- 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 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

CNCHECK Command	2
The NOTE was CP- Deformable-deformable contact pair identific and contact element type 3 has been set up. Nude-surface contact model Contact force based contact model Contact algorithm: Rugmented Lagrange method Contact detection at: modal point (normal to befault contact stiffness factor PNM The resulting contact stiffness Default penetration tolerance factor FIOLM PrictionLess penetration tolerance factor FIOLM PrictionLess penetration tolerance factor PNM MUS- Neerage contact surface length Ruerage contact surface length Befault pinball region factor PINB The resulting pinball region Befault target edge extension factor TOLS -MEMNING: Initial penetration is included.	22.607 TIME 00:11:22 ed by real constant set 1 5 target surface> 1.0000 8.424432+08 8.424432+08 8.10000 9.10511 1.0511 1.0511 1.0000 1.0611 1.0611 1.0.000
www.WOTE www. CP- Max. Initial preservation 1.E-05 was detecte 2730 and target element 2550. Tou may move settre target invrface by : x= - -1.E-05, z= -4.668452477E-13.to reduce initi	22.607 TIME- 00:11:22 ed between contact element -2.652958393E-12, y= ial penetration.
1 CONTACT PAIR IS SELECTED CONTACT PAIR HAVING REAL ID - 1 IS I	INITIALLY CLOSED

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<sup>-5</sup>*, 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.

	Analysis Options	Write Items to Results File
	Large Displacement Static	All solution items
	Calculate prestress effects	C Basic quantities
	Time Control Time at end of loadstep 200 Automatic time steeping Off	Vode Selaces Node Dis Selaces Node Reaction Loads Deward Soldson Deward Node Lad Deward Node L

## 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