

ME 345

Modeling and Simulation

Spring 2025

Case Study 2

Structural Analysis of Beam and Truss

Group 16

Angela Dinh  
Tyler Hoke  
Brandon Kiefer  
Cooper Kinsley  
Jasper Zwemmer

*“We pledge our Honor that we have abided by the Stevens Honor System”*

# Table of Contents

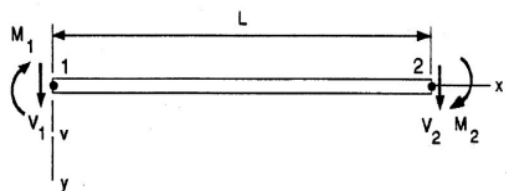
Problem 1: Cantilever Beam Loading Analysis.....	2
Introduction.....	2
Part A: By Hand FEM.....	3
Analysis 1: 4 Nodes .....	3
Analysis 2: 5 Nodes .....	6
Part B: Principles of Superposition.....	9
Part C: SolidWorks Simulation and Comparison of Results .....	10
Part D: Stress Analysis.....	13
Problem 2: Structural Truss Analysis.....	16
Introduction.....	16
Part A1: Zero-Force Members .....	17
Part A2: SolidWorks Analysis of Internal Forces .....	17
Part B: Finite Element Analysis by Hand .....	19
Part C: SolidWorks Analysis of Nodal Displacement.....	21
Part D: Internal Forces .....	23
Part E: MATLAB 2D Finite Element Analysis.....	26
References.....	29
Appendix.....	30
A-1: By Hand FEM - Matlab Code.....	30
A-2: Principle of Superposition - Matlab Code .....	32
A-3: Deflection Analysis Methodology - Additional Screenshots.....	33
A-4: Stress Analysis Methodology - Additional Screenshots .....	34
B-1: Problem 2 Additional Screenshots .....	36
B-2: Part B MATLAB Matrix Code.....	38
B-3: Part E MATLAB Code.....	40
C-1: Contribution Chart .....	42

# Problem 1: Cantilever Beam Loading Analysis

## Introduction

The finite element method is one of the most popular tools for conducting a stress analysis of a model. The method works by breaking down a large, complicated domain into smaller pieces called elements, and statically analyzing those elements individually. Each element within the model is governed by a set of mathematical equations with variables that can be used to determine its mechanical properties. When paired with boundary conditions, those equations can be assembled into a solvable system that can be used as an efficient way to study complex models.

In this problem, the team experimented with the finite element method by following it first “by hand” to solve a generic beam element under loading for displacement. The results of these calculations were then compared to classical methods for solving mechanical properties, as well as to a computer-ran finite element analysis. The beam studied was a cantilever beam, fixed on one end and experiencing loadings elsewhere throughout. Since the calculations in the finite element method require solving a large system of equations, element stiffness matrices are used to simplify the process. The stiffness matrix used for finding linear and angular displacement is shown in Figure 1.1, where  $E$  is the modulus of elasticity of the beam’s material,  $I$  is the beam’s moment of inertia, and  $L$  is the length of an element in the beam.  $V$  and  $M$  are the externally applied shear forces and moments and  $v$  and  $\theta$  are the vertical displacements and angles of twist of each node, respectively.



$$\begin{bmatrix} V_1 \\ M_1 \\ V_2 \\ M_2 \end{bmatrix} = \frac{EI}{L^3} \begin{bmatrix} 12 & 6L & -12 & 6L \\ 6L & 4L^2 & -6L & 2L^2 \\ -12 & -6L & 12 & -6L \\ 6L & 2L^2 & -6L & 4L^2 \end{bmatrix} \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{bmatrix}$$

Figure 1.1: Beam Element & Elemental Stiffness Matrix

For this problem, the team assumes a modulus of elasticity of  $29 \times 10^6$  psi and a moment of inertia of  $12 \text{ in}^4$ . The loading conditions to be used throughout the analysis are shown below in Figure 1.2.

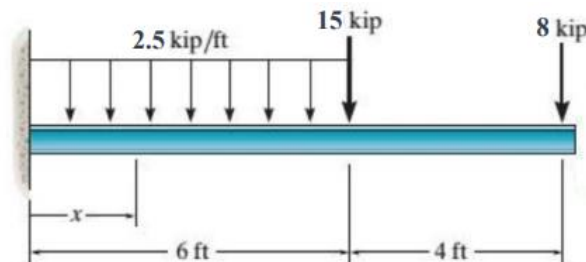


Figure 1.2: Cantilevered beam subject to an applied loading.

## Part A: By Hand FEM

In this section, the displacement of the beam at each node is analyzed using the Finite Element Method (FEM). The process begins by calculating the reaction force and moment at the fixed support. Subsequently, the uniformly distributed load acting along the beam segment from 0 ft to 6 ft is converted into equivalent point loads. The analysis then proceeds by calculating the elemental stiffness matrices based on the defined geometry and loading conditions, assembling these into a global stiffness matrix, applying boundary conditions to create a simplified global stiffness matrix, and finally solving the vertical displacements at each node. Two distinct scenarios are evaluated to examine the influence of element discretization on the accuracy of the solution.

### Analysis 1: 4 Nodes

The first scenario utilizes three beam elements with nodes positioned at  $x = 0, 3, 6,$  and  $10$  ft, as illustrated in Figure 1.3. The second scenario (presented subsequently) further refines the discretization to four beam elements and five nodes to demonstrate how increased discretization affects the computed nodal displacements.

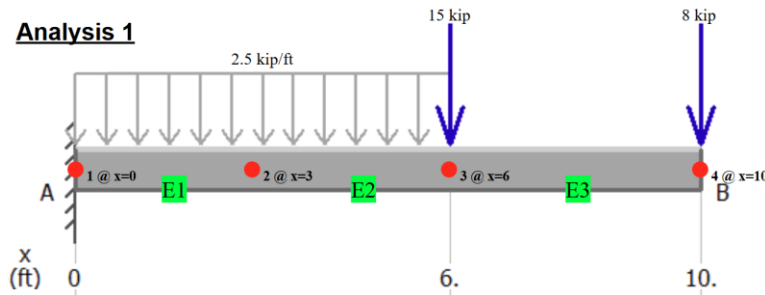


Figure 1.3: Analysis 2 Nodes

Applying sum of forces and sum of moments about point A, the reaction force and moment at the fixed support can be found.

$$R_A = \left(2.5 \frac{\text{kip}}{\text{ft}}\right) (6 \text{ ft}) + 15 \text{ kip} + 8 \text{ kip} = 38 \text{ kip}$$

$$R_A = 38,000 \text{ lbf}$$

$$M_A = \left(2.5 \frac{\text{kip}}{\text{ft}}\right) (6 \text{ ft})(3 \text{ ft}) + (15 \text{ kip})(6 \text{ ft}) + (8 \text{ kip})(10 \text{ ft}) = 215 \text{ kip} \cdot \text{ft}$$

$$M_A = 215,000 \text{ lbf} \cdot \text{ft} \quad (\text{CCW}+)$$

Next, in order to approximate the distributed load with our discrete number of nodes, the distributed load ( $w = 2500 \text{ lbf}$ ) is converted into point loads on nodes 1, 2, and 3 by looking at each element individually.

$$\begin{aligned}
 V_1 &= R_A - \frac{w(L_{E1})}{2} = R_A - \frac{w(L_{E1})}{2} = 34250 \text{ lbf} \\
 V_2 &= -\frac{w(L_{E1})}{2} - \frac{w(L_{E2})}{2} = -\frac{w(3 \text{ ft})}{2} - \frac{w(3 \text{ ft})}{2} = -7500 \text{ lbf} \\
 V_3 &= -\frac{w(L_{E2})}{2} - 15000 \text{ lbf} = -\frac{w(3 \text{ ft})}{2} - 15000 \text{ lbf} = -18750 \text{ lbf} \\
 V_4 &= -8000 \text{ lbf}
 \end{aligned}$$

The moment at node 1 equals the reaction moment calculated earlier. Moments at nodes 2, 3, and 4 are zero.

$$M_1 = M_A = 215,000 \text{ lbf} \cdot \text{ft}, \quad M_2 = M_3 = M_4 = 0$$

For this problem, solving  $M_1$  and  $V_1$  by hand is not necessary, but is shown to validate the MATLAB calculations.  $M_1$  and  $V_1$  are canceled out of the global stiffness matrix when initial conditions are applied later. See the matrix below for a summary of the applied forces and moments.

$$\overline{F}_{3\_Elements} = \begin{bmatrix} V_1 & - \\ M_1 & - \\ V_2 & -7500 \text{ lbf} \\ M_2 & 0 \\ V_3 & -18750 \text{ lbf} \\ M_3 & 0 \\ V_4 & -8000 \text{ lbf} \\ M_4 & 0 \end{bmatrix}$$

*Figure 1.4: Initial Force Vector – 3 Elements*

Next, we can begin solving for the displacements. It is important to note the elemental stiffness matrix shown in Figure 1.1, as well as parameters  $E$ ,  $I$ , and  $L$  which are the elastic modulus, moment of inertia, and length of the beam element respectively.  $E$  and  $I$  are constant for each element, but  $L$  varies. See the values given for  $E$  and  $I$  below. It is important to note the units when performing calculations.

$$E = 29 \times 10^6 \text{ psi}, \quad I = 12 \text{ in}^4, \quad L_1 = 36 \text{ in}, \quad L_2 = 36 \text{ in}, \quad L_3 = 48 \text{ in}$$

Another key aspect of this solution is the boundary conditions of the problem. For this fixed-end cantilever beam, the boundary conditions are deflection at node 1 equals zero and the slope at node 1 equals zero. This is because the end is fixed. It is important to establish these boundary conditions because it allows us to simplify our global stiffness matrix to be solvable.

$$v_1 = 0, \quad \theta_1 = 0$$

Using the elemental stiffness matrix in Figure 1.1, the following elemental stiffness matrices were calculated.

$$\begin{aligned}
 [El_1] \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{bmatrix} &= 10^7 \begin{bmatrix} 0.009 & 0.1611 & -0.009 & 0.1611 \\ 0.1611 & 3.8667 & -0.1611 & 1.9333 \\ -0.009 & -0.1611 & 0.009 & -0.1611 \\ 0.1611 & 1.9333 & -0.1611 & 3.8667 \end{bmatrix} \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{bmatrix} = \begin{bmatrix} V_1 \\ M_1 \\ V_2 \\ M_2 \end{bmatrix} \\
 [El_2] \begin{bmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{bmatrix} &= 10^7 \begin{bmatrix} 0.009 & 0.1611 & -0.009 & 0.1611 \\ 0.1611 & 3.8667 & -0.1611 & 1.9333 \\ -0.009 & -0.1611 & 0.009 & -0.1611 \\ 0.1611 & 1.9333 & -0.1611 & 3.8667 \end{bmatrix} \begin{bmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{bmatrix} = \begin{bmatrix} V_2 \\ M_2 \\ V_3 \\ M_3 \end{bmatrix} \\
 [El_3] \begin{bmatrix} v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{bmatrix} &= 10^7 \begin{bmatrix} 0.0038 & 0.0906 & -0.0038 & 0.0906 \\ 0.0906 & 2.900 & -0.0906 & 1.450 \\ -0.0038 & -0.0906 & 0.0038 & -0.0906 \\ 0.0906 & 1.450 & -0.0906 & 2.900 \end{bmatrix} \begin{bmatrix} v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{bmatrix} = \begin{bmatrix} V_3 \\ M_3 \\ V_4 \\ M_4 \end{bmatrix}
 \end{aligned}$$

Now with the three-elemental stiffness matrices, they can be combined to form a global stiffness matrix.

$$[Gl] \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{bmatrix} = 10^7 \begin{bmatrix} 0.0009 & 0.0161 & -0.0009 & 0.0161 & 0 & 0 & 0 & 0 \\ 0.0161 & 0.3867 & -0.0161 & 0.1933 & 0 & 0 & 0 & 0 \\ -0.0009 & -0.0161 & 0.0018 & 0 & -0.0009 & 0.0161 & 0 & 0 \\ 0.0161 & 0.1933 & 0 & 0.7733 & -0.0161 & 0.1933 & 0 & 0 \\ 0 & 0 & -0.0009 & -0.0161 & 0.0013 & -0.0070 & -0.0004 & 0.0091 \\ 0 & 0 & 0.0161 & 0.1933 & -0.0070 & 0.6767 & -0.0091 & 0.1450 \\ 0 & 0 & 0 & 0 & -0.0004 & -0.0091 & 0.0004 & -0.0091 \\ 0 & 0 & 0 & 0 & 0.0091 & 0.1450 & -0.0091 & 0.2900 \end{bmatrix} \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{bmatrix} = \begin{bmatrix} V_1 \\ M_1 \\ V_2 \\ M_2 \\ V_3 \\ M_3 \\ V_4 \\ M_4 \end{bmatrix}$$

Next, rows 1 and 2 and columns 1 and 2 are removed to form the simplified global stiffness matrix. This can be done because of the boundary conditions.  $v_1 = 0$  and  $\theta_1 = 0$ .

$$[Gl_{IC}] \begin{bmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{bmatrix} = 10^7 \begin{bmatrix} 0.0018 & 0 & -0.0009 & 0.0161 & 0 & 0 \\ 0 & 0.7733 & -0.0161 & 0.1933 & 0 & 0 \\ -0.0009 & -0.0161 & 0.0013 & -0.0070 & -0.0004 & 0.0091 \\ 0.0161 & 0.1933 & -0.0070 & 0.6767 & -0.0091 & 0.1450 \\ 0 & 0 & -0.0004 & -0.0091 & 0.0004 & -0.0091 \\ 0 & 0 & 0.0091 & 0.1450 & -0.0091 & 0.2900 \end{bmatrix} \begin{bmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{bmatrix} = \begin{bmatrix} V_2 \\ M_2 \\ V_3 \\ M_3 \\ V_4 \\ M_4 \end{bmatrix}$$

Finally, the simplified global stiffness matrix is inverted and multiplied by the force vector, giving us the displacement vector. Adding back in  $v_1 = 0$  and  $\theta_1 = 0$  gives the full displacement vector. Displacements are measured in inches, and slope in radians.

$$[Gl_{IC}]^{-1} \begin{bmatrix} V_2 \\ M_2 \\ V_3 \\ M_3 \\ V_4 \\ M_4 \end{bmatrix} = \begin{bmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{bmatrix} = \begin{bmatrix} -4.0388 \\ -0.2031 \\ -13.2617 \\ -0.2927 \\ -28.1566 \\ -0.3191 \end{bmatrix} \rightarrow \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ -4.0388 \\ -0.2031 \\ -13.2617 \\ -0.2927 \\ -28.1566 \\ -0.3191 \end{bmatrix}$$

Lastly, the solution was then verified in MATLAB using sum of forces and sum of moments. The sum of forces and moments are both very close to zero. See Appendix A-1, 4-Node Analysis for the full MATLAB code.

```
SumOfFor = 1.7462e-10
SumOfMom = 1.2107e-08
```

Figure 1.5: Sum of Forces and Moments Result i

## Analysis 2: 5 Nodes

Second, an analysis with four elements and nodes at  $x = 0, 1.5, 4.5, 6$ , and  $10$  ft was conducted to examine the effects of changing the number of nodes and elements on the simulation results.

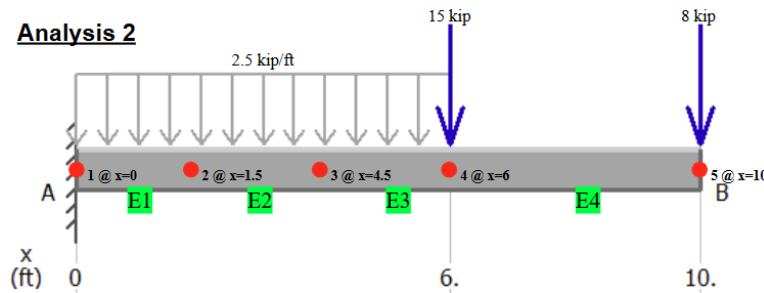


Figure 1.6: Analysis 2 Nodes

Since the loading conditions on the beam are the same, the reaction and moment at point A are also the same.

$$R_A = 38,000 \text{ lbf}, \quad M_A = 215,000 \text{ lbf} \cdot \text{ft} \quad (\text{CCW}+)$$

Next, the distributed load ( $w = 2500 \text{ lbf}$ ) is converted into point loads on nodes 1, 2, 3, and 4 using the same method in Analysis 1.

$$V_2 = -\frac{w(L_{E1})}{2} - \frac{w(L_{E2})}{2} = -\frac{w(1.5 \text{ ft})}{2} - \frac{w(3 \text{ ft})}{2} = -5625 \text{ lbf}$$

$$V_3 = -\frac{w(L_{E2})}{2} - \frac{w(L_{E3})}{2} = -\frac{w(3 \text{ ft})}{2} - \frac{w(1.5 \text{ ft})}{2} = -5625 \text{ lbf}$$

$$V_4 = -\frac{w(L_{E3})}{2} - 15000 \text{ lbf} = -\frac{w(1.5 \text{ ft})}{2} - 15000 \text{ lbf} = -16875 \text{ lbf}$$

$$V_5 = -8000 \text{ lbf}$$

The moment at node 1 is still equal to the reaction moment calculated earlier. Moments at nodes 2, 3, 4, and 5 are zero.

$$M_1 = M_A = 215,000 \text{ lbf} \cdot \text{ft}, \quad M_2 = M_3 = M_4 = M_5 = 0$$

The parameters for this analysis are shown below. The modulus and moment of inertia are the same. There is just an additional length for the fourth element. The boundary conditions are the same as Analysis 1 with  $v_1 = 0$  and  $\theta_1 = 0$ .

$$E = 29 \times 10^6 \text{ psi}, \quad I = 12 \text{ in}^4, \quad L_1 = 18 \text{ in}, \quad L_2 = 36 \text{ in}, \quad L_3 = 18 \text{ in}, \quad L_4 = 48 \text{ in}$$

Using the elemental stiffness matrix in Figure 1.1, the following elemental stiffness matrices were calculated.

$$[El_1] \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{bmatrix} = 10^7 \begin{bmatrix} 0.0072 & 0.0644 & -0.0072 & 0.0644 \\ 0.0644 & 0.7733 & -0.0644 & 0.3867 \\ -0.0072 & -0.0644 & 0.0072 & -0.0644 \\ 0.0644 & 0.3867 & -0.0644 & 0.7733 \end{bmatrix} \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \end{bmatrix} = \begin{bmatrix} V_1 \\ M_1 \\ V_2 \\ M_2 \end{bmatrix}$$

$$[El_2] \begin{bmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{bmatrix} = 10^7 \begin{bmatrix} 0.0009 & 0.0161 & -0.0009 & 0.0161 \\ 0.0161 & 0.3867 & -0.0161 & 0.1933 \\ -0.0009 & -0.0161 & 0.0009 & -0.0161 \\ 0.0161 & 0.1933 & -0.0161 & 0.3867 \end{bmatrix} \begin{bmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \end{bmatrix} = \begin{bmatrix} V_2 \\ M_2 \\ V_3 \\ M_3 \end{bmatrix}$$

$$[El_3] \begin{bmatrix} v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{bmatrix} = 10^7 \begin{bmatrix} 0.0072 & 0.0644 & -0.0072 & 0.0644 \\ 0.0644 & 0.7733 & -0.0644 & 0.3867 \\ -0.0072 & -0.0644 & 0.0072 & -0.0644 \\ 0.0644 & 0.3867 & -0.0644 & 0.7733 \end{bmatrix} \begin{bmatrix} v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \end{bmatrix} = \begin{bmatrix} V_3 \\ M_3 \\ V_4 \\ M_4 \end{bmatrix}$$

$$[El_4] \begin{bmatrix} v_4 \\ \theta_4 \\ v_5 \\ \theta_5 \end{bmatrix} = 10^7 \begin{bmatrix} 0.0038 & 0.0906 & -0.0038 & 0.0906 \\ 0.0906 & 2.900 & -0.0906 & 1.450 \\ -0.0038 & -0.0906 & 0.0038 & -0.0906 \\ 0.0906 & 1.450 & -0.0906 & 2.900 \end{bmatrix} \begin{bmatrix} v_4 \\ \theta_4 \\ v_5 \\ \theta_5 \end{bmatrix} = \begin{bmatrix} V_4 \\ M_4 \\ V_5 \\ M_5 \end{bmatrix}$$

Now with the four elemental stiffness matrices, they can be combined to form a global stiffness matrix of 10x10 size.

$$[Gl] = 10^7 \begin{bmatrix} 0.0072 & 0.0644 & -0.0072 & 0.0644 & 0 & 0 & 0 & 0 & 0 & 0 \\ 0.0644 & 0.7733 & -0.0644 & 0.3867 & 0 & 0 & 0 & 0 & 0 & 0 \\ -0.0072 & -0.0644 & 0.0081 & -0.0483 & -0.0009 & 0.0161 & 0 & 0 & 0 & 0 \\ 0.0644 & 0.3867 & -0.0483 & 1.1600 & -0.0161 & 0.1933 & 0 & 0 & 0 & 0 \\ 0 & 0 & -0.0009 & -0.0161 & 0.0081 & 0.0483 & -0.0072 & 0.0644 & 0 & 0 \\ 0 & 0 & 0.0161 & 0.1933 & 0.0483 & 1.1600 & -0.0644 & 0.3867 & 0 & 0 \\ 0 & 0 & 0 & 0 & -0.0072 & -0.0644 & 0.0075 & -0.0554 & -0.0004 & 0.0091 \\ 0 & 0 & 0 & 0 & 0.0644 & 0.3867 & -0.0554 & 1.0633 & -0.0091 & 0.1450 \\ 0 & 0 & 0 & 0 & 0 & 0 & -0.0004 & -0.0091 & 0.0004 & -0.0091 \\ 0 & 0 & 0 & 0 & 0 & 0 & 0.0091 & 0.1450 & -0.0091 & 0.2900 \end{bmatrix}$$

Next, rows 1 and 2 and columns 1 and 2 are removed to form the simplified global stiffness matrix. This can be done because of the boundary conditions.  $v_1 = 0$  and  $\theta_1 = 0$ .

$$[Gl_{IC}] = 10^7 \begin{bmatrix} 0.0081 & -0.0483 & -0.0009 & 0.0161 & 0 & 0 & 0 & 0 \\ -0.0483 & 1.1600 & -0.0161 & 0.1933 & 0 & 0 & 0 & 0 \\ -0.0009 & -0.0161 & 0.0081 & 0.0483 & -0.0072 & 0.0644 & 0 & 0 \\ 0.0161 & 0.1933 & 0.0483 & 1.1600 & -0.0644 & 0.3867 & 0 & 0 \\ 0 & 0 & -0.0072 & -0.0644 & 0.0075 & -0.0554 & -0.0004 & 0.0091 \\ 0 & 0 & 0.0644 & 0.3867 & -0.0554 & 1.0633 & -0.0091 & 0.1450 \\ 0 & 0 & 0 & 0 & -0.0004 & -0.0091 & 0.0004 & -0.0091 \\ 0 & 0 & 0 & 0 & 0.0091 & 0.1450 & -0.0091 & 0.2900 \end{bmatrix}$$



Finally, the simplified global stiffness matrix is inverted and multiplied by the force vector, giving us the displacement vector. Adding back in  $v_1 = 0$  and  $\theta_1 = 0$  gives the full displacement vector. Displacements are measured in inches, and slope in radians.

$$[Gl_{IC}]^{-1} \begin{bmatrix} V_2 \\ M_2 \\ V_3 \\ M_3 \\ V_4 \\ M_4 \\ V_5 \\ M_5 \end{bmatrix} = \begin{bmatrix} v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \\ v_5 \\ \theta_5 \end{bmatrix} = \begin{bmatrix} -1.1001 \\ -0.1166 \\ -8.2107 \\ -0.2595 \\ -13.1988 \\ -0.2909 \\ -28.0099 \\ -0.3174 \end{bmatrix} \rightarrow \begin{bmatrix} v_1 \\ \theta_1 \\ v_2 \\ \theta_2 \\ v_3 \\ \theta_3 \\ v_4 \\ \theta_4 \\ v_5 \\ \theta_5 \end{bmatrix} = \begin{bmatrix} 0 \\ 0 \\ -1.1001 \\ -0.1166 \\ -8.2107 \\ -0.2595 \\ -13.1988 \\ -0.2909 \\ -28.0099 \\ -0.3174 \end{bmatrix}$$

The resultant displacement vector for the four-node analysis found a maximum displacement at the end of the cantilever beam of 28.0099 inches. The solution was then verified in MATLAB using sum of forces, sum of moments, and using a code given in class. The sum of forces and moments are both very close to zero. See Appendix A-1, 5-Node Analysis for the full MATLAB code.

```
SumOfFor = -9.8953e-10
SumOfMom = -8.8476e-08
```

Figure 1.6: Sum of Forces and Moments Result ii

Figure 1.7 displays the deflection along the length of the cantilever beam for both the 4-node and 5-node variations. Both curves are similar to each other. The greatest difference is between 0 and 2 feet where the additional point in the 5-node simulation deviates from the 4-node simulation. This shows that adding additional nodes can help produce an accurate simulation and give insight into what is happening at all points throughout the beam. Generally, the deflections are very similar between the 4 and 5-node simulations.

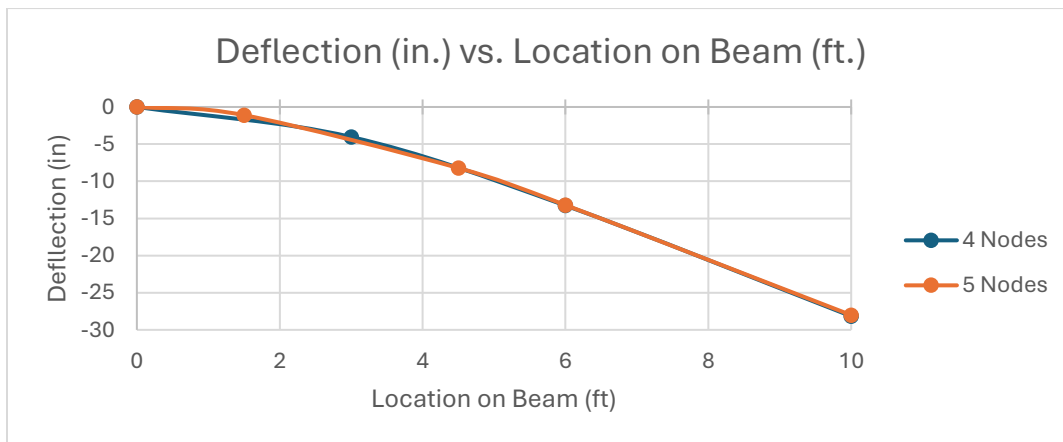


Figure 1.7: By Hand 4 vs. 5 Node Comparison

## Part B: Principles of Superposition

To provide an additional set of results from hand calculations, the principles of superposition were used to find the displacement of the beam at the nodes analyzed in Part A [2]. The principle of superposition can be a useful method for finding a very accurate measurement of displacement in a generic beam element. The method pulls from a list of classical equations used for different loading scenarios. The equations are used to find the displacement at a specific node caused by one of the loadings acting alone. The principles of superposition then declare that the sum of all the displacements caused by each force at a node is the total displacement experienced by the node. When applied to each node on a beam, an accurate model of the displacements experienced throughout the beam can be made.

For this problem specifically, the displacements throughout the entire beam can be defined by three equations specific for a cantilever under loading. The equations calculate the displacement experienced by a node for when the node is to the right, left, or exactly at the location of an applied force being analyzed. The three equations are shown below, where  $P$  is the magnitude of the force,  $E$  is the modulus of elasticity,  $I$  is the moment of inertia,  $x$  is the location of the node being analyzed along the beam, and  $a$  is the location of the force being analyzed along the beam.

$$y = \frac{Px^3}{3EI} \quad \text{for } x = a$$

$$y = \frac{Px^2}{6EI}(3a - x) \quad \text{for } x < a$$

$$y = \frac{Pa^2}{6EI}(3x - a) \quad \text{for } x > a$$

To simplify the analysis, the distributed load in the model is treated as a series of point loads. In other words, the total force applied on the beam throughout the distributed load is divided amongst each of the nodes affected by the distributed load. The simplification will slightly affect the accuracy of the results; however, they will still be a good representation of the actual displacements.

First, to create a comparison for the results of Analysis 1, nodes at  $x = 0, 3, 6,$  and  $10$  feet are analyzed. Using the equations above, a displacement is calculated for each pairing of nodes and applied force. To streamline these calculations, a MATLAB script (see Appendix A-2) was drafted with a matrix holding the magnitude of forces being applied to each node, as well as the location of each node. Then, two *for* loops cycled through combining every possible combination of the location of the node and force applied at a node. A system of *if* statements checks if the force was applied to the left, right, or exactly on the node being analyzed, and the appropriate equation was used to find the effective displacement at the node caused by that force. In the end, the displacements at each node were added up and the results were displayed.

The same process was repeated for Analysis 2, with the nodes here being placed at  $x = 0$ , 1.5, 4.5, 6, and 10 feet. The distributed load was divided up again to fit the new node distribution, and a new starting matrix was defined. The displacement results are shown below in Figure 1.8.

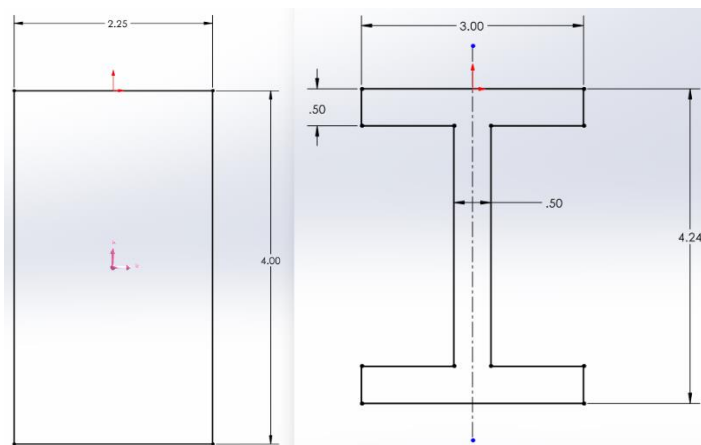
Analysis 1	
Position of Node	Vertical Displacement
0 feet	0 inches
3 feet	-4.039 inches
6 feet	-13.262 inches
10 feet	-28.157 inches

Analysis 2	
Position of Node	Vertical Displacement
0 feet	0 inches
1.5 feet	-1.100 inches
4.5 feet	-8.211 inches
6 feet	-13.199 inches
10 feet	-28.010 inches

*Figure 1.8: Principle of Superposition Displacement Tables*

## Part C: SolidWorks Simulation and Comparison of Results

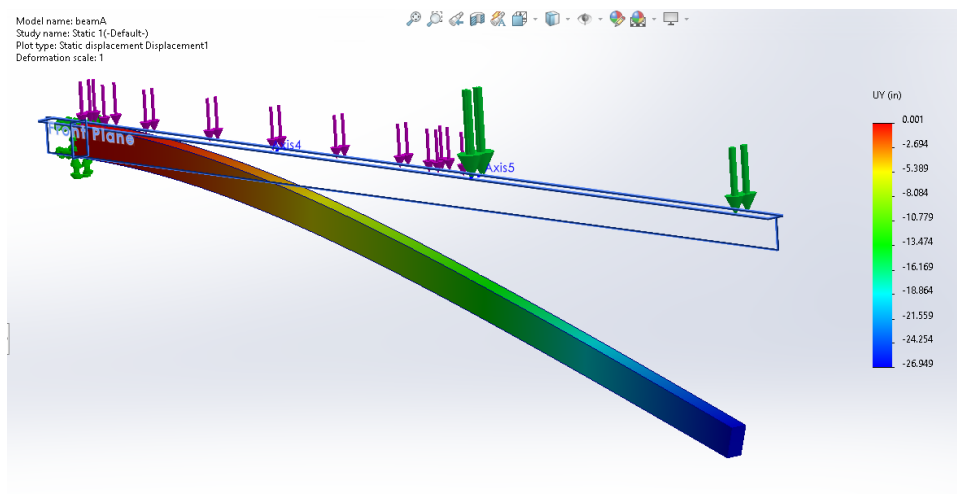
Next, the Part A and Part B solutions were verified using SolidWorks. First, models of the beam were created with two different cross-sections. The team decided to test a standard rectangular cross-section, as well as an I-shaped cross-section. Both cross sections have the same moment of inertia as the analytical solutions ( $I = 12.00 \text{ in}^4$ ). See the figure below for the dimensions of the beam cross sections.



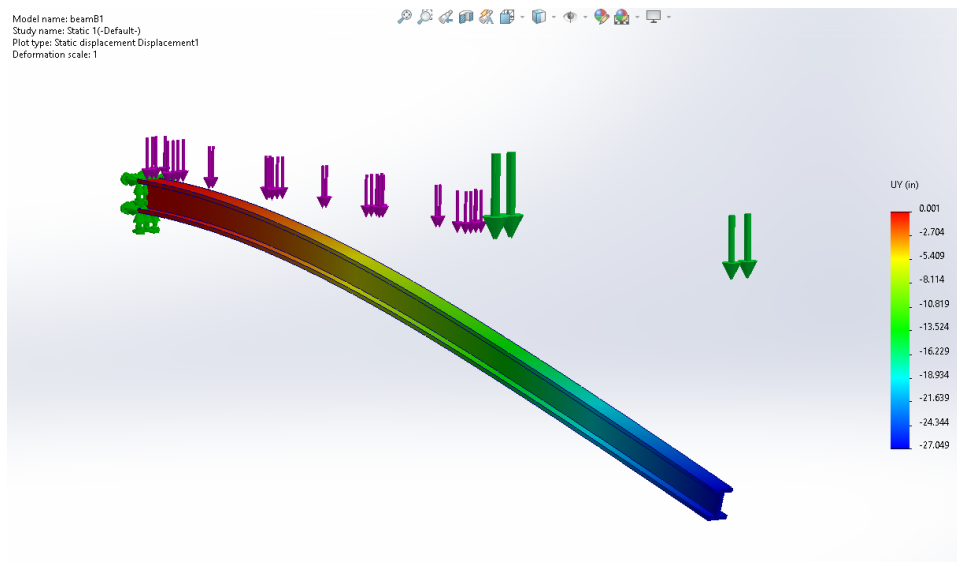
*Figure 1.9: Beam Cross Section Drawings (in.)*

Using SolidWorks' section properties feature, the moment of inertia of each beam was verified as  $12.00 \text{ in}^4$ . Then, both beams were extruded 10 feet, and reference planes were added at the distances examined (1.5ft, 3ft, 4.5ft, 6ft, and 10ft) down the beam. The material was set as AISI 1020 Stainless Steel with an elastic modulus of  $29 \times 10^6 \text{ psi}$ . Now with the models, the static analysis can begin.

The static analysis starts with defining the fixed geometry and the loads applied. The left face of the beams was fixed. The distributed load was applied as 15 kips on the top face of the beam from  $x=0\text{ft}$  to 6 ft. Then the 15 kip load was added on a split line at 6ft and an 8 kip load was added at  $x=10\text{ft}$ . Next, a mesh was created. The mesh was selected with a fine level of detail. This resulted in a good level of detail, but a quick simulation time. Both beams used the same load conditions and simulation settings. See the figure below for the simulation results.



*Figure 1.10: Rectangular Beam Displacement Plot*



*Figure 1.11: I-Beam Displacement Plot*

Using the probe tool, we can examine the displacements along the bottom edge of the beams and create a table to compare the displacements from all solution methodologies so far.

Beam Location (ft)	Method of Solving for Deflection (in.)					
	SolidWorks FEM		Principle of Superposition		By Hand FEM	
	Rectangular	I-Beam	4 Nodes	5 Nodes	4 Nodes	5 Nodes
0	0	0	0	0	0	0
1.5	-1.066	-1.086	X	-1.100	X	-1.100
3	-3.872	-3.913	-4.039	X	-4.039	X
4.5	-7.886	-7.944	X	-8.211	X	-8.211
6	-12.646	-12.745	-13.262	-13.199	-13.262	-13.199
10	-26.763	-26.845	-28.157	-28.010	-28.157	-28.010

*Figure 1.12: Deflection (in.) Comparison Table*

As shown in Figure 1.12, the deflection values across the different methods are generally consistent. Differences in these values are caused by certain model assumptions as well as model detail. For example, one major assumption made for the principle of superposition and the by-hand calculations was approximating the distributed load as multiple point loads at the individual nodes. In these calculations, the distributed load was split up into point loads at each node. For the SolidWorks FEM, the distributed load was applied across the surface of the beam. The point loads are an approximation of the distributed load behavior, but they do not affect the displacement in the same way.

Additionally, the level of detail in the SolidWorks simulation was much higher. The mesh used in SolidWorks was comprised of tens of thousands of individual nodes. In contrast, the principle of superposition method and by-hand FEM calculations used either four or five nodes and a 2D simulation, greatly simplifying the structure. This higher mesh density in SolidWorks allowed for more accurate capture of local deformations and stress concentrations which are typically neglected or approximated in hand calculations. As a result, SolidWorks provided a slightly more conservative deflection. The by-hand methods, while effective for estimating overall behavior, rely on idealized conditions and simplified geometry. Therefore, the increased detail in SolidWorks contributes to a more realistic and comprehensive understanding of the beam's response under the given load.

Looking more specifically at the SolidWorks simulations, the cross-sectional shape of the beam had a minimal effect on the deflection of the beam. Generally, the I-beam deflected more than the rectangular beam, but the deflection at the end of the beam was about one-tenth of an inch greater. With this being a 10ft long beam, a tenth of an inch is insignificant. Since both beams were designed with a moment of inertia of  $12\text{in}^4$ , this is to be expected.

Next, the reactions at the fixed support were verified in SolidWorks. The reaction force for both the rectangular and I-beam was found as 38,000 lbf. This aligns perfectly with our by-hand FEM simulations and the MDSolids verification in Part D.

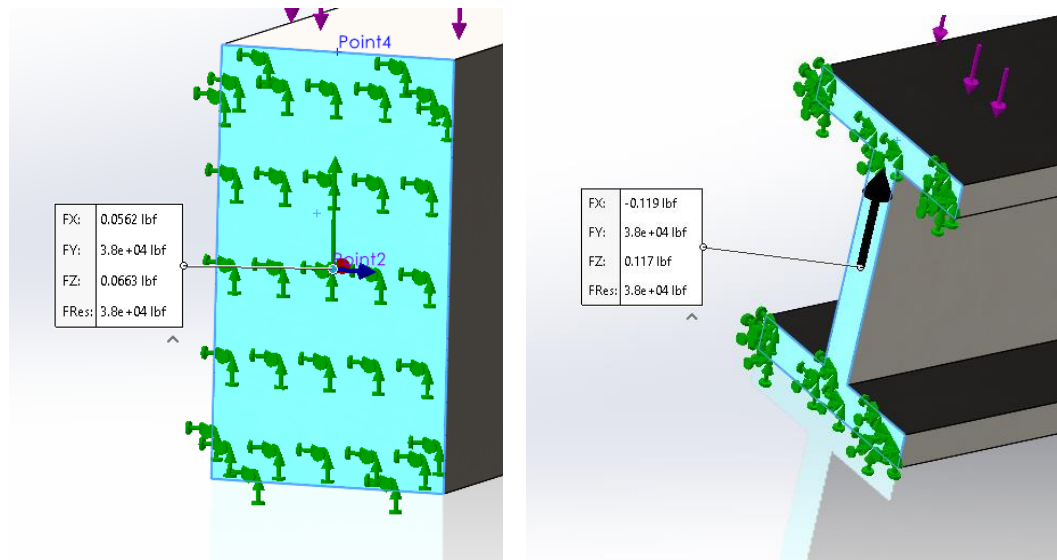


Figure 1.13: SolidWorks Reactions at Fixed Support (lbf)

## Part D: Stress Analysis

The final part of the analysis involves identifying the maximum compressive and tensile stresses in the modelled beams and comparing them to the expected values calculated using classical equations. First, the locations of the minimum and maximum were found using the SolidWorks models by creating stress plots of the normal stresses and labeling the minimum and maximum. The locations of those points are shown in Figure 1.14, with the tensile stresses shaded red and the compressive stresses shaded blue.

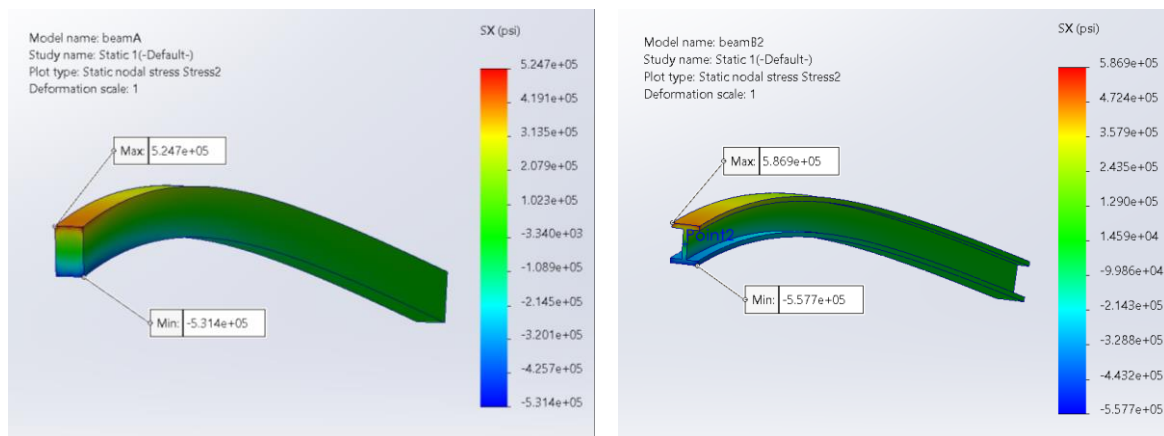


Figure 1.14: Annotated stress plots of beam models.

In both the rectangular beam and I-beam models, the maximum tensile and compressive stresses occur at the fixed plane. Since the normal stresses experienced are directly related to force and moment at the same points in the beam through the axial and bending stress formulas, the peaks in tensile and compressive forces should match the peaks in force and moment. This relationship can be checked by developing the shear and bending moment diagrams for the beam under the same loading scenario and finding where the maximum shear and bending moment occurs. Shear and bending moments are not dependent on the size or cross-sectional area of the beam, so the same diagrams can be used for both the rectangular beam and I-beam models. The shear and bending moment diagrams for the beams were made using MDSolids and are shown in the figure below.

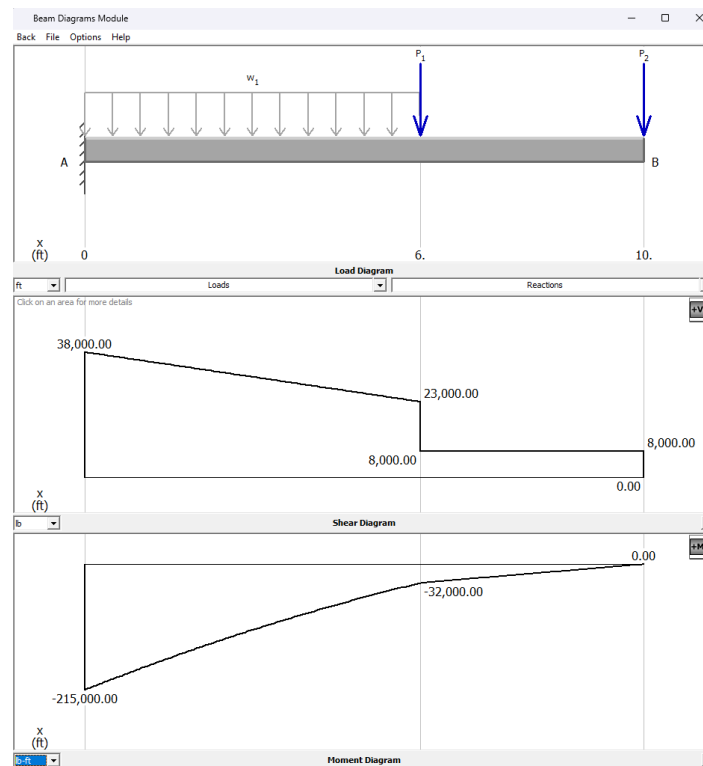


Figure 1.15: MDSolids Shear and Bending Moment Diagrams

At  $x = 0$ , both the shear and bending moment diagrams show large values, corresponding to the fixed support. The shear force is maximum at this location, reflecting the beam's reaction to the applied distributed and point loads. The bending moment also peaks here, reaching a maximum negative value, as expected for a cantilevered beam under this type of loading. On the opposite end of the beam, near  $x = 10$  feet, the diagrams indicate that the moment and shear both approach zero, suggesting this region experiences the least internal stress. This behavior aligns with theoretical expectations, where the maximum stress typically occurs near the fixed support. These findings also match the maximum tensile and compressive forces found in the SolidWorks models, where the maximums normal stress is concentrated near the fixed end of the beam, as shown in the figure below.

The MDSolids model also provides the magnitude of the maximum shear and bending moments to be 38,000 lbf and 215,000 lbf-ft respectively. Since these values occur at the fixed plane, they are equivalent to the wall reactions and can be used to verify the wall reactions used in previous parts of this problem.

Additionally, the 215,000 lbf-ft maximum bending moment can be used to check the magnitudes of the maximum tensile and compressive strengths in the SolidWorks model. Since the bending moment of the beam is the most prominent force acting on the beam normal to the x-axis, it can be used to approximate the normal forces in the beam along the x-axis that are seen in on SolidWorks. Starting with the rectangular beam, the bending stress equation is used below to find the bending stress where it is expected to be maximized: at the top and bottom axes of the fixed plane.

$$\sigma = \frac{Mc}{I} = \frac{(215,000 \text{ lbf} \cdot \text{ft}) \left( \frac{12 \text{ in}}{1 \text{ ft}} \right) (2 \text{ in})}{12 \text{ in}^4} = \pm 430,000 \text{ psi}$$

The equation yields a maximum compressive and tensile stress of 430,000 psi. This value can be compared to the maximums found in the SolidWorks model of the rectangular beam, however, the magnitudes annotated in Figure 1.14 will not be used due to concerns about error. In the SolidWorks model, the fixed plane leads to an uneven distribution of stress concentrations and point loading close to the start of the beam. Normal stress calculations do not account for these changes, and do not expect stresses to vary orthogonal to the original force vectors. To mitigate the effects of these stress concentrations, the probe tool was used to take an average of the stresses experienced at the top and bottom edges of the fixed face of the beam. See Appendix A-4 for detailed screenshots of this methodology. These stresses will be used as the maximum tensile and compressive stresses experienced by the beam. For the rectangular beam, this technique reveals maximum tensile stress and maximum compressive stress of approximately 480,500 and 484,400 psi respectively.

Comparing the hand calculations with the SolidWorks results, there is about an 11% reduction in the maximum stresses between the SolidWorks simulation and the hand calculation. This error is expected because the corners and edges of the rectangular beam create a stress concentration with the fixed support as discussed above. This stress concentration is accounted for in the SolidWorks simulation, but not in the simple bending stress equation below.

With the I-beam, using the same averaging method SolidWorks, the maximum tensile stress is about 509,100 psi and the maximum compressive stress is about 498,300 psi. Using the equation below, the maximum stresses were estimated to be about 455,800 psi, which is a 10% reduction. This error occurs for the same stress concentration reasons described above.

$$\sigma = \frac{Mc}{I} = \frac{(215,000 \text{ lbf} \cdot \text{ft}) \left( \frac{12 \text{ in}}{1 \text{ ft}} \right) (2.12 \text{ in})}{12 \text{ in}^4} = \pm 455,800 \text{ psi}$$



## Problem 2: Structural Truss Analysis

### Introduction

Finite element methods are commonly used to conduct stress analysis for structural trusses. The analysis models the truss and simulates the behavior like deformation and displacement of the structure based on the calculated forces and stresses. These parameters are calculated based on the initial conditions such as applied loads and fixed points. Each node and element can be analyzed to understand how the structure will perform under specific conditions. The results of the analysis can be used to evaluate and design structures.

For this problem, the team modeled and analyzed the structural truss to solve for the 2D truss element. The internal forces of the truss were first analyzed to determine any zero-force members followed by the internal stresses, and finally nodal displacement. For each parameter, the results were calculated by hand initially, then the values were compared to computer ran finite element analysis. Since the calculations in the finite element method require solving a large system of equations, element stiffness matrices are used to simplify the process. The stiffness matrix used for finding nodal displacement is shown in Figure 2.1.  $E$  is the modulus of elasticity of the element material,  $A$  is the cross sectional area, and  $L$  is the length of the element. Inside the global stiffness matrix,  $c$  is  $\cos\theta$  and  $s$  is  $\sin\theta$ . Then in the respective X and Y directions,  $d$  and  $f$  are the nodal displacements and forces respectively.

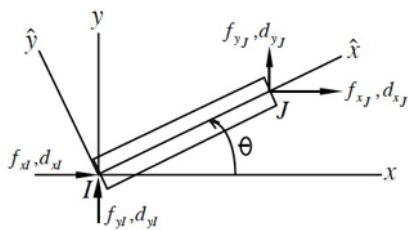
$$\begin{Bmatrix} f_{xI} \\ f_{yI} \\ f_{xJ} \\ f_{yJ} \end{Bmatrix} = \frac{AE}{L} \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \begin{Bmatrix} d_{xI} \\ d_{yI} \\ d_{xJ} \\ d_{yJ} \end{Bmatrix}$$


Figure 2.1: Element stiffness matrix for 2D truss

In this problem, the team assumes the elements are made of aluminum with a Young's Modulus of  $E = 10 \times 10^6$  psi, a cross sectional area  $2.5 \text{ in}^2$ , and a yield strength of 8 ksi. The load and fixed conditions to be used throughout the analysis are shown in Figure 2.2.

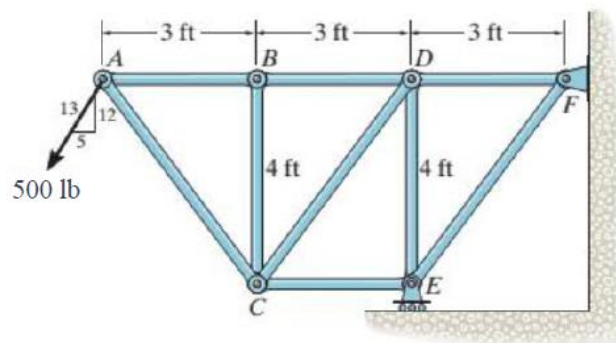
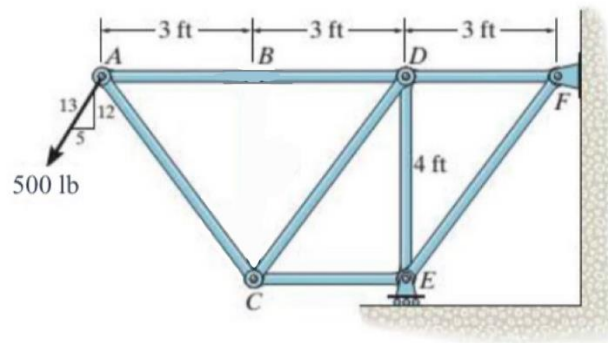


Figure 2.2: Initial Structural Truss for Analysis

## Part A1: Zero-Force Members

Zero-force members are used to increase stability of the truss during construction and to provide added support if the loading is changed, but do not carry any load under specific conditions. Zero-force members can be identified using the Method of Joints. 2 member joints with no collinear members and no external loads or reactions are both members are zero-force members. Three member joints with 2 collinear members and no external loads or reactions, the resulting non-collinear member is a zero-force member. Based on inspection and the Method of Joints, for the initial truss shown in Figure 2.2, the zero-force member identified was member BC. These members do not have any internal forces or stresses based on the examination of joint B. This joint does not have any external loads or reactions, and the non-collinear member was BC.



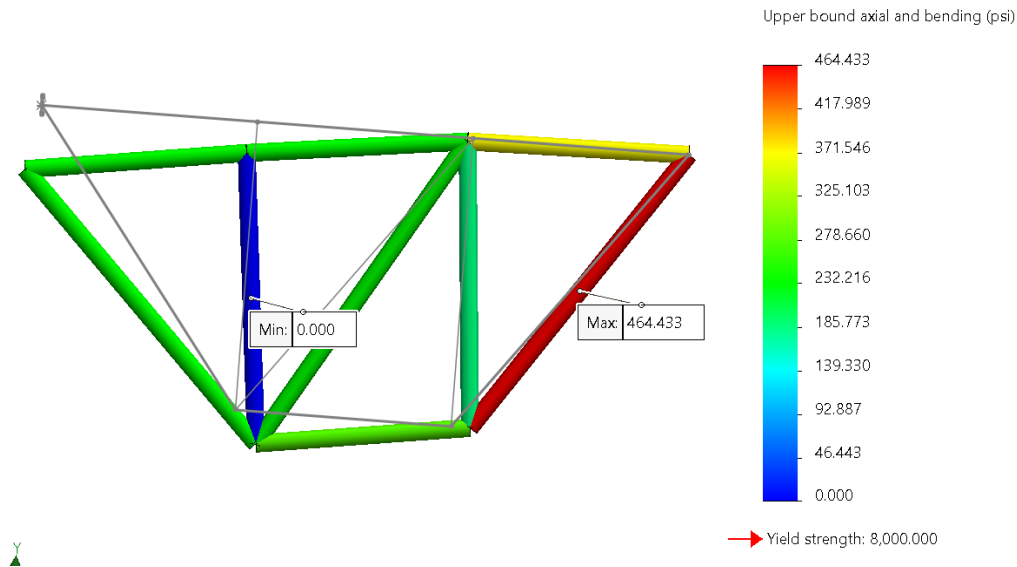
*Figure 2.3: Simplified truss without zero force members*

By eliminating the zero-force member, the truss is simplified for a more efficient truss analysis. The calculations are not as complex with less equations, unknowns, and design optimization can be completed faster. The simplified truss in Figure 2.3, will be used to analyze the internal forces and stresses to determine the structural stability of the truss.

## Part A2: SolidWorks Analysis of Internal Forces

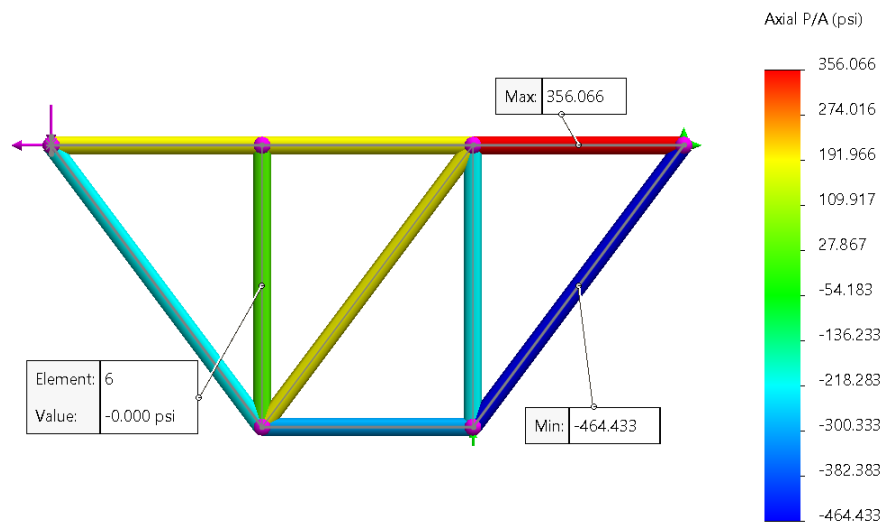
To verify the identified zero-force members, SolidWorks was used to simulate the truss and analyze the corresponding internal stresses. With the given initial conditions and set parameters, SolidWorks Weldments was used to model the truss, and SolidWorks Simulation was used to analyze the loads and internal stresses on the truss and each member. To run the simulation, the truss members were chosen to be AI rectangular tube 4x3 with .1875 walls was chosen because the members would have a calculated cross-sectional area of  $2.48\text{in}^2$  that is close to  $2.5\text{in}^2$  which fits the criteria for this problem. Aluminum alloy was used as the material with  $E = 10 \times 10^6$  psi, and yield strength = 8 ksi. SolidWorks Simulation was used to run a static truss analysis study. Then to complete the initial conditions, node F was fully constrained in all directions, node E was fixed in two directions, able to move along the plane, and the rest of the

nodes were constrained to two degrees of freedom. An external load was applied to node A, and the simulation was run successfully.



*Figure 2.4: Resulting maximum axial and bending stress contour plot*

The full original truss shown in Figure 2.2 was analyzed and used to verify the zero-force member that would be found based on inspection. Figure 2.4 shows the axial and bending stresses found in each member. Member 6 labeled in SolidWorks corresponds to member BC from the original truss and is shown to have no axial or bending stress. Therefore, the member has no internal loads and confirms member BC is a zero-force member.



*Figure 2.5: Resulting Internal Forces Contour Plot for Original Truss*

Member	Force (lbf)
AC	-576.910
AB	538.450
<b>BC</b>	<b>0.000</b>
BD	538.450
CD	576.910
CE	-692.300
DE	-461.530
DF	884.600
EF	-1153.800

Figure 2.6: Beam Forces of Original Truss

Figure 2.6 is a table that shows the values of internal forces in each beam in the original truss. These axial force values were calculated in SolidWorks and the corresponding contour plot is shown in Figure 2.5. As discussed above, member BC experiences no internal force, and is thus a two-force member. All the other members experience an internal force of varying magnitudes, with the smallest non-zero magnitude being the 461.530 lbf experienced by member DE, and the largest force being the 1153.800 lbf experienced by member EF. The table values not only show the force on each member but also indicate whether each member is in tension or compression. Members AB, BD, CD, and DF all have positive force values, indicating they are all in tension. Members AC, CE, DE, and EF all have negative force values, indicating they are all in compression. By visual inspection of the model, these indications make sense, as the force is pulling the members at the top of the model away from the pinned connections while compressing the ones on the bottom, as shown in Figure 2.4.

## Part B: Finite Element Analysis by Hand

The element stiffness matrix for 2D truss shown in Figure 2.2 is shown again the following equation.

$$\begin{bmatrix} f_{xI} \\ f_{yI} \\ f_{xJ} \\ f_{yJ} \end{bmatrix} = \frac{AE}{L} \begin{bmatrix} c^2 & cs & -c^2 & -cs \\ cs & s^2 & -cs & -s^2 \\ -c^2 & -cs & c^2 & cs \\ -cs & -s^2 & cs & s^2 \end{bmatrix} \begin{bmatrix} d_{xI} \\ d_{yI} \\ d_{xJ} \\ d_{yJ} \end{bmatrix}$$

Using this equation seven different matrices were assembled each representing the remaining members of the assembly after the zero force member of BC had been removed.

**Member AD:  $\theta=0^\circ$ ,  $L = 6\text{ft (72in)}$**

$$\begin{bmatrix} f_{xI} \\ f_{yI} \\ f_{xJ} \\ f_{yJ} \end{bmatrix} = \frac{(2.5\text{in}^2)(10e6\text{psi})}{72\text{in}} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} d_{xI} \\ d_{yI} \\ d_{xJ} \\ d_{yJ} \end{bmatrix}$$

**Member AC:  $\theta=53.13^\circ$ , L = 5ft (60in)**

$$\begin{bmatrix} f_{xI} \\ f_{yI} \\ f_{xJ} \\ f_{yJ} \end{bmatrix} = \frac{(2.5in^2)(10e6psi)}{60in} \begin{bmatrix} 0.36 & 0.48 & 0.36 & -0.48 \\ 0.48 & 0.64 & -0.48 & 0.64 \\ -0.36 & -0.48 & 0.36 & 0.48 \\ -0.48 & -0.64 & 0.48 & 0.64 \end{bmatrix} \begin{bmatrix} d_{xI} \\ d_{yI} \\ d_{xJ} \\ d_{yJ} \end{bmatrix}$$

**Member CD:  $\theta=53.15^\circ$ , L = 5ft (72in)**

$$\begin{bmatrix} f_{xI} \\ f_{yI} \\ f_{xJ} \\ f_{yJ} \end{bmatrix} = \frac{(2.5in^2)(10e6psi)}{60in} \begin{bmatrix} 0.36 & 0.48 & -0.36 & -0.48 \\ 0.48 & 0.64 & -0.48 & 0.64 \\ -0.36 & -0.48 & 0.36 & 0.48 \\ -0.48 & -0.64 & 0.48 & 0.64 \end{bmatrix} \begin{bmatrix} d_{xI} \\ d_{yI} \\ d_{xJ} \\ d_{yJ} \end{bmatrix}$$

**Member CE:  $\theta=0^\circ$ , L = 3ft (36in)**

$$\begin{bmatrix} f_{xI} \\ f_{yI} \\ f_{xJ} \\ f_{yJ} \end{bmatrix} = \frac{(2.5in^2)(10e6psi)}{36in} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} d_{xI} \\ d_{yI} \\ d_{xJ} \\ d_{yJ} \end{bmatrix}$$

**Member DE:  $\theta=90^\circ$ , L = 4ft (48in)**

$$\begin{bmatrix} f_{xI} \\ f_{yI} \\ f_{xJ} \\ f_{yJ} \end{bmatrix} = \frac{(2.5in^2)(10e6psi)}{48in} \begin{bmatrix} 1 & 0 & -1 & 0 \\ 0 & 0 & 0 & 0 \\ -1 & 0 & 1 & 0 \\ 0 & 0 & 0 & 0 \end{bmatrix} \begin{bmatrix} d_{xI} \\ d_{yI} \\ d_{xJ} \\ d_{yJ} \end{bmatrix}$$

**Member DF:  $\theta=0^\circ$ , L = 3ft (36in)**

$$\begin{bmatrix} f_{xI} \\ f_{yI} \\ f_{xJ} \\ f_{yJ} \end{bmatrix} = \frac{(2.5in^2)(10e6psi)}{36in} \begin{bmatrix} 0 & 0 & 0 & 0 \\ 0 & 1 & 0 & -1 \\ 0 & 0 & 0 & 0 \\ 0 & -1 & 0 & 1 \end{bmatrix} \begin{bmatrix} d_{xI} \\ d_{yI} \\ d_{xJ} \\ d_{yJ} \end{bmatrix}$$

**Member EF:  $\theta=36.87^\circ$ , L = 5ft (60in)**

$$\begin{bmatrix} f_{xI} \\ f_{yI} \\ f_{xJ} \\ f_{yJ} \end{bmatrix} = \frac{(2.5in^2)(10e6psi)}{60in} \begin{bmatrix} 0.64 & 0.48 & -0.64 & 0.48 \\ 0.48 & 0.36 & -0.48 & -0.36 \\ -0.64 & -0.48 & 0.64 & 0.48 \\ -0.48 & -0.36 & 0.48 & 0.36 \end{bmatrix} \begin{bmatrix} d_{xI} \\ d_{yI} \\ d_{xJ} \\ d_{yJ} \end{bmatrix}$$

When assembling the Global Stiffness Matrix some assumptions need to be made namely, nearly all external forces are assumed to be equal to zero except for  $F_{Ax} = -192.31$  lbf, and  $F_{Ay} = -461.54$  lbf. Additionally,  $F_{Ax}$ ,  $F_{Ay}$ , and  $F_{Az}$  are all unknown values.

$$\begin{bmatrix} f_{ax} \\ f_{ay} \\ f_{cx} \\ f_{cy} \\ f_{dx} \\ f_{dy} \\ f_{ex} \\ f_{ey} \\ f_{fx} \\ f_{fy} \end{bmatrix} = \begin{bmatrix} k_1 + 0.36k_2 & -0.48k_2 & -0.36k_2 & 0.48k_2 & -k_1 & 0 & 0 & 0 & 0 & 0 \\ -0.48k_2 & 0.64k_2 & 0.48k_2 & -0.64k_2 & 0 & 0 & 0 & 0 & 0 & 0 \\ -0.36k_2 & 0.48k_2 & 0.36k_2 + 0.36k_3 + k_4 & -0.48k_2 + 0.48k_3 & -0.36k_3 & -0.48k_3 & -k_4 & 0 & 0 & 0 \\ 0.48k_2 & -0.64k_2 & -0.48k_2 + 0.48k_3 & 0.64k_2 + 0.64k_3 & -0.48k_3 & -0.64k_3 & 0 & 0 & 0 & 0 \\ -k_1 & 0 & -0.36k_3 & -0.48 & k_1 + 0.36k_3 + k_6 & 0.48k_3 & 0 & 0 & -k_6 & 0 \\ 0 & 0 & -0.48k_3 & -0.64k_3 & 0.48k_3 & 0.64k_3 + k_7 & 0 & -k_7 & 0 & 0 \\ 0 & 0 & -k_4 & 0 & 0 & 0 & k_4 + 0.46k_5 & 0.48k_5 & -0.36k_5 & -0.48k_5 \\ 0 & 0 & 0 & 0 & 0 & -k_7 & 0.48k_5 & 0.64k_5 + k_7 & -0.48k_5 & -0.64k_5 \\ 0 & 0 & 0 & 0 & -k_6 & 0 & -0.36k_5 & -0.48k_5 & 0.36k_5 + k_6 & 0.48k_5 \\ 0 & 0 & 0 & 0 & 0 & 0 & -0.48k_5 & -0.64k_5 & 0.48k_5 & 0.64k_5 + k_7 \end{bmatrix} \begin{bmatrix} d_{ax} \\ d_{ay} \\ d_{cx} \\ d_{cy} \\ d_{dx} \\ d_{dy} \\ d_{ex} \\ d_{ey} \\ d_{fx} \\ d_{fy} \end{bmatrix}$$

Each of the matrices above were solved utilizing a MATLAB script, providing the following nodal displacements in inches. Note: guidance from chat gpt was used in the script to solve the matrices.

**Nodal Displacements (inches):**

Node 1:  $u_x = 0.001848$  in,  $u_y = 0.007806$  in

Node 2:  $u_x = 0.001464$  in,  $u_y = 0.004204$  in

Node 3:  $u_x = -0.002546$  in,  $u_y = 0.005708$  in

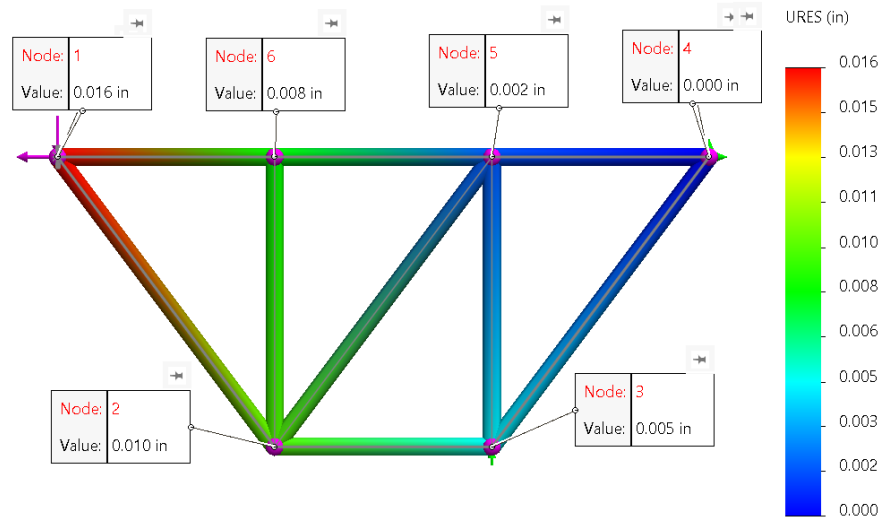
Node 4:  $u_x = 0.001080$  in,  $u_y = 0.000369$  in

Node 5:  $u_x = -0.001923$  in,  $u_y = 0.000000$  in

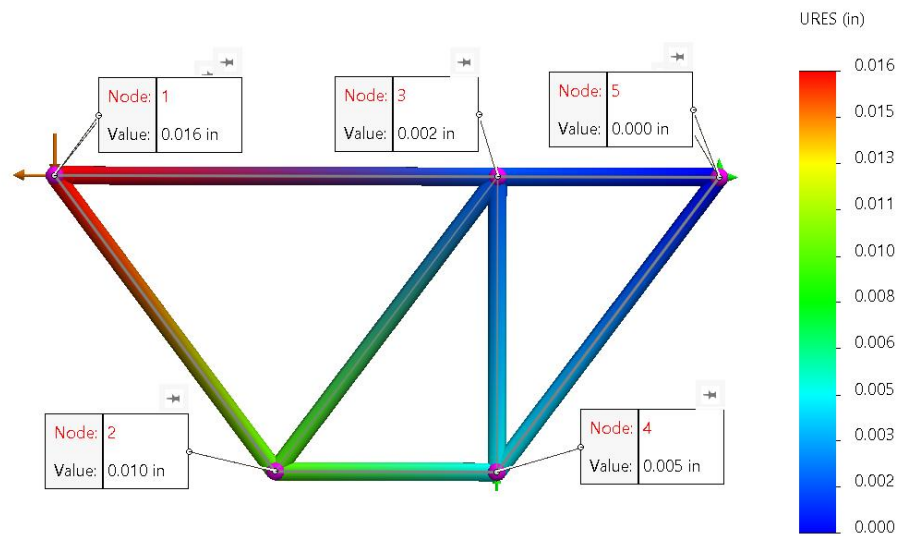
*Figure 2.7: Nodal Displacements Values Produced via MATLAB*

## Part C: SolidWorks Analysis of Nodal Displacement

SolidWorks Simulation was used again to confirm the previous analysis of the displacement for all the nodes. The displacement results were generated from the previously modeled and simulated original truss. The original truss nodal displacements are shown in *Figure 2.8*, and the simplified truss, ignoring the zero-force member, nodal displacements are shown in *Figure 2.9*. The contour plots were generated using the SolidWorks static study simulation with the results specified in resultant displacement.



*Figure 2.8 :Contour Plot of Resultant Nodal Displacement of Original Truss*



*Figure 2.9: Contour Plot of Resultant Nodal Displacement of Simplified Truss*

The results in the contour plots in Figure 2.8 and Figure 2.9 make logical sense since the max displacement is at the node A where the load is applied, and the minimum displacement is at Node F, where it is fully fixed. The applied downward, left load deforms the truss the applied load on the left most point deforms the truss in the downwards direction relative to the fully

fixed node F which has no displacement. Based on the resultant node values shown in the figures above, and the table below, the corresponding displacement measured in inches are the same. Since zero-force members do not contribute deformation or to the overall stiffness of the structure, removing the member will not affect the nodal displacement.

Nodal Displacement				
	Both trusses		Original	Simplified
Node	X (in)	Y (in)	Resultant (in)	
A (1)	-0.0028	-0.0159	0.0160	0.0160
B (6)	-0.0021	-0.0078	0.0000	
C (2)	0.0057	-0.0078	0.0100	0.0100
D (3)	-0.0013	-0.0009	0.0020	0.0020
E (4)	0.0046	0.0000	0.0050	0.0050
F (5)	0.0000	0.0000	0.0000	0.0000

Figure 2.10: Nodal Displacement Comparison

The comparison between the X and Y nodal displacements of the original truss and the simplified truss shown in Figure 2.10 verifies that member BC is a zero force member. The displacement values were simplified and shown only once because all values were equal. The only difference is for the simplified truss, node B is neglected. These results make sense because The results from the SolidWorks Simulation verified the hand calculations made in part B, since the value of the nodal displacements were similar.

## Part D: Internal Forces

To determine the internal forces of different members, first a free body diagram of the entire truss structure must be examined, shown below in Figure 2.11.

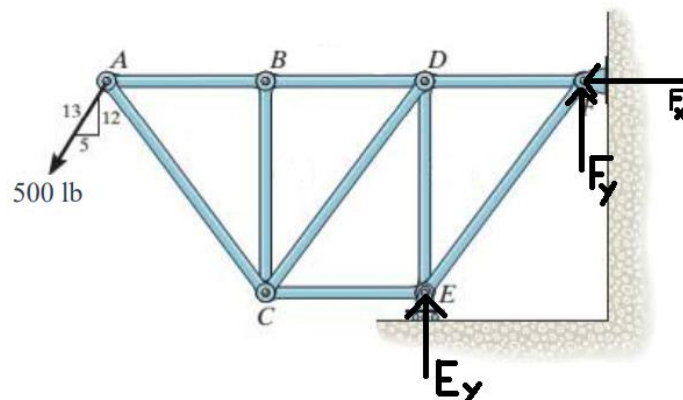


Figure 2.11: Free Body Diagram



From this free body diagram, the reaction force in the x-direction at F can be determined by summing the forces in the x direction and setting it equal to zero.

$$\sum F_x = 0 = -500\left(\frac{5}{13}\right) - F_x$$

$$F_x = -500\left(\frac{5}{13}\right)$$

$$F_x = 192.308 \text{ lbf to the right}$$

The moment about point F can then be set equal to zero to find the reaction force at point E. Since the force on the truss is at an angle, that must be considered when calculating the moment.

$$\sum M_F = 0 = -\left(500\left(\frac{12}{13}\right) \cdot 9\right) + (E_y \cdot 3)$$

$$E_y \cdot 3 = 4153.846$$

$$E_y = 1384.615 \text{ lbf upwards}$$

Using this value, the value of the reaction force in the y direction at point F can be determined by summing the forces in the y direction.

$$\sum F_y = 0 = -\left(500\left(\frac{12}{13}\right)\right) + 1384.615 + F_y$$

$$F_y = 461.538 - 1384.615$$

$$F_y = 923.077 \text{ lbf downwards}$$

Now that all external forces have been defined, individual joints can be examined. This report will focus on joints A and F.

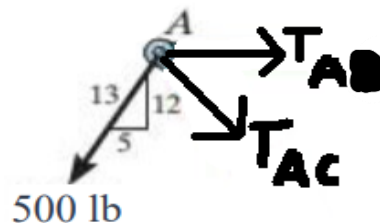


Figure 2.12: Joint A FBD

Since the force in member AD does not have a vertical component, the vertical components can be summed to find the force in member AC.

$$\sum F_y = 0 = -\left(500 \cdot \frac{12}{13}\right) + \left(T_{AC} \cdot \frac{4}{5}\right)$$

$$T_{AC} = -576.9 \text{ lbf}$$

It was found that the internal force in member AC is 576.9 lbf. The negative indicates that the member is in compression, which again lines up with the SolidWorks model. With the force in member AC known, the sum of x components can be set to zero to find the force in member AD.

$$\sum F_x = 0 = -\left(500 \cdot \frac{5}{13}\right) + \left(-576.9 \cdot \frac{3}{5}\right) + T_{AB}$$

$$T_{AB} = 538.4 \text{ lbf}$$

The calculated value of 538.4 lbf again lines up with the SolidWorks model. The positive force indicates that the original assumption of the member being in tension was correct. Due to the fact that member BC is a zero-force member, members AB and BD are essentially the same member, meaning the internal force within both members is the same. Checking against the SolidWorks simulation this checks out. Therefore:

$$T_{BD} = 538.4 \text{ lbf}$$

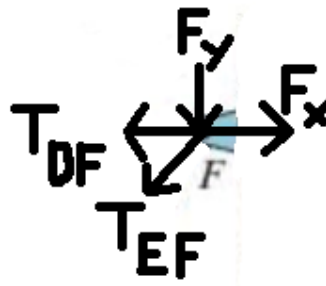


Figure 2.13: Joint F FBD

Like the calculations done on joint A, one member can be isolated. In this case, that is member EF because member BF does not have a y-component.

$$\sum F_y = 0 = -923.077 + \left(T_{EF} \cdot \frac{4}{5}\right)$$

$$T_{EF} = -1153.8 \text{ lbf}$$

The negative value again indicates the assumption of tension was incorrect, meaning it is in tension. Compared to the results of the SolidWorks study, this lines up as does the magnitude. Knowing the force in member EF allows member DF to be solved by setting the x-components to zero.

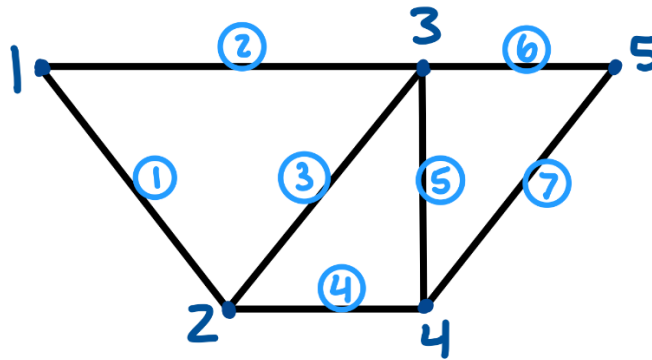
$$\sum F_x = 0 = -T_{DF} \cdot -\left(-1153.8 \cdot \frac{3}{5}\right) + 192.308$$

$$T_{DF} = 884.6 \text{ lbf}$$

Checking with the SolidWorks results, this again lines up, both in magnitude, and the fact that the member is in tension.

## Part E: MATLAB 2D Finite Element Analysis

MATLAB was utilized to perform 2D Finite Element Analysis on the simplified truss. The provided MATLAB code was used to simulate and determine the global stiffness matrix, nodal displacements, and normal stress in each member. The data input into MATLAB included modeling the truss based on node and element locations and initial conditions including applied forces and constraints.



*Figure 2.14: Corresponding labeled nodes and elements used in MATLAB*

To accurately model the problem in MATLAB, the simplified truss was labeled in Figure 2.14, with five nodes and seven elements. The nodes were labeled starting at the top left node, and ending with the top right node, following the previous letter labels A-F, and the elements were also labeled from left to right. Node 1 was identified as the origin (0,0), and with the dimensions in Figure 2.3, the corresponding coordinates for each node were identified in inches. Then Figure 2.14 was used again to identify the connectivity of each element based on the nodes. For example, node 2 has coordinates of (36, -48) and node 3 is (72,0), and member 3 is connected to nodes 2 and 3. The Young's modulus and cross-sectional area were specified to  $10 \times 10^6$  psi and  $2.5 \text{ in}^2$  respectively. Then to apply the nodal constraints, node 5 was fully constrained in both X and Y directions, node 4 was fixed in the Y direction, and the rest of the nodes have two degrees of freedom. Finally, to finish the data input, the applied force vector was set to  $-(2500/13)$  and  $-(6000/13)$  on node 1, in the X and Y directions respectively, and all other nodes were set at 0. It was important to account for the correct units across all parameters and to ensure the correct placement of the values in each set of data to correspond to the correct node.

With the truss accurately represented, the results from the code were used to verify the previously calculated and simulated results.

Nodal Displacement Comparison								
Node	X			Y			Resultant	
	Hand	SW	ML	Hand	SW	ML	SW	ML
A	-0.0028	-0.0028	-0.0028	-0.0158	-0.0159	-0.0158	0.0160	0.0160
B	N/A	N/A	N/A	N/A	N/A	N/A	N/A	N/A
C	0.0056	0.0057	0.0056	-0.0078	-0.0078	-0.0078	0.0100	0.0096
D	-0.0013	-0.0013	-0.0013	-0.0009	-0.0009	-0.0009	0.0020	0.0016
E	0.0046	0.0046	0.0046	0.0000	0.0000	0.0000	0.0050	0.0046
F	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000	0.0000

Figure 2.15: Nodal Displacement Comparison between SolidWorks and MATLAB (psi)

The table above shows the comparison between the SolidWorks and MATLAB nodal displacements. Overall, the MATLAB results closely align with the SolidWorks values, indicating consistency and reliability in the finite element analysis. Small variations are present but fall within an acceptable range, likely due to small differences in rounding or precision differences between software. Notably, the fixed node F shows zero displacement, which is consistent with the applied boundary conditions. The displacements at other nodes follow the expected results based on the applied load and the geometry of the truss, with the largest displacement occurring at node A, where the load was applied.

Member	MATLAB	SolidWorks
AC	-230.769	-232.216
AD	215.385	216.736
CD	230.769	-232.216
CE	-276.923	-278.66
DE	-184.615	-185.773
DF	353.846	356.066
EF	-461.539	-464.433

Figure 2.16: Column Normal Stress for Simplified Truss from MATLAB (psi)

The table above shows the stress values obtained for each member in the truss in MATLAB and SolidWorks. The stress results of the MATLAB and SolidWorks simulation are very similar. Typically, the results are within 1-2% of each other. This validates the accuracy of both approaches. The source of error is due to the slight difference in the cross-sectional area of the selected beam in SolidWorks. When calculating stress, the cross-sectional area of the member is taken into consideration. There was a difference of  $.02in^2$  of the calculated cross-sectional area from the beam members used in SolidWorks compared to the value used in MATLAB. Despite the error, the SolidWorks values were still verified by MATLAB.

Unconstrained Stiffness Matrix is:

1.0e+06 \*

0.4972	-0.2000	-0.1500	0.2000	-0.3472	0	0	0	0	0
-0.2000	0.2667	0.2000	-0.2667	0	0	0	0	0	0
-0.1500	0.2000	0.9944	0	-0.1500	-0.2000	-0.6944	0	0	0
0.2000	-0.2667	0	0.5333	-0.2000	-0.2667	0	0	0	0
-0.3472	0	-0.1500	-0.2000	1.1917	0.2000	0	0	-0.6944	0
0	0	-0.2000	-0.2667	0.2000	0.7875	0	-0.5208	0	0
0	0	-0.6944	0	0	0	0.8444	0.2000	-0.1500	-0.2000
0	0	0	0	0	-0.5208	0.2000	0.7875	-0.2000	-0.2667
0	0	0	0	-0.6944	0	-0.1500	-0.2000	0.8444	0.2000
0	0	0	0	0	0	-0.2000	-0.2667	0.2000	0.2667

*Figure 2.17: MATLAB generated Global Stiffness Matrix*

Figure 2.17 shows the stiffness matrix generated during this step. This stiffness matrix exactly matches the one from Part B. This is to be expected because both calculations are finite element analyses and use the same parameters.

## References

Nisbett, J. Keith, and Richard G. Budynas. *Shigley's Mechanical Engineering Design*. McGraw-Hill, 2024. Accessed 22 Apr. 2025.

*YouTube*, [youtu.be/v0rbpDEbM5g?si=Z0Y8IXwryzyKs2aZ](https://youtu.be/v0rbpDEbM5g?si=Z0Y8IXwryzyKs2aZ). Accessed 22 Apr. 2025.

# Appendix

## A-1: By Hand FEM - Matlab Code

### 4-Node Analysis:

```

E = 29e6; %psi
I = 12; %in4
L1 = 3*12; %in
L2 = 3*12; %in
L3 = 4*12; %in

% Elemental Stiffness Matricies
el1 = (E*I/L1^3)* [ 12 6*L1 -12 6*L1 ; 6*L1 4*L1^2 -6*L1 2*L1^2;...
-12 -6*L1 12 -6*L1 ; 6*L1 2*L1^2 -6*L1 4*L1^2]
el2 = (E*I/L2^3)* [ 12 6*L2 -12 6*L2 ; 6*L2 4*L2^2 -6*L2 2*L2^2;...
-12 -6*L2 12 -6*L2 ; 6*L2 2*L2^2 -6*L2 4*L2^2]
el3 = (E*I/L3^3)* [ 12 6*L3 -12 6*L3 ; 6*L3 4*L3^2 -6*L3 2*L3^2;...
-12 -6*L3 12 -6*L3 ; 6*L3 2*L3^2 -6*L3 4*L3^2]

% Global Stiffness Matrix
gl = zeros(8,8);
gl(1:4,1:4) = el1;
gl(3:6,3:6)= gl(3:6,3:6)+el2;
gl(5:8,5:8)= gl(5:8,5:8)+el3;
gl

% Force Vector, Apply IC
R_wall = 38000
w = 2500 % Distributed Load in lbs/ft

V2 = -w*(L1/12/2)-w*(L2/12/2)
V3 = -w*(L2/12/2)- 15000
V4 = -8000

M2 = 0
M3 = 0
M4 = 0

forceVec_ic = [V2 M2 V3 M3 V4 M4]' % v1 = 0, th1 = 0

gl_ic = gl(3:8, 3:8) % v1 = 0, th1 = 0
inv_gl_ic = inv(gl_ic);

dispvec_ic = inv_gl_ic * forceVec_ic
dispvec = [0; 0; dispvec_ic]
forcevec = gl * dispvec

%Validate
SumOfFor = forcevec(1) + forcevec(3) + forcevec(5) + forcevec(7)
SumOfMom = forcevec(2) + forcevec(6) + forcevec(3)*(L1) + forcevec(5)*(L1 + L2) +
forcevec(7)*(L1+L2+L3)

```

## 5-Node Analysis:

```

E = 29e6; %psi
I = 12; %in4
L1 = 1.5*12; %in
L2 = 3*12; %in
L3 = 1.5*12; %in
L4 = 4*12; %in

% Elemental Stiffness Matrices
e11 = (E*I/L1^3)* [ 12 6*L1 -12 6*L1 ; 6*L1 4*L1^2 -6*L1 2*L1^2;...
-12 -6*L1 12 -6*L1 ; 6*L1 2*L1^2 -6*L1 4*L1^2]
e12 = (E*I/L2^3)* [ 12 6*L2 -12 6*L2 ; 6*L2 4*L2^2 -6*L2 2*L2^2;...
-12 -6*L2 12 -6*L2 ; 6*L2 2*L2^2 -6*L2 4*L2^2]
e13 = (E*I/L3^3)* [ 12 6*L3 -12 6*L3 ; 6*L3 4*L3^2 -6*L3 2*L3^2;...
-12 -6*L3 12 -6*L3 ; 6*L3 2*L3^2 -6*L3 4*L3^2]
e14 = (E*I/L4^3)* [ 12 6*L4 -12 6*L4 ; 6*L4 4*L4^2 -6*L4 2*L4^2;...
-12 -6*L4 12 -6*L4 ; 6*L4 2*L4^2 -6*L4 4*L4^2]

% Global Stiffness Matrix
gl = zeros(10,10);
gl(1:4,1:4) = e11;
gl(3:6,3:6) = gl(3:6,3:6)+e12;
gl(5:8,5:8) = gl(5:8,5:8)+e13;
gl(7:10,7:10) = gl(7:10,7:10)+e14;
gl

R_wall = 38000
w = 2500 % Distributed Load in lbs/ft

V2 = -w*(L1/12/2)-w*(L2/12/2)
V3 = -w*(L2/12/2)-w*(L3/12/2)
V4 = -w*(L3/12/2)- 15000
V5 = -8000

M2 = 0;
M3 = 0;
M4 = 0;
M5 = 0;

forceVec_ic = [V2 M2 V3 M3 V4 M4 V5 M5]' % v1 = 0, th1 = 0

gl_ic = gl(3:10, 3:10) % v1 = 0, th1 = 0
inv_gl_ic = inv(gl_ic);

dispvec_ic = inv_gl_ic * forceVec_ic
dispvec = [0; 0; dispvec_ic]
forcevec = gl * dispvec

%Validate
SumOfFor = forcevec(1) + forcevec(3) + forcevec(5) + forcevec(7) + forcevec(9)
SumOfMom = forcevec(2) + forcevec(3)*(L1) + forcevec(5)*(L1 + L2) +
forcevec(7)*(L1+L2+L3) + forcevec(9)*(L1+L2+L3+L4)

```



## A-2: Principle of Superposition - Matlab Code

```
% PART B - Three Elements
beamA_forces = [ 7500, 36; 18750, 72; 8000, 120];
for i = 1:3
    point = beamA_forces(i,2);
    y = zeros(1,3);

    for j = 1:3
        force = beamA_forces(j,1);
        le = beamA_forces(j,2);

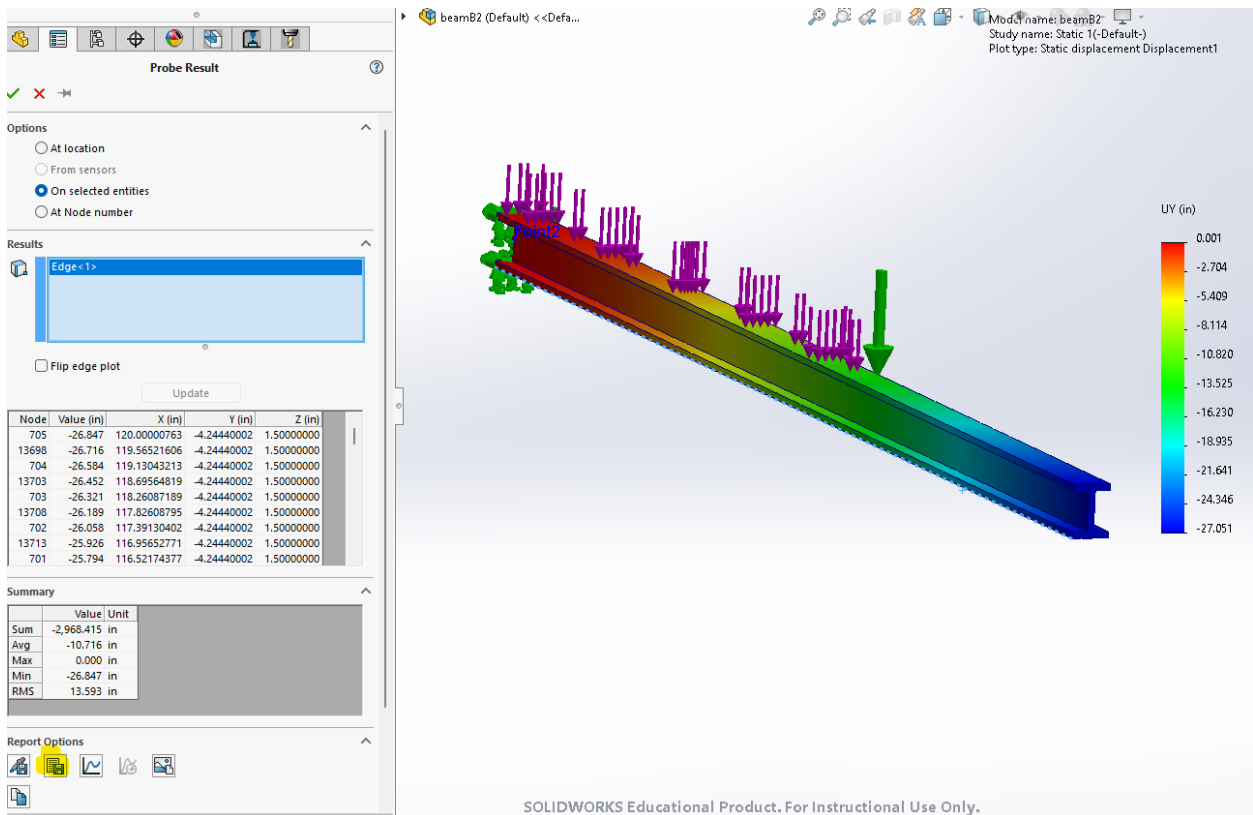
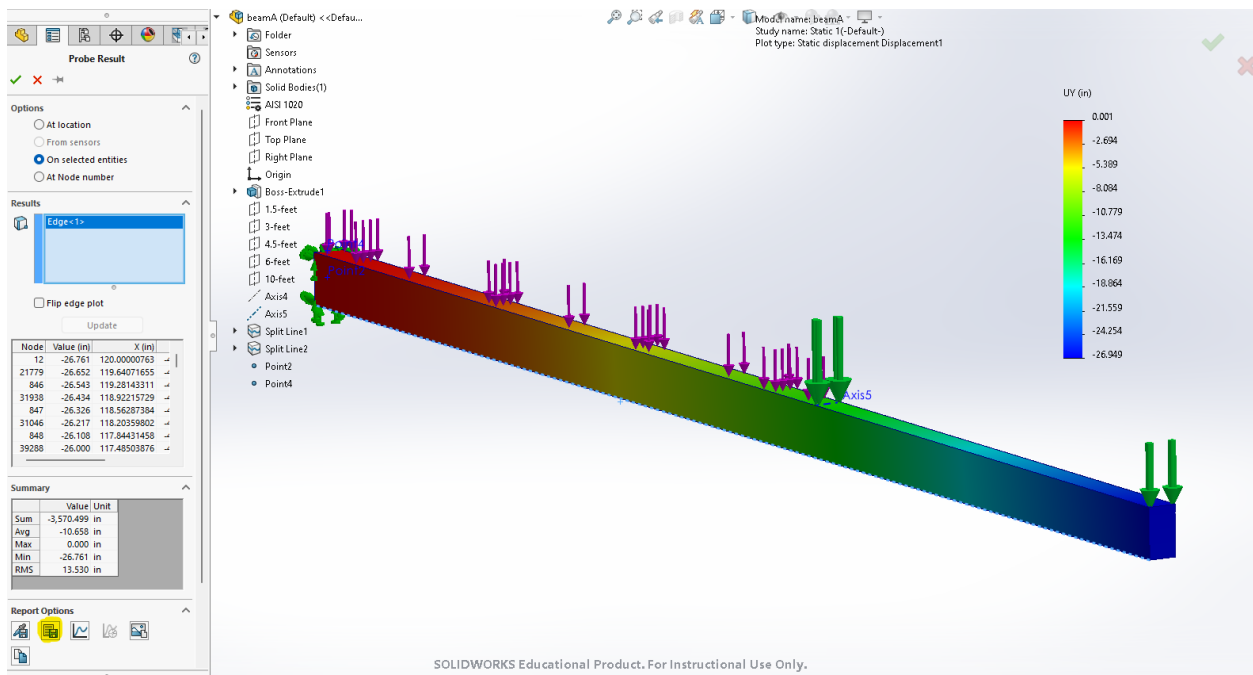
        if (point == le)
            y(1,j) = ((force * (point^3))/(3 * elasticity * inertia)); %00
        end
        if (point < le)
            y(1,j) = -((force * (point^2))/(6 * elasticity * inertia)) * (point - (3
* le)); %A0
        end
        if (point > le)
            y(1,j) = -((force * (le^2))/(6 * elasticity * inertia)) * (le - (3 *
point)); %0B
        end
        PartB_displacementA(i,1) = sum(y);
    end
end
PartB_displacementA

% PART B - Four Elements
beamB_forces = [ 5625, 18; 5625, 54; 16875, 72; 8000, 120];
for i = 1:4
    point = beamB_forces(i,2);
    y = zeros(1,4);

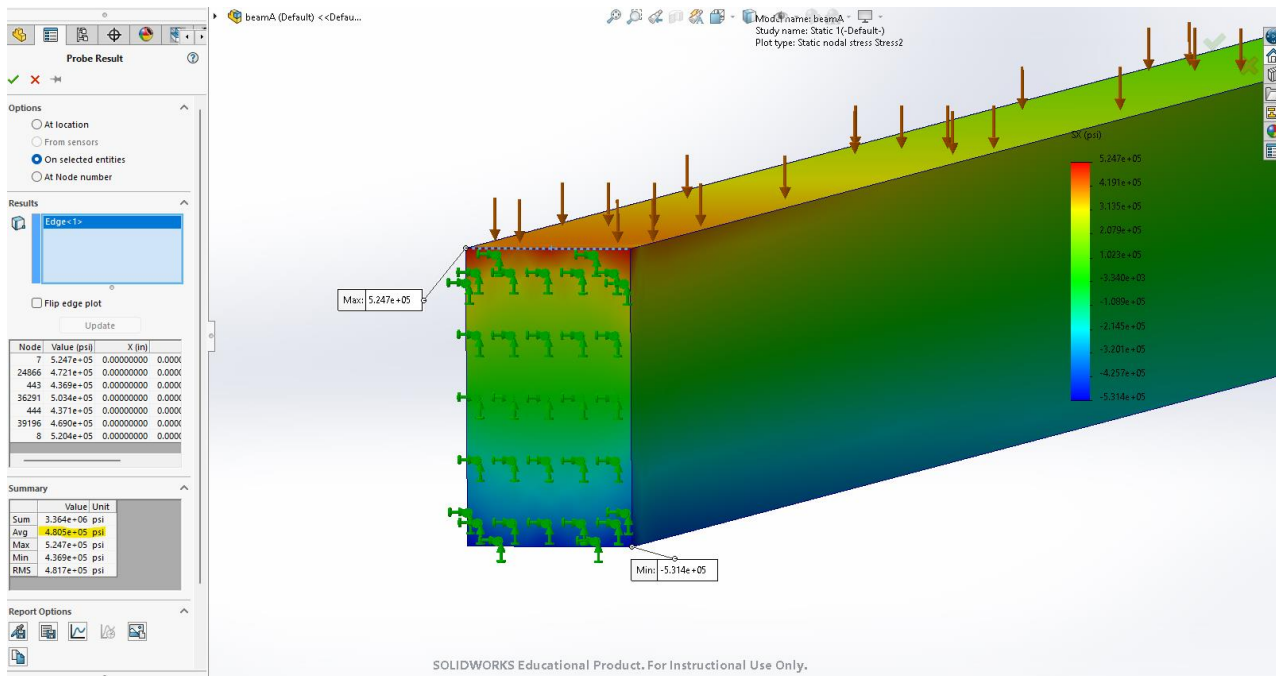
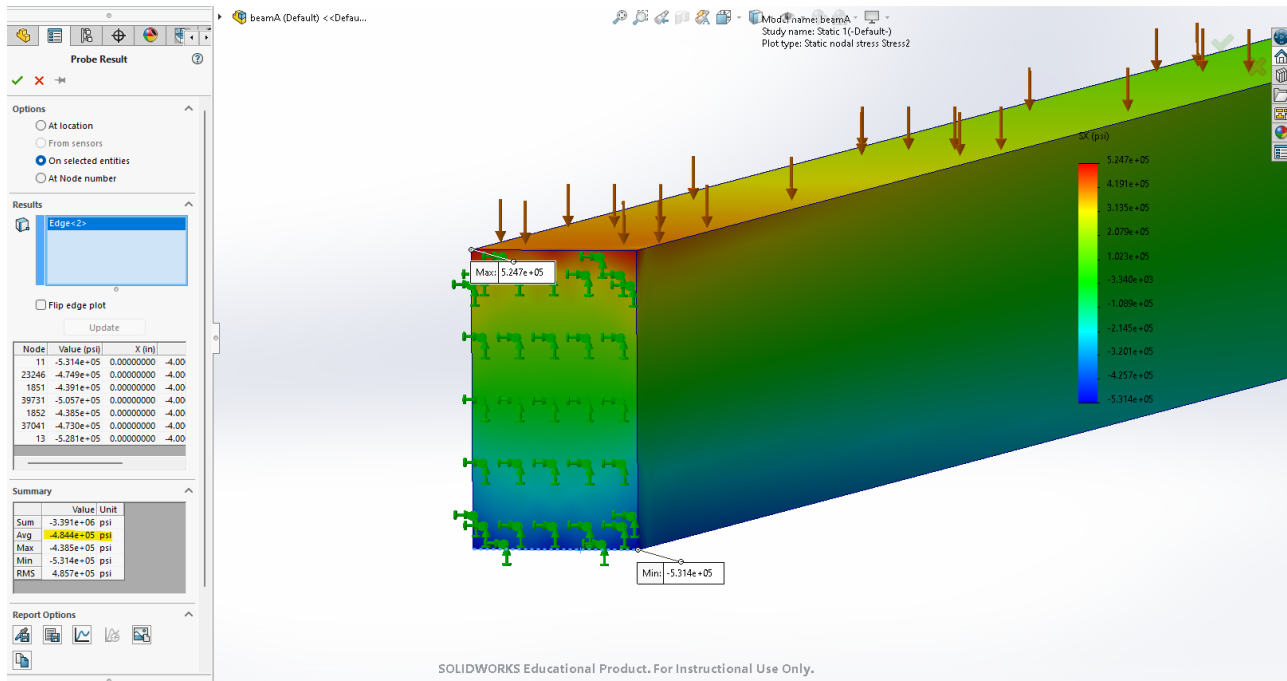
    for j = 1:4
        force = beamB_forces(j,1);
        le = beamB_forces(j,2);

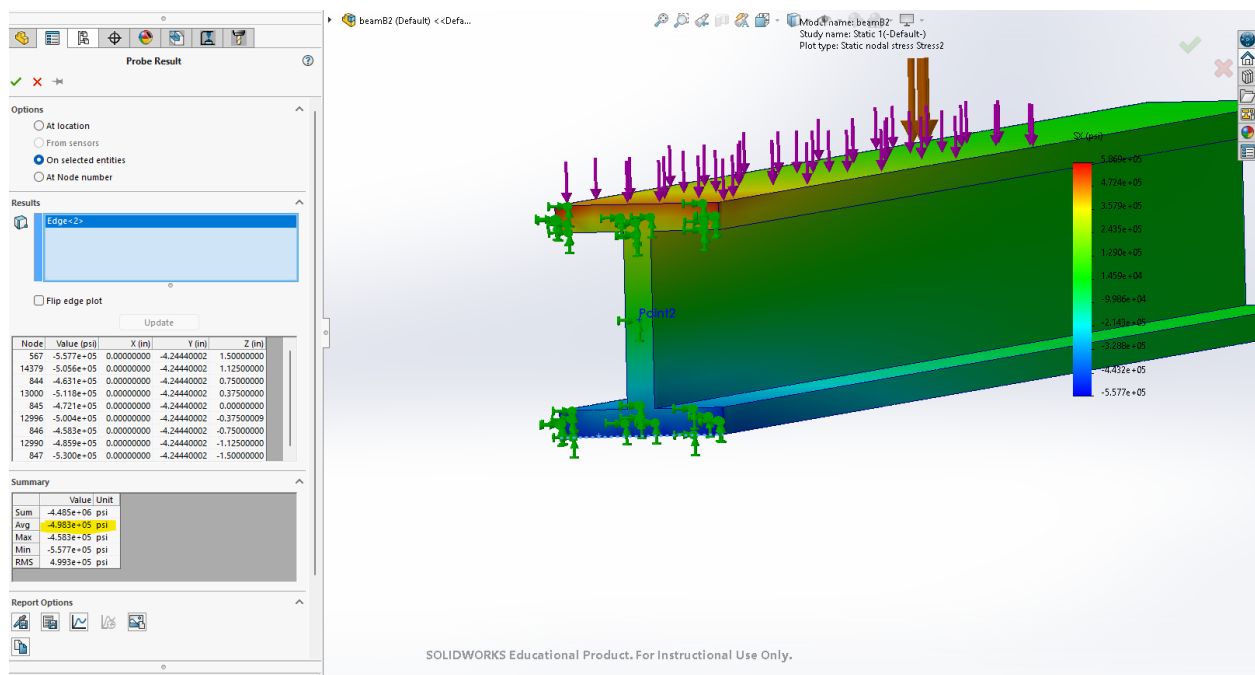
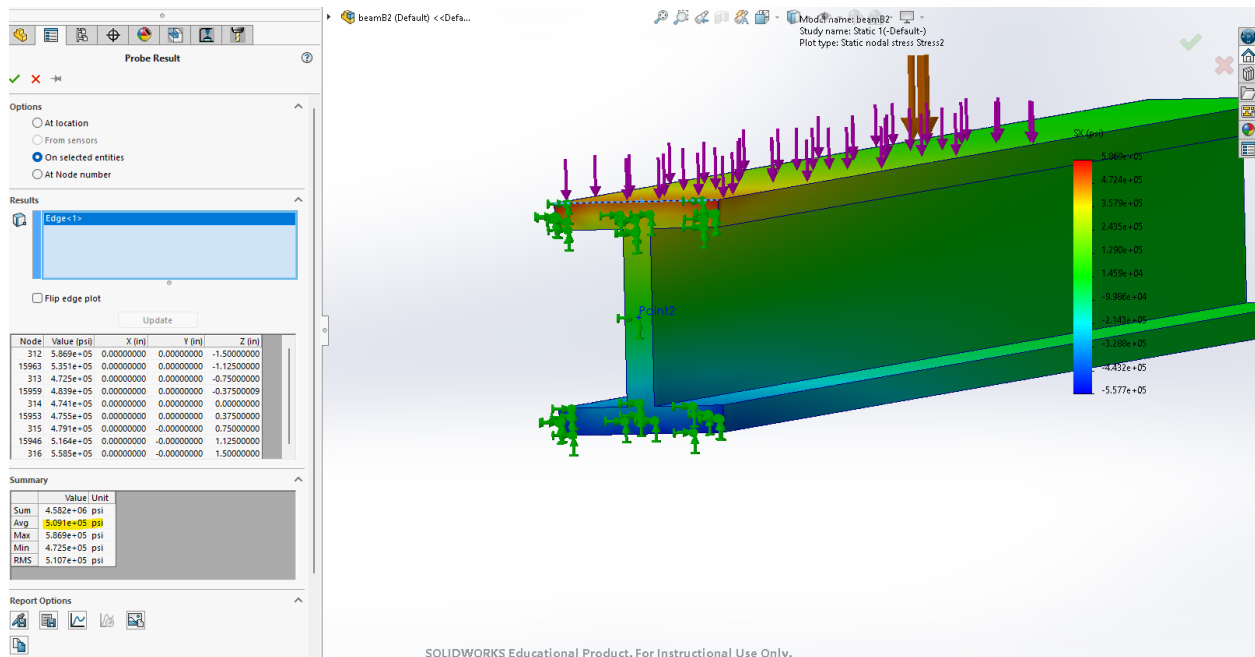
        if (point == le)
            y(1,j) = ((force * (point^3))/(3 * elasticity * inertia)); %00
        end
        if (point < le)
            y(1,j) = -((force * (point^2))/(6 * elasticity * inertia)) * (point - (3
* le)); %A0
        end
        if (point > le)
            y(1,j) = -((force * (le^2))/(6 * elasticity * inertia)) * (le - (3 *
point)); %0B
        end
        PartB_displacementB(i,1) = sum(y);
    end
end
PartB_displacementB
```

## A-3: Deflection Analysis Methodology - Additional Screenshots



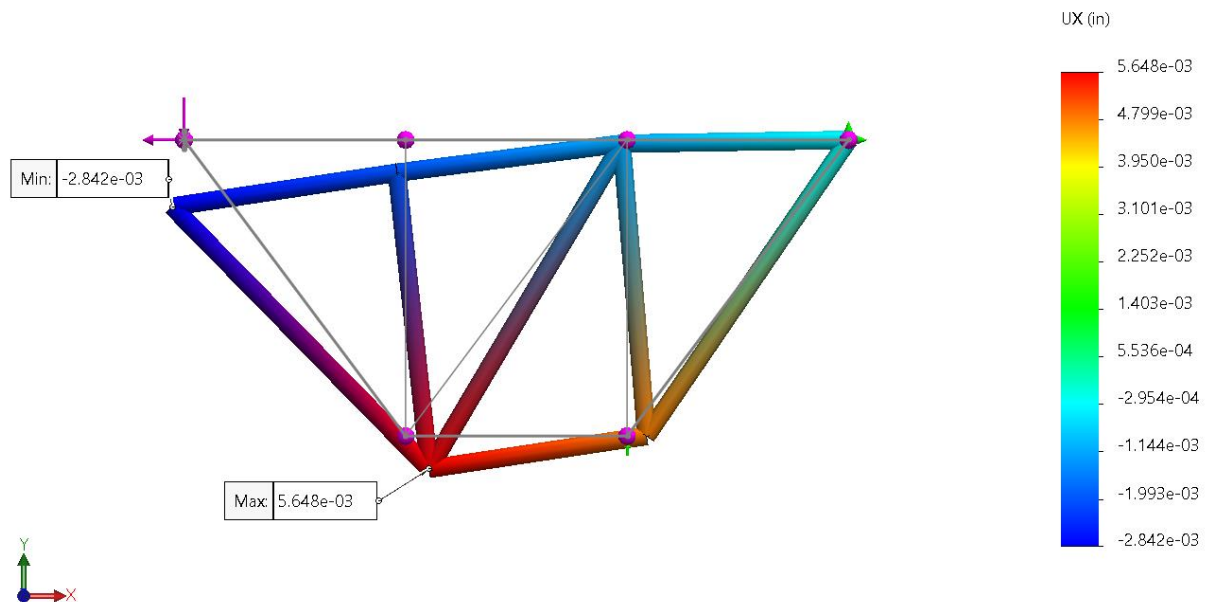
## A-4: Stress Analysis Methodology - Additional Screenshots



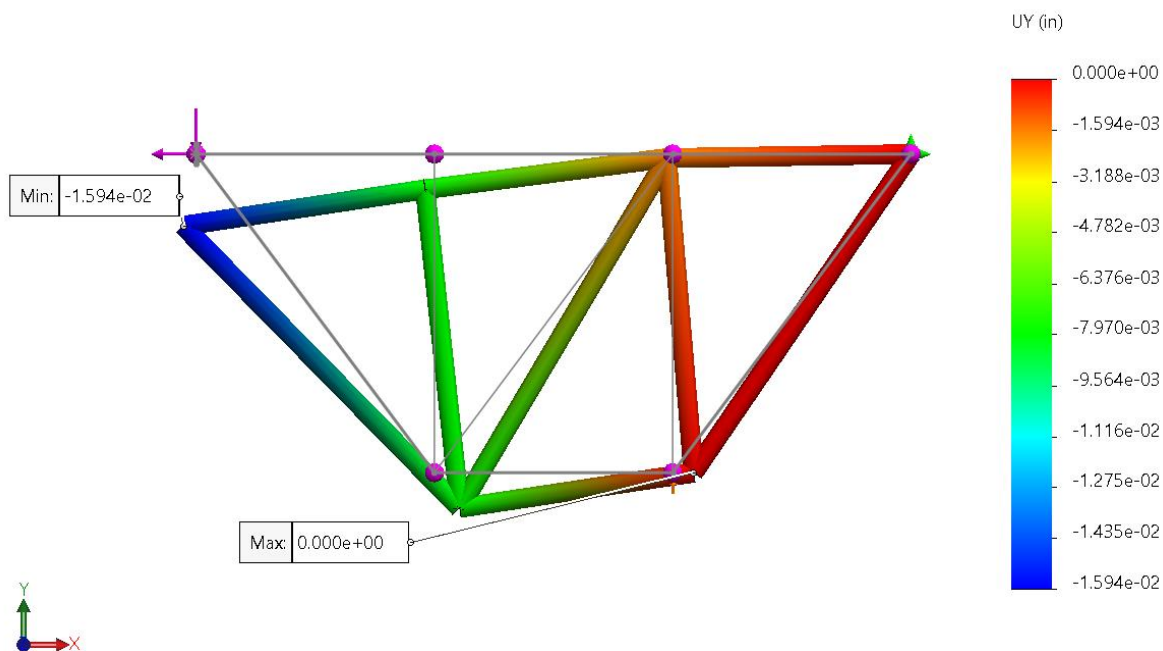


## B-1: Problem 2 Additional Screenshots

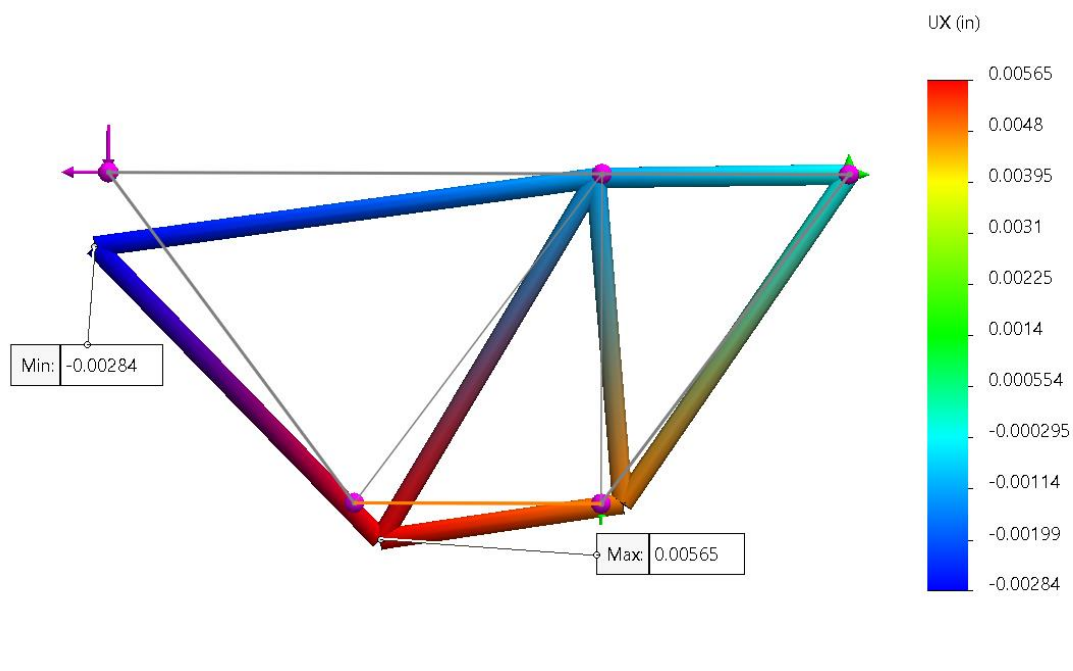
Beam Name	Element	End	Axial (lbf)	Shear1 (lbf)	Shear2 (lbf)	Moment 1 (lbf.in)	Moment 2 (lbf.in)
Beam-1(Al tube rectangular AL TUBE 4 X 3 RECT X 0.1875 WALL(1)(6))	1	1	576.91	0	0	0	0
	1	2	-576.91	0	0	0	0
Beam-2(Al tube rectangular AL TUBE 4 X 3 RECT X 0.1875 WALL(1)(4))	2	1	1,153.8	0	0	0	0
	2	2	-1,153.8	0	0	0	0
Beam-3(Al tube rectangular AL TUBE 4 X 3 RECT X 0.1875 WALL(1)(2))	3	1	-538.45	0	0	0	0
	3	2	538.45	0	0	0	0
Beam-4(Al tube rectangular AL TUBE 4 X 3 RECT X 0.1875 WALL(1)(3))	4	1	-884.6	0	0	0	0
	4	2	884.6	0	0	0	0
Beam-5(Al tube rectangular AL TUBE 4 X 3 RECT X 0.1875 WALL(1)(8))	5	1	-576.91	0	0	0	0
	5	2	576.91	0	0	0	0
Beam-6(Al tube rectangular AL TUBE 4 X 3 RECT X 0.1875 WALL(1)(9))	6	1	7.9792e-05	0	0	0	0
	6	2	-7.9792e-05	0	0	0	0
Beam-7(Al tube rectangular AL TUBE 4 X 3 RECT X 0.1875 WALL(1)(5))	7	1	692.3	0	0	0	0
	7	2	-692.3	0	0	0	0
Beam-8(Al tube rectangular AL TUBE 4 X 3 RECT X 0.1875 WALL(1)(1))	8	1	-538.45	0	0	0	0
	8	2	538.45	0	0	0	0
Beam-9(Al tube rectangular AL TUBE 4 X 3 RECT X 0.1875 WALL(1)(7))	9	1	461.53	0	0	0	0
	9	2	-461.53	0	0	0	0



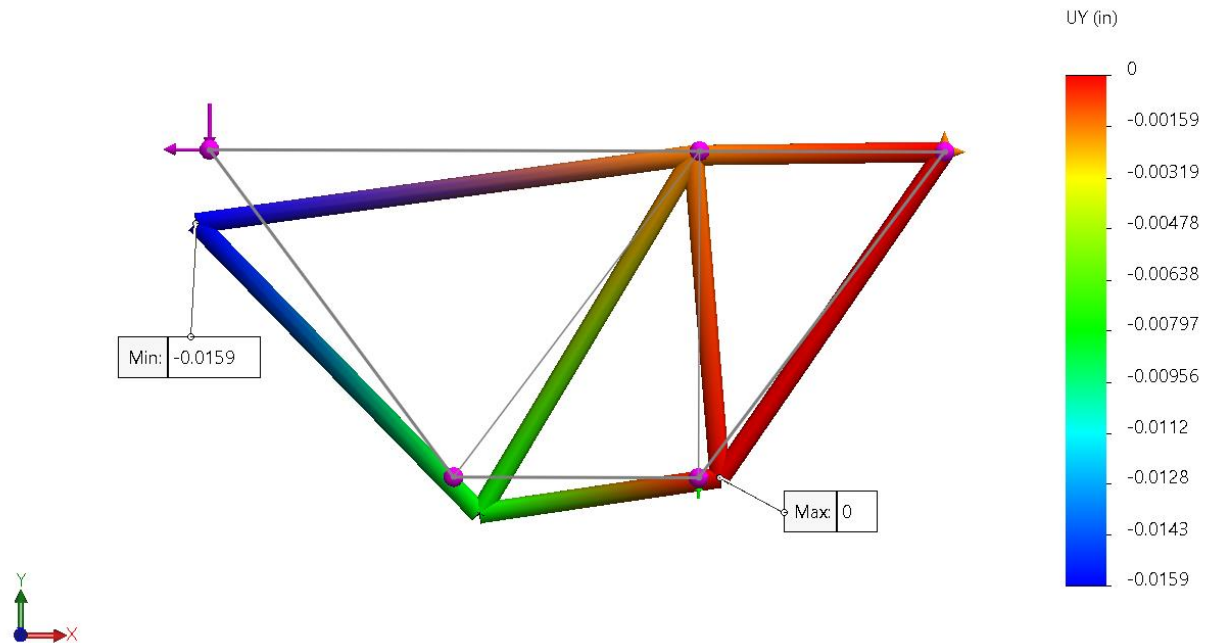
*X displacement of original truss*



*Y displacement for original truss*



*X displacement for simplified truss*



*Y displacement for simplified truss*

```

%*****INPUT DATA*****

numele = 7;                %number of elements
numnod = 5;                %number of nodes
x= [0, 36, 72, 72, 108];   %nodal x-coordinates
y= [0, -48, 0, -48, 0];    %nodal y-coordinates

%%connectivity of elements to nodes
%element 1 is connected to nodes 1&2
%element 2 is connected to nodes 2&3
%element 3 is connected to nodes 3&4
%etc
node= [1,1,2,2,3,3,4;
       2,3,3,4,4,5,5];

%Note: the size of 'young' and 'area' must match numele
young=10*10^6*[1 1 1 1 1 1]; % Young's modulus
area=2.5*[1 1 1 1 1 1];      % cross-sectional areas

%Nodal Constraints
ifix=[zeros(2,numnod)];      %initialize ifix
%ifix=0 is a free DOF, ifix=1 is a constrained DOF
%in this example nodes 1 and 3 fully constrained; others free

ifix=[0,0,0,0,1;
      0,0,0,1,1];

%Applied Force Vector
% force=[Fx1;Fy1;Fx2;Fy2;Fx3;Fy3;Fx4;Fy4;Fx5;Fy5;Fx6;Fy6]
% input only where known, leave as zero in all other slots
force=[-(2500/13);-(6000/13);0;0;0;0;0;0;0;0;0;0];

~      END OF INPUT DATA

```

## B-2: Part B MATLAB Matrix Code

```

E = 10e6;
A = 2.5;
%Node coordinates (in)
coords = [...
0, 0; % Node A-1
36, -48; % Node C-2
72, 0; % Node D-3
72, -48; % Node E-4
108, 0]; % Node F-5
%Member list after removing BC (BC is zero-force)
members = [...
1 2; % AC
1 3; % AD
2 3; % CD
2 4; % CE
3 4; % DE
3 5; % DF
4 5]; % EF
num_nodes = size(coords,1);
num_dof = num_nodes * 2;
K_global = zeros(num_dof);
%global stiffness matrix
for m = 1:size(members,1)
i = members(m,1);
j = members(m,2);
xi = coords(i,1); yi = coords(i,2);
xj = coords(j,1); yj = coords(j,2);
L = sqrt((xj - xi)^2 + (yj - yi)^2);
c = (xj - xi) / L;
s = (yj - yi) / L;
k_local = (A * E / L) * ...
[ c^2 c*s -c^2 -c*s;
c*s s^2 -c*s -s^2;
-c^2 -c*s c^2 c*s;
-c*s -s^2 c*s s^2];
%Global DOF indices
dof = [2*i-1 2*i 2*j-1 2*j];
%Assemble global stiffness matrix
K_global(dof,dof) = K_global(dof,dof) + k_local;
end
%External Force Vector
F = zeros(num_dof,1);
theta_load = atan(5/12);
F(1) = 500 * cos(theta_load); %Fx at Node A
F(2) = 500 * sin(theta_load); %Fy at Node A
%Boundary Conditions:
%Node E: vertical roller support (u_y = 0)
%Node F: pinned support (u_x = 0, u_y = 0)
fixed_dof = [10, 11, 12]; % [E_y, F_x, F_y]
%Free DOF
free_dof = setdiff(1:num_dof, fixed_dof);

```



```

% Solve for displacements
displacements = zeros(num_dof,1);
displacements(free_dof) = K_reduced \ F_reduced;
% Display results
disp('Nodal Displacements (inches):')
for n = 1:num_nodes
fprintf('Node %d: u_x = %.6f in, u_y = %.6f in\n', ...
n, displacements(2*n-1), displacements(2*n));
end
% Display global stiffness matrix
disp('Global Stiffness Matrix [K] (psi*in):')
disp(K_global)

```

### B-3: Part E MATLAB Code

```

% 2D TRUSS PROGRAM
% ME 345
% Stevens Institute of Technology

clear;                                %clear previous runs

%*****INPUT DATA*****

numele = 7;                           %number of elements
numnod = 5;                           %number of nodes
x= [0, 36, 72, 72, 108];              %nodal x-coordinates
y= [0, -48, 0, -48, 0];                %nodal y-coordinates

%%connectivity of elements to nodes
%element 1 is connected to nodes 1&2
%element 2 is connected to nodes 2&3
%element 3 is connected to nodes 3&4
%etc
node= [1,1,2,2,3,3,4;
       2,3,3,4,4,5,5];

%Note: the size of 'young' and 'area' must match numele
young=10*10^6*[1 1 1 1 1 1 1];        % Young's modulus
area=2.5*[1 1 1 1 1 1 1];             % cross-sectional areas

%Nodal Constraints
ifix=[zeros(2,numnod)];               %initialize ifix
%ifix=0 is a free DOF, ifix=1 is a constrained DOF
%in this example nodes 1 and 3 fully constrained; others free

ifix=[0,0,0,0,1;
      0,0,0,1,1];

%Applied Force Vector
% force=[Fx1;Fy1;Fx2;Fy2;Fx3;Fy3;Fx4;Fy4;Fx5;Fy5;Fx6;Fy6]
% input only where known, leave as zero in all other slots
force=[-(2500/13);-(6000/13);0;0;0;0;0;0;0;0;0;0];

```

```

%      END OF INPUT DATA
% *****

% zero bigk matrix to prepare for assembly
ndof=2;
numeqns=numnod*ndof;
bigk=zeros(numeqns);

% loop over elements

for e=1:numele          % compute element stiffness
    nlink=2;           % nlink=number of nodes per element
    x21=x(node(2,e))-x(node(1,e));
    y21=y(node(2,e))-y(node(1,e));
    length=sqrt(x21^2+y21^2);
    c=x21/length;
    s=y21/length;
    ke=area(e)*young(e)/length*[c^2, c*s, -c^2, -c*s;
                                c*s, s^2, -c*s, -s^2;
                                -c^2, -c*s, c^2, c*s;
                                -c*s, -s^2, c*s, s^2];

    % assemble ke into bigk

    n1=ndof-1;
    for i=1:nlink;
        for j=1:nlink;
            rbk=ndof*(node(i,e)-1)+1;
            cbk=ndof*(node(j,e)-1)+1;
            re=ndof*(i-1)+1;
            ce=ndof*(j-1)+1;

            bigk(rbk:rbk+n1,cbk:cbk+n1)=bigk(rbk:rbk+n1,cbk:cbk+n1)+ke(re:re+n1,ce:ce+n1);
        end          %j loop
    end              %i loop
end                  %e loop

disp('Unconstrained Stiffnes Matrix is:')
disp(bigk)
disp('Hit any key to continue')
pause;

% impose support conditions (boundary conditions)
for n=1:numnod
    for j=1:ndof
        if (ifix(j,n) == 1)
            m=ndof*(n-1)+j;
            dummy=bigk(m,m);
            bigk(m,:)=zeros(1,numeqns);
            bigk(:,m)=zeros(numeqns,1);
            bigk(m,m)=1;
            force(m)=0; %zero appropriate rows of force
        end
    end
end
end

```

```

% solve stiffness equations
displace=bigk\force           %nodal displacements

% compute stresses
for e=1:numele
    n1=2*node(1,e)-1;          %get nodes of the element
    n2=2*node(2,e)-1;
    de(1:2)=displace(n1:n1+1); %nodal disp for first node FINAL EXAM
    de(3:4)=displace(n2:n2+1); %nodal disp for second node
    x21=x(node(2,e))-x(node(1,e));
    y21=y(node(2,e))-y(node(1,e));
    lengthvec(e)=sqrt(x21^2+y21^2);
    length=lengthvec(e);
    c=x21/length;
    s=y21/length;
    S=[-c,-s,c,s];
    stress(e)=(young(e)/length)*S*de';
end

stresscol=stress'             %print stresses in elements

```

## C-1: Contribution Chart

Group Number: \_\_\_\_\_

Student names in alphabetical order by last name written here→		Student 1 <u>Angela Dinh</u>	Student 2 <u>Tyler Hoke</u>	Student 3 <u>Brandon Kiefer</u>	Student 4 <u>Cooper Kinsley</u>	Student 5 <u>Jasper Zwemmer</u>
Overall Case Study	Overall Proj. Manag. Lead		x			
	Report Compile Lead	x				
Problem 1	Lead		x			
	Secondary					x
	Tertiary					
	Report Lead					x
	Report Secondary		x			
Problem 2	Lead	x				
	Secondary			x		
	Tertiary				x	
	Report Lead	x				
	Report Secondary			x		
Problem 3 (if needed)	Lead					
	Secondary					
	Tertiary					

	<b>Report Lead</b>					
	<b>Report Secondary</b>					

By signing below, each student verifies that the above information fairly and accurately reflects the contributions of all team members.

\_Tyler Hoke, Angela Dinh, Jasper Zwemmer, Brandon Kiefer, Cooper Kinsley\_  
Student Signatures

