

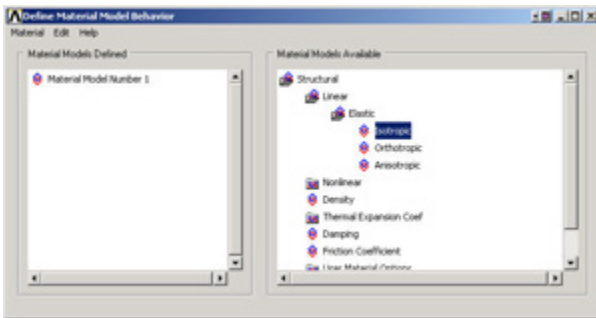
ANSYS - Plate with a Hole - Step 3

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
Problem Set 1

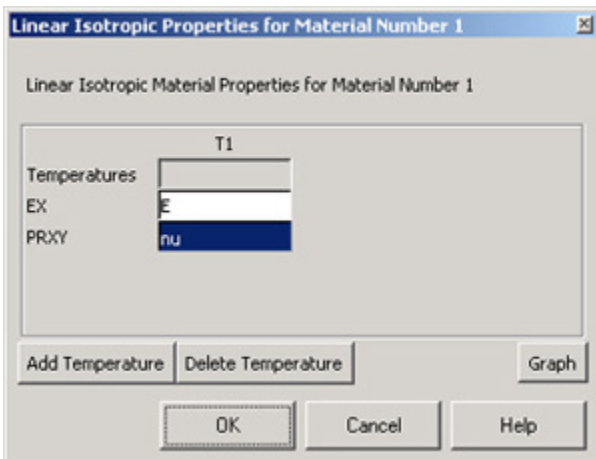
Step 3: Specify material properties

Main Menu > Preprocessor > Material Props > Material Models

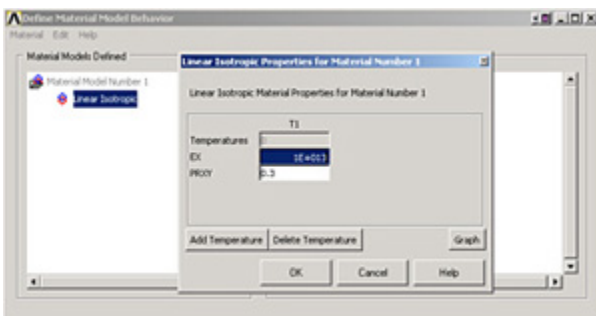
In the Define Material Model Behavior menu, double-click on **Structural**, **Linear**, **Elastic**, and **Isotropic**.



We'll use the previously defined parameter names while specifying the material properties. Enter *E* for Young's modulus **EX**, *nu* for Poisson's Ratio **PRXY**. Click **OK**



To double-check the material property values, double-click on **Linear**, **Isotropic** under **Material Model Number 1** in the Define Material Model Behavior menu. This will show you the current values for EX and PRXY. **Cancel** the Linear Isotropic Properties window.



When you enter parameter names, ANSYS substitutes the corresponding parameter values as soon as you click **OK** or **Apply**.

This completes the specification of *Material Model Number 1*. When we mesh the geometry later on, we'll use the reference no. 1 to assign this material model. Close the *Define Material Model Behavior* menu.

Save your work

Toolbar > SAVE_DB

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

[See and rate the complete Learning Module](#)

[Go to all ANSYS Learning Modules](#)