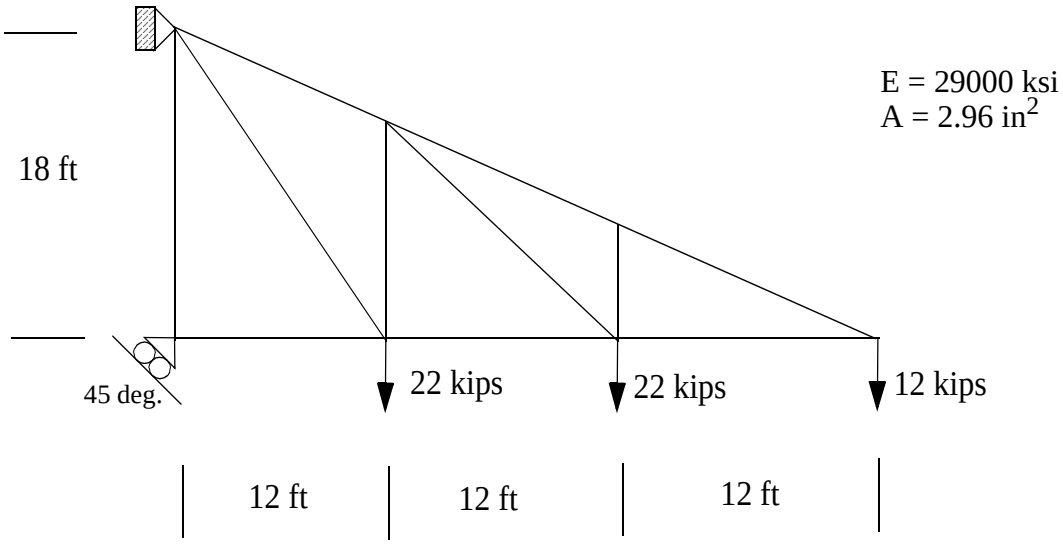
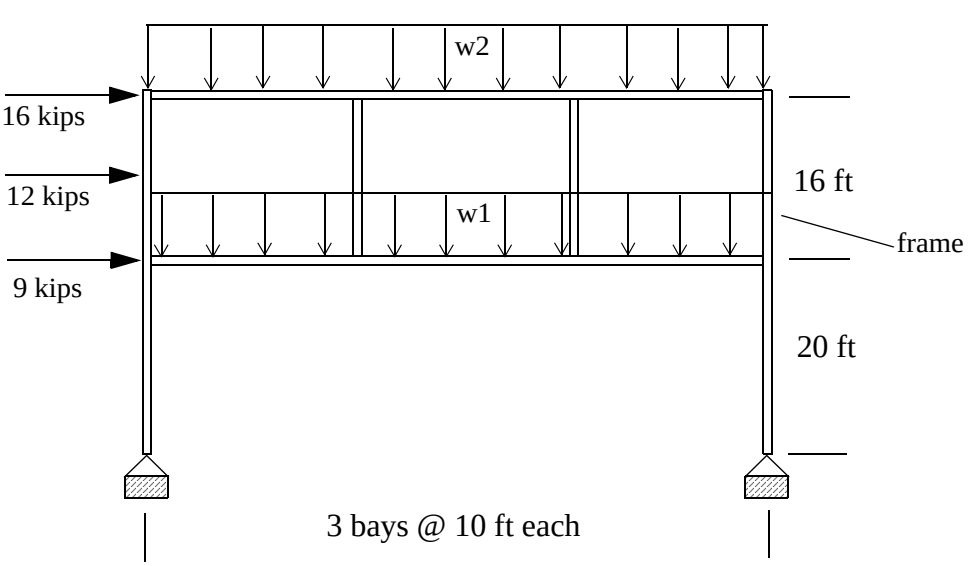
HW #10 assigned 4/15/03, due 4/21/03 (drop off at my office Monday)

This assignment will tackle a variety of modeling issues. In the context of this class, 'modeling' refers to the need to properly represent structures as they exist physically (sometimes trickier than you might think). Each problem addresses a different modeling concept that will be covered in class.

Problem 1) Analyze the following truss structure using VA. Submit a write-up as shown in the example on the web, replacing a moment/shear with a diagram of axial forces. Identify the members with maximum tension and compression.



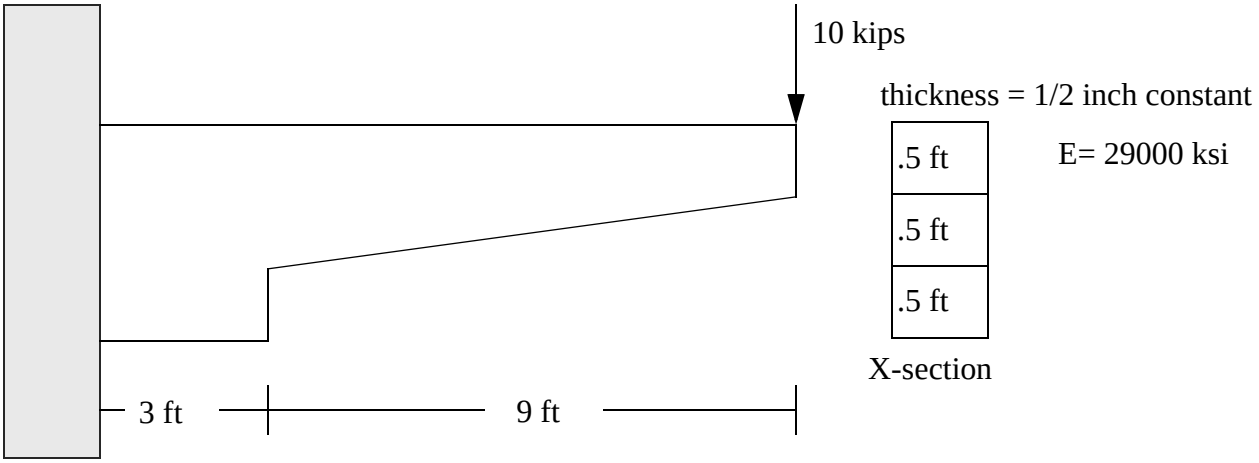
Problem 2)
Use VA to analyze the frame structure below. Do a frame instability (P-DELTA) analysis and a standard linear analysis (the third and first choices, respectively under Static Analysis Type) and present a comparison of the following: 1) maximum lateral displacement in structure, 2) maximum moment at any joint in the structure, 3) and maximum tension in truss members. Present the moment diagrams, displaced shape and model as usual.



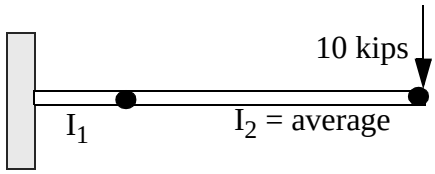
w1 = 5.5 kips/ft
w2 = 7.5 kips/ft
I(column) = 800 in4
I(beam) = 600 in4
A(column) = 6 in2
A(beam) = 3 in2

E = 29000 ksi

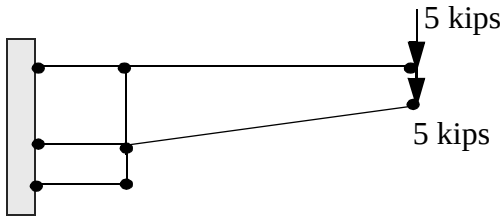
Problem 3) The cantilevered beam below has a constant thickness, and a height that varies linearly after the first 3 feet. The dimensions are given in the cross section figure. Use VA to analyze this beam by trying several models. .



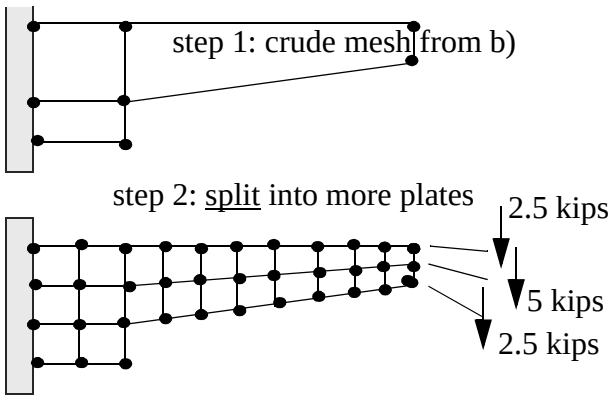
a) Use 2 **beam elements,** one for the constant section, and one representing the average of the tapered section with the 'I' value taken as the average. Report the tip deflection, and location and magnitude of maximum moment.



b) Using 3 **plate** elements, these are under the frame option when you create a new model. Set up the nodes alone, then go to Model -> Create -> New Plate to connect the nodes with plates. Report the tip deflection, and location and magnitude of minimum and maximum Global Stress in X direction (Look at the stress contour plots by using the View Manager after analysis, Plates => MidPlane Stress => Global Sigma X). Note that the point load is to be represented as two loads as shown.



c) This time use 22 **plate** elements. Report the same information as part b). You don't have to create each element one at a time. You can take advantage of the automatic element generation to make this go quickly (Model => Generate Standard), **or** just go to the step-1 figure to the right from part b, left- and then right-click the members and select 'split plate' into as many as you want. Nodes will be created for you (use View -> View Manager -> Nodes -> Names) to see their numbering. Note that the point load is to be represented as three loads as shown.



d) Make a table comparing the maximum displacement results from parts a, b, and c). What does the difference between results for b and c say about the accuracy of these models?

e) Compare the minimum and maximum Global X stresses for parts b and c. Looking at the stress contours, why can we say that the model with more elements is 'better'?