
Master Geometry	1 Edge
Slave Geometry	1 Edge
Master Bodies	Surface Body
Slave Bodies	Surface Body
Definition	
Scope Mode	Manual
Tolerance Type	Value
Tolerance Value	2. mm
Suppressed	No

© 2015 ANSYS, Inc.

F. Connections Worksheet

The connections worksheet contains a number of features that allow users to investigate and evaluate the various connections that may be used in a model.

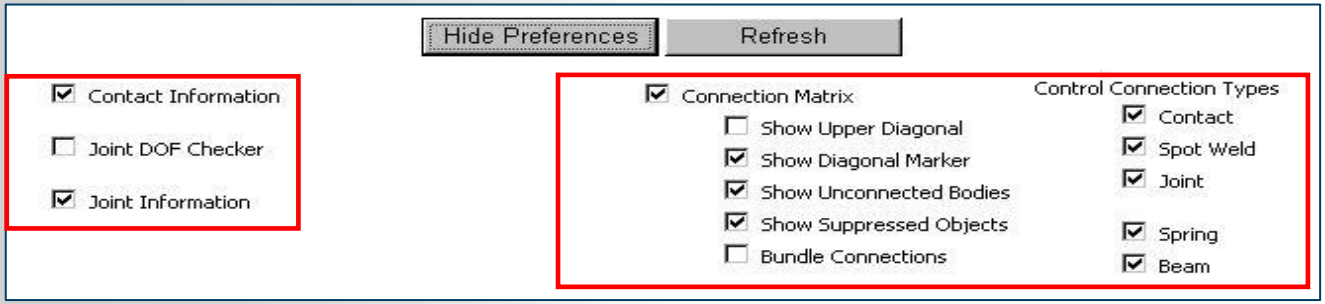
A sample is show here, details are on the following pages.



 © 2015 ANSYS, Inc.

. . . Connections Worksheet

Display in the connections worksheet is controlled via the preferences settings seen below. These preferences are shown upon first activating the worksheet but can be accessed any time using the Show/Hide Preferences button.



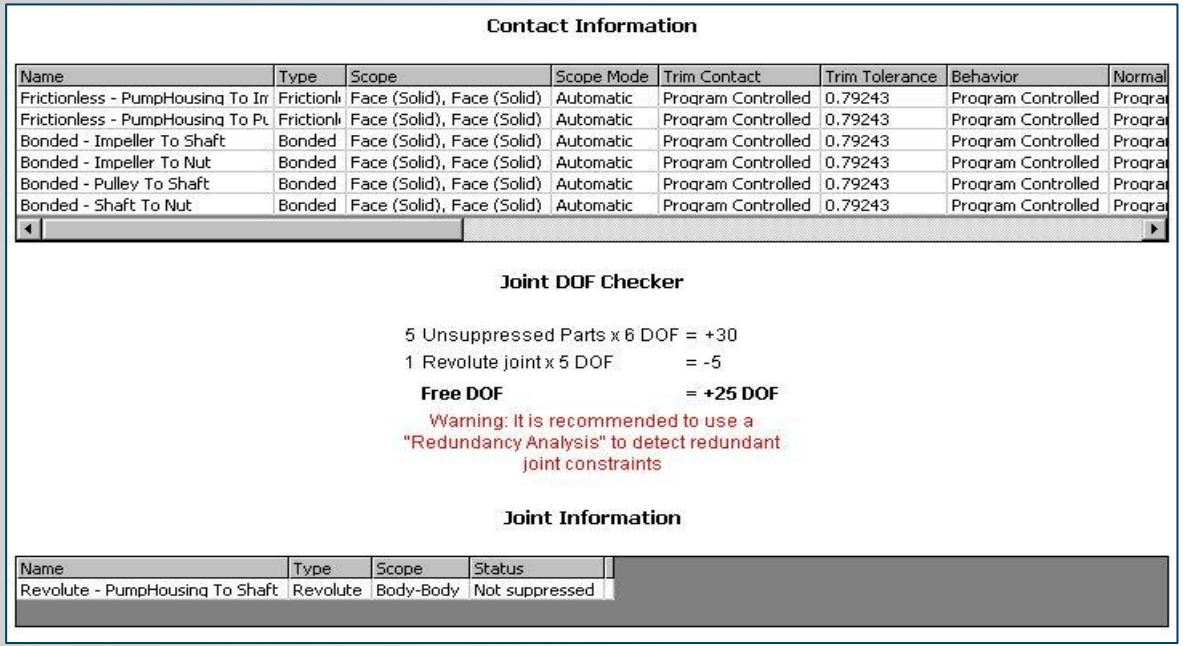
The left hand column is used to indicate whether contact and/or joint information should be displayed.

The right hand column controls the display of the connection matrix section of the worksheet. The connection matrix can be used to display how each body is connected and by which means (i.e. contact, joints, etc.).

 © 2015 ANSYS, Inc.

. . . Connections Worksheet

The contact and joint "information" section provides a list view of these connections along with details about each. The joint DOF checker calculates how many free DOF there are in the model. Note however, this only relates to joints. If contact and other connections exist, they must be accounted for separately.



Contact Information

Name	Type	Scope	Scope Mode	Trim Contact	Trim Tolerance	Behavior	Normal
Frictionless - PumpHousing To Im	Frictionl	Face (Solid), Face (Solid)	Automatic	Program Controlled	0.79243	Program Controlled	Prograı
Frictionless - PumpHousing To Pu	Frictionl	Face (Solid), Face (Solid)	Automatic	Program Controlled	0.79243	Program Controlled	Prograı
Bonded - Impeller To Shaft	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	0.79243	Program Controlled	Prograı
Bonded - Impeller To Nut	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	0.79243	Program Controlled	Prograı
Bonded - Pulley To Shaft	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	0.79243	Program Controlled	Prograı
Bonded - Shaft To Nut	Bonded	Face (Solid), Face (Solid)	Automatic	Program Controlled	0.79243	Program Controlled	Prograı

Joint DOF Checker

5 Unsuppressed Parts x 6 DOF = +30
1 Revolute joint x 5 DOF = -5
Free DOF = +25 DOF

Warning: It is recommended to use a "Redundancy Analysis" to detect redundant joint constraints

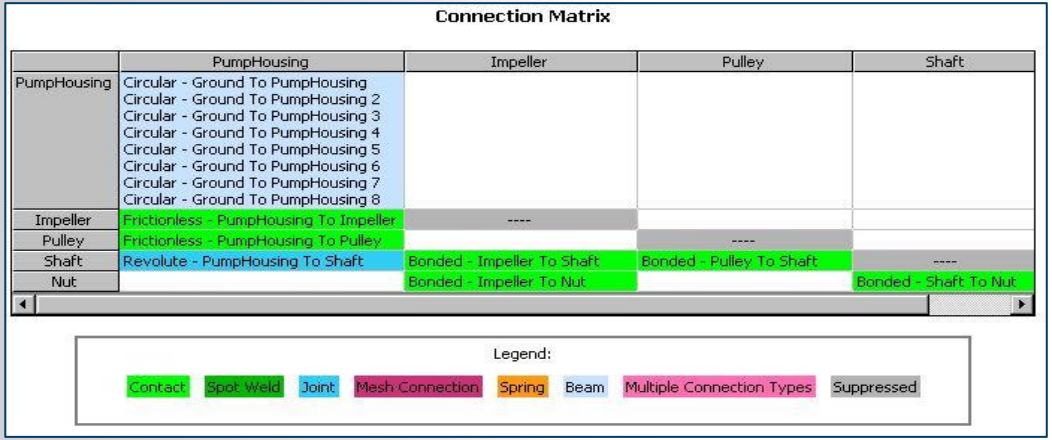
Joint Information

Name	Type	Scope	Status
Revolute - PumpHousing To Shaft	Revolute	Body-Body	Not suppressed

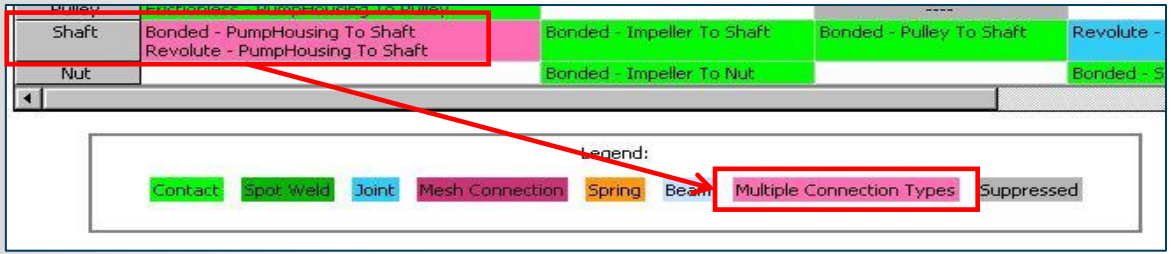
© 2015 ANSYS, Inc.

. . . Connections Worksheet

The Connection matrix lists parts along the top and left side with rows and columns displaying various connections for each (color coded).



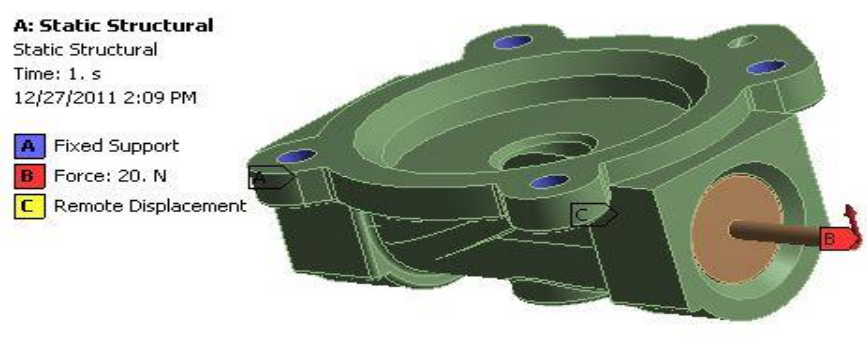
The connection matrix can be particularly useful in finding over constraint situations. Where multiple connection types are detected, they are flagged in the matrix.

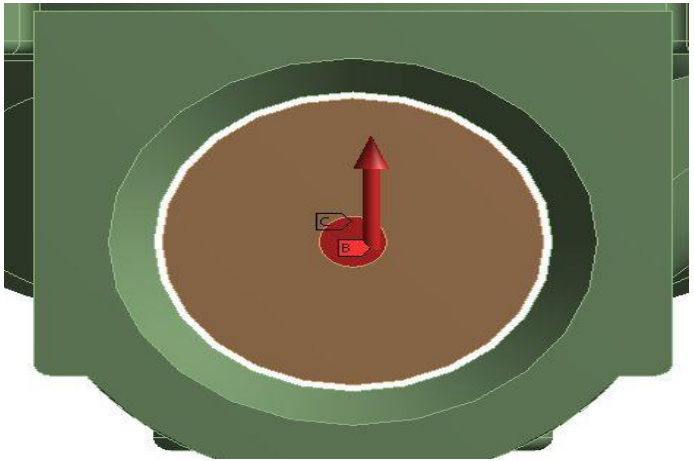


 © 2015 ANSYS, Inc.

G. Workshop 5.1

- **Workshop 5.1 – Contact Offset Control**





 © 2015 ANSYS, Inc.

H. Joint Definitions

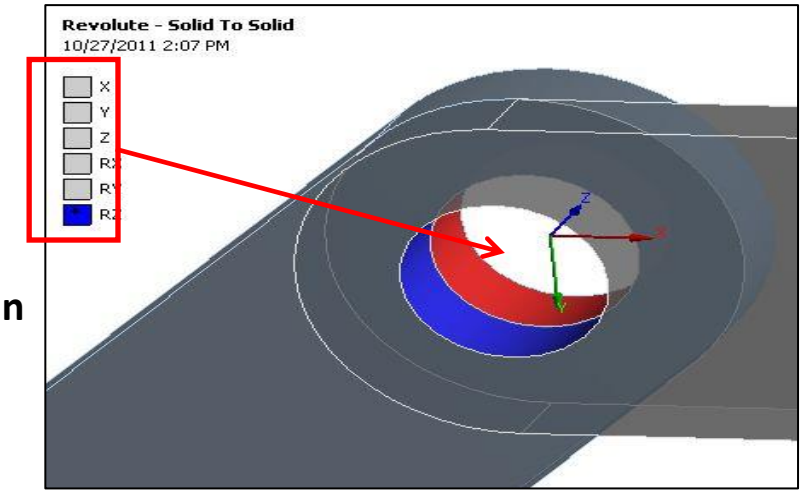
The joint feature in Mechanical provides an alternative to contact when simulating the interaction between bodies or to ground (fixed) locations:

There are 9 joint types available in Mechanical which can be either body to body or body to ground.

The reference and mobile regions are color coded.

The legend displays the joints degrees of freedom with respect to its reference coordinate system. Colored DOF are free, gray indicates a fixed DOF.

Note, this material is meant to be an introduction to the joint feature. The rigid dynamic training course contains in depth coverage.

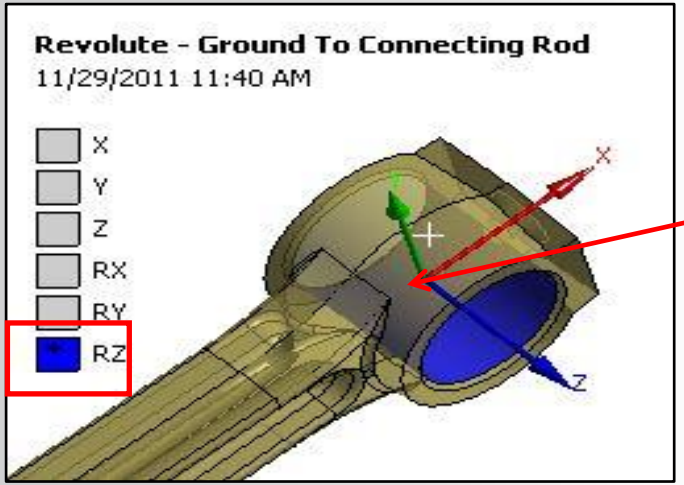


 © 2015 ANSYS, Inc.

. . . Joint Definitions

Revolute Example:

- **The legend shows the “RZ” or rotation about Z is free.**
- **Degrees of freedom shown in grey are constrained.**
- **The “Reference Coordinate System” is listed in the details and displayed graphically.**



Details of "Revolute - upper flange To lower flange"

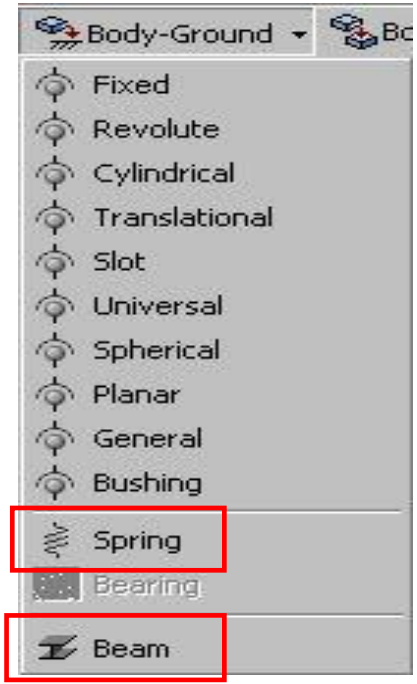
Definition	
Connection Type	Body-Ground
Type	Revolute
Torsional Stiffness	0. N·mm/°
Torsional Damping	0. N·mm·s/°
Suppressed	No
Reference	
Coordinate System	Reference Coordinate System
Mobile	
Scoping Method	Geometry Selection
Applied By	Remote Attachment
Scope	1 Face
Body	lower flange
Initial Position	Unchanged
Behavior	Rigid
Pinball Region	All
Stops	

© 2015 ANSYS, Inc.

I. Springs and Beams

Springs and beams can be defined as body to body or body to ground like joints:

- **Springs and beams are found in the Body-Ground and Body-Body menus.**
- **Ground locations refer to a coordinate system as the ground location.**
- **Note, springs and beams are a form of remote condition and have Behavior and Pinball Region controls (these topics will be covered in chapter 7).**



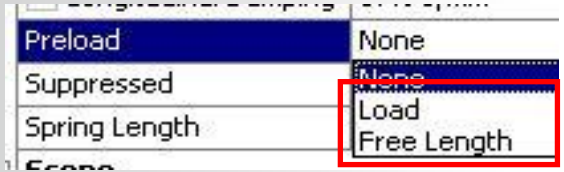
Behavior	Rigid
Pinball Region	All

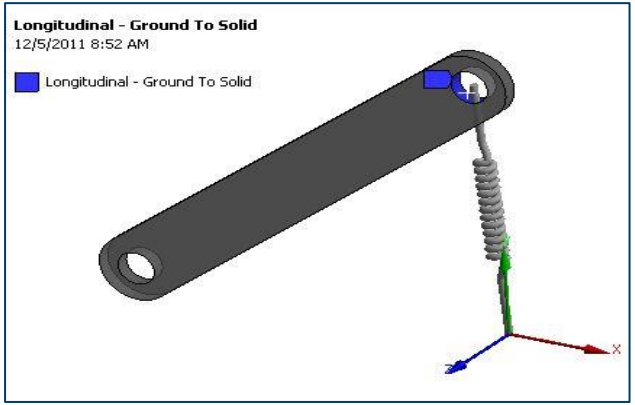
 © 2015 ANSYS, Inc.

. . . Springs and Beams

Springs:

- **Springs are assumed to be in their free state (unloaded) by default.**
- **Spring behavior is both tensile and compressive.**
- **Damping may be added to the spring's definition.**
- **Grounded springs refer to a local coordinate system as the ground location.**
- **A preload may be added using either a free length or load value.**



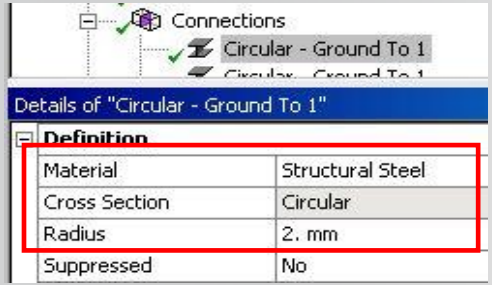


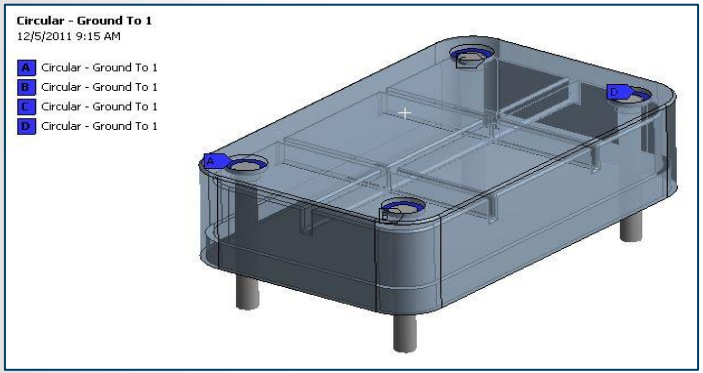
© 2015 ANSYS, Inc.

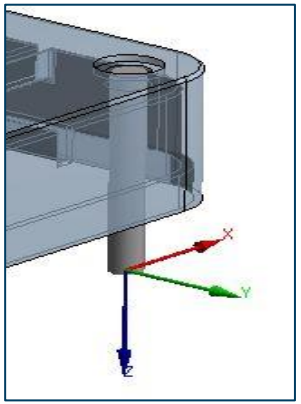
. . . Springs and Beams

Beams:

- **Beams are assumed to have a circular cross section. The radius is set in the beam details.**
- **A beam's material is set in the details (Engineering Data materials).**
- **Grounded beams refer to a local coordinate system as the ground location.**





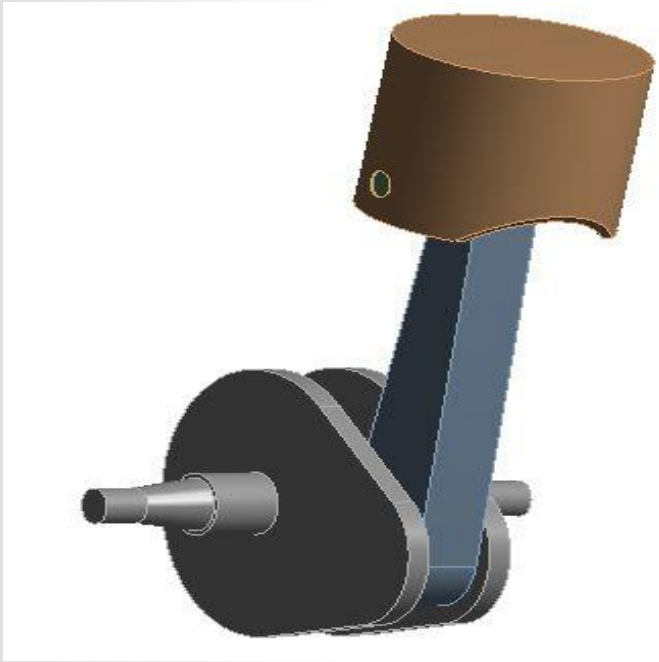


Although not limited to this purpose, beams are often useful in simulating fasteners.

© 2015 ANSYS, Inc.

J. Workshop 5.2

- **Workshop 5.2, Using Joints**



 © 2015 ANSYS, Inc.

K. APPENDIX

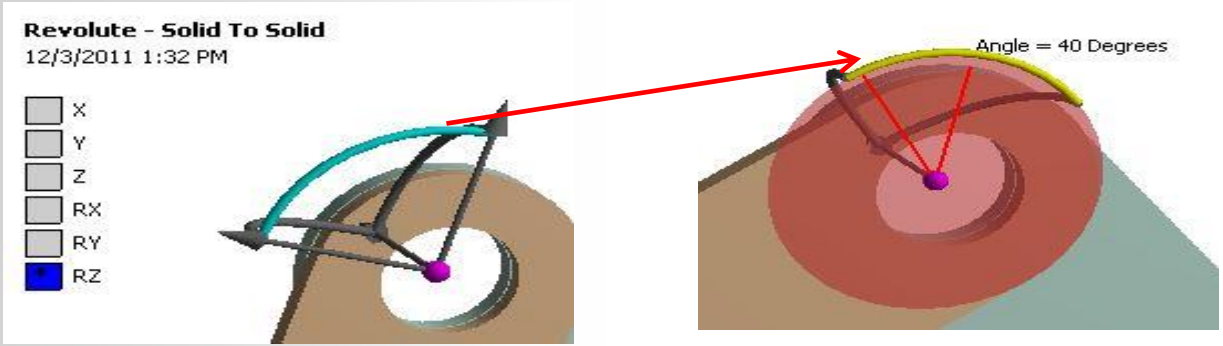
- Joint Configuration
- Joint Stops and Locks

 © 2015 ANSYS, Inc.

Joint Configuration

Configuring a joint allows its initial state (configuration) to be changed:

- **Begin by highlighting the joint to be configured in the tree.**
- **Now click the “Configure” icon in the context menu.**
- **When a joint is in configure mode its position can be changed by dragging the DOF handle shown below.**



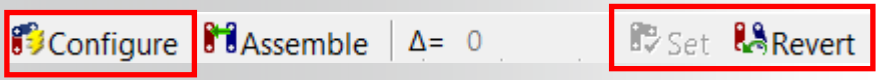
© 2015 ANSYS, Inc.

. . . Joint Configuration

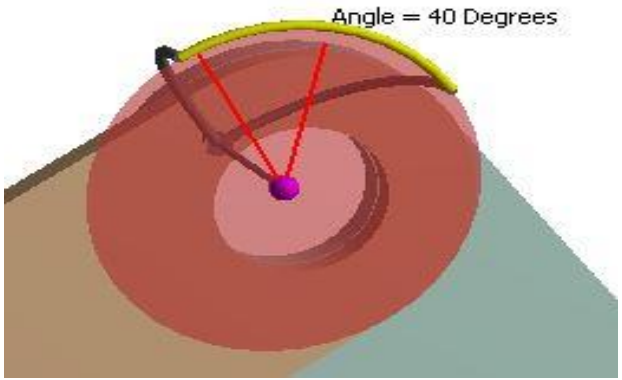
Joint configuration may be used to merely "test" the effect of the joint's motion. Toggle off the configure tool and the joint will return to its original configuration.

A joint can be locked into a new position if desired:

- After setting a new configuration for the joint, choose "Set" from the context menu. The new configuration becomes the starting position when solved.
- The "Revert" icon can be used to cancel the operation.



- In addition to manually configuring a joint, a value can be entered into the field next to the configure icon.



© 2015 ANSYS, Inc.

. . . Joint Configuration

Since a joint's motions are determined according to its coordinate system it will sometimes be necessary to reorient these systems to obtain the correct joint behavior.

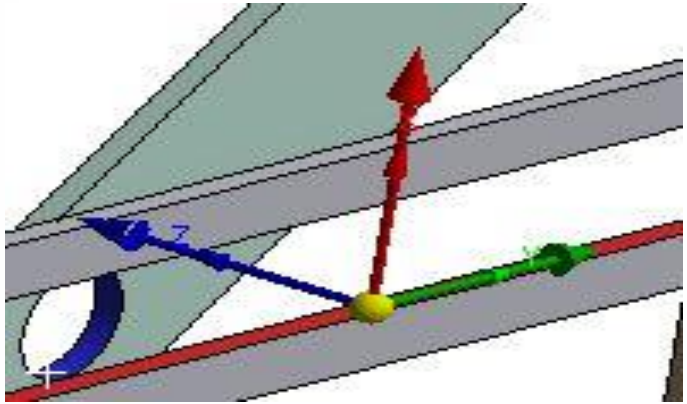
- **Click in the Coordinate System field in the joint's details to place it in edit mode. Notice the CS graphically expands while editing.**

Details of "Translational - Solid To Solid"

Definition	
Reference	
Scoping Method	Geometry Selection
Scope	1 Face
Body	Solid
Coordinate System	Reference Coordinate System

Details of "Translational - Solid To Solid"

Definition		
Reference		
Scoping Method	Geometry Selection	
Scope	1 Face	
Body	Solid	
Coordinate System	Apply	Cancel
Mobile		

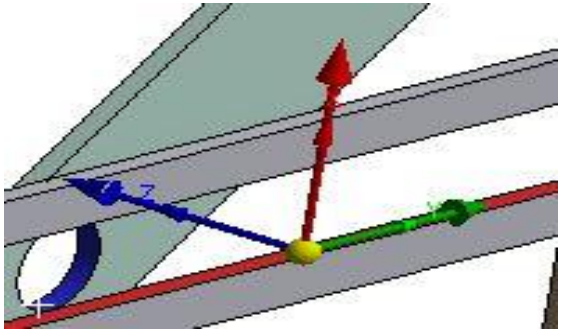


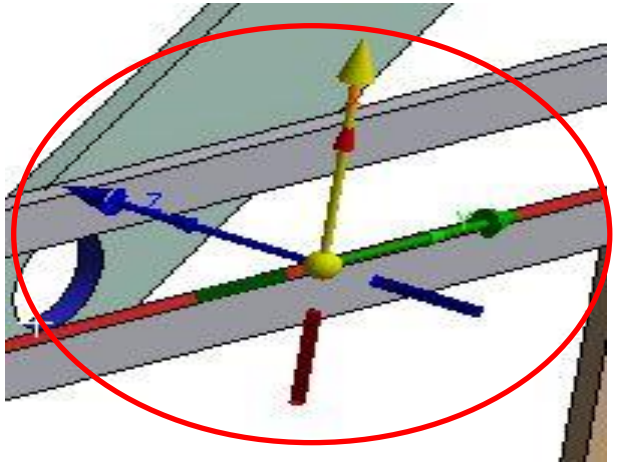
 © 2015 ANSYS, Inc.

. . . Joint Configuration

While in edit mode click on the CS axis to be modified:

- **With that axis “active” you can click on another axis, edge, face, etc. to establish a new direction.**
  - **Note the negative axes show as well while editing directions.**
- **Complete the change by pressing the “Apply” button in the joint details.**





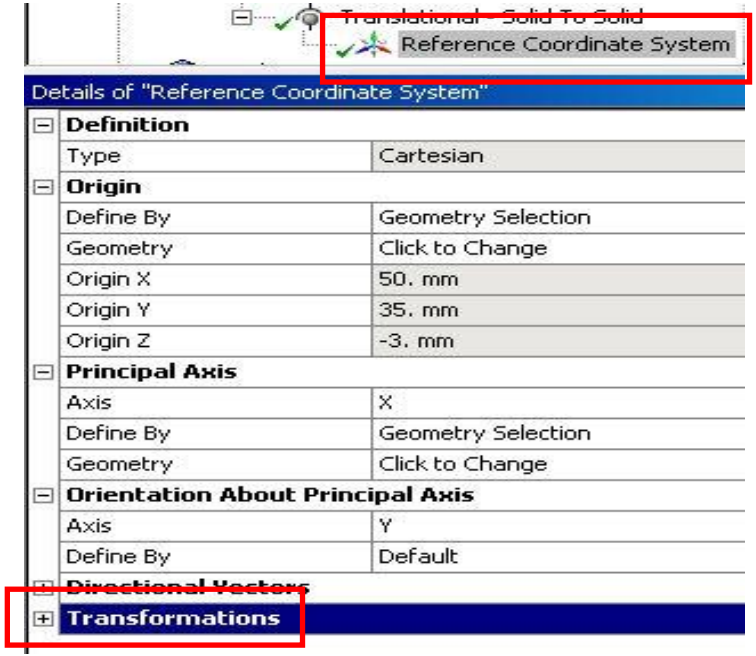
Details of "Translational - Solid To Solid"

Definition		
Reference		
Scoping Method	Geometry Selection	
Scope	1 Face	
Body	Solid	
Coordinate System	Apply	Cancel
Mobile		

© 2015 ANSYS, Inc.

. . . Joint Configuration

In addition to manually reorienting a joint coordinate system the same transforms used in creating and modifying local coordinate systems are available.

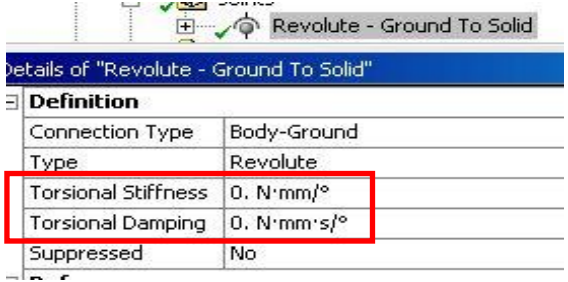


 © 2015 ANSYS, Inc.

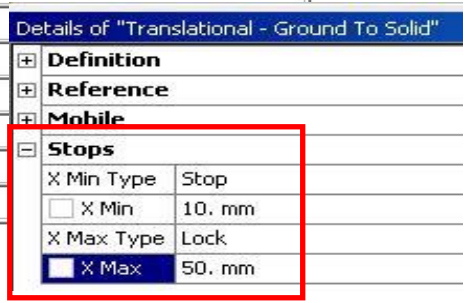
Joint Stops and Locks

For the Revolute and Cylindrical joint types a torsional stiffness and/or damping can be defined in the joint's details.

Most joints can also employ stops and/or locks to limit the range of joint motion (see table below).



Joint Type	Stop/Lock
Revolute	Yes
Cylindrical	Yes
Translational	Yes
Slot	Translational
Universal	Yes
Spherical	No
Planar	Yes
General	Translational



 © 2015 ANSYS, Inc.