ANSYS - Plate with a Hole - Step 7

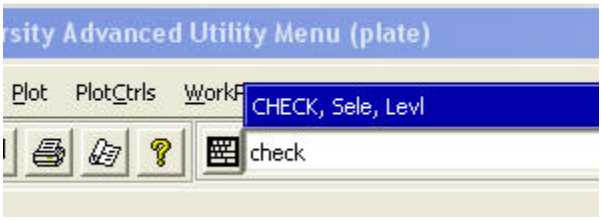
Problem Specification
1. Start-up and preliminary set-up
2. Specify element type and constants
3. Specify material properties
4. Specify geometry
5. Mesh geometry
6. Specify boundary conditions
7. Solve
8. Postprocess the results
9. Validate the results
Problem Set 1

Step 7: Solve!

Enter solution module:

Main Menu > Solution

Enter `check` in the *Input* window and press Enter.



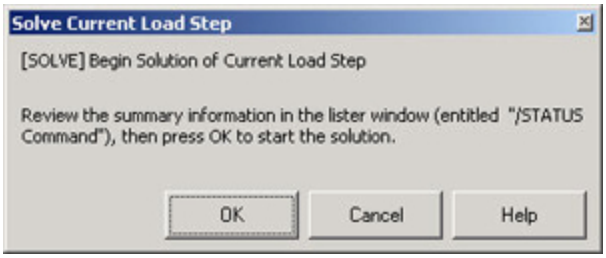
If the problem has been set up correctly, there will be no errors or warnings reported. If you look in the *Output* window, you should see the message: The analysis data was checked and no warnings or errors were found.

Main Menu > Solution > Solve > Current LS

Recall from the truss tutorial that this solves the current load step (LS) i. e. the current loading conditions. In this problem also, there is only one load step.

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 window should pop up saying "Solution is done!". Congratulations! Close the window.

Verify that ANSYS has created a file called *plate.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