# ANSYS - Truss Step 7



## Step 7: Solve!

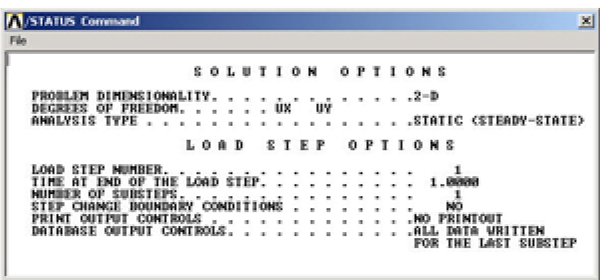
### Enter Solution Module

**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

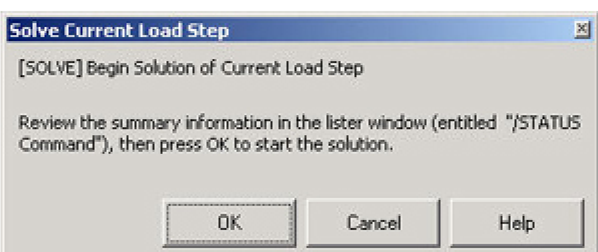
Review the information in the */STAT Command window*. This is a summary of the problem that ANSYS is about to solve.




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