University of California, Berkeley
CEE C133/ME C180, Engineering Analysis Using the Finite Element Method
Spring 2009
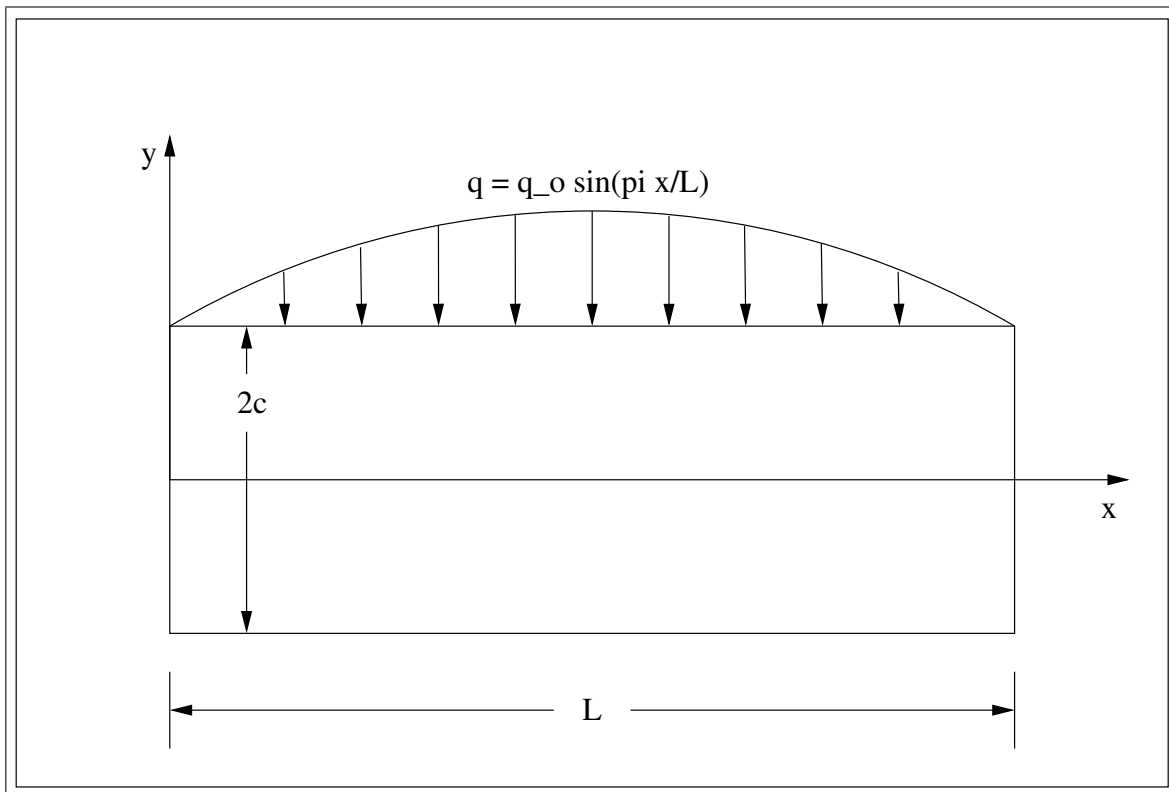Instructor: S. Govindjee
GSI: N. Hodge

Lab 10

- Consider the following figure. It consists of a two-dimensional beam subject to transverse sinusoidal loading. Model this in COMSOL, with the boundary conditions similar to the exact solution given below.

- The exact solution of the y-displacement along the centerline of a beam with the boundary conditions

$$u(0,0) = v(0,0) = v(L,0) = 0$$

and the condition $L \gg c$ is

$$v(x, 0) = -\frac{3q_0 L^4}{2c^3 \pi^4 E} \sin\left(\frac{\pi x}{L}\right) \left[1 + \frac{1 + \nu}{2} \frac{\pi c}{L} \tanh\left(\frac{\pi c}{L}\right)\right].$$

As a reference, you might try the values $L = 20$, and $c = 1$, so that the total length of the beam is 10 times the width.

Also, note that to apply the boundary conditions, you need to be able to access *points* on the left and right end of the beam, i.e., the BCs should *not* be specified over the whole edges. A reasonable way to do this would be to draw points at the center of each end of the beam, and then use the command "Coerce to Solid", with both the point and the rectangle selected, so that the point is merged into the boundary of the beam.

• Generate three plots, one each containing all three of (the FE solution using triangles, the FE solution using quads, and the exact solution, as a function of x), with each plot representing a single value of *nel* for both the triangles and the quads (or, at least, close to the same value of nel). What do you notice?

To create the quad mesh, it will be useful to use the controls "Decrease Mesh Size" and "Mesh All (Mapped)" on the Mesh toolbar.

To get the data out, it will probably be useful to run this from MATLAB as a .m file, with the following commands to extract the finite element data:

```
x = linspace(0,L,100);
y = 0*x;
coords = [x;
          y];
a = postinterp(fem,'v',coords);
```