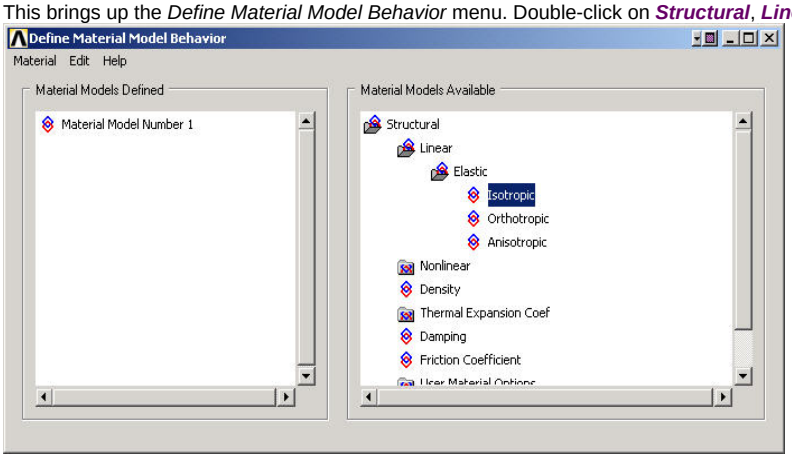
ANSYS - Disks in Point Contact - Step 3

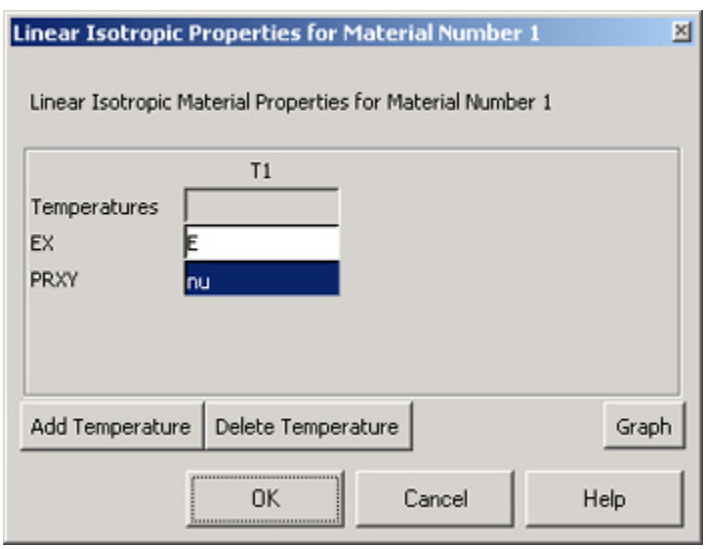
Step 3: Specify material properties

Main Menu > Preprocessor >Material Props > Material Models ....

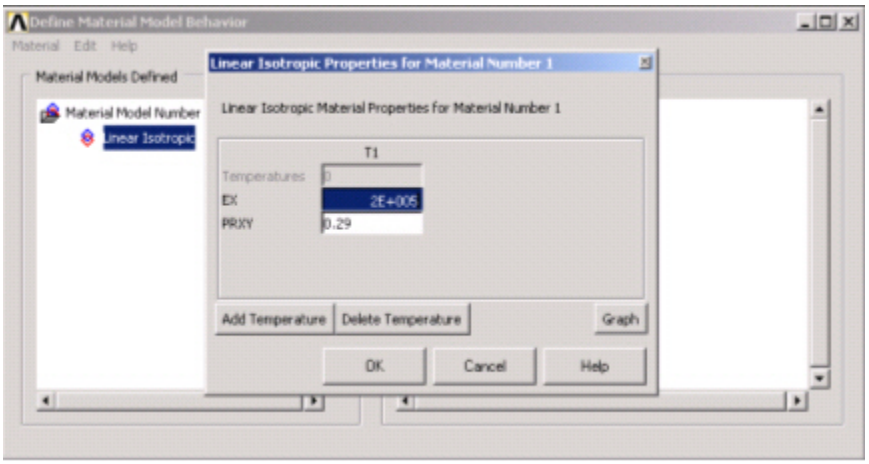
This brings up the *Define Material Model Behavior* menu. Double-click on ***Structural***, ***Linear***, ***Elastic***, and ***Isotropic***.



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