

# ANSYS - Truss 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  
Problem Set 2

## Step 7: Solve!

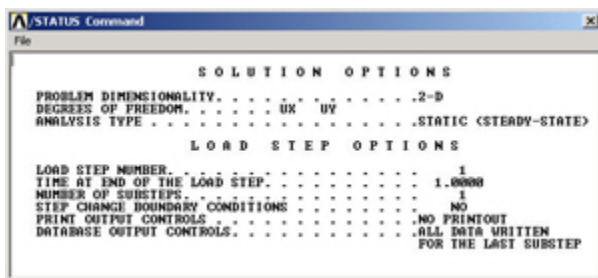
### Enter Solution Module

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

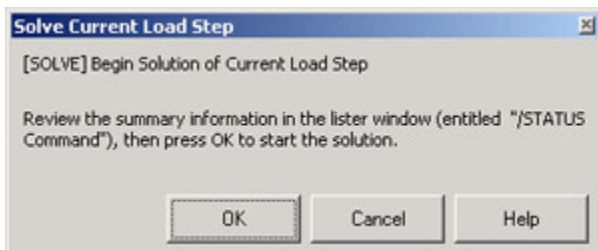
Review the information in the */STATUS Command window*. This is a summary of the problem that ANSYS is about to solve.



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](#)