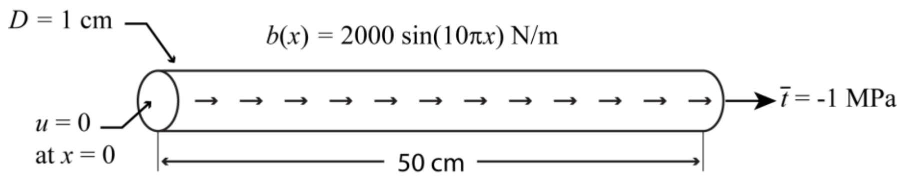
MAE 404/598 Finite Elements in Engineering
Project #1: **Due: March 3, 2015**

An aluminum rod (E = 70 GPa) is fixed on one end and loaded by a sinusoidal body force and a applied traction at the other end. Write a finite element program in MATLAB to approximate the displacement field and compare with analytical solutions and an approximate solution from ABAQUS.



Report format
Reports must be typed and submitted by hard copy either in lecture or at ECG 205 (TA office) prior to the due date. Each report should have the following sections:

1. Analytical solution (use of symbolic math software is okay).
2. MATLAB solution
3. ABAQUS solution
4. Comparison of methods
5. Appendix: MATLAB code(s)

Grading scale:
Points will be awarded for correct completion of each of the tasks listed below with partial credit awarded for incomplete or incorrect attempts. Plots must contain axis labels and units as appropriate for full credit.

MAE 404: Grade calculated on 100 points. **MAE 598:** Grade calculated on 200 points.

General points (30 possible)
- [10] Report is clearly written.
- [10] Plot displacement fields for analytical, ABAQUS and MATLAB solutions all on the same plot.
- [10] Plot strain fields for analytical, ABAQUS and MATLAB solutions all on the same plot.

Analytical solution points (20 possible)
- [10] Analytical solution for the displacement field.
- [10] Analytical solution for the strain field.

MATLAB solution points (155 possible)
- [10] Code follows clean and consistent style with comments included as appropriate.
- [10] Plot of the displacement field for linear element solution with two different element sizes.
- [10] Plot of the strain field for linear element solution with two different element sizes.
- [15] Plot of the displacement field for quadratic element solution with two different element sizes.
- [15] Plot of the strain field for quadratic element solution with two different element sizes.
- [15] Comparison of times to (a) assemble and (b) solve the system of equations with 1000 elements for full storage and sparse storage of the stiffness matrix. (*Hint: Use tic() and toc() MATLAB functions for timing*)
- [20] Plot of L2 displacement error norm vs element size on a log-log plot and computed rate of convergence while using linear elements.
- [20] Plot of L2 displacement error norm vs element size on a log-log plot and computed rate of convergence while using quadratic elements.
- [20] Plot of energy error norm vs element size on a log-log plot and computed rate of convergence with linear elements.
- [20] Plot of energy error norm vs element size on a log-log plot and computed rate of convergence with quadratic elements.

ABAQUS solution points (30 possible)
- [10] Plot displacement and strain fields for linear elements.
- [20] Plot displacement and strain fields for quadratic elements.

MAE 404/598 Finite Elements in Engineering
Project #1: **Due: March 3, 2015**

Hints for solving the problem

How to compute convergence rate (pseudocode):

```
h = [];  % array of element length for each solution.
e = [];  % array of L2 error norm values for each solution.
for nn = [11, 41, 161, 641, 2565]
    d  = solve_fem(mesh);  % function to solve FEM problem with a given mesh.
    h(end+1) = mesh.x(2) – mesh.x(1);
    e(end+1) = L2_error_norm(mesh, d);  % function to compute L2 error norm.
end
plot(log10(h), log10(e), ‘ro-’);
```

How to plot solution (needed for quadratic elements):

```
d  = solve_fem(mesh);  % function to solve fem problem with a given mesh.
xx = [];  % Coordinates to plot solution on.
uu = [];  % Solution values to plot.
for c = mesh.conn
    % For linear elements, c is 2x1; for quadratic 3x1.
    xe = mesh.x(c);  % Element node coordinates, same dimensions as c.
    de = d(c);       % Element node displacements, same dimensions as c.
    for xi = linspace(-1, 1, 10)
        N = shape(xi);     % Evaluate shape functions [1x2] or [1x3].
        uu(end+1) = N*de;  % Interpolate displacement at local coord xi.
    end
end
plot(xx, u, ‘k-’, mesh.x, d, ‘ro’);
```