

ANSYS 11 - Crank Step 3

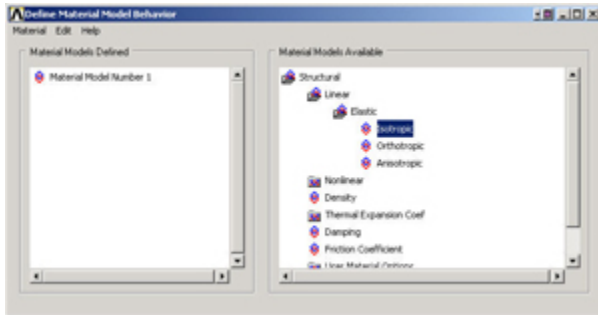
Problem Specification

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

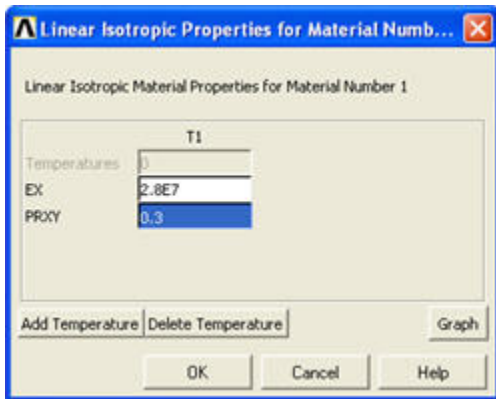
Step 3: Specify material properties

Main Menu > Preprocessor > Material Props > Material Models

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



Enter 2.8E7 for Young's modulus **EX**, 0.3 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.

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](#)