

ANSYS - Vibration Analysis of a Frame - Step 3

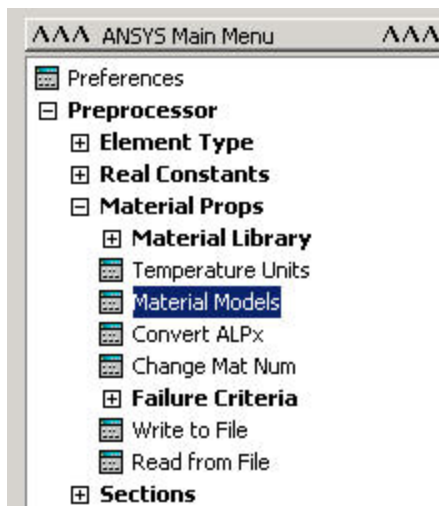
Problem Specification

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

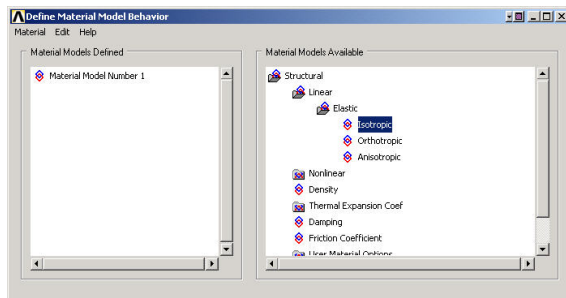
Step 3: Specify material properties

Enter the Define Material Model Behavior menu

Select Main Menu > Preprocessor > Material Props > Material Models

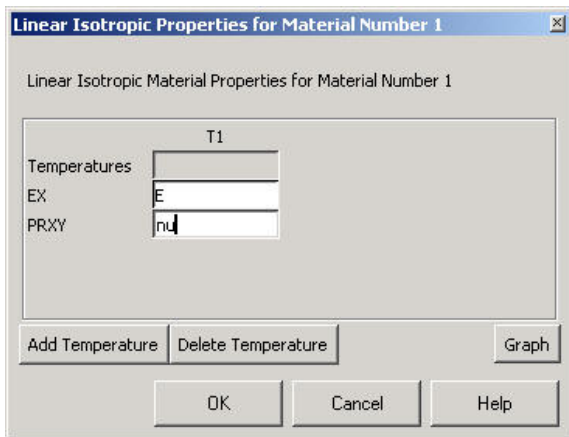


In the *Define Material Model Behavior* menu, double-click on *Structural, Linear, Elastic, and Isotropic*.



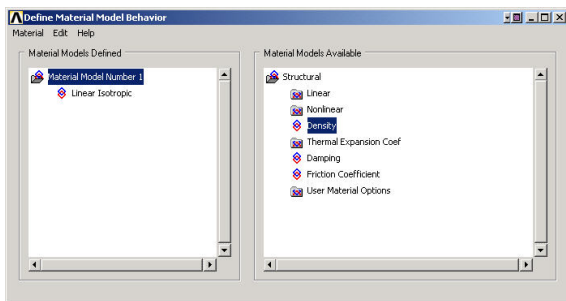
Specify Material properties

Enter E for Young's modulus *EX*, nu for Poisson's Ratio *PRXY*.

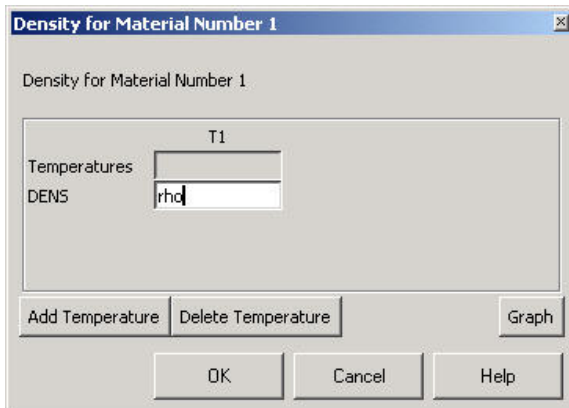


Click **OK**.

Double-click on **Density** under **Structural**.



Enter rho for **DENS**.



Click **OK**.

This completes the specification for Material Model #1. Close the *Define Material Model Behavior* menu.

Save your work

Click on the **SAVE_DB** button in the *ANSYS Toolbar*.

Go to [Step 4: Specify geometry](#)

Go to [all ANSYS Learning Modules](#)