# ANSYS - Disks in Point Contact - Step 7



## Step 7: Solve!

As mentioned in step 1, the definition of initial contact is perhaps the most important aspect of building a contact analysis model. Before we start the solution, we'll verify the initial contact status. Issue the following command in the *ANSYS Command Input window*: *******cncheck

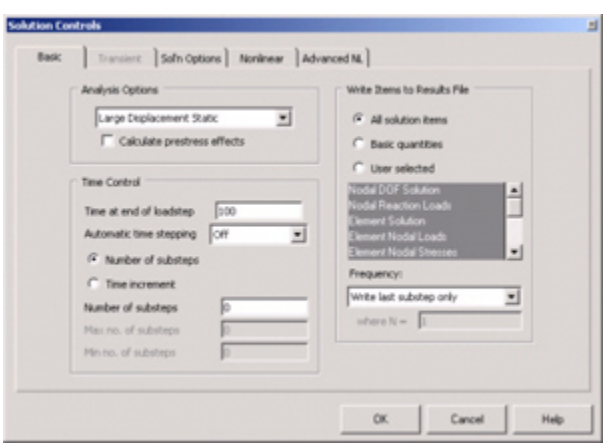

This feature lists the initial status of contact pairs and provides a summary of the contact problem we have set up. Note that an initial penetration of 1E-5 has been detected. This is consistent with the value of parameter *inter=1e*$^{-5}$, which we set up at the beginning of the tutorial. Also, note that 1 contact pair is selected.

### Define Solution Control Options

We'll specify the analysis option as a static analysis in which large deformation effects are to be included. To do this, we'll use the Solution Controls menu.

*Main Menu > Solution > Analysis Type > Sol'n Controls

This brings up the *Solution Controls* menu. Select ***Large Displacement Static*** for ***Analysis Options***. Under ***Time Control***, enter `100` for ***Time at end of loadstep*** and select from the ***off Automatic time stepping*** drop-down menu. Click ***OK***.




### Solve

We are now ready to solve:

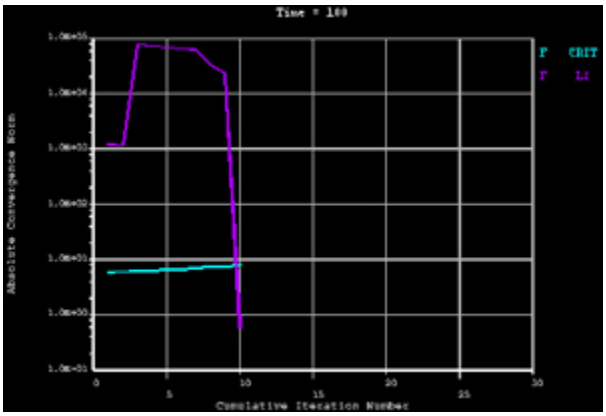
**Main Menu > Solution**

Issue `check` in the *ANSYS Command Input window*. If the problem has been set up correctly, there will be no errors or warnings reported. If you look in the *Output* window, you should see the message: `The analysis data was checked and no warnings or errors were found.`

Main Menu > Solution > Solve > Current LS

Review the information in the */STATUS Command* window. Close this window. Click ***OK*** in *Solve Current Load Step* menu.

ANSYS performs the solution and a yellow window should pop up saying "Solution is done!". Close the yellow window. You should get the following screen, which shows that the solution has converged.




Verify that ANSYS has created a file called *diskscontact.rst* in your working directory. This file contains the results of the (previous) *solve*.

Go to Step 8: Postprocess the results

Go to all ANSYS Learning Modules