# ANSYS - Vibration Analysis of a Frame - Step 7



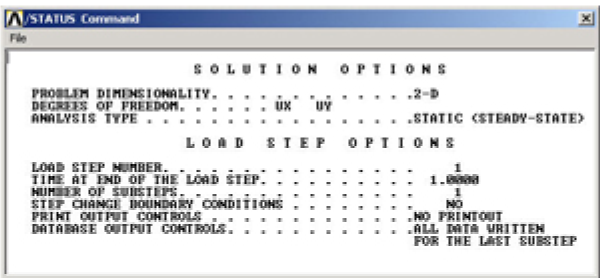
## Step 7: Solve!

### Enter Solution Module

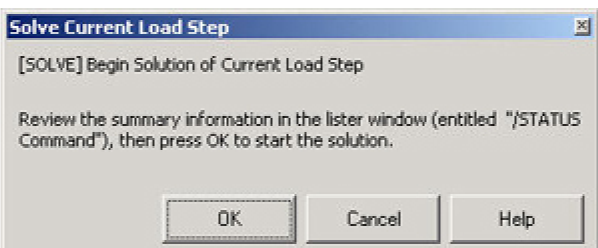
Select in *Main Menu:*

**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