

# ANSYS - Truss Step 3

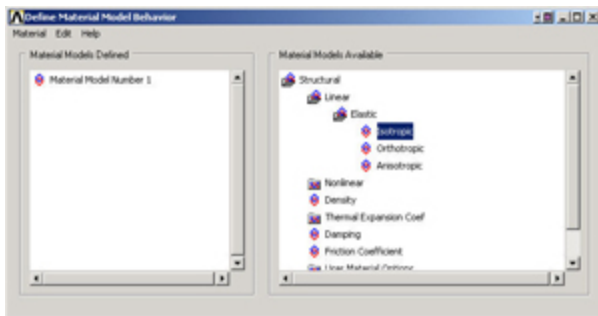
## Problem Specification

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

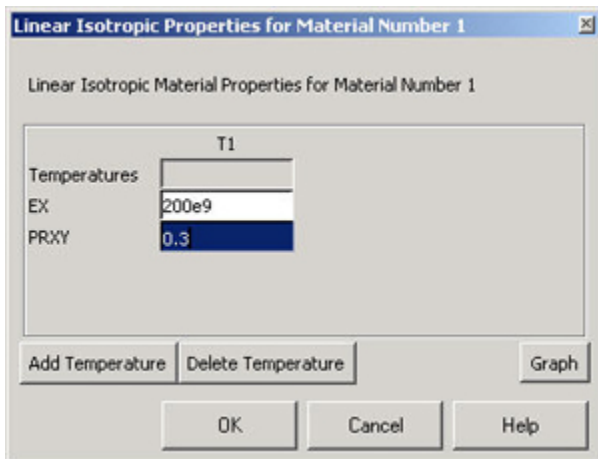
**Main Menu > Preprocessor > Material Props > Material Models**

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](#)