ANSYS - Truss Step 7

Step 7: Solve!

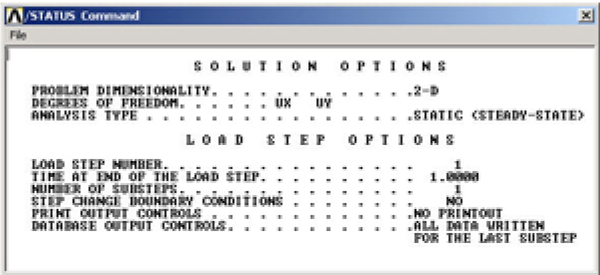
Enter Solution Module

Main Menu > Solution > Solve > Current LS

This solves the current load step (LS) i. e. the current loading conditions. In our problem, there is only one load step; ANSYS allows for multiple load steps that can be solved sequentially without leaving the *Solution* module.

Review the Problem

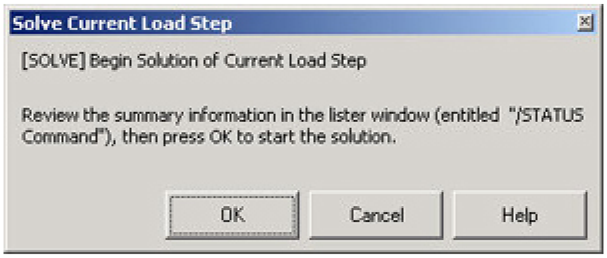
Review the information in the */STAT Command window*. This is a summary of the problem that ANSYS is about to solve.



Close this window.

Perform Solution

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



ANSYS performs the solution and a window should pop up saying "Solution is done!". Congratulations! You just obtained your first ANSYS solution.

Close the window.

In preparation for the postprocessing step to be undertaken next, exit the solution module by closing the *Solution* menu.

Verify that ANSYS has created a file called *truss.rst* in your working directory. This file contains the results of the (previous) *solve*. The *rst* extension in the filename stands for results from a structural analysis. The *truss.db* file contains only steps 1-6. To resume your work subsequent to exiting ANSYS, you'll have to first resume from the *jobname.db* file and then read in the results from the *jobname.rst* file using

Main Menu > General Postproc > Read Results > First Set

This is one of the many ANSYS quirks you'll encounter as you work with the program.

Go to Step 8: Postprocess the results

See and rate the complete Learning Module

Go to all ANSYS Learning Modules