Modify the myfea program as necessary to solve the problems below. Turn in the requested information, a print of your new element m-files and model files, printed output of nodal and element values.

1. Add a thermal stress capability to your truss2d and truss3d Matlab M-Files. Include the thermal expansion coefficient and temperature change as additional element properties in the model input file. Then use your program to solve Problems 2 and 3 from CP1 (restated below). Assume that the 2D truss in F&B Problem 2.7 is aluminum, and the 3D truss in Problem 3 is steel. Determine the deflection of each node and stress in each element when the temperature is raised 100C. Include the applied loads given in the problem statements. In the 3D truss problem assume that node 7 is fixed in x,y, and z. Make a plot of the deformed and undeformed structure.

Solve Problems 2 and 3 from CP1 using ABAQUS. These problems are restated below. Compare your results with those obtained with your MATLAB code.

1. Develop an analysis capability to evaluate truss elements in 2 dimensions. Add a new element called truss2d) which has 2 nodes and 2 dofs per node to your MATLAB finite element program. The element properties are the cross-sectional area and Young's modulus. You can check your program with F&B Example 2.3, pg. 31. Use your program to solve F&B Problem 2.7. Make plot of deformed and undeformed structure.
2. Develop an analysis capability to evaluate truss elements in 3 dimensions. Add a new element called truss3d which has 2 nodes and 3 dofs per node to your MATLAB finite element program. The element properties are the cross-sectional area and Young's modulus. Use your program to evaluate the space truss shown below. Compute nodal displacements and element stresses. Make a plot of the undeformed and deformed structure.

