CE 222 HW # Solution

Dan Simkins

April 9, 2001

1 Problem 1

Analyze the given cantilever for two different boundary conditions.

First, we must compute the equivalent nodal loads. Since we are using constant strain triangles with linear variations of shape functions along the edge, uniform distributed loading always yields equal distribution of loads to the two nodes on the element side that is loaded. Thus, computing the nodal loads is easy as the following example of uniform shear traction on end of cantilever:

$$w = \frac{20 \text{ kip}}{10 \text{ in}} = 2 \text{ kip/in}$$

Assume there are 5 elements through the depth (the beam model requires an even number of elements through the depth).

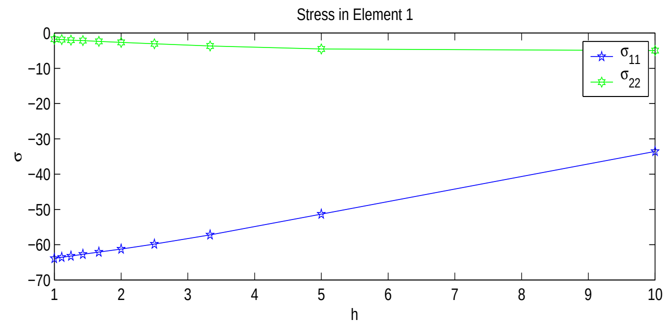
$$\text{element size} \qquad h = \frac{10}{5} = 2 \text{ in}$$

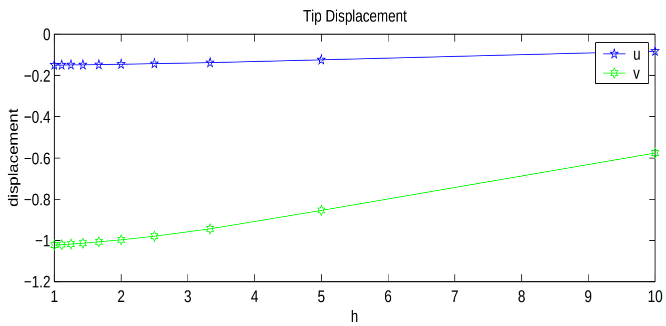
$$\text{element nodal load} \qquad p_e = \frac{w\dot{h}}{2} = 2 \text{ kip/node}$$

Note that interior nodes get two contributions, one from each adjoining element, and the corner nodes only get the single contribution. So, in this example, the top and bottom nodes get 2 kip, and the nodes through the depth get 4 kip each.

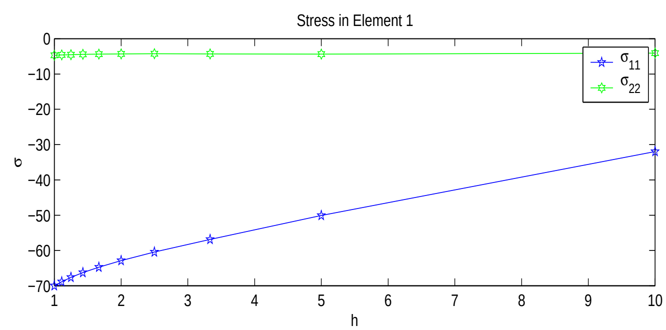
I have provided 3 MATLAB files. The first two are the model files for the two boundary conditions, based on the *beam.m* model file provided as an example. The third file runs the calculations and generates graphs. I chose to keep an aspect ratio of 1:1 for my elements.

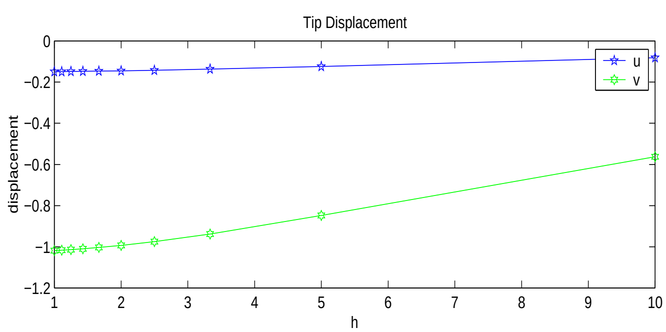
The following figure is a plot of the bottom tip displacement as a function of the element size, h, and the normal stresses at the bottom base of the beam for the case with tractions specified on the LH edge.





And for the displacements specified on LH edge





The displacements for each case are:

	Traction BC	Displacement BC
x-disp	-0.1495	-0.1492
y-disp	-1.0221	-1.0188

We can see that the case restraining the displacements is slightly stiffer than the case with tractions specified. The reason is that, due to Poisson's Ratio, the material wants to expand or contract in the orhtogonal directions to the main longitudinal strain. In the traction boundary condiditon case, the beam is free to move laterally at the support, whereas in the displacement boundary condition case it is not. You can see in the above graphs that the σ_{22} component of stress at the fixed end is different in the two cases.

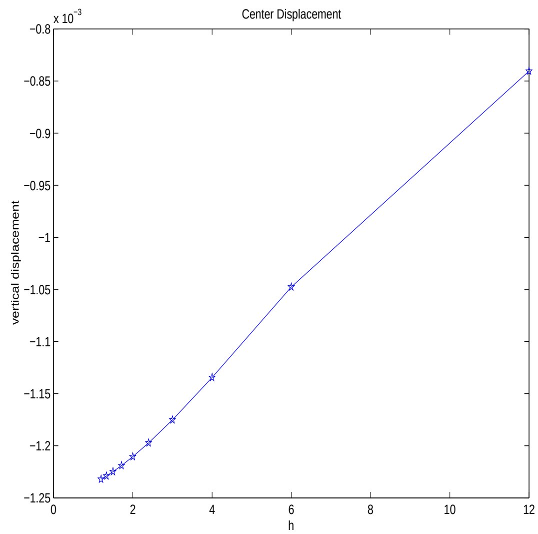
The convergence should be quadratic for the displacement, and linear for stress/strain, as discussed in the lecture notes. The graphs seem to show this behavior, although the displacment graph is rather flat.

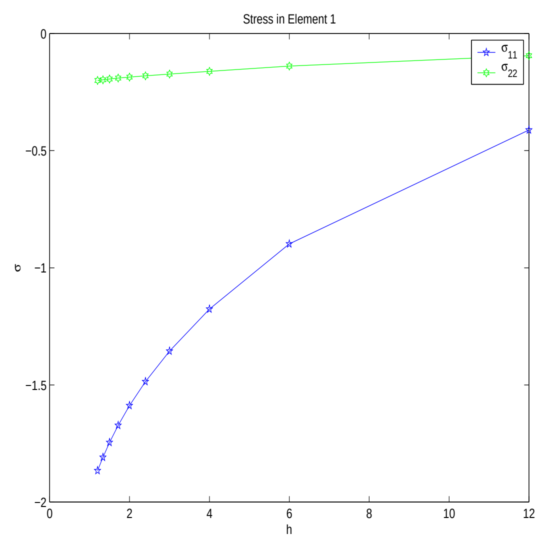
2 Problem 2

You are asked to solve the fixed-fixed beam by using symmetry. This is done by splitting the beam in half and putting roller supports preventing horizontal displacements along the cut edge.

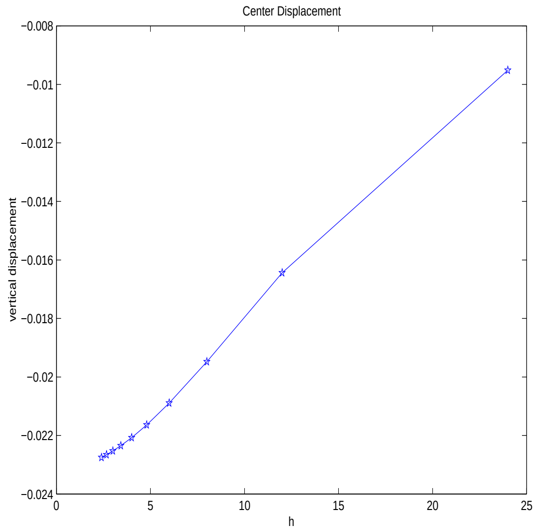
The displacement at the bottom center of the beam and stresses in element 1, the element at the bottom of the fixed support, are plotted below.

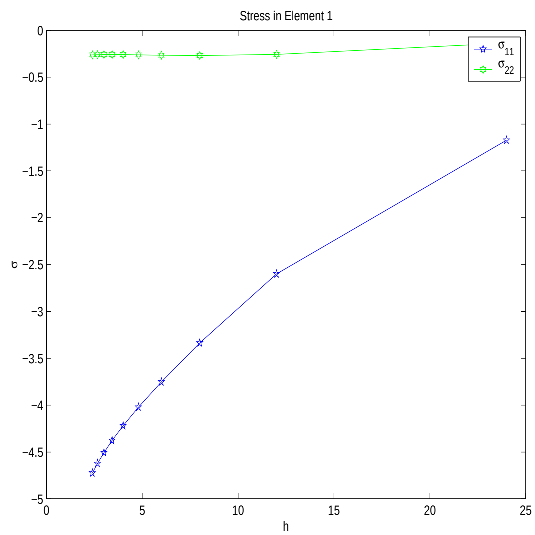
The 72 in beam,





And for 240 in beam



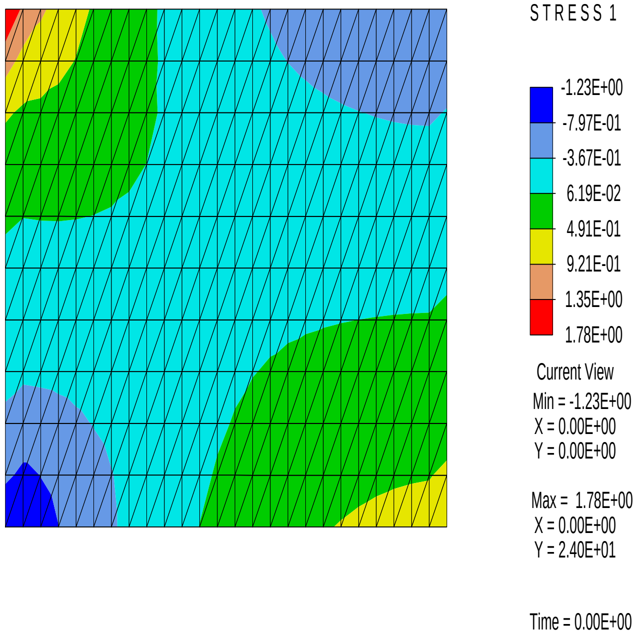


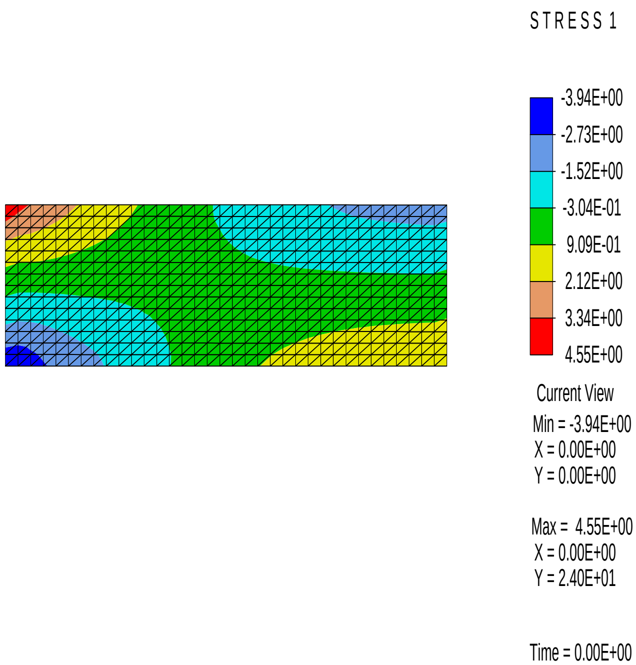
I have provided two files, p2.m and beam.m, that I used to compute these graphs.

The displacements for each case are:

	L=72	L=240
x-disp	0.0	0.0
y-disp	−0.001232	−0.022752

The following show the bending stress in the short beam and long beam, respectively, as computed by FEAP. I used FEAP only because of the problem the MATLAB contouring package seems to exhibit, the calculation were done with FEDEASlab, and compared with FEAP.





2.1 Discussion

The bending stress distribution as shown above fits with our intuition for these two beams - the beams are bent in double curvature, and we see the bending stress changing sign from one end to the other, with the largest absolute values at the top and bottom of the beam.

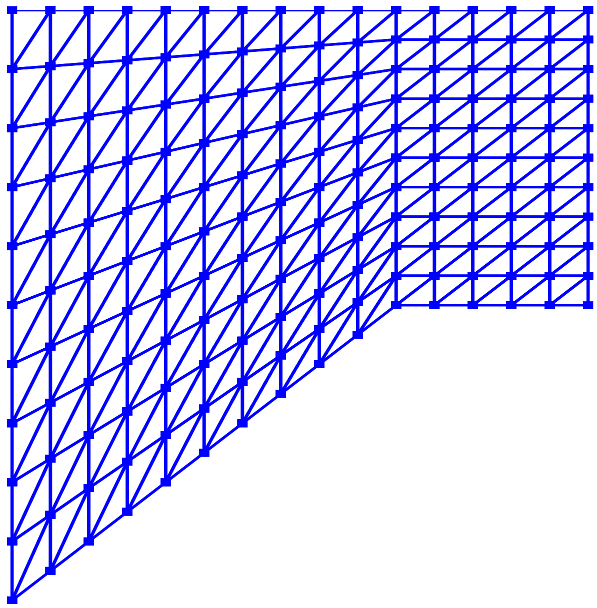
The displacement plots show the convergence of the solution, which is quadratic for the displacement. Why is the graph not exactly quadratic? Recall, the error is proportional to h^2, but that constant depends on the element size, which is what we see for the largest element size. What about the stress? See the discussion for Problem 3 for an explanation.

3 Problem 3

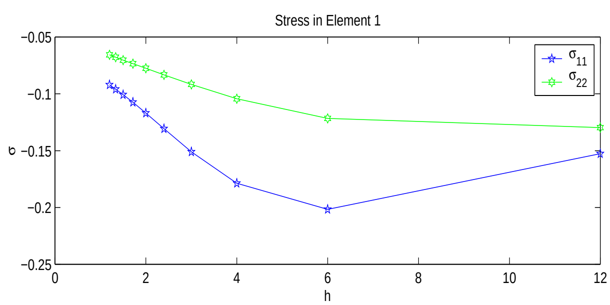
The haunched beam can be meshed by using two *block*'s, one a trapezoid and one a rectangle and *tie*-ing them together. I have provided two MATLAB files, *p3.m* and *beam.m* that solves this problem.

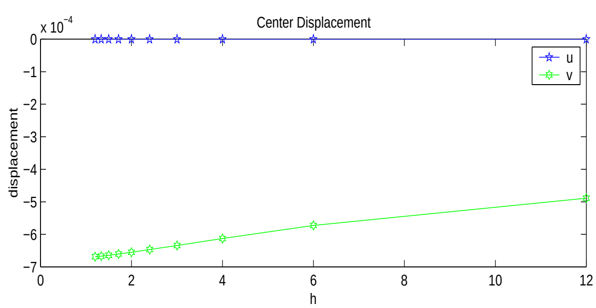
Note, there is a bug in the *tie* command that didn't properly connect these meshes, it has been corrected.

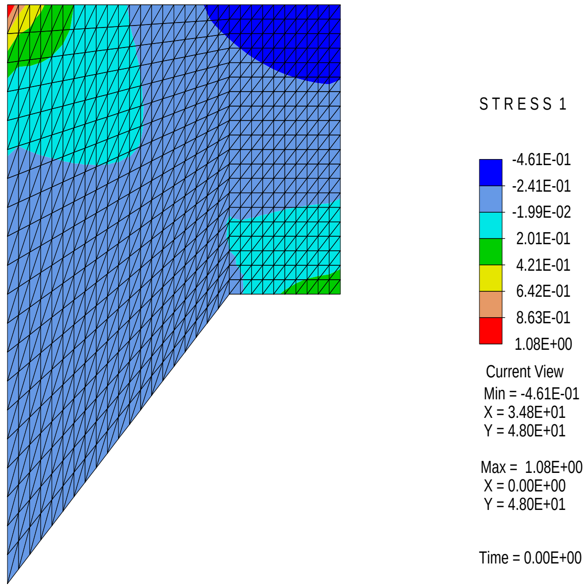
The mesh looks like:

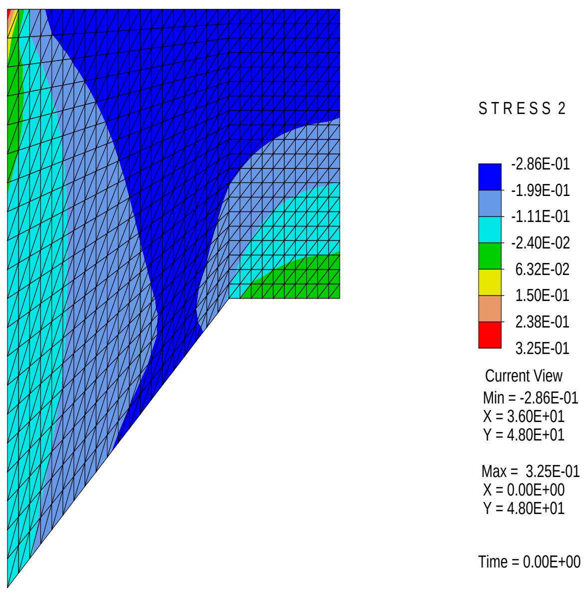


The stresses and displacements:









The displacements are:

x-disp	0.0
y-disp	$-6.6898 \cdot 10^{-4}$

Notice that the bending stress at the support doesn't show the concentrated compression as it did in the rectangular beam case. Also, note that there is some stress concentration at the re-entrant corner.

You should be suspicious when you see a quantity, such as the bending stress in the above graph, change non-monotonically during mesh refinement. However, in this case, it can be understood by realizing that the stress reported is the stress at the center of the element. Since relatively large elements were used at first, the point that the stress is reported at changes quite abit for the first few refinements. Thus, it isn't surprising to see this behavior, since we aren't comparing stress at the same point! However, once the elements start to get fairly small, we don't expect to see such large changes. FEAP does the same thing, but it reports stresses at the first Gauss point, rather than the center.