# ANSYS - Disks in Point Contact - Step 9



## 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/mm$^2$.

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