

ANSYS - Plate with a Hole - Step 3

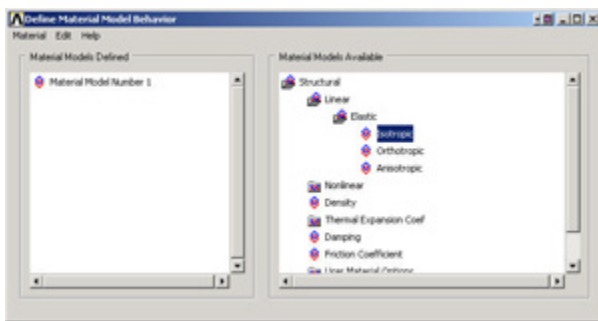
Problem Specification

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

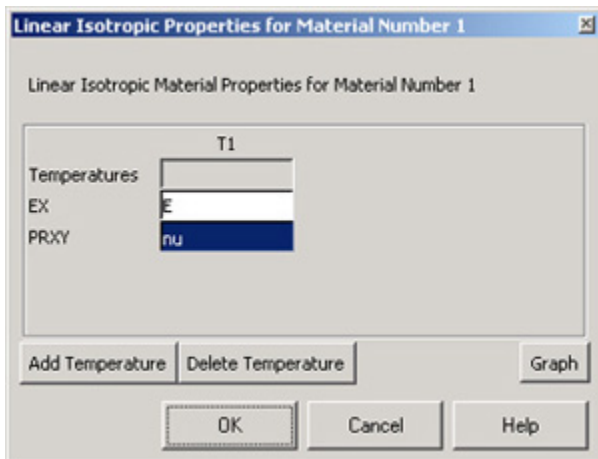
Step 3: Specify material properties

Main Menu > Preprocessor > Material Props > Material Models

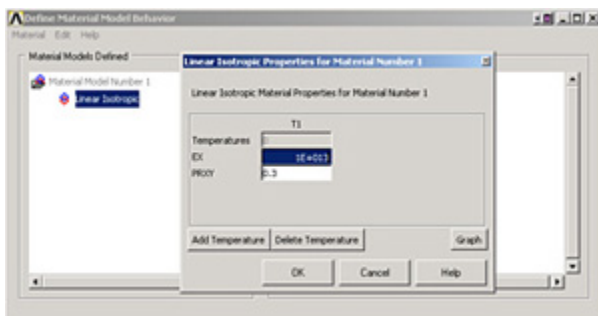
In the Define Material Model Behavior menu, double-click on **Structural, Linear, Elastic**, and **Isotropic**.



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](#)