

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

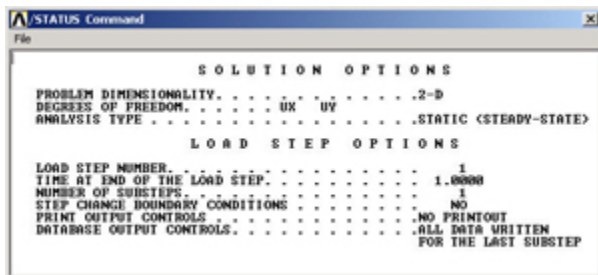
Step 7: Solve!

Enter Solution Module

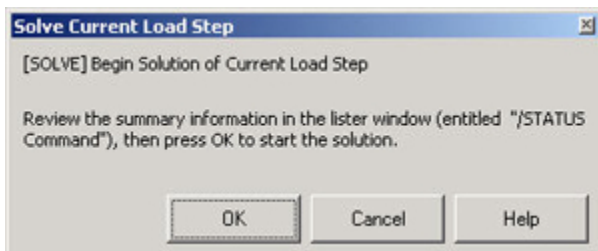
Select in *Main Menu*:

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](#)