Finite Element Analysis Homework 2

Author: Anshul Yadav

Date: 2nd October 2021

Abstract

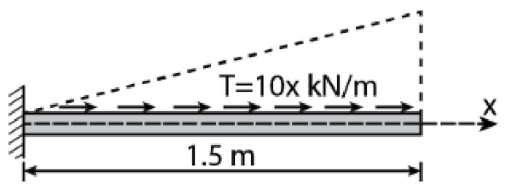
This report is written in response to the two homework problems. Problem 1 is about a bar on which a linearly varying axial load is applied. We were asked to calculate an exact solution for axial displacement and stress and to compare that with a few finite element models. I used ANSYS to generate the finite element models and calculated the exact solution on my own. The differences in displacements between the exact solution and the models are minimal though the stress differences are not.

For the second problem we are a symmetric truss geometry with forces and are asked to find the deformation and stresses using the finite element model. Here, I also used ANSYS to generate the model.

Method:

Problem 1

Problem 1:
For the bar subjected to linear varying axial load shown in the figure below, determine the nodal displacements and axial stress distribution using (a) single LINK 180 element (b) two equal-length LINK 180 elements, (c) four equal-length LINK 180 elements, and (d) eight equal-length LINK 180 elements. Plot the displacement and axial stress as functions of x for these four FE models. Additionally, plot the exact solutions (analytical solutions) of displacement and axial stress in the same plots. Compare the results and discuss your observations, including the accuracy of FE solutions for different number of elements, and reasons for any discrepancies or differences in solutions. Let the cross-sectional area of the bar A=10 cm^2 and Young's modulus E=210 GPa.



The Exact Solution:

The exact solution for displacement of the free end involves an integral relation that integrates strain with respect to the x coordinate. E represents the young's modulus and A represents the bar's cross-sectional area. L is the length of the bar. Equation was provided in lecture in EMA 306.

$$\delta_{AB} = \delta_B = \int_0^L F(x)/EA\, dx \qquad (1)$$

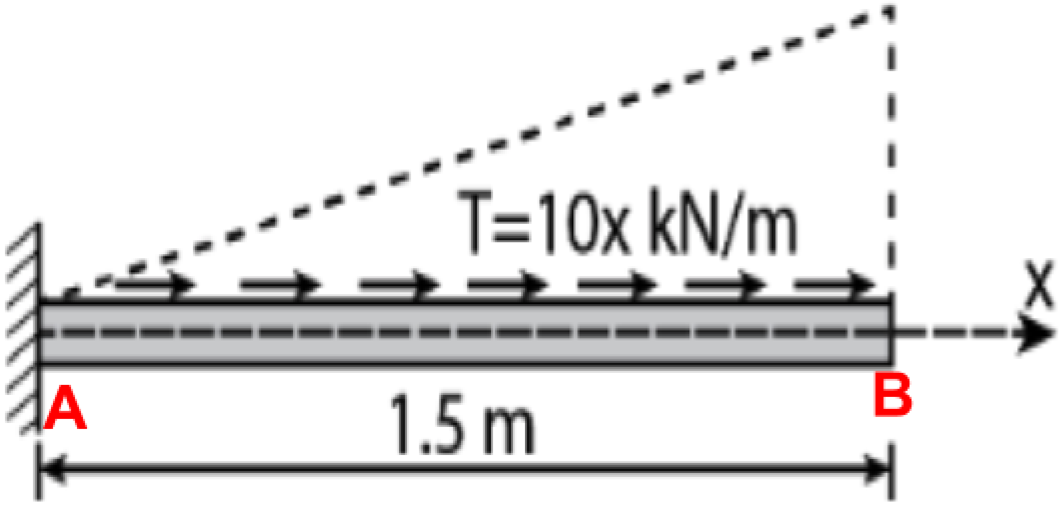


Figure 1

Using a free body diagram with an internal cut one can determine the force function F(x). As indicated in the figure 2, I make a cut at an arbitrary location and use the right hand side of the FBD to find the force function F(x). I use the right side cut as the Left side cut would have contained the reaction force at point A which is also unknown.

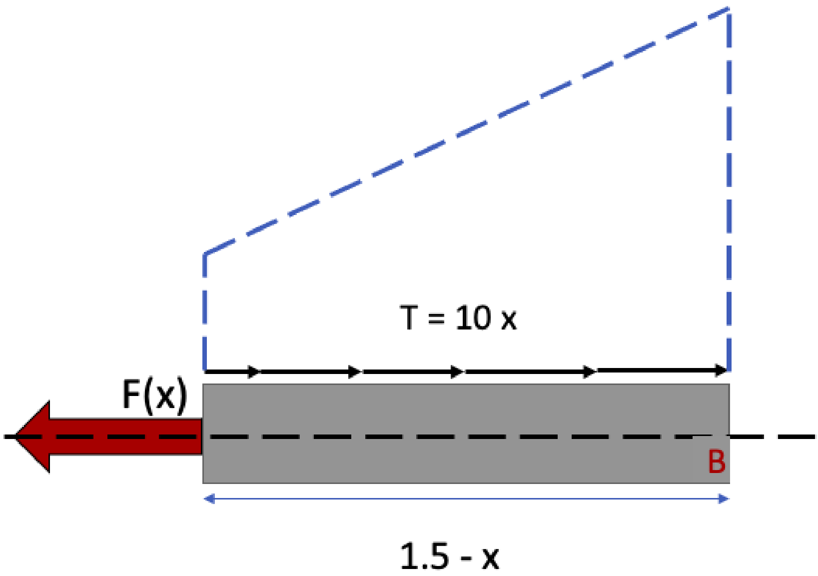


Figure 2

Using Equilibrium in x direction

$F(x) = 1/2(1.5\, m * 10\, KN/m * 1.5\, m) - 1/2(x\, m * 10\, KN/m * x\, m) = (2250 - 10000x^2)/2N$
(2)

Substituting to find the overall displacement.

$$\delta_{AB} = 1/(2EA)\int_0^L (22500 - 10000x^2)\,dx = 1/2EA\,[22500x - 10000x^3/3]_0^{1.5}$$

$$= 22500/2EA = 22500/(2 * 210 * 10^9 * 10^{-3}) = 3/56000\,m = 5.35714 * 10^{-5} m$$

To calculate displacement to an intermediate x-value, one only needs to change the bounds of integration,

$$\delta_{Ax} = 1/(2EA)\int_0^x (22500 - 10000b^2)\,db = (27x - 4x^3)/504000$$

(3)

The exact solution for the stress function is just the force function divided by cross-sectional area.

$\sigma(x) = F(x)/A =$

$(22500 - 10000x^2)/(2 * 0.001)\,Pa = (1.125 - 0.5x^2) * 10^7 Pa$ (4)

Finite Element Model:

4 finite element models were constructed for the four parts of question 1. Each model was made with a Link 180 element. All models were assumed to be elastic, linear and isotropic. All units were in SI - Forces in Newtons, Young's Modulus in Pa, displacements in m, area in m^2.The material properties for the model were as follows: E=210 GPa and $\nu = 0.3$.

For the first model, I used a single element element. Nodal forces were calculated according to equation 5, which was provided in lecture. All DOF for node 1 were set 0, whereas only y,z displacement for node 2 were set to 0.

$$\{f_{1x}, f_{2x}\}^T = \int_o^L \{1 - x/L,\ x/L\}^T\{cx, cx\}^T dx = \{cx^2/2 - cx^3/3L,\ cx^3/3L\}^T|_0^L = \{cL^2/6, cL^2/3\}$$

$= \{1/3F, 2/3F\}$(5)

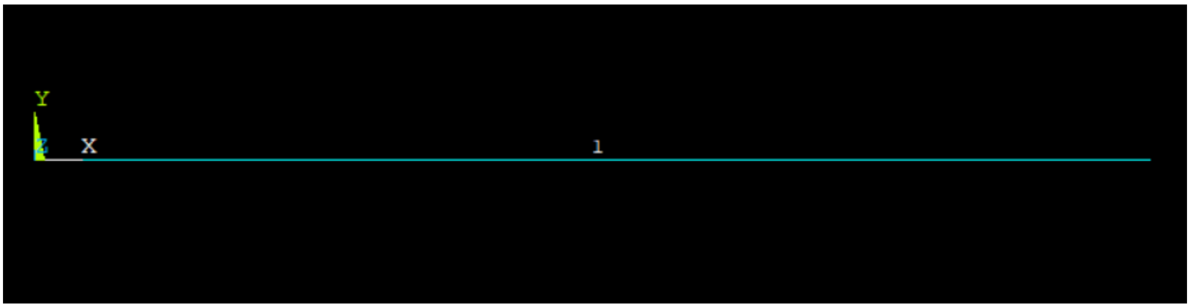


Figure 3

The solution for nodal Displacement and elemental Stresses as provided by ansys.

Element	Node Number	Observed	Observed X	Applied Force

Number		Stress (Pa)	Displacement (m)	(Newton)
1	1	0.75000E+00 7	0.0000	3750
	2	0.75000E+00 7	0.53571E-004	7500

Table 1

For the second model, I used 2 elements. Nodal forces were also calculated according to equation 5. For nodal forces at nodes inside the model (e.g. node 3) forces were calculated according to equation 3.10.33 in Daryl's Textbook (ref1). All DOF for keypoint 1(node 1) were set 0, whereas only y,z displacement for keypoint 2 (node 2(rightmost node)) were set to 0.

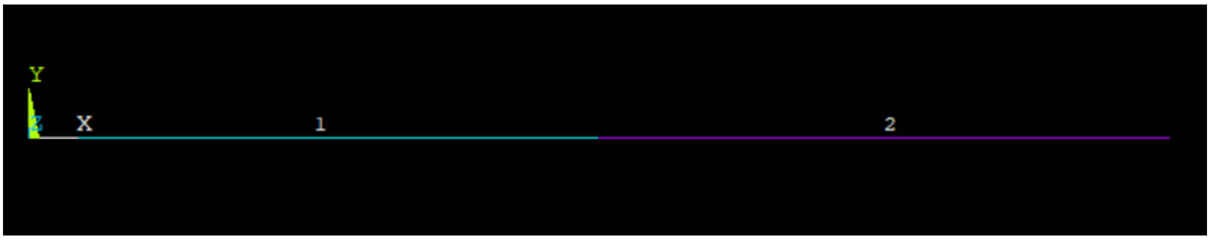


Figure 4

Element Number	Node Number	Observed Stress (Pa)	Observed X Displacement (m)	Applied Force (Newton)
1	1	0.10312E008	0	937.5
	3	0.10312E008	0.36830E-004	4687.5
2	3	0.46875E+007	0.36830E-004	4687.5
	2	0.46875E+007	0.53571E-004	5625

Table 2

For the third model, I used 4 elements. Nodal forces were calculated according to equation 5 and equation 3.10.33 in Daryl's textbook (ref1). All DOF for keypoint 1(node 1) were set 0, whereas only y,z displacement for keypoint 2 (node 2(rightmost node)) were set to 0.

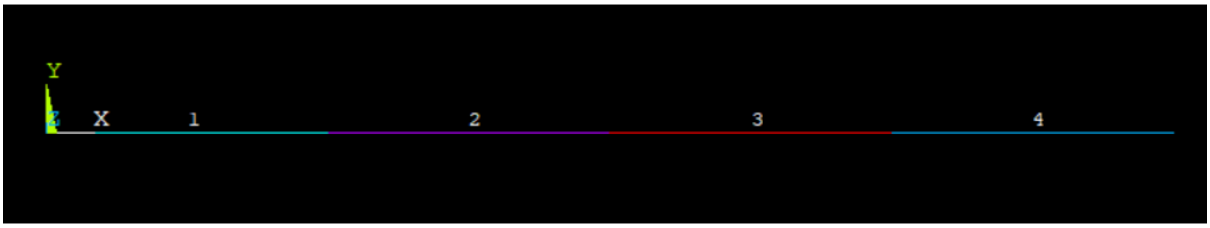


Figure 5

Element Number	Node Number	Observed Stress (Pa)	Observed X Displacement (m)	Applied Force (Newton)
1	1	0.11016E+008	0.0000	234.375
	3	0.11016E+008	0.19671E-004	1406.2
2	3	0.96094E+007	0.19671E-004	1406.2
	4	0.96094E+007	0.36830E-004	2812.5
3	4	0.67969E+007	0.36830E-004	2812.5
	5	0.67969E+007	0.48968E-004	4218.8
4	5	0.25781E+007	0.48968E-004	4281.8
	2	0.25781E+007	0.53571E-004	2578.1

Table 3

For the fourth model, I used 8 elements. Nodal forces were calculated according to equation 5 and equation 3.10.33 in Daryl's textbook (ref1). All DOF for keypoint 1(node 1) were set 0, whereas only y,z displacement for keypoint 2 (node 2(rightmost node)) were set to 0.

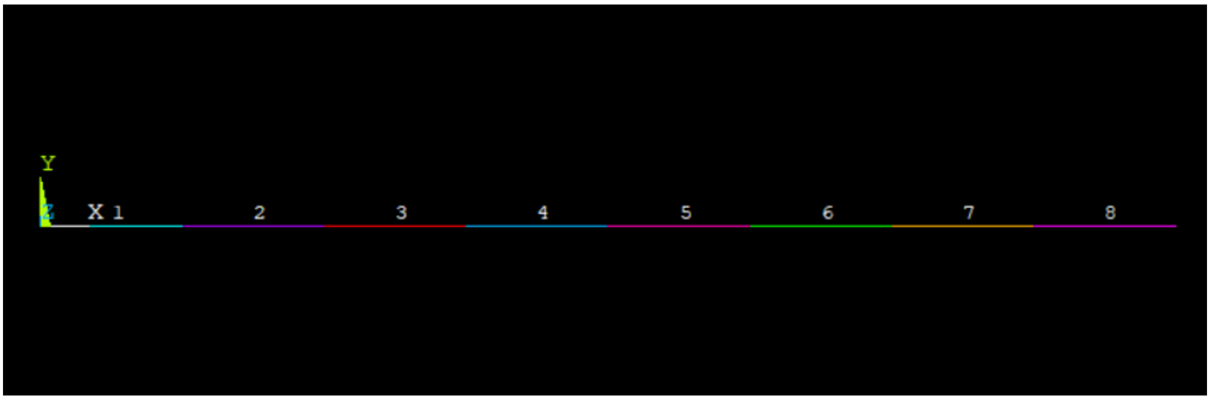


Figure 6

Element	Node Number	Observed	Observed X	Applied Force

Number		Stress (Pa)	Displacement (m)	(Newton)
1	1	0.11134E+008	0.0000	58.59
	3	0.11134E+008	0.99414E-005	351.56
2	3	0.10783E+008	0.99414E-005	351.56
	4	0.10783E+008	0.19569E-004	703.125
3	4	0.10080E+008	0.19569E-004	703.125
	5	0.10080E+008	0.28569E-004	1054.7
4	5	0.90250E+007	0.28569E-004	1054.7
	6	0.90250E+007	0.36627E-004	1406.2
5	6	0.76188E+00	0.36627E-004	1406.2
	7	0.76188E+00	0.43429E-004	1757.8
6	7	0.58610E+007	0.43429E-004	1757.8
	8	0.58610E+007	0.48662E-004	2109.4
7	8	0.37516E+007	0.48662E-004	2109.4
	9	0.37516E+007	0.52012E-004	2406.9
8	9	0.13447E+007	0.52012E-004	2406.9
	2	0.13447E+007	0.53571E-004	1347.7

Table 4

Discussion of Results:

Figure 7 shows the theoretical solution for displacement and the finite element approximations by the fours models.

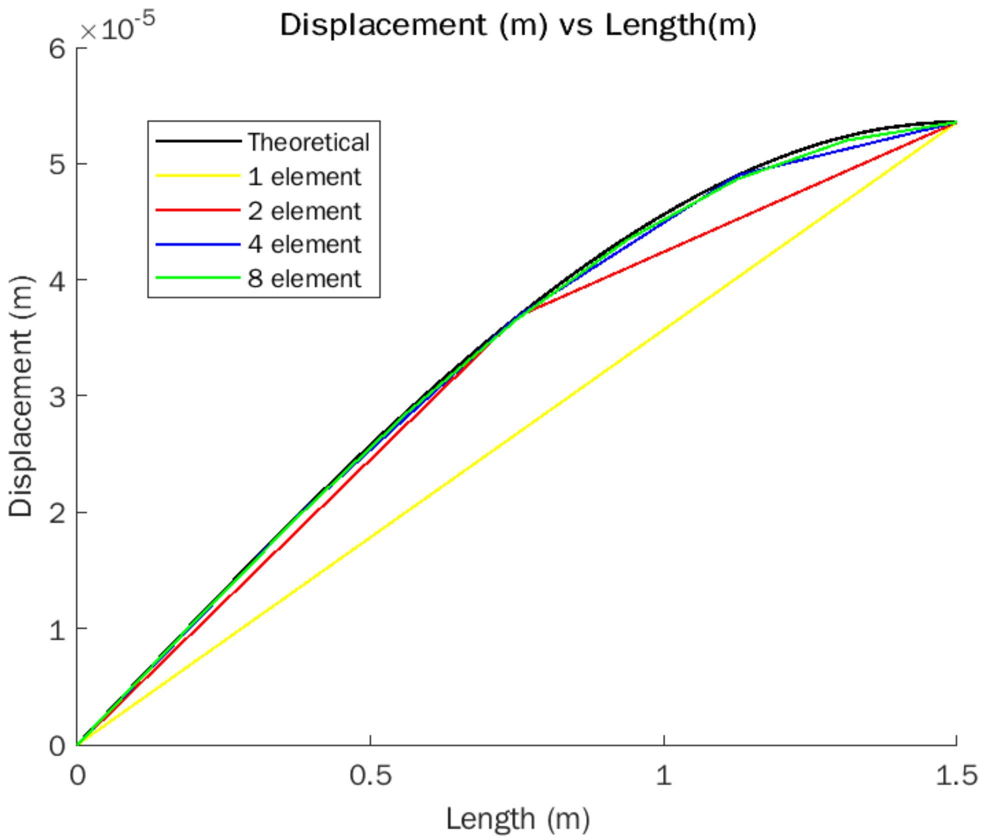


Figure 7

Although even the 8 element model deviates slightly from the exact solution, one can see that as the number of elements increases the accuracy of the model goes up. This notion is further justified by taking a closer look as done in Figure 8.

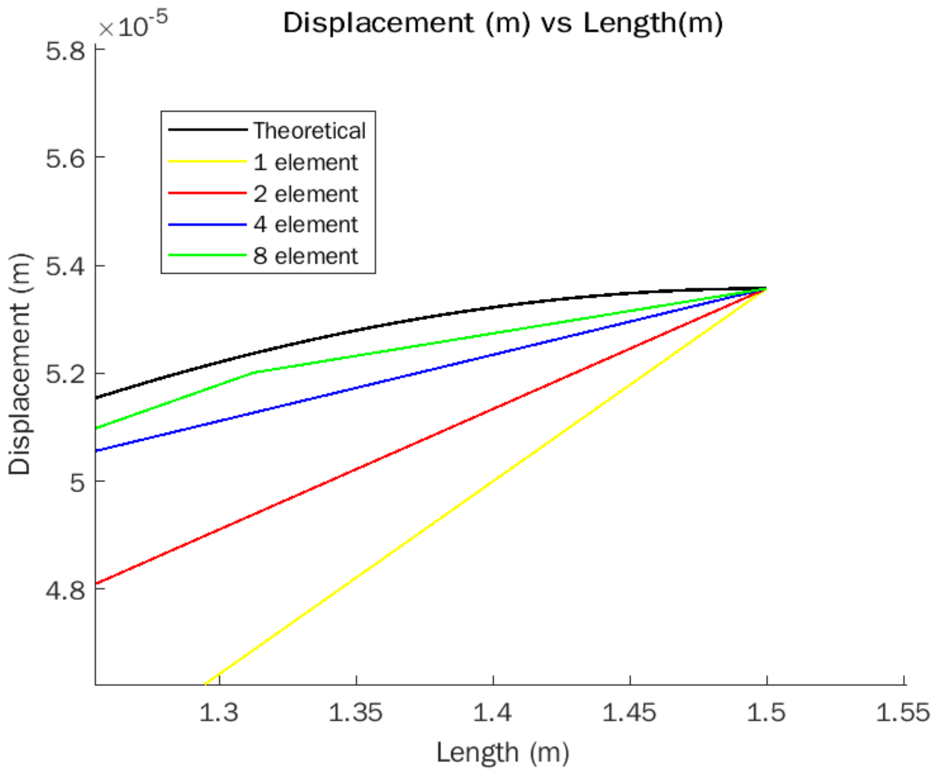


Figure 8

Figure 9 shows the theoretical solution for stresses and the finite element approximations by the fours models.

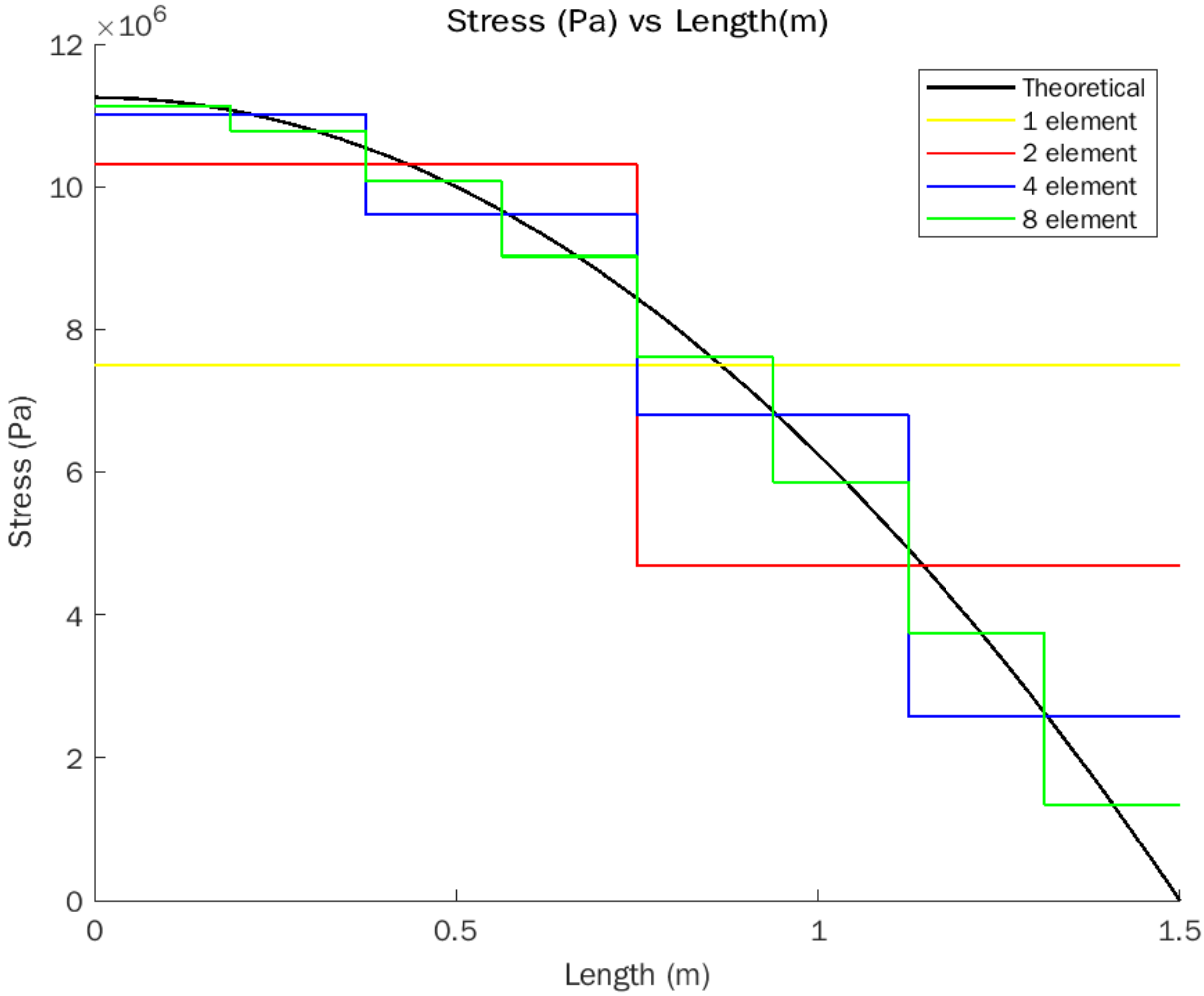
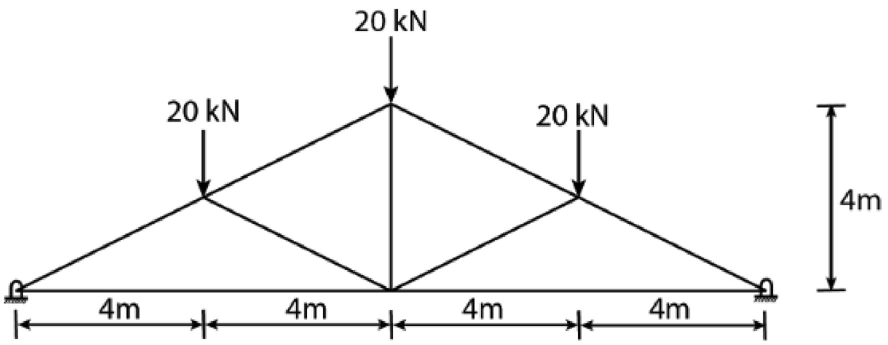


Figure 9

Once again it is clear from the graph that the theoretical value is better approximated by the model as the number of elements goes up. It bears mentioning that all models have significant deviation from theoretical stress values which is not the case with the displacement curves. None of the models predict that the stress at the free end is 0.

Problem 2

Problem 2:
Use symmetry and LINK 180 elements to model the roof truss shown in the figure below, and determine the displacements of the nodes and the stress in each element. All elements have E=210 GPa and A=1×10^{-3} m^2.



Finite Element Model:

A finite element model consisting of 4 nodes and 5 Link 180 elements was constructed. The material properties for the model were as follows: E=210 GPa and $\nu = 0.3$. Due to symmetry in the model about the y axis, only the left half was considered.

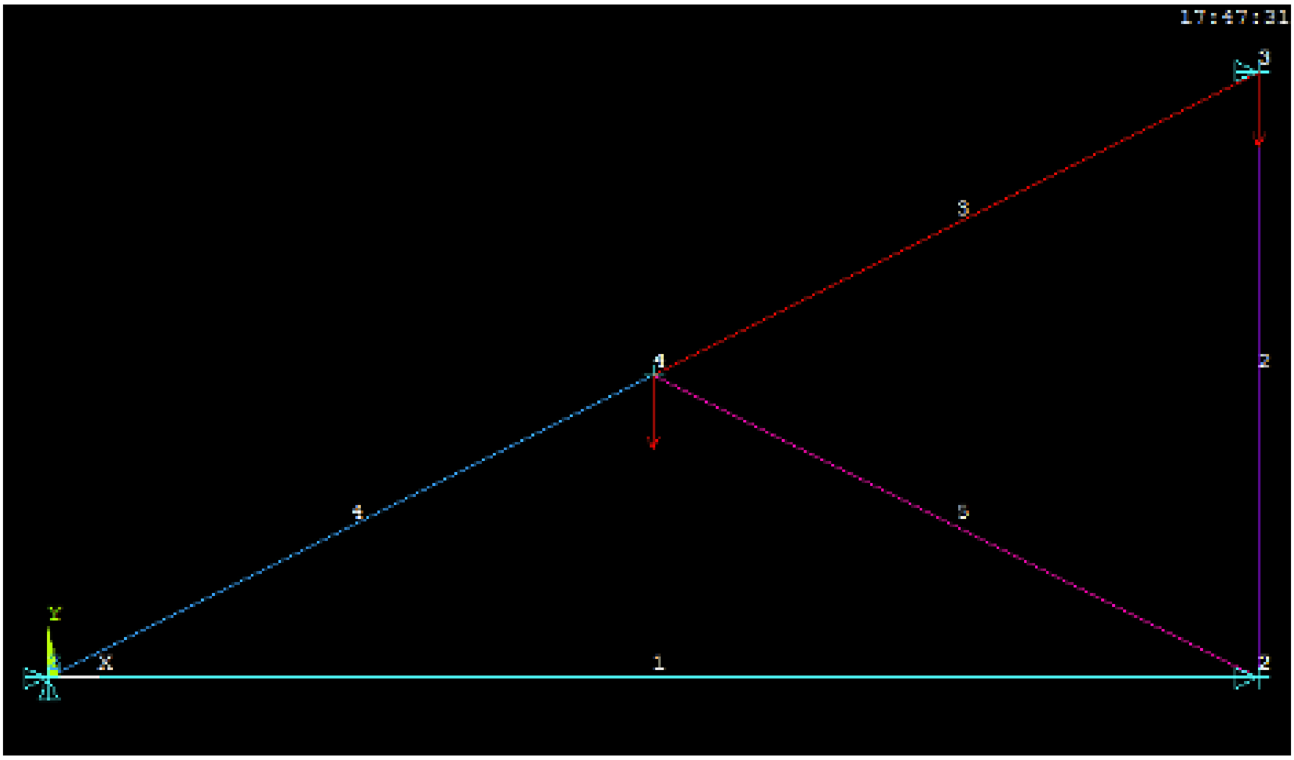


Figure 10

As seen in the model, nodes 2 and 3 were constrained to displace only in y direction because displacement in x direction would be impossible due to symmetry in the model.

The following Table lists results that were obtained from ANSYS.

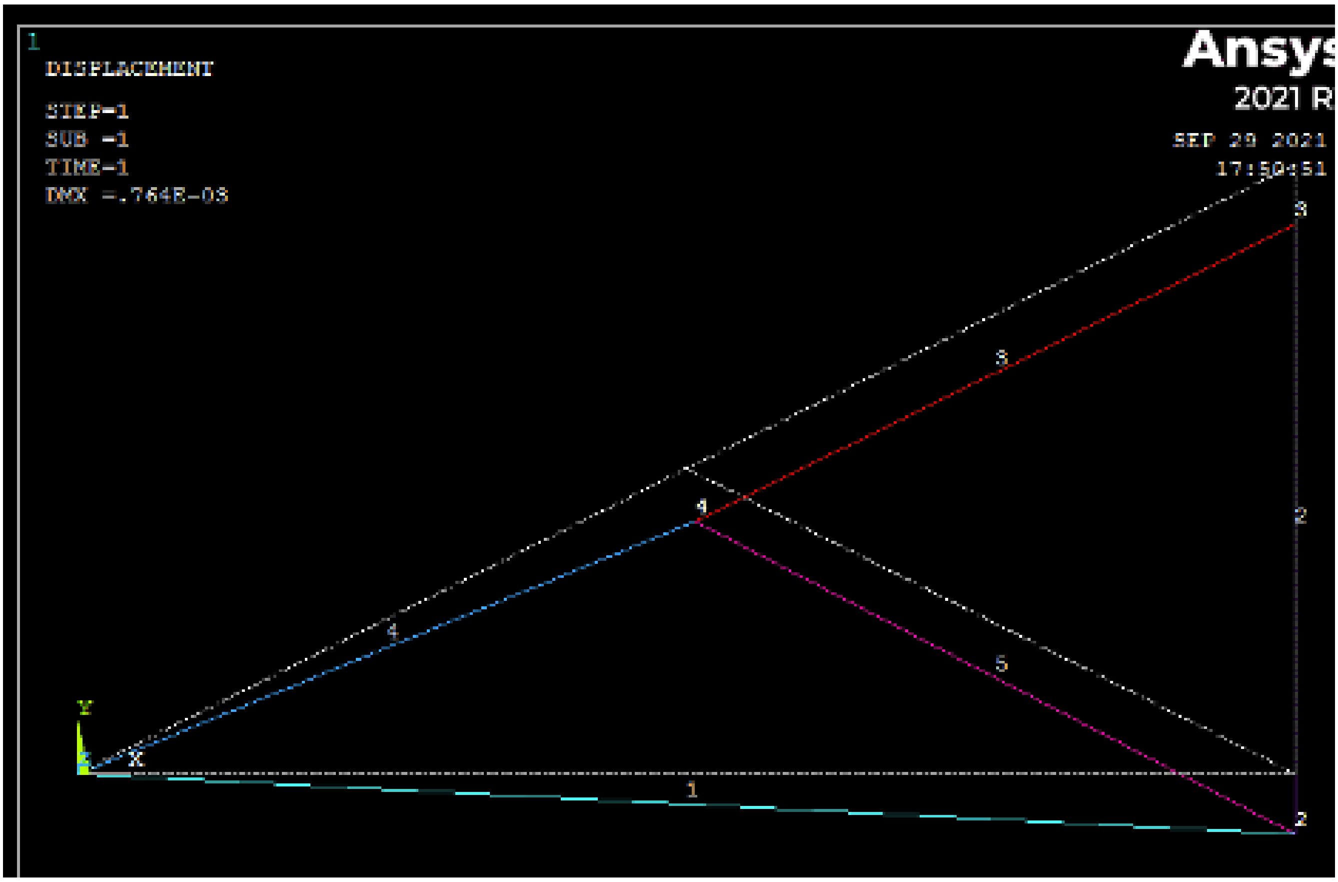
Element Number	Node Number	Observed Stress (X) (Pa)	Observed Stress (Y) (Pa)	Observed X Displacement (m)	Observed Y Displacement (m)	Applied Nodal Force (N)
1	1	0	0	0	0	0
	2	0	0	0	-0.76440 E-003	0
2	2	0.1E7	0	0	-0.76440 E-003	0
	3	0.1E7	0	0	-0.74436 E-003	20000
3	3	-0.67082 E+007	0	0	-0.74436 E-003	20000
	4	-0.67082 E+007	0	0.1124E-003	-0.64840 E-003	20000
4	4	-0.89443 E+007	0	0.1124E-003	-0.64840 E-003	20000
	1	-0.89443 E+007	0	0	0	0
5	4	-0.22361 E+007	0	0.1124E-003	-0.64840 E-003	20000
	2	-0.22361 E+007	0	0	-0.76440 E-003	0

Table 5

Discussion of Results

One interesting thing to note is that there is no stress on any element in the y direction. In reality there would be some stress on the elements in the y direction too due to the presence of a house below the roof. Furthermore, all shear stresses are 0. In reality, there would be some non-zero shear stress in elements too.

Figure 10 shows the deformation of the model under the applied loads.



Conclusion:

For problem 1, the prediction of models went up as the number of elements went up. For all models the stress values differed significantly from observed values. Taking this into account, it is difficult to say whether the displacement and stress values for problem 2 are accurate or not. Furthermore, it must be said that link 180 may not be the best element for problem 2 as displacements are in both x and y directions.

References:

(1) *A first course in the Finite Element Method 6th edition* by Daryl L. Logan. Equation 3.10.33 Pages: 124-133.