

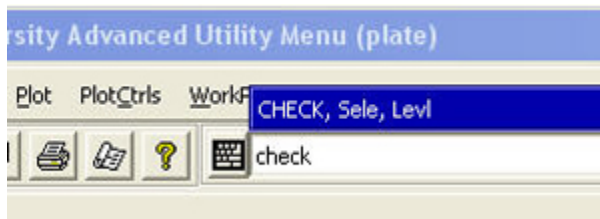
ANSYS 11 - Crank Step 7

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!

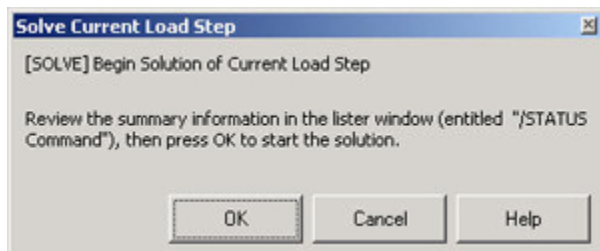
Before we start the solution, we should check our model for errors. Enter `check` in the *Input* window and press Enter.



All warnings and errors found will be displayed in the Output Window. There are no errors but you will see the warnings regarding element shapes that we encountered before. So we're finally ready to kick back and let ANSYS do some of the work: assembling the local and global stiffness matrices and inverting the global system to determine the displacements at the nodes.

Main Menu > Solution > Solve > Current LS

Click **OK** in *Solve Current Load Step* menu.



ANSYS should cheerfully report "Solution is done!"

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

[Go to Step 8: Postprocess the results](#)

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)