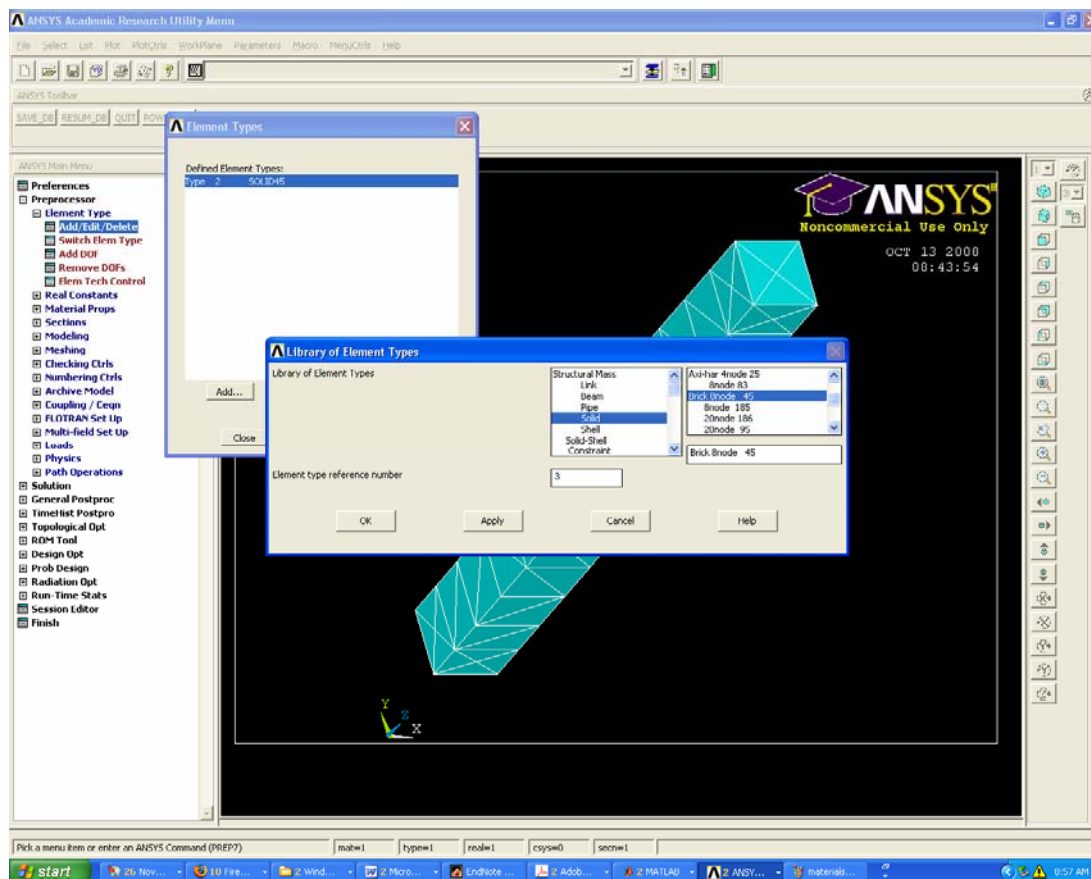


MECH3230 FEM Project #1: Hints

For those of you who are having a hard time using ANSYS for the project, I suggest that you go through the ANSYS Tutorials. Click on the Ansys 'Help' menu and select 'ANSYS Tutorials'. I then recommend that you complete Chapter 2. Structural Tutorial and Chapter 8. Contact Tutorial. Although you do not need to use Contact Elements, the Contact Tutorial uses 3-D elements.

I've also written some hints for you below:

To select 8 node brick element (Solid45) see the figure below and follow the same links:

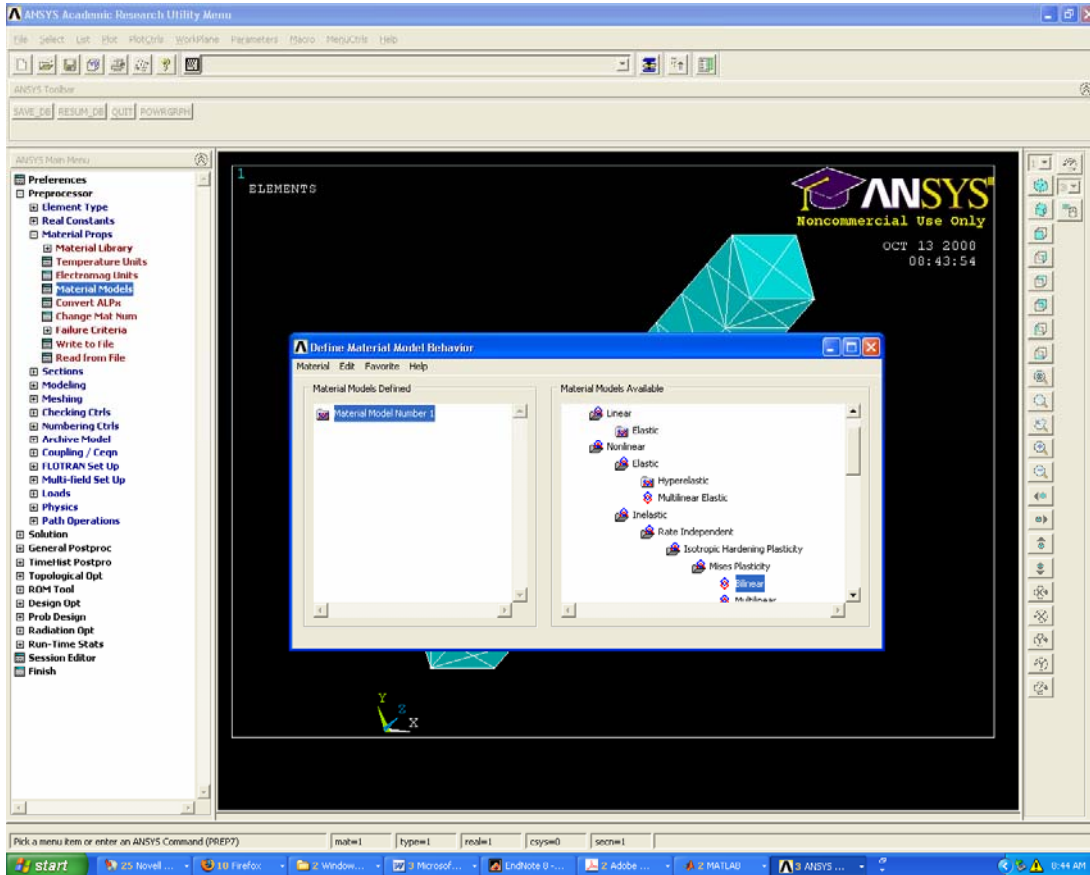


To set material properties for elastic perfectly plastic:

Follow the following series of selections: Material Prop -> Material models -> Structural -> Nonlinear -> Inelastic -> Rate Independent -> Isotropic -> Hardening Plasticity -> Mises Plasticity -> Bilinear

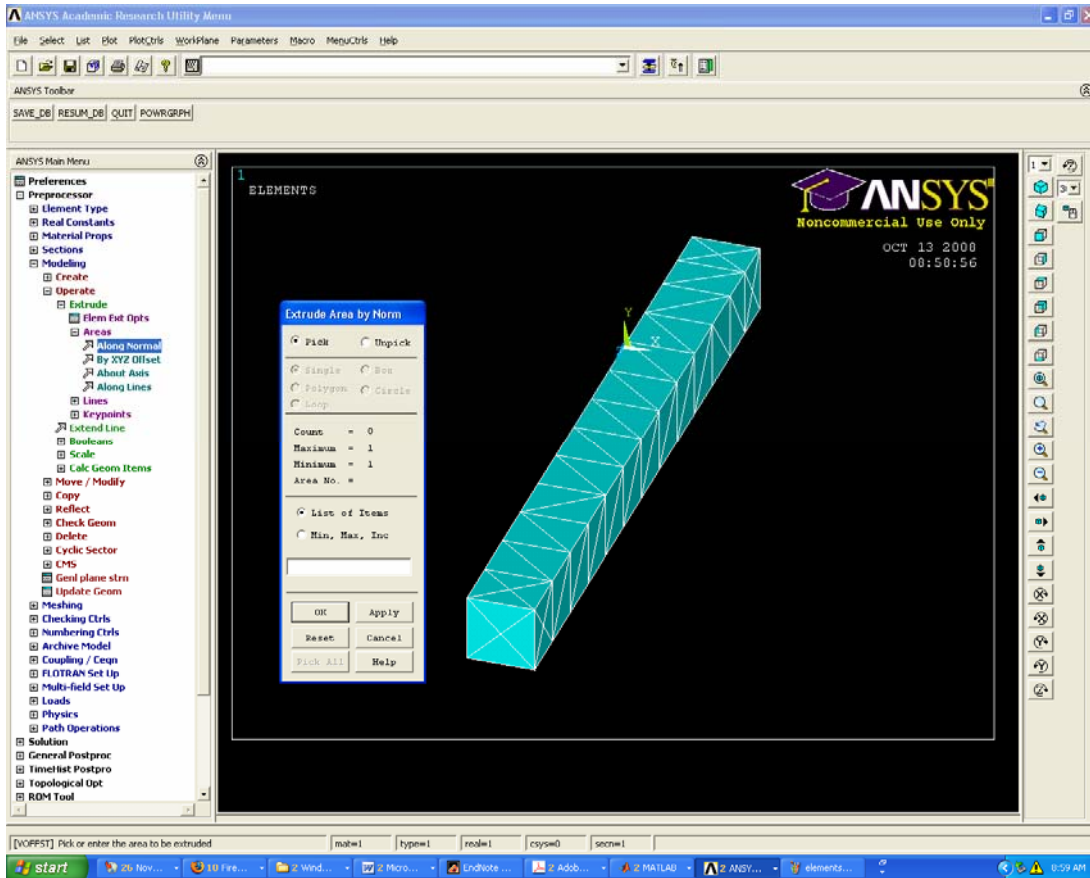
Then enter the yield strength and enter 0 for the Tang Mod (Tangent Modulus). A tangent modulus greater than 0 would result in hardening (it defines the slope of the

stress-strain curve after the yield strength is reached). If you do have problems with convergence of the model, you can apply a low Tang Mod to help (like 1 or 2% of E).



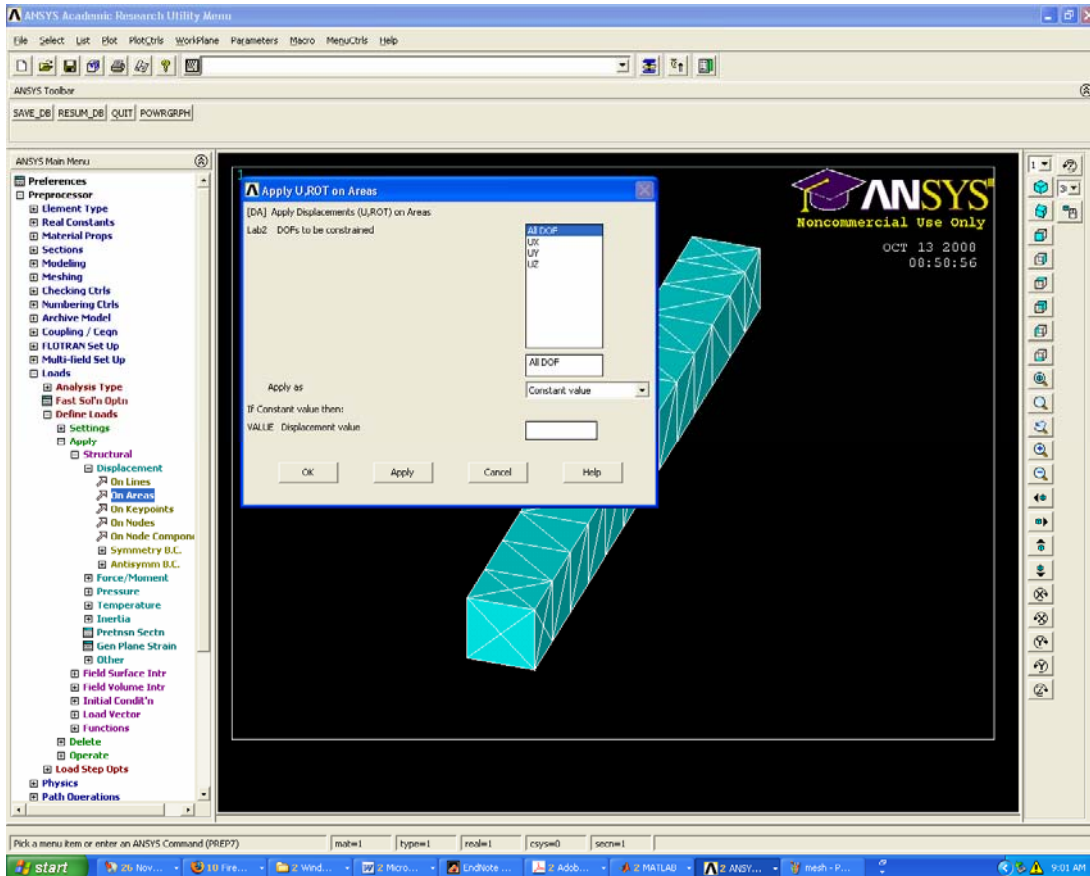
Probably the best way to create your beam is to create the cross section using rectangles. Then use the 'Add' command under Modeling->Booleans to create a single area.

Then Use the 'extrude' command under the Modeling->Operate menu. Then extrude 'Along Normal', as shown in the next picture.



Then just mesh your volume and add the boundary conditions and loads.

To apply the zero displacement condition at one end to make the beam a cantilever beam here is a picture of the menu:



Then you have to apply your load at the end of the beam. This can be tricky, since if you add just a point load at the end of the beam it will give you unrealistically high loads. You apply the displacement to all the nodes at the end of the right side of the beam. You can figure how to do this on your own.

Once everything is properly set-up, tell ANSYS to solve the problem by going to Solution -> Solve -> Current LS

Use a white background for all plots. You can do this by following from the menu bar: PlotCtrls -> Hard Copy -> To File. Then make sure 'Reverse Video' is checked.

When you are including plasticity and you are having trouble getting it to converge and give you an answer, try using a Tangent Modulus in your material properties of 1 or 2% of the elastic modulus. This will effect the solution because then you will have some hardening in your model. Not everyone will have this problem due to differences in meshing and how you apply your load.

This all should be very easy to do. There are also multiple ways to do this so that there is not only one answer. Hopefully everyone's project will not look the same, and that is probably OK. This is a design problem similar to what you might see in industry, so you need to gain experience in open ended problems such as these.