

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 Stress
Theory	0.0258 mm	-1667N/mm ²
ANSYS	0.0252 mm	-1574.9N/mm ²

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/mm².

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 Stress
Theory	0.0258 mm	-1667N/mm ²
ANSYS Coarse	0.0252 mm	-1574.9N/mm ²
ANSYS Fine	0.0258 mm	-1694.5N/mm ²

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](#)