Lab #3
Design 2D Truss Structure with SolidWorks

The City College of New York
Department of Mechanical Engineering

ME 37100 Computer Aided Design
Section 1EF

Yesim Kokner
Sarah Liu

7 October 2024

Table of Content

Abstract

A two-dimensional analysis of the nodal stresses, displacements and reaction force of an Aluminum alloy 1060 truss was done to determine maximum loading before failure at specified locations. Such calculations compared through a SolidWorks simulation and theoretical calculations yielded results on linkages not needed for providing support for force equilibrium but still needed for structural support.

Introduction

Truss structures are composed of several 2 force member elements connected by a series of nodes. Such structure is used to withstand applied transverse forces. Truss structures can be seen in bridges, within roofs, aircraft hangers, etc., to provide a strong support composed of element acting as a whole structure. Truss can take on several different configurations, adapting to the specific situation in which the truss is to be used.

Analysis of a truss structure can be done in 2-Dimensional planes by taking advantage of th global stiffness matrix. Such a method allows for finding the element stress, displacements and forces as well as the reaction forces at the supports. FEA can also be used to find the same values by applying appropriate constraints on the truss. Both methods provide a way to find proper geometrical configurations needed or maximum loading to avoid failing.

Theoretical Background

In the following section, background information and equations will be given for 2-dimensional truss as the figure below, which will be used in this lab.

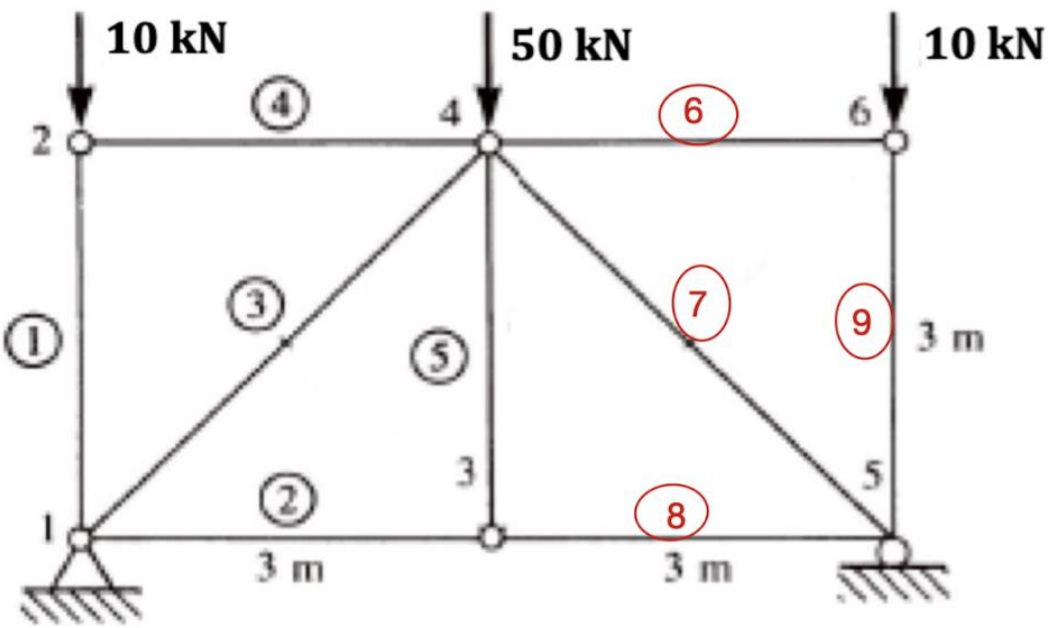

Figure1: 2D Truss structure analyzed in this lab. 5 elements are analyzed (numbered in black) and nodes found through symmetry (numbered in red) with a total of 6 nodes.

It can be noted that symmetry can be used for this problem so only the labeled elements will be solved in the calculations. The rods of the truss are said to be solid and has a cross-sectional area of the elements are $A = 3\ x\ 10^{-4}\ m^2$ Aluminum Alloy has the following material properties:

$$Elastic\ modulus\ (E) = 69\ GPa$$
$$Yield\ Strength\ (S_y) = 62\ MPa$$

Each local element stiffness matrix can be obtained with the following equation:

$$k = \frac{AE}{L}\begin{bmatrix} C^2 & CS & -C^2 & -CS \\ CS & S^2 & -CS & -S^2 \\ -C^2 & -CS & C^2 & CS \\ -CS & -S^2 & CS & S^2 \end{bmatrix} \text{(Equation 1)}$$

Where L is the length of the bar, C = cosine, S = sine of the angle of the bar in relative to the global coordinate system.

The local stiffness matrix can then be assembled in the global coordinates and related with the global forces and displacements by adding up all the local stiffness matrix to obtain [K]. The global matrix is as follows:

$$\{F\} = [K]\{d\} \text{ (Equation 2)}$$

Boundary conditions lead to simplifications and to make such problems solvable. Nodal displacement will then be solved which can then be used to calculated nodal stresses with the following equation:

$$\sigma_{a-b} = \frac{E}{L}[-C \quad -S \quad C \quad S]\begin{bmatrix} u_a \\ v_a \\ u_b \\ v_b \end{bmatrix} \text{(Equation 3)}$$

With the stresses at each node, elemental forces can be calculated using:

$$F = \sigma_{a-b}\ x\ A \text{ (Equation 4)}$$

Factor safety (n) can be used to determine the maximum loading onto the truss by comparing with the yield strength value of the truss material. For failure to occur, n would be under 1 Such calculations can be done with the following equation:

$$n = \frac{S_y}{\sigma} \text{(Equation 5)}$$

Manual Analysis and Calculations

In the manual calculations for the nodal stresses, displacements and the forces acting on each link, and method of assembling a global stiffness matrix was used based on the configurations of each element. From the truss below, several local stiffness matrices were formed corresponding with the labeled parts in the problem. (Note: all results will also be tabulated with the results from FEA in the Graphical Demonstrations of SolidWorks Results sections).

Using equation 1, the local element stiffness matrix are as follows:

$$k_{1-2}=\frac{AE}{3}\begin{bmatrix}0&0&0&0\\0&1&0&-1\\0&0&0&0\\0&-1&0&1\end{bmatrix}\quad k_{1-3}=\frac{AE}{3}\begin{bmatrix}1&0&-1&0\\0&0&0&0\\-1&0&1&0\\0&0&0&0\end{bmatrix}\quad k_{3-4}=\frac{AE}{3}\begin{bmatrix}0&0&0&0\\0&1&0&-1\\0&0&0&0\\0&-1&0&1\end{bmatrix}$$

$$k_{2-4}=\frac{AE}{3}\begin{bmatrix}1&0&-1&0\\0&0&0&0\\-1&0&1&0\\0&0&0&0\end{bmatrix}\quad k_{1-4}=\frac{AE}{3\sqrt{2}}\begin{bmatrix}.5&.5&-.5&-.5\\.5&.5&-.5&-.5\\.5&-.5&.5&-.5\\-.5&-.5&.5&.5\end{bmatrix}$$

From figure 1, several boundary conditions are given to help reduce the global stiffness matrix.

$$\boldsymbol{u_1}=\boldsymbol{v_1}=0$$

$$\boldsymbol{u_3}=\boldsymbol{u_4}=0$$

$$F_{2y}=\ -10\ kN\ (F_{6y}=\ -10\ kN)$$

$$F_{4y}=\ -25\ kN$$

(Due to symmetry half the applied load on node 4 will be used for the calculations)

Thus, the remainder of the terms of the local stiffness matrix, placed into the global stiffness matrix and relating such to the global forces (equation 2) can be written as:

$$\begin{matrix}0\\-10{,}000\\0\\-25{,}000\end{matrix}=AE\begin{bmatrix}1/3&0&-1&0\\0&1/3&0&0\\-1&0&1/3&0\\0&0&0&0.4512\end{bmatrix}\begin{matrix}u_2\\v_2\\v_3\\v_4\end{matrix}$$

Plugging in the A and E values, the matrix yields the following answers:

$$\boldsymbol{u_2}\ =\boldsymbol{u_6}\ =\ 0\ m$$

$$\boldsymbol{v_2}\ =\boldsymbol{v_6}\ =\ \ -1.449\ E-3\ m$$

$$\boldsymbol{v_3}=\boldsymbol{v_4}\ =\ -1.0248\ E-2\ m$$

From determining the nodal displacement, the nodal stresses can be calculated from equation 3. The calculations for each nodal stress are as follows:

$$\sigma_{1-2}^{(1)}=\frac{E}{3}[0\quad -1\quad 0\quad 1\quad]\begin{bmatrix}0\\0\\0\\-1.449\ E-3\end{bmatrix}$$

$$\sigma_{1-3}^{(2)}=\frac{E}{3}[-1\quad 0\quad 1\quad 0\quad]\begin{bmatrix}0\\0\\0\\-1.0248\ E-2\end{bmatrix}$$

$$\sigma_{1-4}^{(3)} = \frac{E}{3\sqrt{2}}\begin{bmatrix} \frac{-\sqrt{2}}{2} & \frac{-\sqrt{2}}{2} & \frac{\sqrt{2}}{2} & \frac{\sqrt{2}}{2} \end{bmatrix}\begin{bmatrix} 0 \\ 0 \\ 0 \\ -1.0248\,E-2 \end{bmatrix}$$

$$\sigma_{2-4}^{(4)} = \frac{E}{3}\begin{bmatrix} -1 & 0 & 1 & 0 \end{bmatrix}\begin{bmatrix} 0 \\ -1.449\,E-3 \\ 0 \\ -1.0248\,E-2 \end{bmatrix}$$

$$\sigma_{3-4}^{(5)} = \frac{E}{3}\begin{bmatrix} 0 & -1 & 0 & 1 \end{bmatrix}\begin{bmatrix} 0 \\ -1.0248\,E-2 \\ 0 \\ -1.0248\,E-2 \end{bmatrix}$$

Solving for the matrices yields:

$$\boldsymbol{\sigma_{1-2}^{(1)}} = \boldsymbol{\sigma_{5-6}^{(9)}} = 33.3333MPa(C) \quad \boldsymbol{\sigma_{1-3}^{(2)}} = \boldsymbol{\sigma_{3-5}^{(8)}} = 0\ MPa$$

$$\boldsymbol{\sigma_{1-4}^{(3)}} = \boldsymbol{\sigma_{4-5}^{(7)}} = 117.851\ MPa\ (C) \quad \boldsymbol{\sigma_{2-4}^{(4)}} = \boldsymbol{\sigma_{4-6}^{(6)}} = 0\ Mpa \quad \sigma_{3-4}^{(5)} = 0\ MPa$$

Element forces can then be calculated using equation 4 for each node as follows:

$$F_{1-2}^{(1)} = -\,33.3333\ MPa\ x\ 3E-4\ m^2$$

$$F_{1-3}^{(2)} = 0\ MPa\ x\ 3E-4\ m^2$$

$$F_{1-4}^{(3)} = 117.851\ MPa\ x\ 3E-4\ m^2$$

$$F_{2-4}^{(4)} = 0\ MPa\ x\ 3E-4\ m^2$$

$$F_{3-4}^{(5)} = 0\ MPa\ x\ 3E-4\ m^2$$

From the calculations, the following element forces were obtained:

$$\boldsymbol{F_{1-2}^{(1)}} = \boldsymbol{F_{5-6}^{(9)}} = -10\ kN \quad \boldsymbol{F_{1-3}^{(2)}} = \boldsymbol{F_{3-5}^{(8)}} = 0\ kN \quad \boldsymbol{F_{1-4}^{(3)}} = \boldsymbol{F_{4-5}^{(7)}} = -35.355\ kN$$

$$\boldsymbol{F_{3-4}^{(5)}} = 0\ kN$$

To find the reaction force at node 1, force equilibrium can be used to determine the x and y components of the reaction force. Only elements 1, 3 and 4 will affect node 1.

$$\sum F_x = -\ 35.355\ cos(45)\ kN + \boldsymbol{F_{2-4}^{(4)}} = 0$$

$$\sum F_y = -\,10\ kN - 35.355\ sin(45)\ kN + R_{1y} = 0$$

Yielding the following reaction force components:

$$\boldsymbol{F_{2-4}^{(4)}} = \boldsymbol{F_{4-6}^{(6)}} = 25\ kN \quad R_{1y} = R_{5y} = 35\ kN$$

To determine which link causes failure, equation 5 can be used. It can be found that only links 3 and 7 will have a factor of safety under 1 at n = 0.5261.

Graphical Demonstrations of SolidWorks Results

In the SolidWorks FEA portion of the lab, weldments module was used with rod dimensions outlined in figure 1 and A utilized to find the diameter of the rod. Note that the node values that will be explained in the writing match that of figure 1 and will not be mentioned as that in the images.

Note: Due to the discrepancy of the yield strength of Aluminum Alloy 1060 material, in the SolidWorks simulations, such are replaced by a custom material that has the yield strength and elastic modulus outlined in the problem statement.

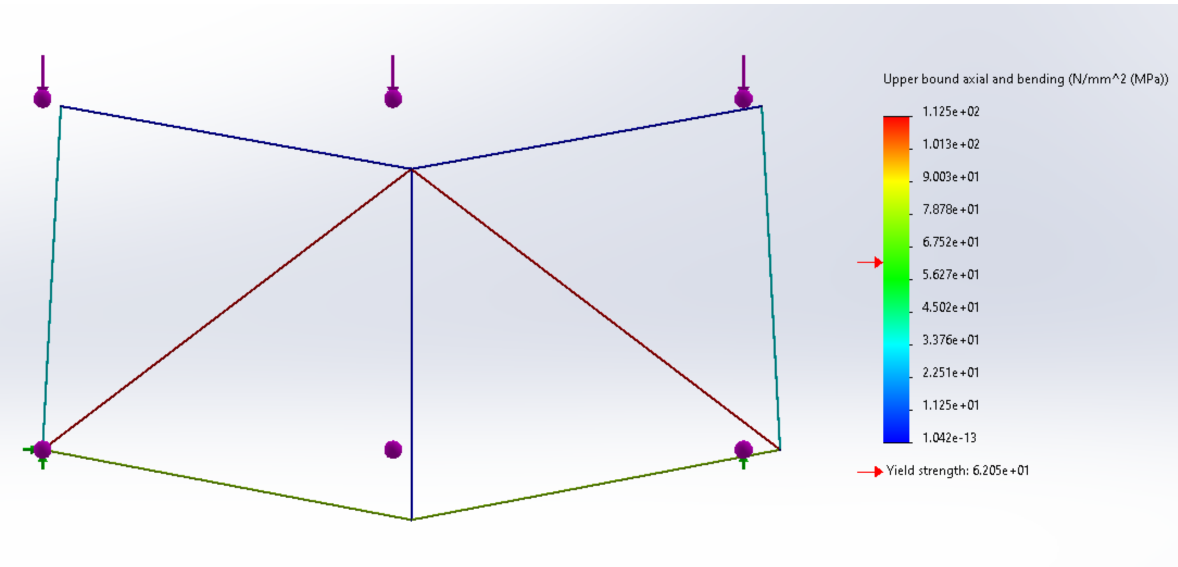

Figure 2: Truss structure under the loading displayed in figure 1. A material of Aluminum alloy 1060 with a yield strength of 27.57 MPa is used.

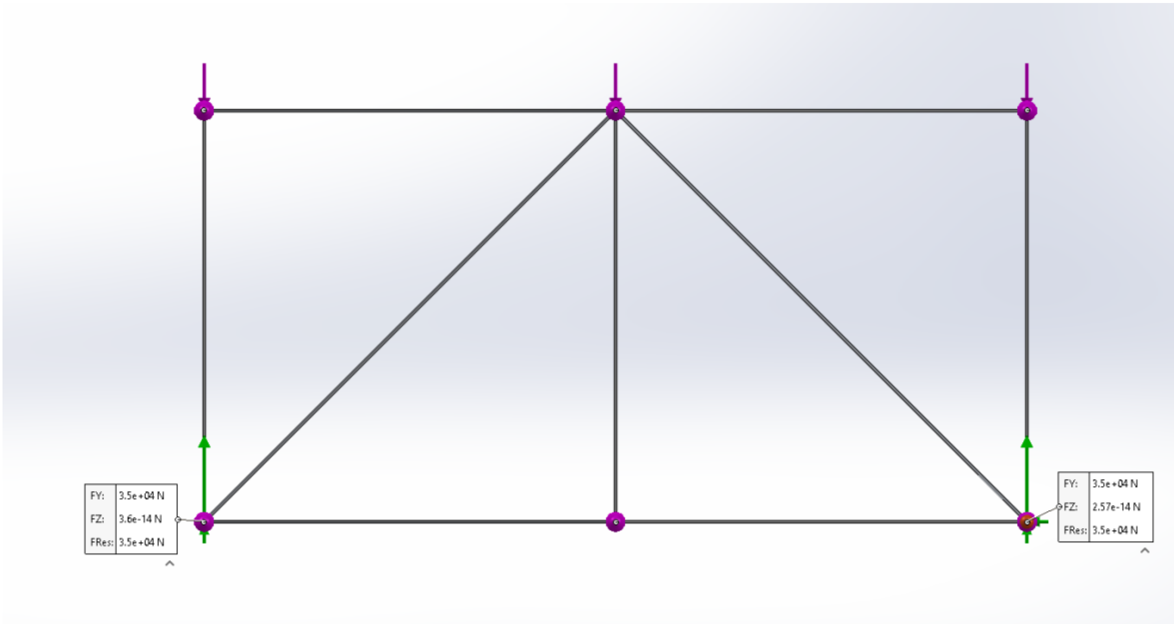

Figure 3: Aluminum Alloy 1060 structure under loading and dimension in figure 1, reaction forces found at node 1 and 5.

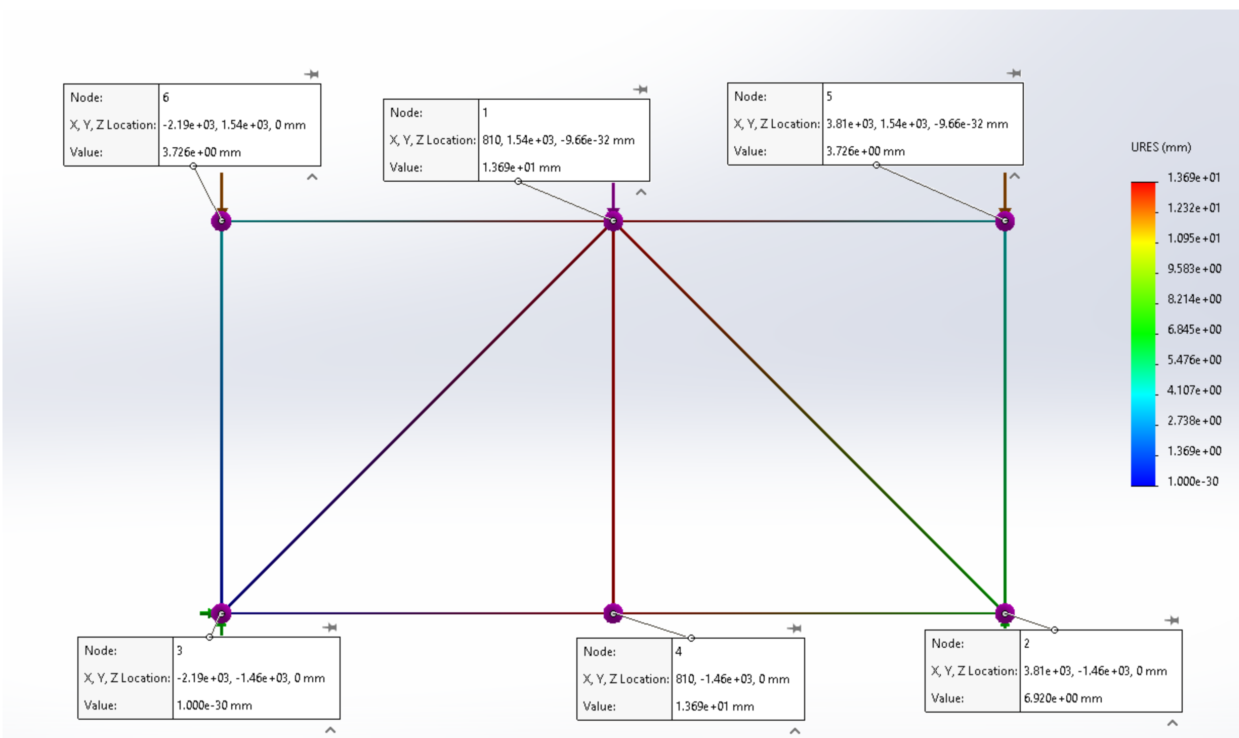

Figure 4: Aluminum Alloy 1060 structure under loading and dimension in figure 1, displacement found at nodes 1-6.

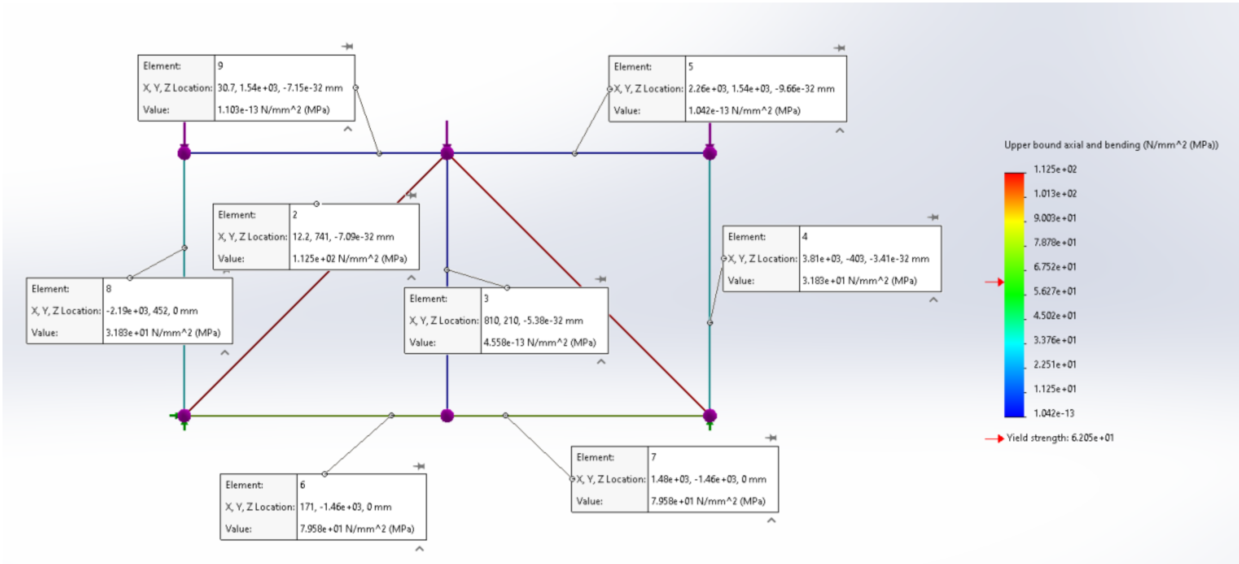

Figure 4: Aluminum Alloy 1060 structure under loading and dimension in figure 1, stress found at element 1-9.

Element	Solidworks FEA Stress (MPa)	Hand Calculations Stress (MPa)
1	-31.83	-33.333
2	79.58	83.33
3	-112.5	-117.851
4	5.687E-13	0
5	2.233E-12	0
6	5.584E-13	0
7	-112.5	-117.851
8	79.58	83.33
9	-31.83	-33.333

Table 1: Tabulated Stress values of each element labeled in figure 1 of FEA results and results from hand calculations.

Element	Solidworks FEA Forces (kN)	Hand calculations Forces (kN)
1	-10	-10
2	25	25
3	-35.355	-35.355
4	7.8074E-8	0
5	3.0658E-7	0
6	7.6668E-9	0
7	-35.355	-35.355
8	25	25
9	-10	-10

Table 2: Tabulated force values of each element labeled in figure 1 of FEA results and results from hand calculations.

Node	Solidworks FEA Displacement (m)	Hand calculations Displacement (m)
1	$v_1 = 1.583E-3$	$v_1 = 0$
2	$v_2 = 8.527E-3$	$v_2 = -1.449\,E-3$
3	$v_3 = 3.133E-2$	$v_3 = 1.0248\,E-2$
4	$v_4 = 3.133E-2$	$v_4 = -1.0248\,E-2$
5	$v_5 = 1E-23$	$v_5 = 0$
6	$v_6 = 8.527E-3$	$v_6 = -1.449\,E-3$

Table 3: Tabulated displacement values of each node labeled in figure 1 of FEA results and results from hand calculations.

Force applied at center (kN)	Maximum Stress in Truss (MPa)	Factor of safety (n)
50	112.5395	0.5509
30	67.5237	0.9182
27.5	61.8967	1
18.6	41.8647	1.4896
0	31.8310	1.9478

Table 4: Values of maximum stress and corresponding calculated factor of safety (equation 5) using yield strength of 27.57 MPa.

Node	Support Reaction Force (kN) with FEA	Support Reaction Force (kN) Through Hand Calculations
1	35	35
5	35	35

Table 5: Values of Support reaction force via FEA and Hand Calculations

Discussions and Interpretations of Results

By solving for nodal elements displacement through the use of global stiffness matrix, displacement in the x direction in node 3 and 4 was assumed to be 0 in order to solve the problem. However, it can be seen that there was slight displacement not found in hand calculations in the x direction of those nodes. Such results are negligible for hand calculations to make the problem solvable. Due to the constraints of the support, this is a reasonable assumption. The yielding strength (Figure 2) for aluminum alloy 1060 is 25.75 MPa, however such was not the given yield strength therefore a custom material of the same elastic modulus (69 GPa) and yield strength (62 MPa) was used. By applying equation 5, elements 3,7, 2 and 8 will all fail under the applied load. The factor of safety was considered as 1 for failure as it will allow the maximum stress to be that of the yield strength. Yield strength is the point of stress that separates the elastic and plastic regions. Having a factor of safety below 1 indicates that the stress is beyond the yield strength. This result shows such truss structure is not balanced at its current configuration with the applied load that is currently placed on the truss.

To find the maximum safe load to apply to the structure before failure, the yield strength of 62 MPa is used in equation 5, replacing the stress with F/A and solving for the force to yield a maximum load of 18.6 kN. With the current loading on the truss, the truss would fail. Through the SolidWorks simulation to find the max load, changing the 50 kN force in the center of the truss to a lower value (Table 4). Compared with the theoretical maximum load, it was found that around a load of 27.5 kN would be the max load as it yields a factor of safety of 1. The differences in the hand calculation thus give us an error of 32.4%. Such high error can be due to the fact that such overall load is not considering the load concentric to node 4.

Removing any of the links will compromise the equilibrium of the truss. Element 4, 6 and 5 was seen to have no forces acting upon them (Table 2). Such elements do not play a role in providing support for the structure in terms of force equilibrium but rather play a role as structural support. Thus, by removing those linkages, the force will still be supported through the other elements.

Conclusion

In this lab, an analysis of each elemental stress and forces were found through SolidWorks and through manual calculations. Nodal displacement was also found, in which through manual calculation serval nodes were considered to have no displacement in certain directions, but through FEA, it was found such assumptions were mostly correct but still has a value. FEA accounted for the real-world factors which was assumed to not be present through manual analysis. It was found through the SolidWorks analysis the maximum safe load applied at the center of the truss was not accurate as such configuration will fail under any loading. No link can be removed safely for the structure as although several members do not experience any force or stress, they are integral for the structure of the truss.

References

[1] A First Course in the Finite Element Method, Enhanced Edition, 6th Ed., by Daryl Logan (2022), ISBN-10: 0357884140