ANSYS - Disks in Point Contact - Step 6

Step 6: Specify boundary conditions

As in previous tutorials, we'll apply the loads to the geometry rather than the mesh.

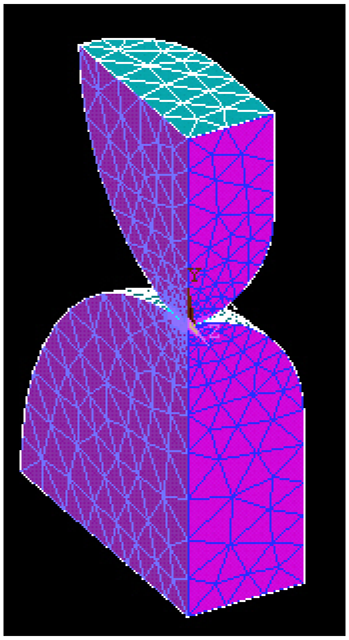
One important change to note is that instead of applying force P to the lower disk, we'll constrain the lower surface of the lower disk in the vertical (Y) direction in order to sufficiently constrain the model and avoid rigid body motion.

Apply Symmetry Boundary Conditions

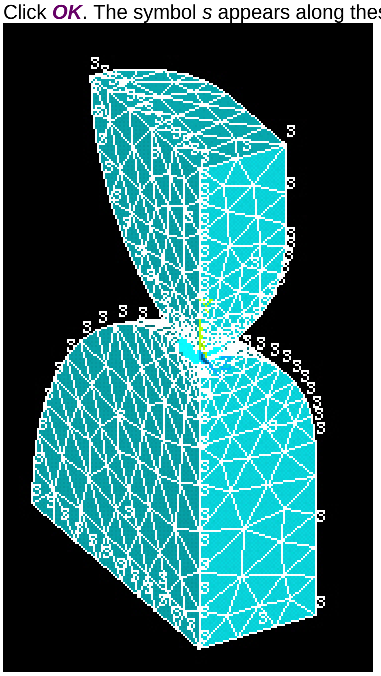
We'll apply symmetry boundary conditions along the planes of symmetry.

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > Symmetry B.C. > On Areas

Select the four areas that define the planes of symmetry by clicking on them.



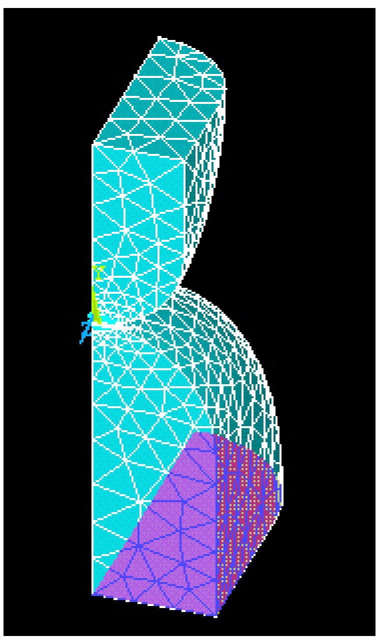
Click ***OK***. The symbol *s* appears along these areas indicating that symmetry B.C.s have been applied.



Apply Displacement Constraints

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Displacement > On Areas

This brings up the *Apply U,Rot on Areas* pick menu. Select the bottom area of the lower disk (x-z plane). Click ***OK***.

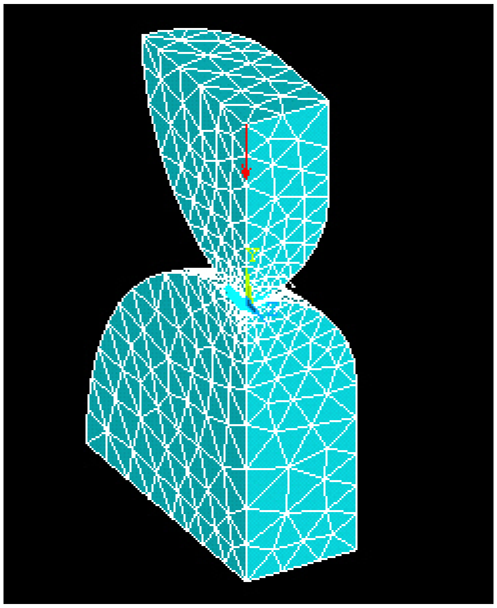


This brings up the *Apply U,Rot on Areas* menu. Select UY for the DOFs to be constrained and enter `0` for the ***Displacement value***. Click ***OK***.

Apply Force

Main Menu > Preprocessor > Loads > Define Loads > Apply > Structural > Force/Moment > On Keypoints

This brings up the *Apply F/M on KPs* pick menu. Select keypoint 8 on the upper body and click ***OK***. This brings up the *Apply F/M on Keypoints* menu. Select ***FY*** for the ***Direction of force/mom***. Enter `p/4` for ***Force/Moment value*** and click ***OK***. A single red arrow denotes the force and the direction in which it is acting. Note that we have divided the total force P=4500N by four to account for the fact that only a quarter of the volumes are being modeled.



Apply Coupled BCs

We will apply a coupled boundary condition to the upper area of the upper disk to ensure that all nodes attached to this area move equally as a result of the applied load.

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 6 is selected. Area 6 is the upper area (x-z plane) of the upper disk. Click ***OK***.

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 ***OK***.

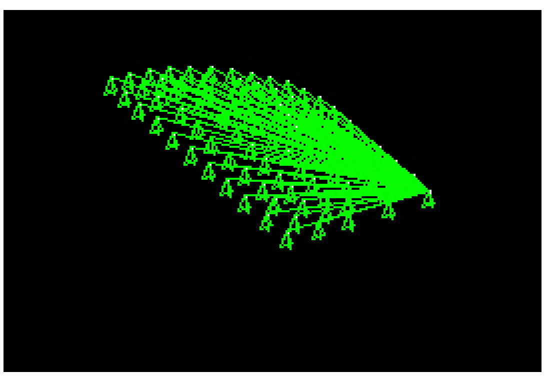
Verify that only nodes attached to area 6 are currently selected: **Utility Menu > Plot > Nodes**

We'll now apply a coupled boundary condition to the selected nodes.

Main Menu > Preprocessor > Coupling/Ceqn > Couple DOFs

This brings up the *Define Coupled DOFs* pick menu. In the *Input* window, ANSYS tells you to "Pick or enter nodes to be coupled". Since we have already selected the nodes, click ***Pick All***.

This brings up the *Define Coupled DOFs* menu. Enter `1` for ***Set reference number*** which is an arbitrary number. Select ***UY*** for ***Degree-of-freedom label***as we want to couple the movement of the nodes in the y direction. This step ensures that all nodes on the upper surface will move equally as a result of the applied load. Click ***OK***.



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

Save Your Work

Toolbar > SAVE_DB

Go to Step 7: Solve

Go to all ANSYS Learning Modules