# ME309 Homework #1

**OBJECTIVES:**
- Inside view of a finite element code.
- Exercise with structural stiffness matrix compilation.
- Introduction to a commercial finite element code.

**1.) (30%) Comparing results from various solution approaches.**
In this problem you will use three different methods to solve the problem described in Figure 1. The three methods are;

(A) Use the formula for deformation of an axially loaded bar from an undergraduate mechanics/strength of materials course (see books by Gere, Hibbeler, Beer & Johnston, etc.).
(B) Compile the appropriate stiffness matrix and load vector by hand and solve for the nodal displacements.
(C) Build a finite element model using the bar elements in ANSYS.

**(1.1)** Use each of the three methods listed above to find the displacements at the four nodal points. Clearly discuss your method in each case, and any differences (if any) you found among the results from the three methods.

**(1.2)** Based on the displacements found in (1.1), what are the stresses in each of the bar segments based on a hand calculation? Now, review your ANSYS stress results. Report both the nodal results (nodal averaged) and the element results (nodal un-averaged). Discuss your results in each case, and any differences (if any) you found among the results.

**(1.3)** Would you expect your answers to be any different if you had used the ANSYS BEAM3[1] (2-D elastic Beam) elements? Why or why not? Would you expect your answers to be different if you had used twice as many elements in methods (B) or (C) above? Why or why not?

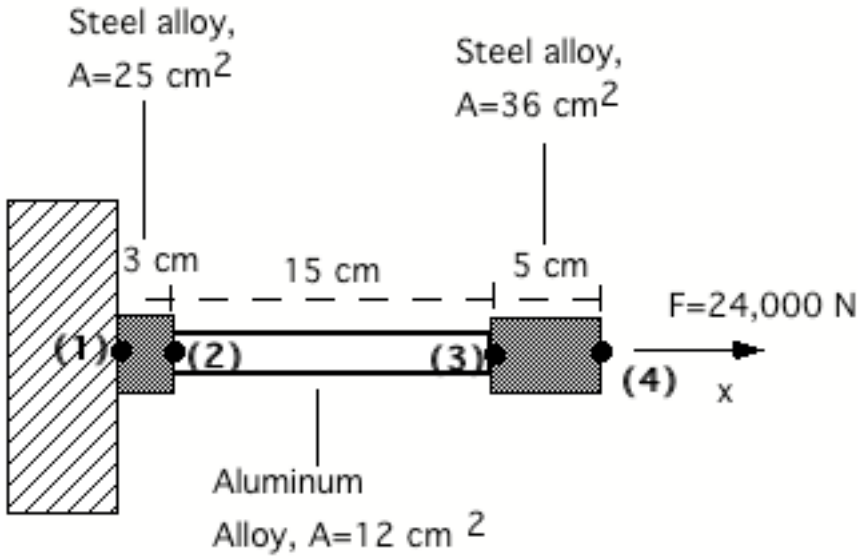


**Figure 1**

[1] Information on the BEAM3 element can be found in the HELP feature in ANSYS

**2.) (30%) Comparing hand calculations with hand-generated finite element results.**

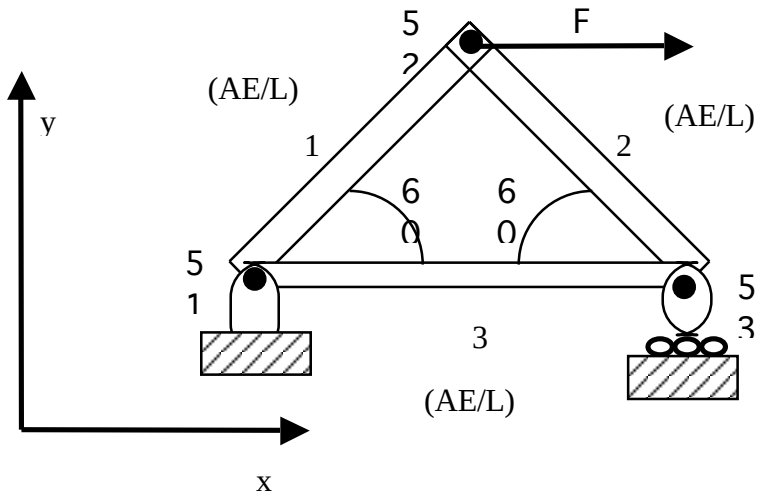


**Figure 2**

For the truss shown in Figure 2 determine the stress in bar elements 1, 2 and 3 using two different methods;
**(2.1)** use the method of joints or method of sections (as found in a statics textbook).

**(2.2)** follow the steps outlined below (resulting in the formation of the stiffness matrix for the truss).

**Step #1** Write the element stiffness matrix for Element #1 in terms of
-- the bar's axis system
-- the global x-y.

**Step #2** Repeat Step #1 for element #2.

**Step #3** Repeat Step #1 for element #3.

**Step #4** Assemble the global stiffness matrix in terms of global x and y deformations at Nodes 51, 52, and 53, denoted by;

$(U_{51},V_{51})$= x and y deflection at global node 51
$(U_{52},V_{52})$= x and y deflection at global node 52
$(U_{53},V_{53})$= x and y deflection at global node 53.

**Step #5** Consider the boundary conditions shown in the figure. Write these boundary conditions out. Also write the appropriate force vector that reflects these boundary conditions.

**Step #6** Use the stiffness matrix created in Step #4 and the force vector in Step #5 to find the displacements. (hint: you can de-couple the equations that involve the reaction forces from those that involve the unknown displacements). Also, find the reactions forces at nodes 51 and 53.

**Step #7** Use the displacements found in Step #7 to find the stress in bar elements 1, 2 and 3.

**(2.3)** Compare the calculated values of stress from **(2.1)** and **(2.2)**. Are they the same?

**3.) (20%) Presenting FEM results.**
Reconsider the beam problem presented in ANSYS Workshop (Tutorial #2).

**(3.1)** For Part I, analysis V report:

--brief description of model (e.g., number of elements, boundary conditions, load description)
--plot of deformed configuration of the beam
--a check of the reaction forces
--maximum displacement and rotation, noting locations
--maximum bending stress, noting location
--a check of maximum displacement/rotation and stress with comparison to the theoretical solution.
--edited log file with BRIEF comments explaining a few of the commands. (Hint: Try typing "nedit yourfilename.log" in a xterm window.)

**(3.2)** Within the ANSYS HELP feature, find the description for a BEAM4 (3-D Elastic Beam) and print out the first two pages of the element description. Would your answers in (a) be changed if you had used BEAM4 elements instead of BEAM3 elements? Why or why not? Also, compare the Degrees of Freedom of the two element types.

**4.) (20%) More on Presenting FEM results.**
Reconsider the Plate problem presented in ANSYS Workshop (Tutorial #2). For the mapped mesh (II.6) report:

--brief description of model (e.g., number of elements, boundary conditions, load description)
--deformed configuration of structure
--a check of reaction forces
--stress contour plot (normal component in direction of loading)
--maximum stress, noting location
--stress concentration factor at edge of hole as determined with FEM (including supporting calculations and assumptions)
--comments on stress concentration as compared to value reported in literature (use Peterson table included in this handout).
--edited log file with your BRIEF explanatory comments

from "Stress Concentration Design Factors,"
by R. E. Peterson, John-Wiley