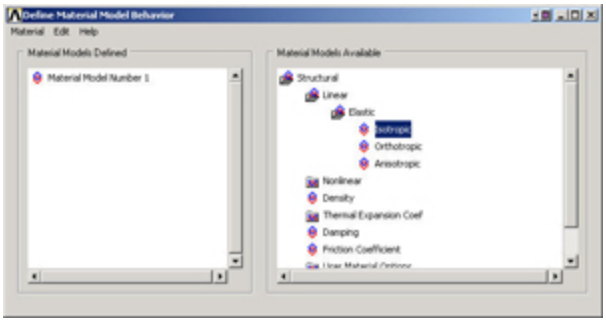
ANSYS - Truss Step 3

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
Problem Set 2

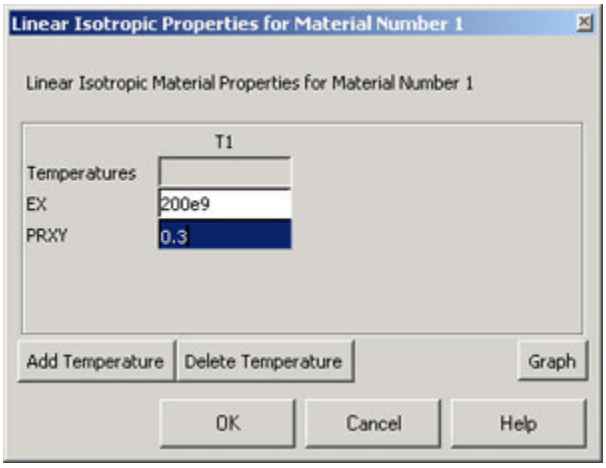
Step 3: Specify material properties

Main Menu > Preprocessor > Material Props > Material Models

In the ***Material Models Available*** Frame of the *Define Material Model Behavior* window, double-click on ***Structural***, ***Linear***, ***Elastic***, and ***Isotropic***.



Enter `200e9` for *Young's Modulus* ***EX***.
Enter `0.3` for *Poisson's Ratio* ***PRXY***.



Click ***OK***. 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

Utility Menu > File > Save as Jobname. db

This saves all the relevant data into one file called *truss.db* in your working directory, *truss* being taken from the jobname and *db* being an abbreviation for database. Verify that ANSYS has created this "database file" in your working directory. You can restart from your last save at any time using

Utility Menu > File > Resume Jobname. db or ANSYS Toolbar > RESUM_DB

Each time you successfully finish a series of steps, you should save your work. Unfortunately, ANSYS doesn't have an undo button (though that is the first thing I needed while learning ANSYS!) and one way to recover from mistakes is to resume from your last save.

Go to Step 4: Specify geometry

See and rate the complete Learning Module

Go to all ANSYS Learning Modules