

ANSYS - Disks in Point Contact - Step 3

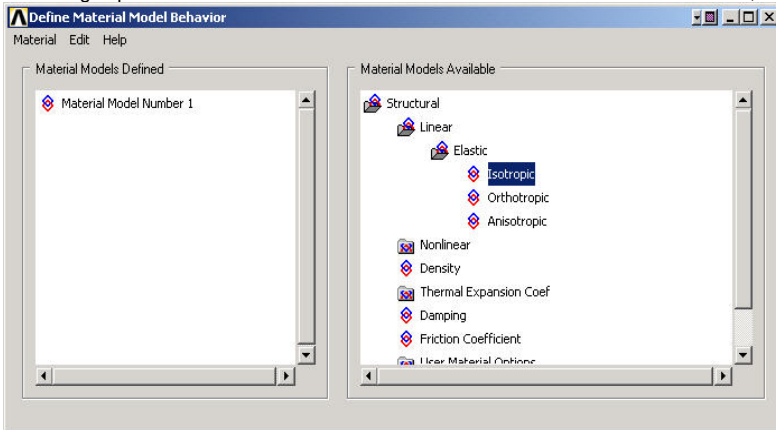
Problem Specification

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

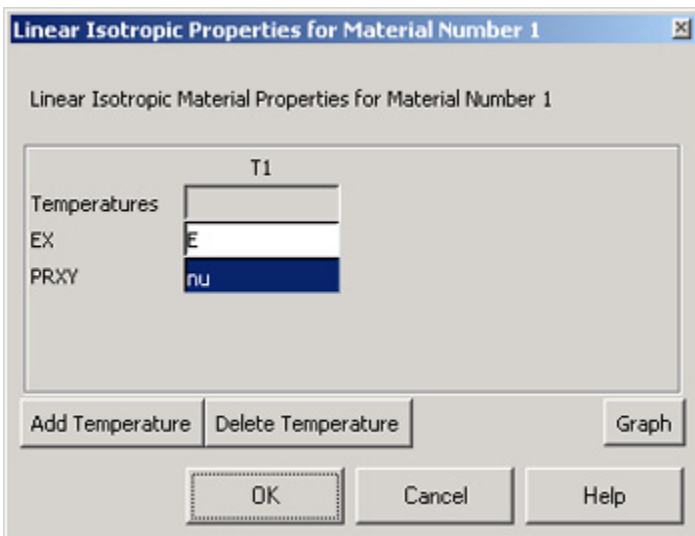
Step 3: Specify material properties

Main Menu > Preprocessor > Material Props > Material Models

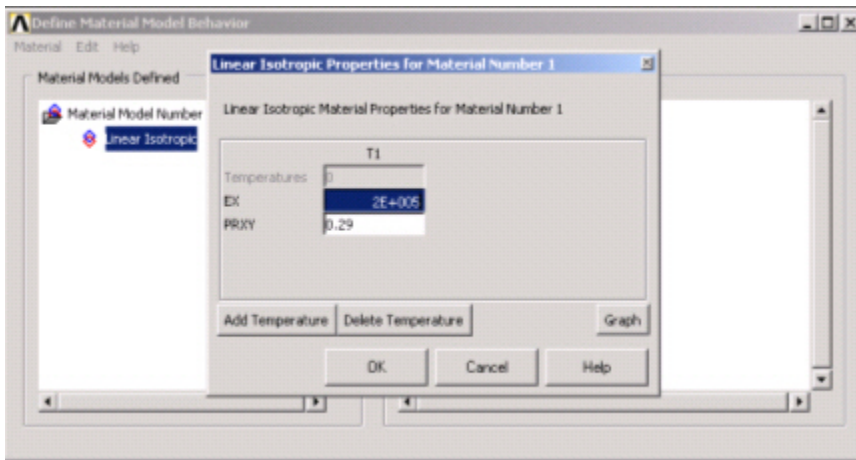
This brings up the *Define Material Model Behavior* menu. Double-click on **Structural**, **Linear**, **Elastic**, and **Isotropic**.



We'll use the previously defined parameter names to specify 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*. Close the *Define Material Model Behavior* menu.

Save your work

Toolbar > SAVE_DB

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

Go to [all ANSYS Learning Modules](#)