ANSYS - Disks in Point Contact - Step 1

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 1: Start-up and preliminary set-up

Create a folder

Create a folder called *contact* at a convenient location. We'll use this folder to store files created during the ANSYS session.

Start ANSYS

Start > Programs > ANSYS 7.1 > Configure ANSYS Classic

Enter the location of folder *contact* as your ***Working directory*.**

Specify *diskscontact* as your ***Initial jobname***. Click on ***Run***.
Resize the ANSYS windows as shown in this screen arrangement so that you can read instructions in the browser window and implement them in ANSYS.

Set Preferences

Main Menu > Preferences

In the *Preferences for GUI Filtering* dialog box, click on the box next to ***Structural*** so that a tick mark appears in the box. Click ***OK***.

> Recall that this is an optional step that customizes the graphical user interface so that only menu options valid for structural problems are made available during the ANSYS session.

Enter Parameters

For convenience, we'll create scalar parameters corresponding to the principal radii of curvature of the upper and lower disks R_1, R_1', R_2 *and* R_2', load *p,* material properties *E, n=nu* and the initial interference between the disks, *inter.*

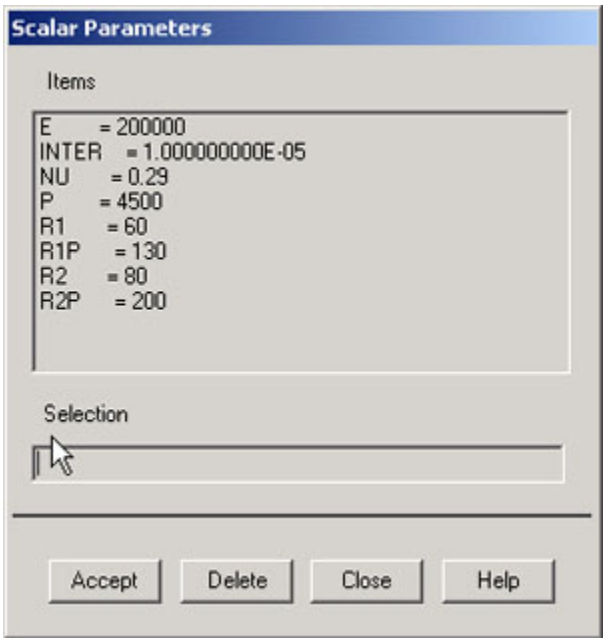
> In static analysis, rigid body motion occurs when a body is not sufficiently restrained. As a result, impractical and excessively large displacements may occur. Furthermore, in problems where rigid body motions are constrained only by the presence of contact, it is critical to ensure that the bodies (and contact pairs) are in contact in the initial geometry. In other words, you want to build your model so that the bodies (and contact pairs) are "just touching." The definition of initial contact is perhaps the most important aspect of building a contact analysis model. For this reason, we will define a scalar parameter *(inter)* to specify the amount of initial interference between the disks and build the geometry so that the disks are just touching.

Utility Menu > Parameters > Scalar Parameters

Enter the parameter value for R_1:
`R1=60`
Click ***Accept***.

Similarly, enter the other parameter values and click ***Accept*** after each.

```
R1p=130
R2=80
R2p=200
inter=1e-5
p=4500
E=2e5
nu=0.29
```



Close the *Scalar Parameters* window. Note that we have specified the radii in mm, the load *p* in N and the Young's modulus in N/mm^2. Therefore, the displacement results we obtain will be in mm and the stresses in N/mm^2.

We can now enter these variable names instead of the corresponding values as we set up the problem in ANSYS. This is also helpful in carrying out parametric studies where one looks at the effect of changing a parameter.

Go to Step 2: Specify element type and constants

Go to all ANSYS Learning Modules