

ANSYS - Truss Step 3

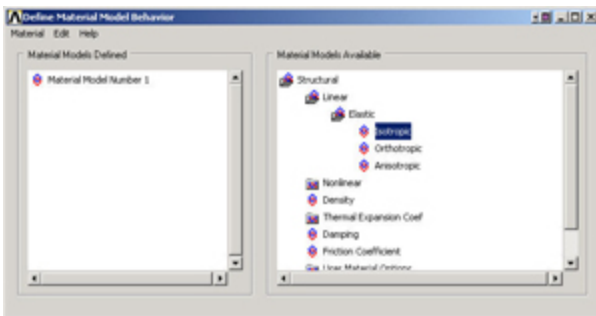
Problem Specification

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

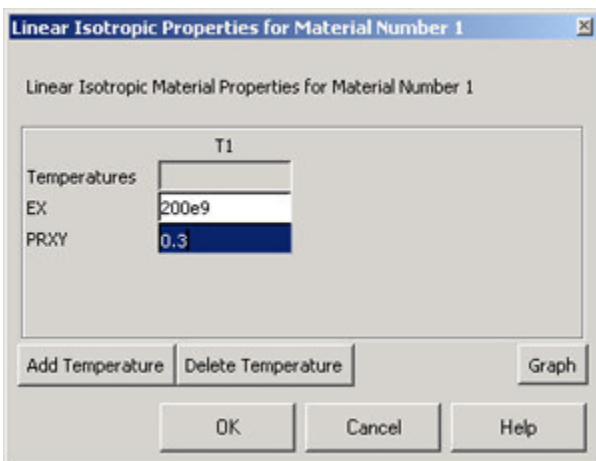
Main Menu > Preprocessor > Material Props > Material Models

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](#)