

ANSYS - Vibration Analysis of a Frame - 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

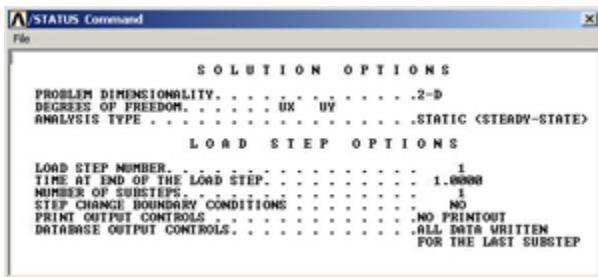
Step 7: Solve!

Enter Solution Module

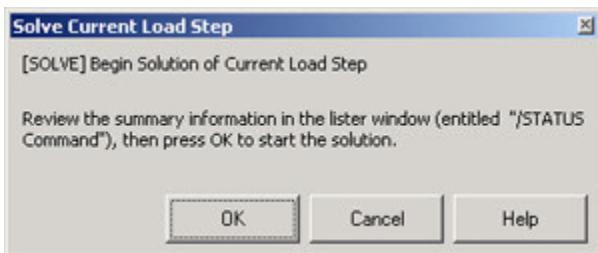
Select in *Main Menu*:

Solution > Solve > Current LS

Review the information in the /STAT Command window.



Close this window.



Click **OK** in *Solve Current Load Step* dialog box.—

ANSYS performs the solution and a yellow window should pop up saying "Solution is done!"

Save your work

Click on **SAVE_DB** in the *ANSYS Toolbar* to save the database.

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

Go to [all ANSYS Learning Modules](#)