Finite Element Trusses

3.0   Trusses Using FEA

We started this series of lectures looking at truss problems. We limited the discussion to statically determinate structures and solved for the forces in elements and reactions at supports using basic concepts from statics.

In this section, we will apply basic finite element techniques to solve general two dimensional truss problems. The technique is a little more complex than that originally used to solve truss problems, but it allows us to solve problems involving statically indeterminate structures.

3.1   Local and Global Coordinates

We start by looking at the beam or element shown in the diagram below. This element attaches to two nodes, 1 and 2. In the Figure we are showing two coordinate systems. One is a one dimensional coordinate system that aligns with the length of the element. We will call this the local coordinate system. The other is a two dimensional coordinate system that does not align with the element. We will call this the global coordinate system. The $\langle x', y' \rangle$ coordinates are the local coordinates for the element and $\langle x, y \rangle$ are the global coordinates.
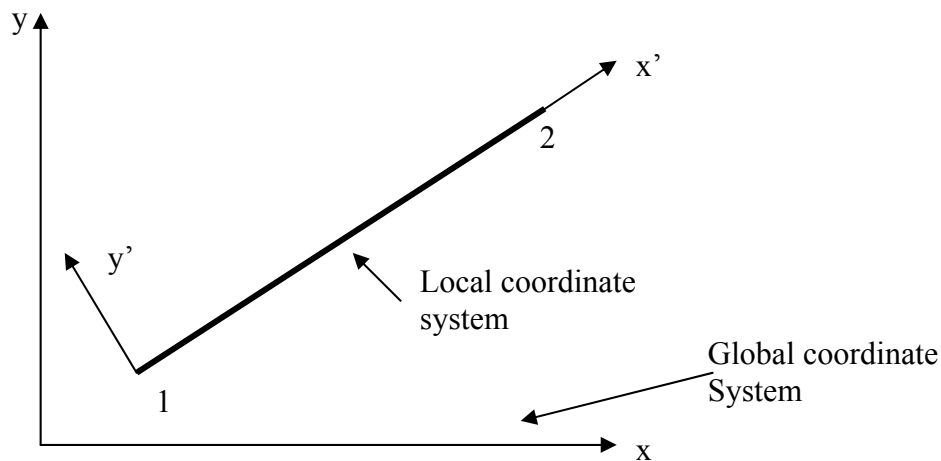

Figure 1 - Local and global coordinate systems

We can convert the displacements shown in the local coordinate system by looking at the following diagram. We will let q_1' and q_2' represent displacements in the local coordinate system and q_1, q_2, q_3, and q_4 represent displacements in the x-y (global) coordinate system. Note that the odd subscripted displacements are in the x direction and the even ones are in the y direction as shown in the following diagram.
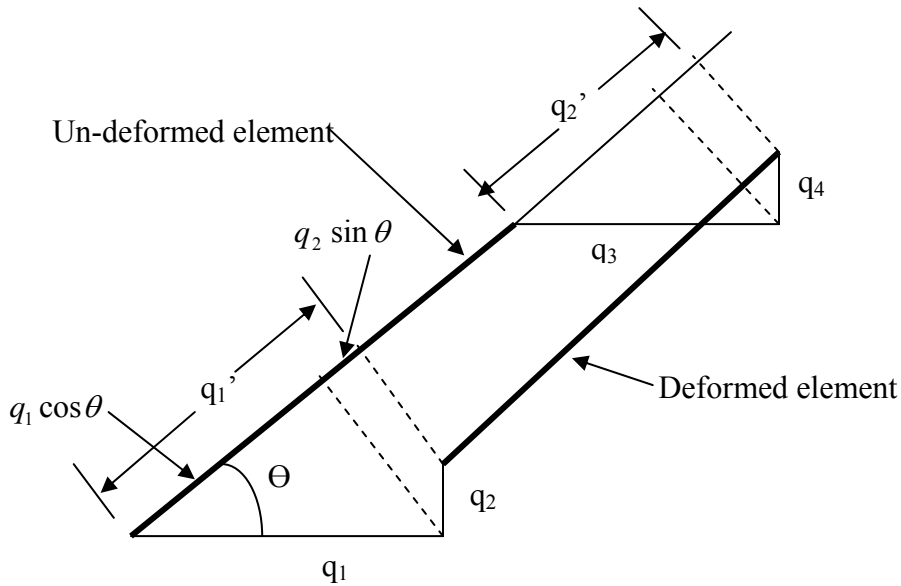
Figure 2 - The deformation of an element in both local and global coordinate systems.

We know that for small deformations in tension or compression a beam, acts like a spring. The amount of deformation is linearly proportional to the force applied to the beam. As the beam is stretched or compressed, we are added potential energy to the beam. This energy is called strain energy and it can be modeled with Hook's law. The law states that the force is directly proportional to the deformation.

$$F = k\Delta x \tag{3.1}$$

We can compute the energy by integrating over the deformation

$$u = k\int_0^Q x dx = \frac{1}{2}kQ^2 \tag{3.2}$$

where $k = \dfrac{AE}{L}$ the element stiffness, A = the cross sectional area of the element, E = Young's modulus for the material, and L = the length of the element. Q is the total change in length of the element. Note that we are assuming the deformation is linear over the element. All equal length segments of the element will deform the same amount. We call this a constant strain deformation of the element.

We can rewrite this change in length as

$$Q = (q_2' - q_1') \tag{3.3}$$

Substituting this into equation (3.2) gives us

$$u = \frac{1}{2}k(q_2' - q_1')^2 \tag{3.4}$$

or expanding

$$u = \frac{1}{2} k (q_2'^2 - 2 q_2' q_1' + q_1'^2) \tag{3.5}$$

Rewriting this in vector form we let

$$q' = \begin{Bmatrix} q_1' \\ q_2' \end{Bmatrix} \tag{3.6}$$

and

$$k' = \frac{AE}{L} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \tag{3.7}$$

With this we can rewrite equation (3.5) as:

$$u = \frac{1}{2} q'^T k' q' \tag{3.8}$$

We can do the indicated operations in (3.8) to see how the vector notation works. We do this by first expanding the terms then doing the multiplication.

$$u = \frac{AE}{2L} \{ q_1' \quad q_2' \} \begin{bmatrix} 1 & -1 \\ -1 & 1 \end{bmatrix} \begin{Bmatrix} q_1' \\ q_2' \end{Bmatrix} \tag{3.9}$$

$$u = \frac{AE}{2L} \{ q_1' - q_2' \quad -q_1' + q_2' \} \begin{Bmatrix} q_1' \\ q'_2 \end{Bmatrix} \tag{3.10}$$

$$u = \frac{AE}{2L} \left(q_1'(q_1' - q_2') + q_2'(q_2' - q_1') \right) \tag{3.11}$$

$$u = \frac{AE}{2L} \left(q_1'^2 - q_1' q_2' + q_2'^2 - q_1' q_2' \right) \tag{3.12}$$

$$u = \frac{AE}{2L} (q_1'^2 - 2 q_1' q_2' + q_2'^2) \tag{3.13}$$

Which is the same as equation (3.5).

Equation (3.7) is the stiffness matrix for a one dimensional problem.

6.2   Two Dimensional Stiffness Matrix

We know for local coordinates that

$$q' = \begin{Bmatrix} q'_1 \\ q'_2 \end{Bmatrix} \qquad (3.6)$$

and for global coordinates (See Figure 2)

$$q = \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \end{Bmatrix} \qquad (3.14)$$

We can transform the global coordinates to local coordinates with the equations

$$q'_1 = q_1 \cos\theta + q_2 \sin\theta \qquad (3.15)$$

and

$$q'_2 = q_3 \cos\theta + q_4 \sin\theta \qquad (3.16)$$

This can be rewritten in vector notation as:

$$q' = Mq \qquad (3.17)$$

where

$$M = \begin{bmatrix} c & s & 0 & 0 \\ 0 & 0 & c & s \end{bmatrix}, \qquad (3.18)$$

$c = \cos\theta$, and $s = \sin\theta$.

Using

$$u = \frac{1}{2} q'^T k' q' \qquad (3.8)$$

we can substitute in equation (3.17)

$$u = \frac{1}{2} q^T \left[M^T k' M \right] q \qquad (3.19)$$

Now we will let

$$k = M^T k'M \tag{3.20}$$

and doing the multiplication, k our stiffness matrix for global two dimensional coordinates becomes

$$k = \frac{AE}{L} \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \tag{3.21}$$

where:

E = Young's modulus for the element material
A = the cross sectional area of the element
L = the length of the element
$c = \cos\theta$
$s = \sin\theta$

3.3 Stress Computations

The stress can be written as

$$\sigma = E\varepsilon \tag{3.22}$$

where ε is the strain, the change in length per unit of length. We can rewrite this as:

total deformation

$$\sigma = E\frac{q_2' - q_1'}{L} \tag{3.23}$$

length of element

In vector form we can write the equation as

$$\sigma = \frac{E}{L}\{-1 \quad 1\}\begin{Bmatrix} q_1' \\ q_2' \end{Bmatrix} \tag{3.24}$$

From our previous discussion, we know that in local coordinates

$$q' = \begin{Bmatrix} q_1' \\ q_2' \end{Bmatrix} \tag{3.6}$$

and in global coordinates

$$q = \begin{Bmatrix} q_1 \\ q_2 \\ q_3 \\ q_4 \end{Bmatrix} \qquad (3.14)$$

From equation (3.17) we know that

$$q' = Mq \qquad (3.17)$$

where

$$M = \begin{bmatrix} c & s & 0 & 0 \\ 0 & 0 & c & s \end{bmatrix} \qquad (3.18)$$

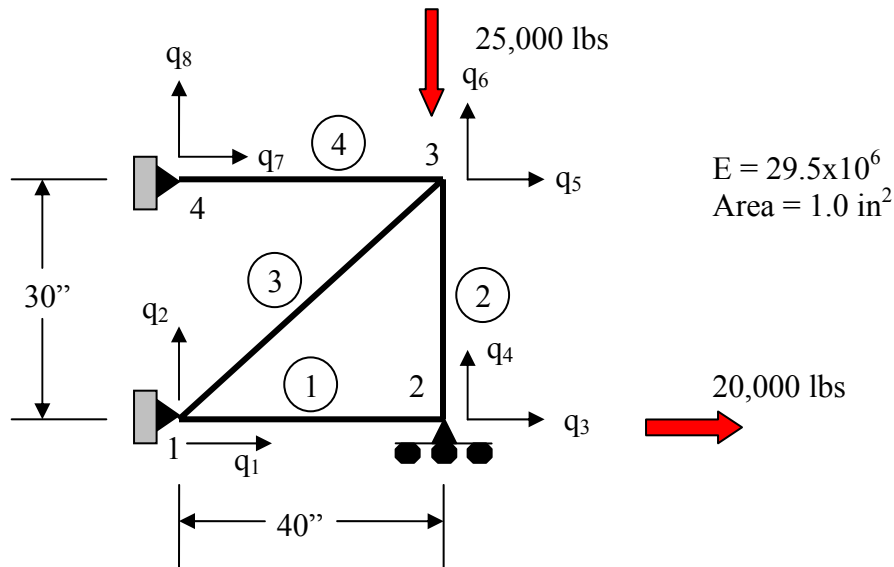Substituting this in to the equation (3.24) yields

$$\sigma = \frac{E}{L}\{-1 \quad 1\}Mq \qquad (3.25)$$

Now we multiply M by the vector

$$\sigma = \frac{E}{L}\{-c \quad -s \quad c \quad s\}q \qquad (3.26)$$

3.4   Truss Example

We can now use the techniques we have developed to compute the stresses in a truss. Consider

Computing Displacements

There are 4 nodes and 4 elements making up the truss. We are going to do a two dimensional analysis so each node is constrained to move in only the X or Y direction. We call these directions of motion degrees of freedom or **dof** for short. There are 4 nodes and 8 degrees of freedom (two degrees of freedom for each node). We can number the degrees of freedom with the formulas:

Vertical degree of freedom $dof = 2*node$ (3.27)
Horizontal degree of freedom $dof = 2*node - 1$ (3.28)

where *node* is the node number.

We can locate each node by its coordinates. The table below shows the coordinates of the nodes in the problem we are solving. We can use these coordinates to determine the lengths and angles of the elements.

Node	X	Y
1	0	0
2	40	0
3	40	30
4	0	30

Table 1 - Coordinates of the nodes in the truss.

Each element can be described as extending from one node to another. This also can be defined in a table below.

Element	From Node	To Node
1	1	2
2	3	2
3	1	3
4	4	3

Table 2 - The elements and the nodes they connect in the truss.

From these two tables we can derive the lengths of each element and the cosine and sine of their orientation. This is shown in the table below.

Element	Length	Cosine	Sine
1	40	1	0
2	30	0	-1
3	50	0.8	0.6
4	40	1	0

Table 3 - Elements with sines and cosines to be used in the stiffness matrix.

In the previous sections we developed the stiffness matrix for an element. This is shown in equation (3.21) below.

$$k = \frac{AE}{L} \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \tag{3.21}$$

This stiffness matrix is for an element. The element attaches to two nodes and each of these nodes has two degrees of freedom. The rows and columns of the stiffness matrix correlate to those degrees of freedom.

Using the equation shown in (3.21) we can construct that stiffness matrix for element 1 defined in the table above. The stiffness matrix is:

Global dof

$$k_1 = \frac{29.5x10^6}{40} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix} \tag{3.29}$$

Element 2

$$k_2 = \frac{29.5x10^6}{30}\begin{matrix} 5 & 6 & 3 & 4 \\ \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} & \begin{matrix} 5 \\ 6 \\ 3 \\ 4 \end{matrix} \end{matrix}$$

(3.30)

Element 3

$$k_3 = \frac{29.5x10^6}{50}\begin{matrix} 1 & 2 & 5 & 6 \\ \begin{bmatrix} .64 & .48 & -.64 & -.48 \\ .48 & .36 & -.48 & -.36 \\ -.64 & -.48 & .64 & .48 \\ -.48 & -.36 & .48 & .36 \end{bmatrix} & \begin{matrix} 1 \\ 2 \\ 5 \\ 6 \end{matrix} \end{matrix}$$

(3.31)

Element 4

$$k_4 = \frac{29.5x10^6}{40}\begin{matrix} 7 & 8 & 5 & 6 \\ \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} & \begin{matrix} 7 \\ 8 \\ 5 \\ 6 \end{matrix} \end{matrix}$$

(3.32)

The next step is to add the stiffness matrices for the elements to create a matrix for the entire structure. We can facilitate this by creating a common factor for Young's modulus and the length of the elements.

For element 1, we divide the outside by 15 and multiply each element of the matrix by 15. Multiplying and dividing by the same number is the same as multiplying and dividing by 1.

$$k_1 = \frac{29.5x10^6}{600}\begin{matrix} 1 & 2 & 3 & 4 \\ \begin{bmatrix} 15 & 0 & -15 & 0 \\ 0 & 0 & 0 & 0 \\ -15 & 0 & 15 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} & \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix} \end{matrix}$$

(3.33)

We multiply and divide element 2 by 20.

$$k_2 = \frac{29.5x10^6}{600} \begin{array}{c} \begin{array}{cccc} 5 & 6 & 3 & 4 \end{array} \\ \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 20 & 0 & -20 \\ 0 & 0 & 0 & 0 \\ 0 & -20 & 0 & 20 \end{bmatrix} \begin{array}{c} 5 \\ 6 \\ 3 \\ 4 \end{array} \end{array} \qquad (3.34)$$

Multiply and divide element 3 by 12.

$$k_3 = \frac{29.5x10^6}{600} \begin{array}{c} \begin{array}{cccc} 1 & 2 & 5 & 6 \end{array} \\ \begin{bmatrix} 7.68 & 5.76 & -7.68 & -5.76 \\ 5.76 & 4.32 & -5.76 & -4.32 \\ 7.68 & 5.76 & 7.68 & 5.76 \\ -5.76 & -4.32 & 5.76 & 4.32 \end{bmatrix} \begin{array}{c} 1 \\ 2 \\ 5 \\ 6 \end{array} \end{array} \qquad (3.35)$$

We do the same for element 4 by multiplying and dividing it by 15.

$$k_4 = \frac{29.5x10^6}{600} \begin{array}{c} \begin{array}{cccc} 7 & 8 & 5 & 6 \end{array} \\ \begin{bmatrix} 15 & 0 & -15 & 0 \\ 0 & 0 & 0 & 0 \\ -15 & 0 & 15 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{array}{c} 7 \\ 8 \\ 5 \\ 6 \end{array} \end{array} \qquad (3.36)$$

The coefficient for each stiffness matrix is the same so we can easily add the matrices. We add the degree of freedom for each element stiffness matrix into the same degree of freedom in the structural matrix. The resulting structural stiffness matrix is shown below.

$$K = \frac{29.5\times10^6}{600} \begin{array}{c} \begin{array}{cccccccc} 1 & 2 & 3 & 4 & 5 & 6 & 7 & 8 \end{array} \\ \begin{bmatrix} 22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\ 5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0 \\ -15.0 & 0 & 15.0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & 20.0 & 0 & -20.0 & 0 & 0 \\ -7.68 & -5.76 & 0 & 0 & 22.68 & 5.76 & -15 & 0 \\ -5.76 & 4.32 & 0 & -20.0 & 5.76 & 24.32 & 0 & 0 \\ 0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \end{bmatrix} \begin{array}{c} 1 \\ 2 \\ 3 \\ 4 \\ 5 \\ 6 \\ 7 \\ 8 \end{array} \end{array} \qquad (3.37)$$

Remembering our basic equation

$$KQ = F \tag{3.38}$$

where K is the structural or global stiffness matrix, Q is the displacement of each node, and F is the external force matrix. This results in

$$\frac{29.5 \times 10^6}{600}
\begin{bmatrix}
22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\
5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0 \\
-15.0 & 0 & 15.0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 20.0 & 0 & -20.0 & 0 & 0 \\
-7.68 & -5.76 & 0 & 0 & 22.68 & 5.76 & -15 & 0 \\
-5.76 & 4.32 & 0 & -20.0 & 5.76 & 24.32 & 0 & 0 \\
0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0
\end{bmatrix}
\begin{Bmatrix}
q_1 \\ q_2 \\ q_3 \\ q_4 \\ q_5 \\ q_6 \\ q_7 \\ q_8
\end{Bmatrix}
=
\begin{Bmatrix}
0 \\ 0 \\ 20{,}000 \\ 0 \\ 0 \\ -25{,}000 \\ 0 \\ 0
\end{Bmatrix}
\tag{3.39}$$

We have boundary conditions at the fixed supports. Our assumption is that these joints will not move in the constrained direction. We remove these from our matrix. The constrained displacements are dof 1, 2, 4, 7, and 8. The lines in equation (3.40) show the rows and columns that are removed.

$$\frac{29.5 \times 10^6}{600}
\begin{bmatrix}
22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\
5.76 & 4.32 & 0 & 0 & 5.76 & 4.32 & 0 & 0 \\
-15.0 & 0 & 15.0 & 0 & 0 & 0 & 0 & 0 \\
0 & 0 & 0 & 20.0 & 0 & -20.0 & 0 & 0 \\
-7.68 & -5.76 & 0 & 0 & 22.68 & 5.76 & -15 & 0 \\
-5.76 & 4.32 & 0 & -20.0 & 5.76 & 24.32 & 0 & 0 \\
0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0
\end{bmatrix}
\begin{Bmatrix}
q_1 \\ q_2 \\ q_3 \\ q_4 \\ q_5 \\ q_6 \\ q_7 \\ q_8
\end{Bmatrix}
=
\begin{Bmatrix}
0 \\ 0 \\ 20{,}000 \\ 0 \\ 0 \\ -25{,}000 \\ 0 \\ 0
\end{Bmatrix}
\tag{3.40}$$

The resulting matrix is:

$$\frac{29.5 \times 10^6}{600}
\begin{bmatrix}
15 & 0 & 0 \\
0 & 22.68 & 5.76 \\
0 & 5.76 & 24.32
\end{bmatrix}
\begin{Bmatrix}
q_3 \\ q_5 \\ q_6
\end{Bmatrix}
=
\begin{Bmatrix}
20{,}000 \\ 0 \\ -25{,}000
\end{Bmatrix}
\tag{3.41}$$

We can use Gaussian elimination or any number of other solution techniques to solve the system of equations shown above. Doing so yields

$$\begin{Bmatrix} q_3 \\ q_5 \\ q_6 \end{Bmatrix} = \begin{Bmatrix} 27.12 \times 10^{-3} \\ 5.65 \times 10^{-3} \\ -22.25 \times 10^{-3} \end{Bmatrix} \qquad \text{inches} \tag{3.42}$$

Computing Stresses

Previously we showed that

$$\sigma = \frac{E}{L} \{-c \quad -s \quad c \quad s\} q \tag{3.26}$$

We use this equation to compute the stress in each element.

$$\sigma_1 = \frac{29.5 \times 10^6}{40} \{-1 \quad 0 \quad 1 \quad 0\} \begin{Bmatrix} 0 \\ 0 \\ 27.12 \times 10^{-3} \\ 0 \end{Bmatrix} \begin{matrix} 1 \\ 2 \\ 3 \\ 4 \end{matrix} \quad \overleftarrow{\text{dof}} \tag{3.43}$$

or

$$\sigma_1 = 20,000 \, psi \tag{3.44}$$

$$\sigma_2 = \frac{29.5 \times 10^6}{30} \{0 \quad 1 \quad 0 \quad -1\} \begin{Bmatrix} 5.65 \times 10^{-3} \\ -22.25 \times 10^{-3} \\ -27.12 \times 10^{-3} \\ 0 \end{Bmatrix} \begin{matrix} 5 \\ 6 \\ 3 \\ 4 \end{matrix} \tag{3.45}$$

$$\sigma_2 = -21,875 \, psi \tag{3.46}$$

Using a similar technique we get

$$\sigma_3 = -5,208 \, psi \tag{3.47}$$

and

$$\sigma_4 = 4,167 \, psi \tag{3.48}$$

Computing the Reactions

The last step is to compute the support reactions. We need to determine the reaction forces along dof 1, 2, 3, 7, and 8 which correspond to the fixed supports. These are obtained by substituting Q into the original finite element equation.

$$R = KQ - F \tag{3.48}$$

We only need to use those rows of the structural stiffness matrix that correspond to the fixed supports. At these supports, we are not supplying an external force so F=0. Our equation becomes

$$R = KQ \tag{3.50}$$

or

$$
\begin{Bmatrix} R_1 \\ R_2 \\ R_4 \\ R_7 \\ R_8 \end{Bmatrix} = \frac{29.5 \times 10^6}{600}
\begin{bmatrix}
22.68 & 5.76 & -15.0 & 0 & -7.68 & -5.76 & 0 & 0 \\
5.76 & 4.32 & 0 & 0 & -5.76 & -4.32 & 0 & 0 \\
0 & 0 & 0 & 20 & 0 & -20 & 0 & 0 \\
0 & 0 & 0 & 0 & -15.0 & 0 & 15.0 & 0 \\
0 & 0 & 0 & 0 & 0 & 0 & 0 & 0
\end{bmatrix}
\begin{Bmatrix}
0 \\ 0 \\ 0 \\ 27.12 \times 10^{-3} \\ 0 \\ 5.65 \times 10^{-3} \\ -22.25 \times 10^{-3} \\ 0 \\ 0
\end{Bmatrix}
\tag{3.51}
$$

We multiply the stiffness matrix K and the deformation vector Q to get the reactions. They are shown in the following equation.
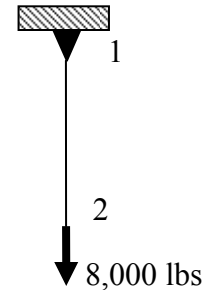
$$
\begin{Bmatrix} R_1 \\ R_2 \\ R_4 \\ R_7 \\ R_8 \end{Bmatrix} =
\begin{Bmatrix} -15,833.3 \\ 3,126 \\ 21,879 \\ -4,167 \\ 0 \end{Bmatrix}
\tag{3.52}
$$

Problems

1.   Element area = 1.5 in^2 E=30,000,000

   Element length = 5 feet

   Write the stiffness matrix for the structure. The
bar is vertical. Show all work.

2.   Using a different load, the element shown in
Problem 1 deforms by 0.02 inches in length. What is the stress in the material? Use a
finite element approach to solve the problem. Show all work.

3.   Use a finite element approach, solve for the stress, joint displacement, and
reaction force on the element shown in Problem 1. Use the 8,000 lbs force as shown in
the diagram. Show all work.

4.   The structure shown in the diagram results in the stiffness matrix shown in the
table. Manually solve for the displacement of node 4. Show all work.

1.0e+006 *

0.6293	0.4720	0	0	0	0	-0.6293	-0.4720
0.4720	0.3540	0	0	0	0	-0.4720	-0.3540
0	0	0.6146	0	0	0	-0.6146	0
0	0	0	0	0	0	0	0
0	0	0	0	0.6293	-0.4720	-0.6293	0.4720
0	0	0	0	-0.4720	0.3540	0.4720	-0.3540
-0.6293	-0.4720	-0.6146	0	-0.6293	0.4720	1.8733	0
-0.4720	-0.3540	0	0	0.4720	-0.3540	0	0.7080
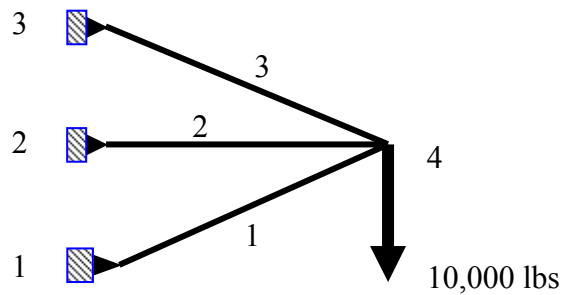
Element	Area	E
1	2 in^2	29.5e6
2	1 in^2	29.5e6
3	2 in^2	29.5e6

Node	X feet	Y feet
1	0	0
2	0	3
3	0	6
4	4	3

5. Element area = 1 in^2 Material = steel

Node	X	Y
1	0	40
2	30	0
3	60	40
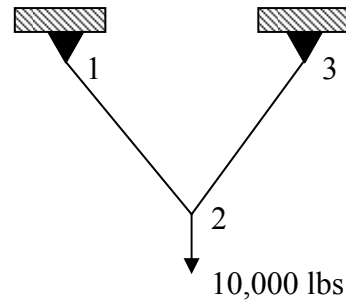
A. Find the joint displacements
B. Find the stress in the elements
C. Find the reactions

1

3

2

10,000 lbs

6. Element area = 1 in^2 Material = steel

2   4

5,000 lbs

1   3

D. Find the joint displacements
E. Find the stress in the elements
F. Find the reactions
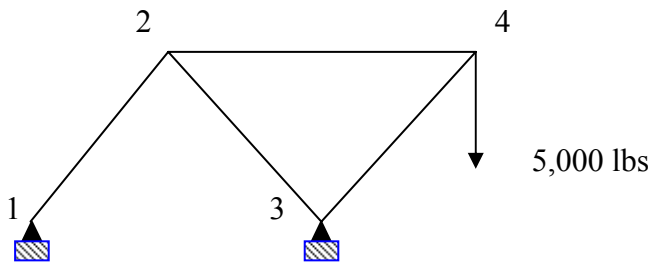
Node	X	Y
1	0	0
2	4	3
3	8	0
4	12	3
Element	From Node	To Node
1	1	2
2	2	3
3	2	4
4	3	4

Write a Matlab program that uses the finite element technique discussed in class to solve for the displacements, stresses, and reactions in a finite element truss. You may want to modify the static stress program you wrote earlier to create this new program. The two programs should be able to use the same input file.

Solve the problem shown above to turn in. Use both this new program and the static truss program to run the data file. Compare the results.