# ANSYS - Disks in Point Contact - Step 9

**Problem Specification** 

- 1. Start-up and preliminary set-up
- 2. Specify element type and constants
- 3. Specify material properties
- 4. Specify geometry
- 5. Mesh geometry
- 6. Specify boundary conditions
- 7. Solve
- 8. Postprocess the results
- 9. Validate the results

## Step 9: Validate the results

#### **Theoretical Results**

We will compare the solution obtained with ANSYS with the expected theoretical values.

|           | Approach     | Max Prin<br>Stress       |
|-----------|--------------|--------------------------|
| Theory    | 0.0258<br>mm | -1667N/mm <sup>2</sup>   |
| ANSY<br>S | 0.0252<br>mm | -1574.9N/mm <sup>2</sup> |

As we can see, the value for the approach obtained with ANSYS compares well with the theoretical value (less than 3% variation). The max principal stress also compares well with the theoretical value (less than 6% variation).

#### **Refine Mesh**

We will repeat the calculations on a mesh with a refined element size level near the point of contact to check the validity of the solution.

Main Menu > Preprocessor > Meshing > Mesh Tool

Select Clear under Mesh and Pick All in the pick menu. The mesh is deleted.

In addition to deleting the mesh, we need to delete the target and contact elements and associated nodes.

Main Menu > Preprocessor > Modeling > Delete > Elements

This brings up the Delete Elements pick menu. Select Pick All.

Main Menu > Preprocessor > Modeling > Delete > Nodes

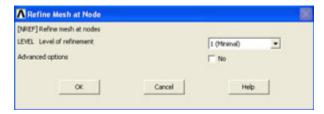
This brings up the Delete Nodes pick menu. Select Pick All.

We will now re-mesh the geometry.

In the MeshTool menu, make sure Global is selected under Element Attributes and click on Set. Select 1 SOLID92 for Element type number. Then, make sure the overall element size level is set to 1 under SmartSize. Also, make sure Volumes is selected in the drop-down list next to Mesh and the default options of Tet and Free meshing are selected under Shape. Click on the Mesh button and click on Pick All.

In the MeshTool, select KeyPoints in the drop-down list next to Refine at:. Click on the Refine button. This brings up the Refine mesh at Keypoints pick menu. Enter 1 under List of items and hit Enter (keyboard). Then, enter 5 and click OK. Recall that keypoints 1 and 5 are located at the point of contact. Keypoint 1 belongs to the lower disk and keypoint 5 to the upper disk. We selected both keypoints as we want to refine the mesh on the upper and lower disks.

This brings up the *Refine Mesh at keypoint* menu. This menu allows us to select the level of refinement we want to achieve. We'll use the minimal option of *1(Minimal)*, which is the default. Click *OK*.



The mesh is now refined around the selected nodes.

We now need to re-mesh the target and contact surfaces. To do this, repeat the steps described in step 5. We also need to re-apply the coupled boundary condition to the upper area of the upper disk. To do this, repeat the steps described in step 6.

Obtain a new solution: Main Menu > Solution > Solve > Current LS

You should get screen similar to the one presented in step 7, showing that the solution has converged.

List the principal stresses: Main Menu > General Postproc > List Results > Nodal Solution

Select Stress from the left list, Principals SPRIN from the right and click OK.

Scroll all the way down in this window. You will find that the new maximum principal stress is -1694.5N/mm<sup>2</sup>.

List the principal maximum displacements: Main Menu > General Postproc > List Results > Nodal Solution

Select DOF solution from the left list, Translation UY from the right and click OK.

You will find that the new displacement for nodes attached to the upper area of the upper disk is again the same and has a value of -0.25800E-01 =0.0258 mm.

We can now compare the new results with the results previously obtained.

|                 | Approach     | Max Prin<br>Stress       |
|-----------------|--------------|--------------------------|
| Theory          | 0.0258<br>mm | -1667N/mm <sup>2</sup>   |
| ANSYS<br>Coarse | 0.0252<br>mm | -1574.9N/mm <sup>2</sup> |
| ANSYS Fine      | 0.0258<br>mm | -1694.5N/mm <sup>2</sup> |

The new value obtained for the approach is the same as the theoretical value. Also, the new value for the max principal stress varies by less than 2% with respect to the theoretical value. This indicates that even though both meshes provide adequate resolution, the finer mesh generates more accurate results.

#### **Exit ANSYS**

Utility Menu > File > Exit

Select Save Everything and click OK.

### Reference

Boresi, A.P., and Schmidt, R.J., Advanced Mechanics of Materials, Sixth Edition, John Wiley and Sons, Inc., 2003.

Go to all ANSYS Learning Modules