Problem 1 - Analysis of a three bar structure (hand calculation)

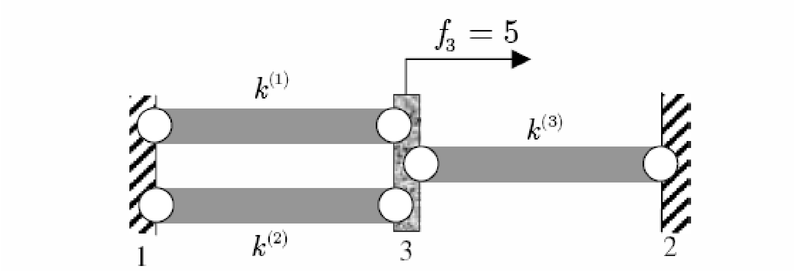


Figure 1: Three-bar problem

Three bars are joined as shown in Figure 1. The left and right ends are both constrained, i.e. prescribed displacement is zero at both ends. There is a force of 5 N acting on the middle node. The nodes are numbered starting with the nodes where displacements are prescribed. You may consider $k^{(e)}$ as the spring constant and is known.

a. Assemble the global stiffness and force matrix.
b. Partition the system and solve for the nodal displacements u_3.
c. Compute the reaction forces.

Solution:

The element stiffness matrices are

$$\mathbf{K}^{(1)} = \overset{\begin{matrix}[1] & \quad [3]\end{matrix}}{\begin{bmatrix} k^{(1)} & -k^{(1)} \\ -k^{(1)} & k^{(1)} \end{bmatrix}} \begin{matrix}[1] \\ [3]\end{matrix}, \quad \mathbf{K}^{(2)} = \overset{\begin{matrix}[1] & \quad [3]\end{matrix}}{\begin{bmatrix} k^{(2)} & -k^{(2)} \\ -k^{(2)} & k^{(2)} \end{bmatrix}} \begin{matrix}[1] \\ [3]\end{matrix}, \quad \mathbf{K}^{(3)} = \overset{\begin{matrix}[3] & \quad [2]\end{matrix}}{\begin{bmatrix} k^{(3)} & -k^{(3)} \\ -k^{(3)} & k^{(3)} \end{bmatrix}}$$

where the global numbers corresponding to the element nodes are indicated above each column and to the right of the row.

By direct assembly, the global stiffness matrix is

$$\mathbf{K} = \overset{\begin{matrix}[1] & \qquad [2] & \qquad [3]\end{matrix}}{\begin{bmatrix} k^{(1)} + k^{(2)} & 0 & -k^{(1)} - k^{(2)} \\ 0 & k^{(3)} & -k^{(3)} \\ -k^{(1)} - k^{(2)} & -k^{(3)} & k^{(1)} + k^{(2)} + k^{(3)} \end{bmatrix}} \begin{matrix}[1] \\ [2] \\ [3]\end{matrix}$$

The displacement and force matrices for the system are

$$\mathbf{d} = \begin{bmatrix} 0 \\ 0 \\ u_3 \end{bmatrix}, \qquad \mathbf{f} = \begin{bmatrix} 0 \\ 0 \\ 5 \end{bmatrix}, \qquad \mathbf{r} = \begin{bmatrix} r_1 \\ r_2 \\ 0 \end{bmatrix}$$

The global system of equation is given by

$$\begin{bmatrix} k^{(1)} + k^{(2)} & 0 & -k^{(1)} - k^{(2)} \\ 0 & k^{(3)} & -k^{(3)} \\ -k^{(1)} - k^{(2)} & -k^{(3)} & k^{(1)} + k^{(2)} + k^{(3)} \end{bmatrix} \begin{bmatrix} 0 \\ 0 \\ u_3 \end{bmatrix} = \begin{bmatrix} r_1 \\ r_2 \\ 5 \end{bmatrix}.$$

As the first two displacements are prescribed, we partition after two rows and columns

$$\left[\begin{array}{cc|c} k^{(1)}+k^{(2)} & 0 & -k^{(1)}-k^{(2)} \\ 0 & k^{(3)} & -k^{(3)} \\ \hline -k^{(1)}-k^{(2)} & -k^{(3)} & k^{(1)}+k^{(2)}+k^{(3)} \end{array}\right]\left[\begin{array}{c} 0 \\ 0 \\ \hline u_3 \end{array}\right]=\left[\begin{array}{c} r_1 \\ r_2 \\ \hline 5 \end{array}\right]$$

or

$$\begin{bmatrix} \mathbf{K}_{\mathrm{E}} & \mathbf{K}_{\mathrm{EF}} \\ \mathbf{K}_{\mathrm{EF}}^{\mathrm{T}} & \mathbf{K}_{\mathrm{F}} \end{bmatrix}\begin{bmatrix} \bar{\mathbf{d}}_{\mathrm{E}} \\ \mathbf{d}_{\mathrm{F}} \end{bmatrix}=\begin{bmatrix} \mathbf{r}_{\mathrm{E}} \\ \mathbf{f}_{\mathrm{F}} \end{bmatrix},$$

Where

$$\mathbf{K}_{\mathrm{E}}=\begin{bmatrix} k^{(1)}+k^{(2)} & 0 \\ 0 & k^{(3)} \end{bmatrix} \qquad \mathbf{K}_{\mathrm{F}}=\begin{bmatrix} k^{(1)}+k^{(2)}+k^{(3)} \end{bmatrix} \qquad \mathbf{K}_{\mathrm{EF}}=\begin{bmatrix} -k^{(1)}-k^{(2)} \\ -k^{(3)} \end{bmatrix}$$

$$\bar{\mathbf{d}}_{\mathrm{E}}=\begin{bmatrix} 0 \\ 0 \end{bmatrix} \qquad \mathbf{d}_{\mathrm{F}}=[u_3] \qquad \mathbf{f}_{\mathrm{F}}=[5] \qquad \mathbf{r}_{\mathrm{E}}=\begin{bmatrix} r_1 \\ r_2 \end{bmatrix}$$

The reduced system of equations is given by

$$(k^{(1)}+k^{(2)}+k^{(3)})u_3=5,$$

which yields

$$u_3=\frac{5}{k^{(1)}+k^{(2)}+k^{(3)}}.$$

The unknown reaction vector is

$$\mathbf{r}_E=\begin{bmatrix} r_1 \\ r_2 \end{bmatrix}=\mathbf{K}_E\bar{\mathbf{d}}_E+\mathbf{K}_{EF}\mathbf{d}_F=\begin{bmatrix} -k^{(1)}-k^{(2)} \\ -k^{(3)} \end{bmatrix}\frac{5}{k^{(1)}+k^{(2)}+k^{(3)}}$$

Problem 2 - Analysis of a plane truss structure (hand calculation)

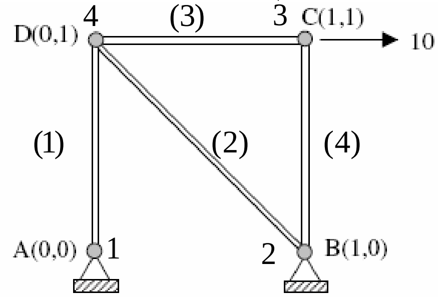


Figure 2: A plane truss structure

Consider the truss structure given in Figure 2. Nodes A and B are fixed. A force equal to 10 N acts in the positive x-direction at node C. Coordinates of joints are given in meters. Young's modulus E and the cross-sectional area A constants.

a. Assemble the global stiffness and force matrix.
b. Partition the system and solve for the nodal displacements.
c. Compute the stresses and reactions.

Solution:

Element 1:

Element 1 is numbered with global nodes 1 and 4.

$$\cos 90^\circ = 0, \quad \sin 90^\circ = 1, \quad l^{(1)} = 1, \quad k^{(1)} = EA$$

$$K^{(1)} = EA\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

Element 2:

Element 2 is numbered with global nodes 2 and 4.

$$\cos 135^\circ = -\frac{1}{\sqrt{2}}, \quad \sin 135^\circ = \frac{1}{\sqrt{2}}, \quad l^{(2)} = \sqrt{2}, \quad k^{(2)} = \frac{EA}{\sqrt{2}}$$

$$K^{(2)} = \frac{EA}{\sqrt{2}}\begin{bmatrix} \frac{1}{2} & -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} \\ -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} & -\frac{1}{2} \\ -\frac{1}{2} & \frac{1}{2} & \frac{1}{2} & -\frac{1}{2} \\ \frac{1}{2} & -\frac{1}{2} & -\frac{1}{2} & \frac{1}{2} \end{bmatrix}$$

Element 3:

Element 3 is numbered with global nodes 3 and 4

$$\cos 180^\circ = -1, \quad \sin 0^\circ = 0, \quad l^{(3)} = 1, \quad k^{(3)} = EA$$

$$K^{(3)} = EA\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix}$$

Element 4:

Element 4 is numbered with global nodes 2 and 3

$$\cos 90^\circ = 0, \quad \sin 90^\circ = 1, \quad l^{(4)} = 1, \quad k^{(4)} = EA$$

$$K^{(4)} = EA\begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix}$$

Direct assembly:

$$\mathbf{K} = EA\begin{bmatrix} 0 & 0 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & 0 & 0 & 0 & 0 & -1 \\ 0 & 0 & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} \\ 0 & 0 & -\frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} & 0 & -1 & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\ 0 & 0 & 0 & 0 & 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & -1 & 0 & 1 & 0 & 0 \\ 0 & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} & -1 & 0 & 1+\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\ 0 & -1 & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} & 0 & 0 & -\frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} \end{bmatrix}$$

and

$$\mathbf{d} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ u_{3x} \\ u_{3y} \\ u_{4x} \\ u_{4y} \end{bmatrix} \quad \mathbf{f} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \\ 10 \\ 0 \\ 0 \\ 0 \end{bmatrix} \quad \mathbf{r} = \begin{bmatrix} r_{1x} \\ r_{1y} \\ r_{2x} \\ r_{2y} \\ 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

Once again notice that if the external force component at a node is prescribed, then the corresponding displacement component at that node is unknown. On the other hand if a displacement component at a node is prescribed, then the corresponding force component at that node is an unknown reaction.

The global system is partitioned after four rows and four columns:

$$\begin{bmatrix} \mathbf{K_E} & \mathbf{K_{EF}} \\ \mathbf{K_{EF}^T} & \mathbf{K_F} \end{bmatrix} \begin{bmatrix} \bar{\mathbf{d}}_\mathbf{E} \\ \mathbf{d_F} \end{bmatrix} = \begin{bmatrix} \mathbf{r_E} \\ \mathbf{f_F} \end{bmatrix}$$

$$\bar{\mathbf{d}}_\mathbf{E} = \begin{bmatrix} 0 \\ 0 \\ 0 \\ 0 \end{bmatrix}, \quad \mathbf{d_F} = \begin{bmatrix} u_{3x} \\ u_{3y} \\ u_{4x} \\ u_{4y} \end{bmatrix}, \quad \mathbf{f_F} = \begin{bmatrix} 10 \\ 0 \\ 0 \\ 0 \end{bmatrix}, \quad \mathbf{r_E} = \begin{bmatrix} r_{3x} \\ r_{3y} \\ r_{4x} \\ r_{4y} \end{bmatrix}$$

$$\mathbf{K_F} = \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 1 & 0 & 0 \\ -1 & 0 & 1+\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\ 0 & 0 & -\frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} \end{bmatrix}, \quad \mathbf{K_{EF}} = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 \\ 0 & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} \\ 0 & -1 & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \end{bmatrix}$$

The unknown displacement matrix if found from the solution of the reduced system of equations:

$$EA\begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 1 & 0 & 0 \\ -1 & 0 & 1+\frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \\ 0 & 0 & -\frac{1}{2\sqrt{2}} & 1+\frac{1}{2\sqrt{2}} \end{bmatrix}\begin{bmatrix} u_{3x} \\ u_{3y} \\ u_{4x} \\ u_{4y} \end{bmatrix} = \begin{bmatrix} 10 \\ 0 \\ 0 \\ 0 \end{bmatrix}$$

and is given by

$$\begin{bmatrix} u_{3x} \\ u_{3y} \\ u_{4x} \\ u_{4y} \end{bmatrix} = \frac{1}{EA}\begin{bmatrix} 20+20\sqrt{2} \\ 0 \\ 10+20\sqrt{2} \\ 10 \end{bmatrix}$$

The unknown reaction matrix $\mathbf{r}$ is

$$\mathbf{r_E} = \begin{bmatrix} r_{3x} \\ r_{3y} \\ r_{4x} \\ r_{4y} \end{bmatrix} = \mathbf{K_E}\bar{\mathbf{d}}_\mathbf{E} + \mathbf{K_{EF}}\mathbf{d_F} = \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 0 & 0 & -1 \\ 0 & 0 & -\frac{1}{2\sqrt{2}} & \frac{1}{2\sqrt{2}} \\ 0 & -1 & \frac{1}{2\sqrt{2}} & -\frac{1}{2\sqrt{2}} \end{bmatrix}\begin{bmatrix} 20+20\sqrt{2} \\ 0 \\ 10+20\sqrt{2} \\ 10 \end{bmatrix} = \begin{bmatrix} 0 \\ -10 \\ -10 \\ 10 \end{bmatrix}$$

It can be easily verified that the equilibrium equations are satisfied:

$$\sum F_x = 0, \quad \sum F_y = 0, \quad \sum M_2 = 0$$

Finally, in the postprocessing step the stresses in the four elements are computed as follows:

$$\sigma^e = E^e \frac{u'^e_{2x} - u'^e_{1x}}{l^e} = \frac{E^e}{l^e}\begin{bmatrix}-1 & 0 & 1 & 0\end{bmatrix}\begin{bmatrix}u'^e_{1x}\\ u'^e_{1y}\\ u'^e_{2x}\\ u'^e_{2y}\end{bmatrix} = \frac{E^e}{l^e}\begin{bmatrix}-1 & 0 & 1 & 0\end{bmatrix}\mathbf{T^e d^e}$$

$$= \frac{E^e}{l^e}\begin{bmatrix}-\cos\phi^e & -\sin\phi^e & \cos\phi^e & \sin\phi^e\end{bmatrix}\mathbf{d^e}.$$

For element 1, we have

$$\phi^{(1)} = 90^\circ \quad \left(\cos\phi^{(1)} = 0, \sin\phi^{(1)} = 1\right),$$

$$\mathbf{d}^{(1)} = \begin{bmatrix}u_{1x}\\ u_{1y}\\ u_{4x}\\ u_{4y}\end{bmatrix} = \frac{1}{EA}\begin{bmatrix}0\\ 0\\ 10+20\sqrt{2}\\ 10\end{bmatrix}$$

$$\sigma^{(1)} = \begin{bmatrix}0 & -1 & 0 & 1\end{bmatrix}\begin{bmatrix}0\\ 0\\ 10+20\sqrt{2}\\ 10\end{bmatrix}\frac{1}{A} = \frac{10}{A}$$

For element 2, we have

$$\phi^{(2)} = 135^\circ \quad \left(\cos\phi^{(2)} = -\frac{1}{\sqrt{2}}, \sin\phi^{(2)} = \frac{1}{\sqrt{2}}\right),$$

$$\mathbf{d}^{(2)} = \begin{bmatrix}u_{2x}\\ u_{2y}\\ u_{4x}\\ u_{4y}\end{bmatrix} = \frac{1}{EA}\begin{bmatrix}0\\ 0\\ 10+20\sqrt{2}\\ 10\end{bmatrix}$$

$$\sigma^{(2)} = \frac{1}{\sqrt{2}}\begin{bmatrix}\frac{1}{\sqrt{2}} & -\frac{1}{\sqrt{2}} & -\frac{1}{\sqrt{2}} & \frac{1}{\sqrt{2}}\end{bmatrix}\begin{bmatrix}0\\ 0\\ 10+20\sqrt{2}\\ 10\end{bmatrix}\frac{1}{A} = \frac{-10\sqrt{2}}{A}$$

For element 3, we have

$$\phi^{(3)} = 180^\circ \quad \left(\cos\phi^{(3)} = -1, \sin\phi^{(3)} = 0\right),$$

$$\mathbf{d}^{(3)} = \begin{bmatrix}u_{3x}\\ u_{3y}\\ u_{4x}\\ u_{4y}\end{bmatrix} = \frac{1}{EA}\begin{bmatrix}20+20\sqrt{2}\\ 0\\ 10+20\sqrt{2}\\ 10\end{bmatrix}$$

$$\sigma^{(3)} = \begin{bmatrix} 1 & 0 & -1 & 0 \end{bmatrix} \begin{bmatrix} 20+20\sqrt{2} \\ 0 \\ 10+20\sqrt{2} \\ 10 \end{bmatrix} \frac{1}{A} = \frac{10}{A}$$

For element 4, we have

$$\phi^{(4)} = 90^\circ \quad \left(\cos\phi^{(1)} = 0, \sin\phi^{(1)} = 1\right),$$

$$\mathbf{d}^{(4)} = \begin{bmatrix} u_{2x} \\ u_{2y} \\ u_{3x} \\ u_{3y} \end{bmatrix} = \frac{1}{EA} \begin{bmatrix} 0 \\ 0 \\ 20+20\sqrt{2} \\ 0 \end{bmatrix}$$

$$\sigma^{(1)} = \begin{bmatrix} 0 & -1 & 0 & 1 \end{bmatrix} \begin{bmatrix} 0 \\ 0 \\ 20+20\sqrt{2} \\ 0 \end{bmatrix} \frac{1}{A} = 0$$

Problem 3 - Analysis of two plane truss structures (MatLab)

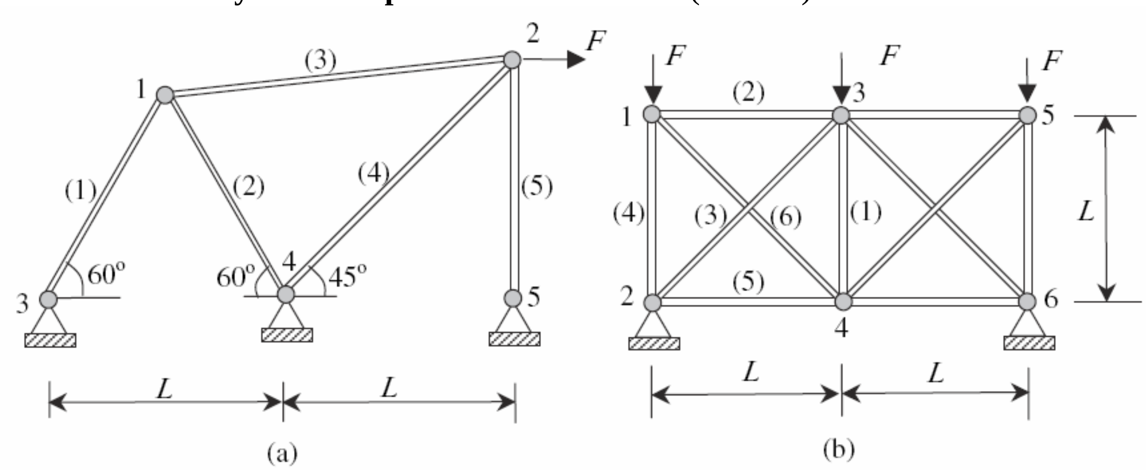


Figure 3: Two plane truss structures

Using the MATLAB finite element code, find the displacements and stresses in the two truss structures given in Figure 3. Plot the deformed structure with MATLAB. For truss structure (b), exploit the symmetry. For the two trusses, check the equilibrium at node 1. The Young's modulus $E = 10^{11}$ Pa, cross sectional areas of all bars $A = 10^{-2} m^2$, forces $F = 10^3$ N and $L = 2$ m.

Solution:

(a) After modifying the input data and grid,

```
f(3) = P;
E = ones(nel,1)*1e11;
A = ones(nel,1)*1e-2;
debc = [5, 6, 7, 8, 9, 10];
etc.

Nodes =    [1,        4, 0, 2, 4;
             sqrt(3), 2, 0, 0, 0];

Elems=[1, 1, 1, 2, 2
       3, 4, 2, 4, 5];
```

we obtain the following results:

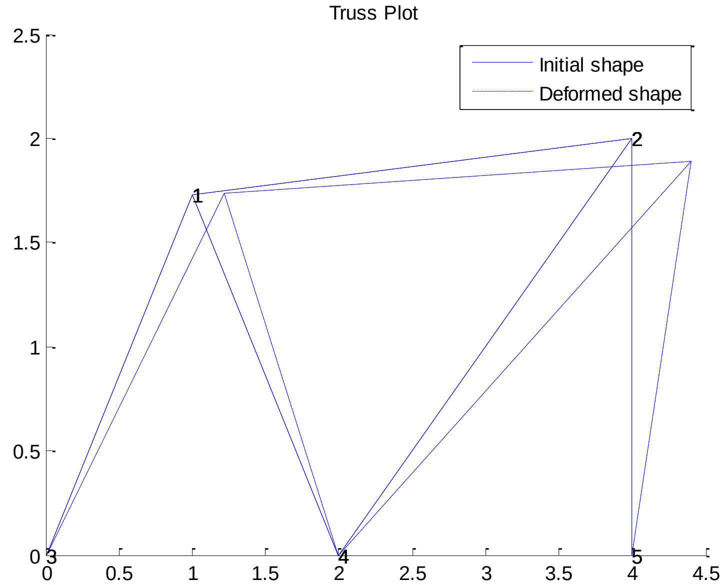
Node#	x-displacement(m)	y-displacement(m)
1	2.082758e-006	6.200815e-008
2	3.763022e-006	-1.051633e-006
3	0	0
4	0	0
5	0	0

Node#	x-reaction force (Nt)	y-reaction force (Nt)
3	-273.769936	-474.183439
4	-726.230064	-51.633121
5	0.000000	525.816561

Element #	Strain	Stress (Pa)	Force (nt)
1	5.475399e-007	54753.987269	547.539873
2	-4.938392e-007	-49383.923641	-493.839236
3	5.227623e-007	52276.230829	522.762308
4	6.778473e-007	67784.733254	677.847333
5	-5.258166e-007	-52581.656067	-525.816561

It is easy to verify that the overall equilibrium is satisfied and also at node 1.

The deformed shape is



(b) Taking advantage of symmetry, we will split the whole structure in two sub-structures (left and right). We need to be sure that this is done properly so when we add left + right sub-structures, we do obtain the original structure with the correct loads. Let us concentrate on the left sub-structure.

At first, the force acting at node 3 is taken only 500 N (half of the original force for the whole structure). This makes sense as when adding left + right sub-structures, we need to recover on node 3 a force of 1000N.

Also, the nodes 3 and 4 cannot move in the x-direction (if they do, the left and the right structures will not symmetric!).

In addition, the cross sectional area of element 1 is halved. If we don't do that, then adding left and right sub-structures will give us not one but two vertical truss elements of total area twice the original one. You can basically think that the middle bar is two bars in parallel each with area A/2 and each taking a load F/2!

After modifying the input data and grid,

```
P1 = 1e3;
P2 = 500;
f(2) = -P1;
f(6) = -P2;
E = ones(nel,1)*1e11;
A = [0.5e-2; ones(nel-1,1)*1e-2];
debc = [3, 4, 5, 7];
….

Nodes =     [0, 0, 2, 2;
```

```
                    2, 0, 2, 0];

Elems=[3, 1, 2, 1, 2, 1;
        4, 3, 3, 2, 4, 4];
….
```

we obtain the following results:

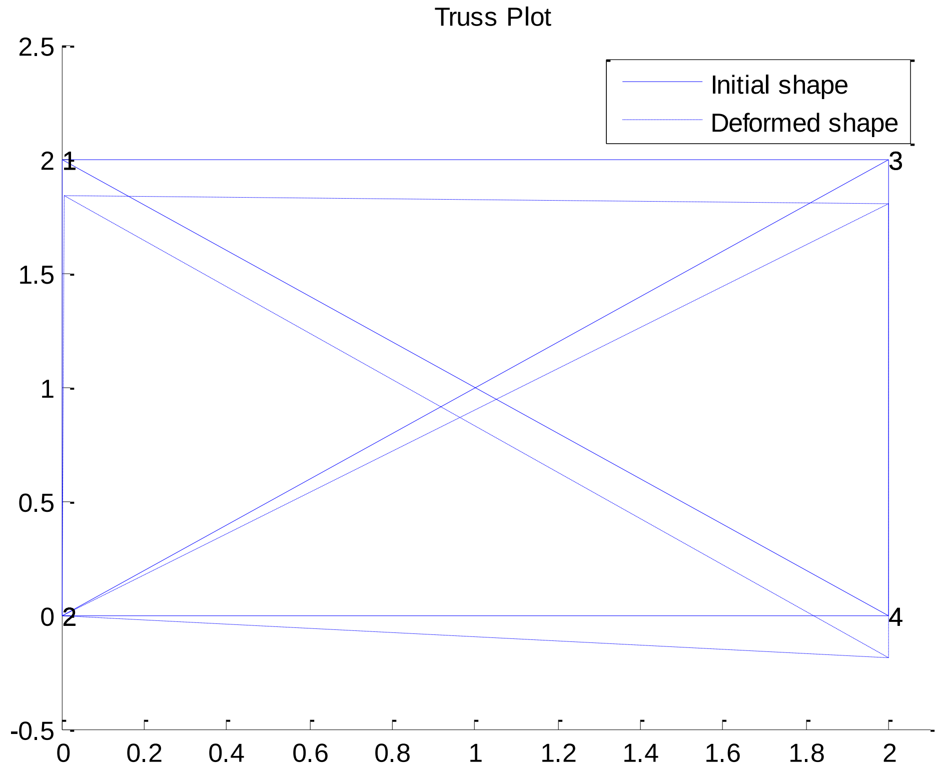
Node#	x-displacement(m)	y-displacement(m)
1	8.578644e-008	-2.085786e-006
2	0	0
3	0	-2.585786e-006
4	0	-2.414214e-006

Node#	x-reaction force (Nt)	y-reaction force (Nt)
2	457.106781	1500.000000
3	-500.000000	0
4	42.893219	0

Element #	Strain	Stress (Pa)	Force (nt)
1	-8.578644e-008	-8578.643763	-42.893219
2	-4.289322e-008	-4289.321881	-42.893219
3	-6.464466e-007	-64644.660941	-646.446609
4	-1.042893e-006	-104289.321881	-1042.893219
5	0	0	0
6	6.066017e-008	6066.017178	60.660172

It is easy to verify that the overall equilibrium is satisfied as well as equilibrium at node 1.

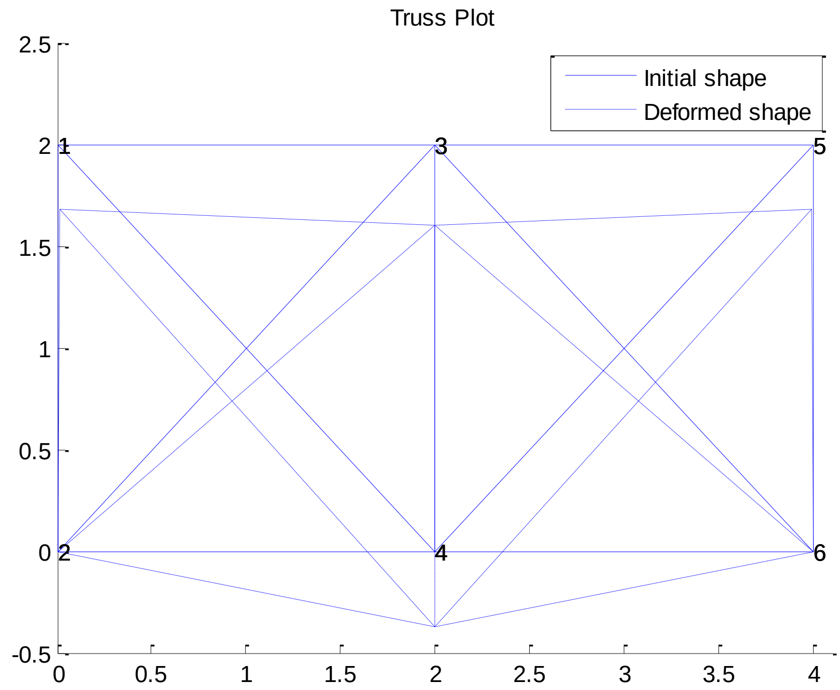
The deformed shape is



Here are the results from the full structure.

Node#	x-displacement(m)	y-displacement(m)
1	8.578644e-008	-2.085786e-006
2	0	0
3	6.908279e-023	-2.585786e-006
4	8.957883e-023	-2.414214e-006
5	-8.578644e-008	-2.085786e-006
6	0	0

The deformed shape is



Therefore, the two results are the same.

Problem 4 (Ansys) Repeat problem 3 using ANSYS

Solution:

The truss problem can be solved by following the steps in the tutorial at
https://confluence.cornell.edu/display/simulation/truss

Detailed commands are listed below step by step:

1. Use the APDL Product Launcher to open ANSYS;
2. Set preferences to show structural elements in GUI;
3. Add element type;
   →Add ≫ Structural Mass ≫ Link ≫ 2D spar1;
4. Set area:
   Preprocessor≫ Element Type ≫ Add/Edit/Delete
   →Add ≫ Link1 ≫ Area = 0.01;
5. Specify material properties:
   Preprocessor≫ Material Props ≫ Material Models
   → Material Model 1 ≫ Structural ≫ Linear ≫ Elastic ≫ Isotropic
   EX=1e11, PRXY=0.3;

6. Define nodes:
   Preprocessor≫ Modeling≫ Create≫ Nodes≫ In Active CS
   Input node numbers and corresponding coordinates in the XY global coordinate system;
7. Define elements:
   Preprocessor ≫ Modeling ≫ Create ≫ Elements ≫ Auto Numbered ≫ Thru Nodes
8. Set boundary conditions:
   Preprocessor ≫Loads ≫ Define Loads ≫ Apply ≫ Structural ≫ Displacement ≫ On Nodes
   Apply zero displacement in X and Y directions for relevant nodes, or zero displacement in the X direction for relevant nodes under symmetry conditions;
9. Start Loading:
   Preprocessor ≫ Loads ≫ Define Loads ≫ Apply ≫ Structural ≫ Force/Moment ≫On Nodes
   Input force value and direction;
10. Run Solver:
    Solution ≫ Solve ≫ Current LS;
11. Plot deformed structure:
    General Postproc ≫ Plot Results ≫ Deformed Shape;
12. List displacements:
    General Postproc ≫ List Results ≫ Nodal Solution
    →Nodal Solution ≫ DOF Solution ≫ Displacement vector sum;
13. List axial stresses:
    General Postproc ≫ Element Table ≫ Define Table ≫ Add ≫ By Sequence No ≫ LS, 1
    General Postproc ≫ Element Table ≫ List Elem Table ≫ LS, 1

Outputs for problem 4a:

Nodal displacements obtained from ANSYS:

```
 PRINT U    NODAL SOLUTION PER NODE
   ***** POST1 NODAL DEGREE OF FREEDOM LISTING *****
   LOAD STEP=     0  SUBSTEP=     1
   TIME=    1.0000      LOAD CASE=   0
   THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL
COORDINATE SYSTEM
     NODE      UX           UY           UZ           USUM
       1  0.20829E-05 0.61996E-07  0.0000     0.20838E-05
       2  0.37631E-05-0.10516E-05  0.0000     0.39073E-05
       3   0.0000      0.0000      0.0000      0.0000
```

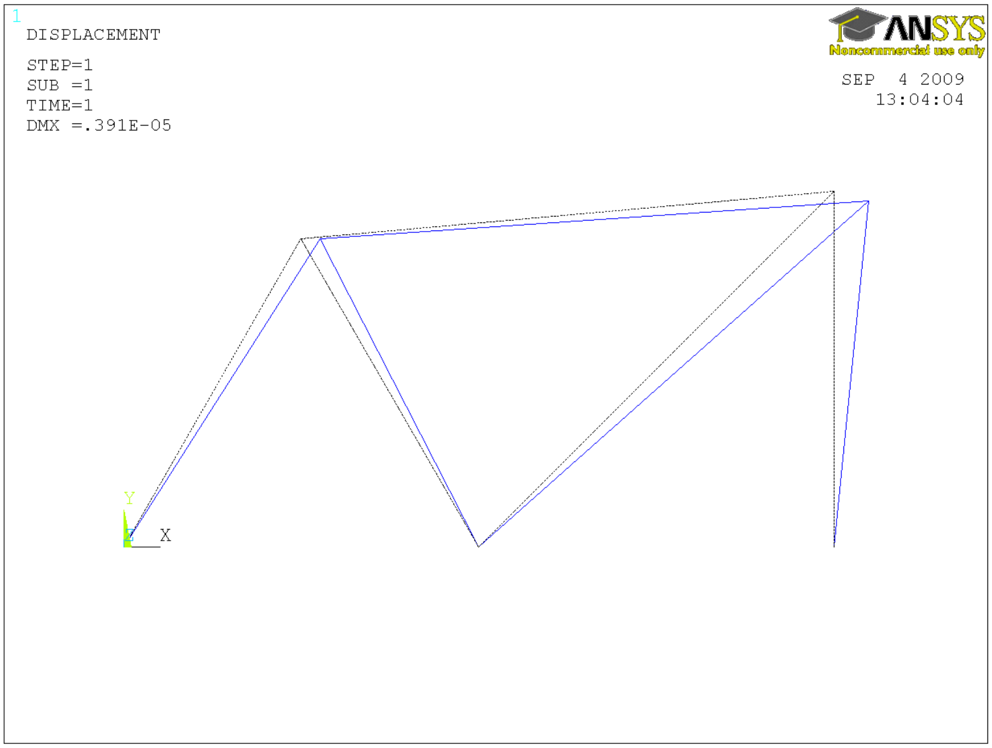
```
      4   0.0000      0.0000      0.0000      0.0000
      5   0.0000      0.0000      0.0000      0.0000
 MAXIMUM ABSOLUTE VALUES
 NODE          2           2           0           2
 VALUE   0.37631E-05-0.10516E-05  0.0000     0.39073E-05
```

Summary: The x- and y-displacements, together with UX and UY, agree well with the results from the MATLAB code. Stresses and member forces can be obtained from the element table for the LINK1 element. The sequence numbers for these items are LS, 1 and SMISC, 1 respectively. The values obtained from ANSYS are:

```
 PRINT ELEMENT TABLE ITEMS PER ELEMENT

  ***** POST1 ELEMENT TABLE LISTING *****

    STAT    CURRENT     CURRENT
    ELEM    STRESS      FORCE
       1   54754.      547.54
       2  -49385.     -493.85
       3   52275.      522.75
       4   67786.      677.86
       5  -52582.     -525.82

 MINIMUM VALUES
 ELEM          5           5
 VALUE   -52582.     -525.82

 MAXIMUM VALUES
 ELEM          4           4
 VALUE    67786.      677.86
```

Summary: The stress and member force values are consistent with the MATLAB code results. It is easy to verify that the overall equilibrium is satisfied and also at node 1.

Deformed shape from ANSYS (in dark blue):



Outputs for problem 4b:

The truss problem can be solved by following the same steps as part (a). Because of symmetry, only the left half needs to be modeled. The force at node 3 is only 500 N and the nodes 3 and 4 cannot move in the x direction (UX=0).

The x- and y-displacements, UX and UY, respectively, match the results from the MATLAB code closely.

```
PRINT U    NODAL SOLUTION PER NODE
   ***** POST1 NODAL DEGREE OF FREEDOM LISTING *****
   LOAD STEP=     1  SUBSTEP=     1
   TIME=    1.0000      LOAD CASE=   0

  THE FOLLOWING DEGREE OF FREEDOM RESULTS ARE IN THE GLOBAL
COORDINATE SYSTEM

    NODE       UX          UY          UZ          USUM
       1  0.85786E-07-0.20858E-05  0.0000     0.20875E-05
```

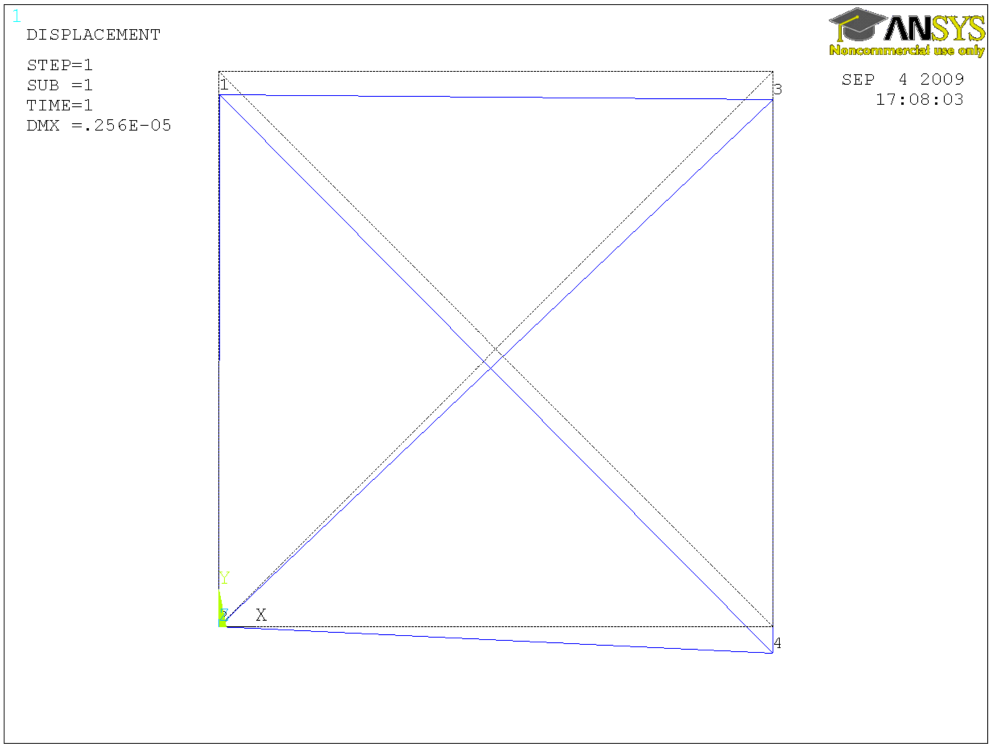
```
       2   0.0000      0.0000      0.0000      0.0000
       3   0.0000    -0.25858E-05  0.0000     0.25858E-05
       4   0.0000    -0.24142E-05  0.0000     0.24142E-05

 MAXIMUM ABSOLUTE VALUES
 NODE          1           3           0           3
 VALUE   0.85786E-07-0.25858E-05  0.0000     0.25858E-05
```

Stresses and member forces can be obtained from the element table for the LINK1 element. The sequence numbers for these items are LS,1 and SMISC,1 respectively. The values obtained from ANSYS are:

```
 PRINT ELEMENT TABLE ITEMS PER ELEMENT

  ***** POST1 ELEMENT TABLE LISTING *****

    STAT    CURRENT
    ELEM    AXIAL_ST
       1  -8578.6
       2  -4289.3
       3  -64645.
       4 -0.10429E+06
       5   0.0000
       6   6066.0

 MINIMUM VALUES
 ELEM          4
 VALUE  -0.10429E+06

 MAXIMUM VALUES
 ELEM          6
 VALUE    6066.0
```

Summary: The stress and member force values are consistent with the MATLAB code results. It is easy to verify that the overall equilibrium and also equilibrium at node 1 are satisfied.

Deformed shape from ANSYS (in dark blue):


1
DISPLACEMENT
STEP=1
SUB =1
TIME=1
DMX =.256E-05
ANSYS
Noncommercial use only
SEP 4 2009
17:08:03
1
3
2
4
Y
X