ANSYS - Disks in Point Contact - Step 5

Step 5: Mesh geometry

We'll start by meshing the upper and lower disks using *SOLID92* elements . Then, we'll mesh the target and contact surfaces using *TARGE170* elements and *CONT175* elements respectively.

Main Menu > Preprocessor > MeshTool

This brings up the *MeshTool*.

Set Meshing Parameters

We'll now specify the element type, real constant set and material property set to be used in the meshing of the upper and lower volumes. Make sure ***Glob al*** is selected under ***Element Attributes*** and click on ***Set***.

This brings up the *Meshing Attributes* menu. You will see that *PLANE92* and material number 1 are already selected. Also, recall that no real constants need to be specified for *PLANE92* element type. Click ***OK***.

Set Mesh Size

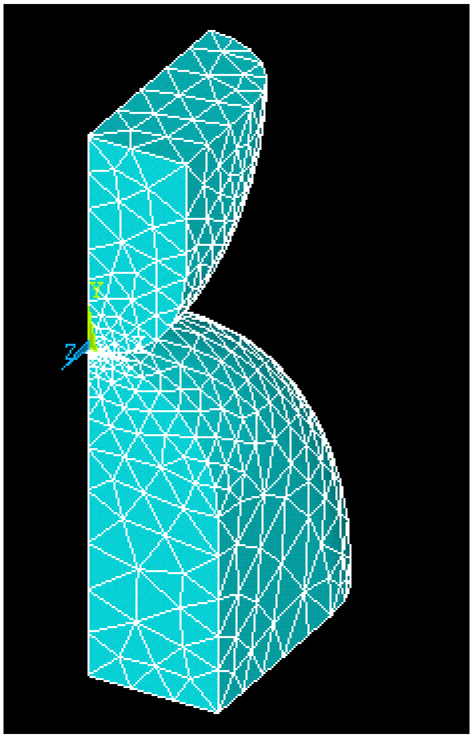
We'll use the *SmartSize* option which enables automatic element sizing. Click on the *SmartSize* checkbox so that a tickmark appears in it. Change the setting for the overall element size level to ***1*** by moving the slider under ***SmartSize*** to the left.

Mesh Volumes

In the *MeshTool*, make sure ***Volumes*** is selected in the drop-down list next to ***Mesh***. We'll use tetrahedral elements, so make sure the default option of ***Tet*** is selected under ***Shape***. We'll also use the default of ***Free*** meshing.

Click on *the* ***Mesh*** button. This brings up the *Mesh Volumes* pick menu.

In the *Input* window, ANSYS tells you to "pick or enter volumes to be meshed". Since both volumes are to be meshed, click on ***Pick All***. The geometry is meshed and the elements are plotted in the *Graphics* window.



Mesh Target Surface

Before meshing the target surface, we need to select the nodes attached to the surface of the lower disk that are expected to come into contact with the upper disk. Since only a small area of the lower disk is expected to come into contact with the upper disk, we will select only the nodes near the point of contact and define the target surface with these nodes. To do this we will use "select logic".

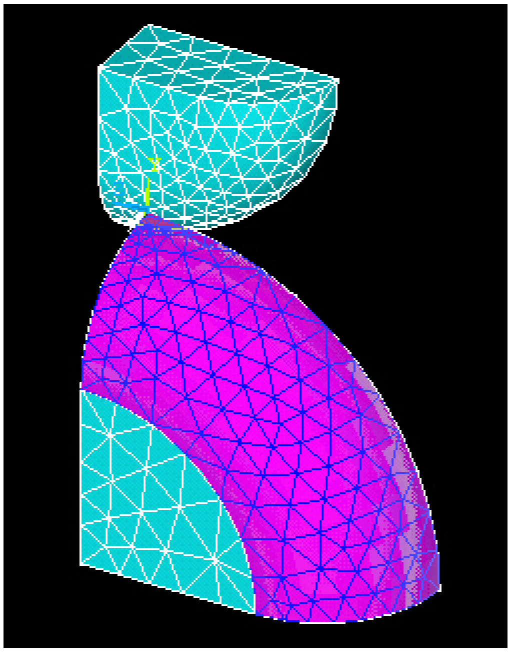
First, we'll select the target area and then the nodes attached to this area and located above the y=-2 plane. As we will see in the results, only a subset of the nodes located above the y=-2 plane actually come into contact with the upper disk.

> In contact problems, the target surface is modeled through a set of *target segments/elements*. Several target elements comprise one target surface. These target elements will define the surface of the lower disk that is expected to come into contact with the upper disk.

Utility Menu > Select > Entities

Select ***Areas*** from the pull-down menu at the top. Make sure ***By Num/Pick*** is selected below that. Click ***Apply***.

Hold down the left mouse button until area 8 is selected. Area 8 belongs to the lower disk and is the curved area that will come into contact with the upper disk once the force is applied. You might need to rotate the view to be able to select this area. Click OK in the pick menu.



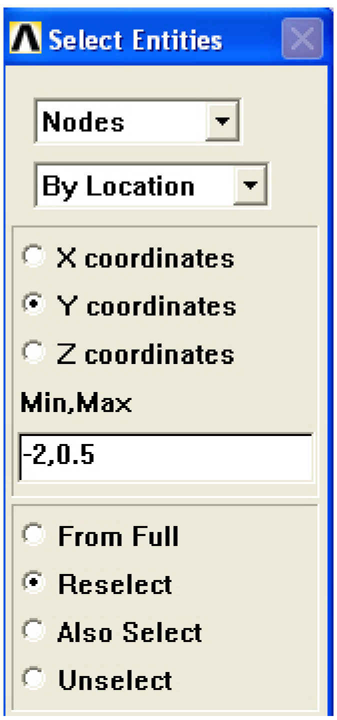
Verify that area 8 has been selected: **Utility Menu > Plot > Areas**.

Next we'll select the nodes attached to this area. In the *Select Entities* menu, select ***Nodes*** from the pull-down menu at the top and ***Attached to*** below that. Select ***Areas, All*** below that. Click ***Apply***.

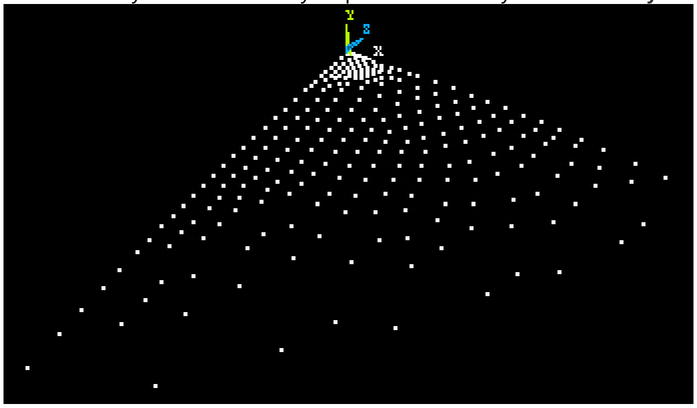
Check that only nodes attached to area 8 are currently selected: **Utility Menu > Plot > Nodes**

Next we'll select the nodes located above the y=-2 plane.

In the *Select Entities* menu, make sure ***Nodes*** is selected in the pull-down menu at the top and select ***By Location*** below that. Select ***Y coordinates*** below that and enter `-2,0.5` as the ***Min,Max***. Then select ***Reselect*** below that since we want to select a subset of the already selected nodes. Click ***OK***.



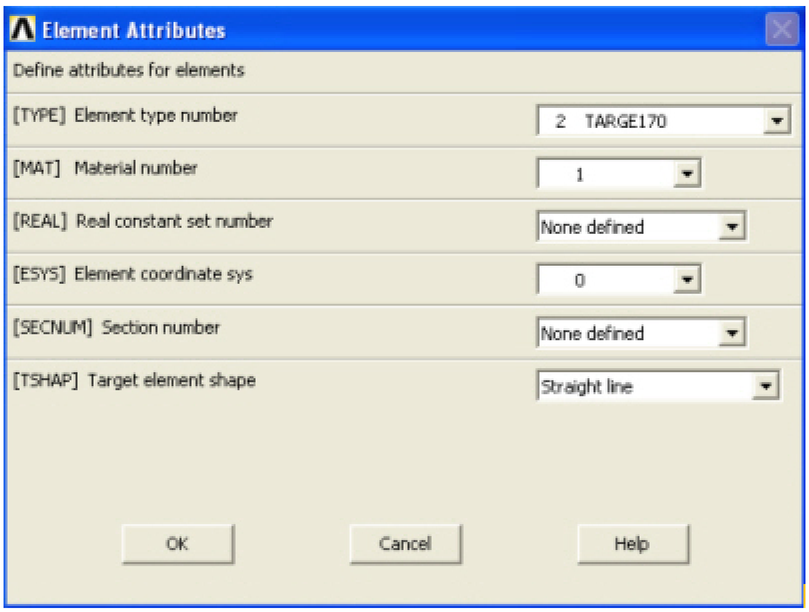
Check that only nodes above the y=-2 plane are currently selected: **Utility Menu > Plot > Nodes**



We'll now mesh the selected nodes using *TARGE170* elements.

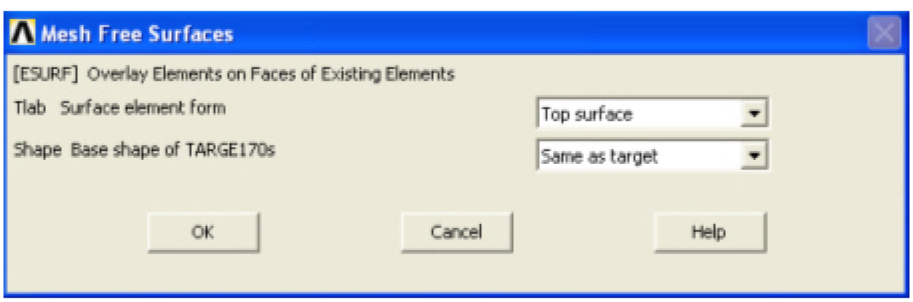
Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes

This brings up the *Element Attributes* menu. Select ***2 TARGE170*** for ***Element type number***. Note that the material number is defaulted to 1 as this the only one available. Also, recall that we did not define any real constants for this element. Click ***OK***. We have now specified the element type to be used for the meshing of the target surface.



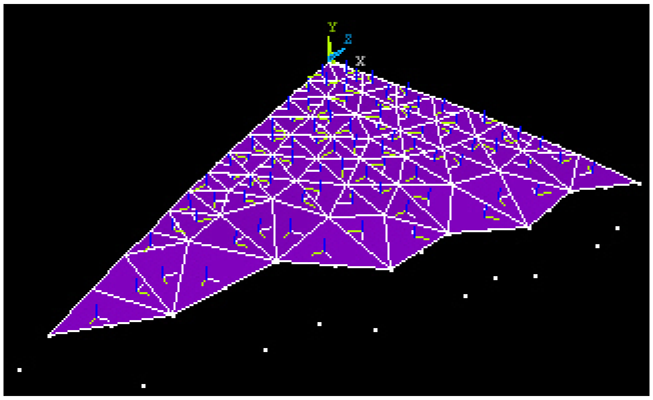
Main Menu > Preprocessor > Modeling > Create > Elements > Surf /Contact > Surf to Surf

This brings up the *Mesh Free Surface* menu. We'll use the default settings. Click ***OK***.



This brings up the *Mesh free Surfaces* pick menu. In the *Input* window, ANSYS tells you to "pick or enter node for contact elements ". Since we have already selected the nodes, click ***Pick All***.

The target surface is meshed and the elements are plotted in the *Graphics* window.



In solving contact problems, where you expect large displacements or where you don't know where contact might occur between bodies, you want to start by selecting as many nodes as possible to capture all regions where contact may occur. At the same time, you want to keep the number of selected nodes as small as possible to reduce the time to generate a solution. Contact problems are highly nonlinear and require significant computer resources to solve. In most cases, it is best to use an iterative approach in order to reach an appropriate number of nodes and build an efficient model.

Mesh Contact Surface

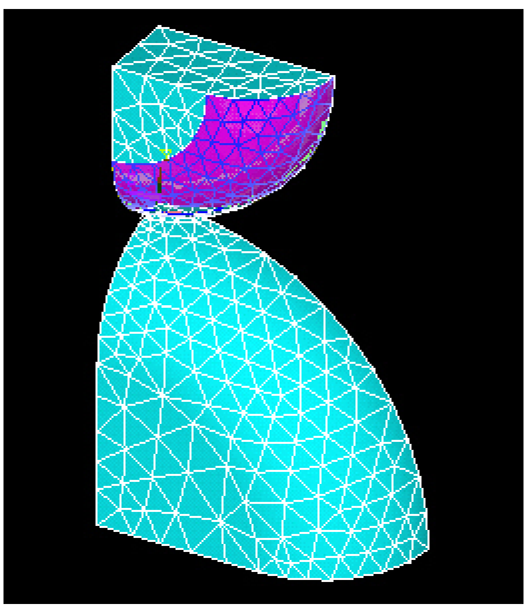
We'll now mesh the contact surface. Again, we'll start by selecting the nodes attached to the surface of the upper disk that are expected to come into contact with the lower disk. In this case, we'll select the contact area first and then the nodes attached to this area and located below the y=1.5 plane.

First, we need to undo the selections of areas and nodes we made in the previous step. Select everything: **Utility Menu > Select > Everything.**

Utility Menu > Select > Entities

Select ***Areas*** from the pull-down menu at the top and ***By Num/Pick*** below that. Select ***From Full*** below that. Click ***Apply***.

Hold down the left mouse button until area 4 is selected. Area 4 belongs to the lower body and is the curved area that will be in contact with the lower /target body once the force is applied. You might need to rotate the view to be able to select this area. Click ***OK*** in the pick menu.



Verify that area 4 has been selected: **Utility Menu > Plot > Areas**.

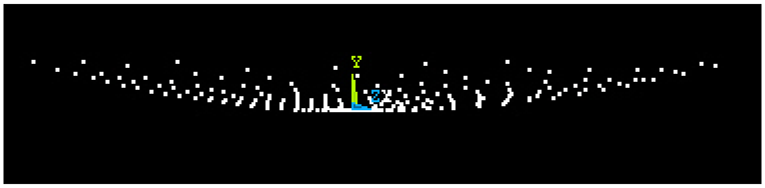
Next we'll select the nodes attached to this area. In the *Select Entities* menu, select ***Nodes*** from the pull-down menu at the top and ***Attached to*** below that. Select ***Areas, All*** and ***From Full*** below that. Click ***Apply***.

Check that only nodes attached to area 4 are currently selected: **Utility Menu > Plot > Nodes**

Next we'll select the nodes located below the y=1.5 plane.

In the *Select Entities* menu, make sure ***Nodes*** is selected in the pull-down menu at the top and select By Location below that. Select ***Y coordinates*** below that and enter `-0.5,1.5` as the ***Min,Max***. Then select ***Reselect*** below that since we want to select a subset of the already selected nodes. Click ***OK***.

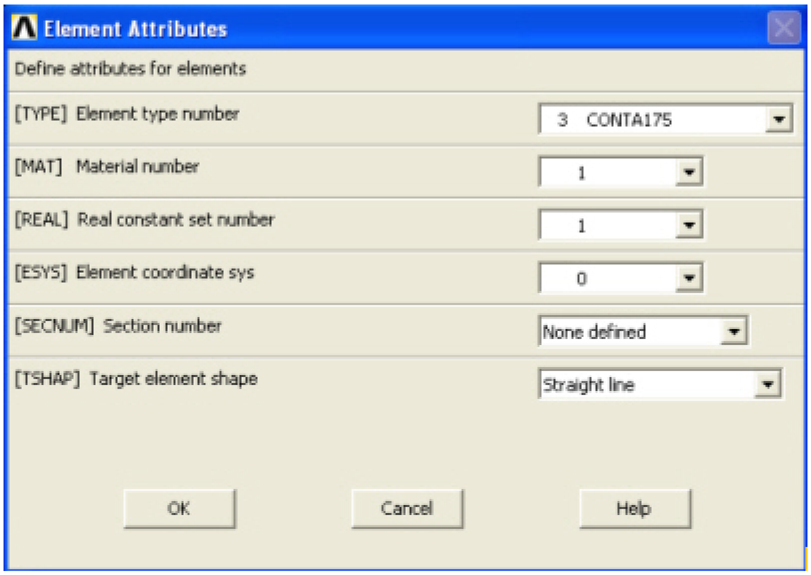
Check that only nodes below the y=1.5 plane are currently selected: Utility Menu > Plot > Nodes



We'll now mesh the selected nodes using *CONTA175* elements.

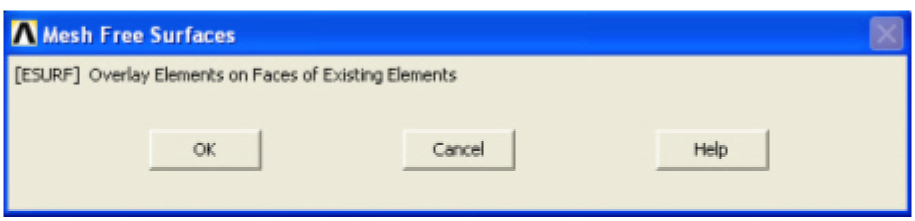
Main Menu > Preprocessor > Modeling > Create > Elements > Elem Attributes

This brings up the *Element Attributes* menu. Select ***3 CONTA175*** for ***Element type number***. Note that the material number and the real constant set number are both set to 1. ANSYS has set the various real constants to their default values and created real constant set 1. Click ***OK. Close*** the warning message that appears. We have now specified the element type to be used for the meshing of the contact surface.



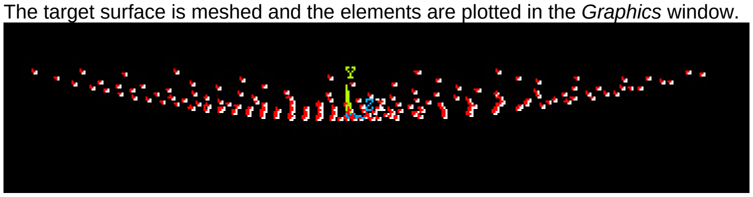
Main Menu > Preprocessor > Modeling > Create > Elements > Surf /Contact > Node to Surf

This brings up the *Mesh Free Surface* menu. Click ***OK***.



This brings up the *Mesh free Surfaces* pick menu. In the *Input* window, ANSYS tells you to "pick or enter node for contact elements ". Since we have already selected the nodes, click ***Pick All***.

The target surface is meshed and the elements are plotted in the *Graphics* window.



Before we move to the next step, we need to undo the selections of areas and nodes we have made. Select everything: **Utility Menu > Select > Everything**.

Save Your Work

Toolbar > SAVE_DB

Go to Step 6: Specify boundary conditions

Go to all ANSYS Learning Modules