UNIVERSITY OF COLORADO BOULDER

ASEN 3112: STRUCTURES

Lab 2: 16 - Bay Truss

Author:
NICK BARBATTINI[1]

Author:
JACOB GRECO[2]

Author:
KATE KOSMICKI[3]

Author:
RISHAB PALLY[4]

Author:
LUIS ROMO[5]

Author:
YUZHOU SHEN[6]

Professor: Francisco Lopez Jimenez

Due: Nov 4, 2022

[1]SID: 109407259
[2]SID: 109784574
[3]SID: 109843119
[4]SID: 109519936
[5]SID: 109381915
[6]SID: 109520557

Smead Aerospace
UNIVERSITY OF COLORADO BOULDER

Contents

This paper provides a structural analysis of materials with various complex configurations using the ANSYS application which aids with computing and modeling the stresses/strains in the system. With this analysis, users can better understand how to develop real life engineering applications much more efficiently. By running the ANSYS simulation, we were able to understand and compare the inaccuracies between structural analysis and actual experimental results. Using FEM (finite element method), we were able to determine whether it is structurally stable enough for a real world application. When cross checking using the ANSYS simulation, we received a deflection value that is 39% greater than the actual experimental value. Therefore the FEM is an effective option when it comes to approximating and analyzing the performance of complex structures but comes along with some inaccuracies.

1 Results

1.1 Question 1: Experimental Results

The plots depicting the displacement, internal, and reaction force measurements from the experimental data are shown below. All plots contain a linear line of best fit for linear regression analysis.

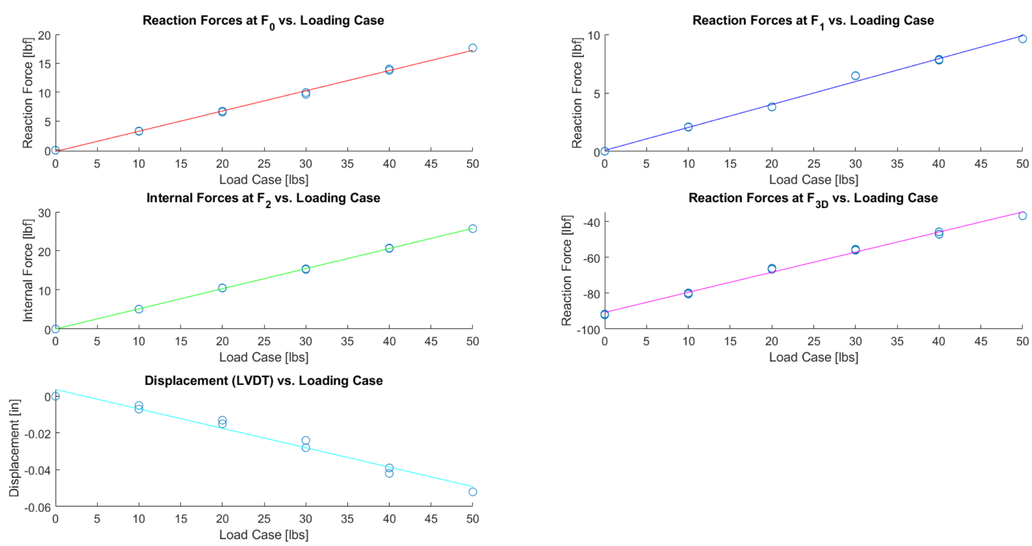


Figure 1: Displacement, internal, reaction force, and linear regression lines over different loading cases

The linear regression lines in Figure (1) display the close linear trend of the measured data. The residuals for all measurements are shown in the table below. With Figure (1) and Table (1) combined, the assumption of a linear trend among the experimental data measurements is valid.

Residuals from Experimental Data and Linear Regression				
F_0	F_1	F_2	F_3D	$LVDT$
2.9602	2.6290	1.2699	14.7902	0.0312

Table 1: Residuals from Analysis of Experiemnetal Data Measurements and Linear Regression

1.2 Question 2: Analytical and FEM Results

When modeling the truss in ANSYS FEM, the group made a couple of assumptions simplifying the model. One simplification the group assumed is that the inline load cell from the actual experiment had the same stiffness as the overall truss material. This load cell would have a different stiffness which would effect the way it reacts to the forces. Since the stiffness is unknown, it is assumed to be the same. Another simplification made was regarding the boundary conditions of the truss. In this FEM, a boundary condition was set on each side of the bar on the elements between the y and z axis that connect to the support. This fixes those elements in their place, meaning there will be no displacements at those nodes. Another simplification made was about friction because it was assumed that there was no friction in the x direction at the end with the roller support. As seen in the following figures, these simplifications resulted in discrepancies between the experimental and modeled truss deformations and forces.

Figures that depict the displacement, internal forces, and reaction forces from the ANSYS simulation are shown below.

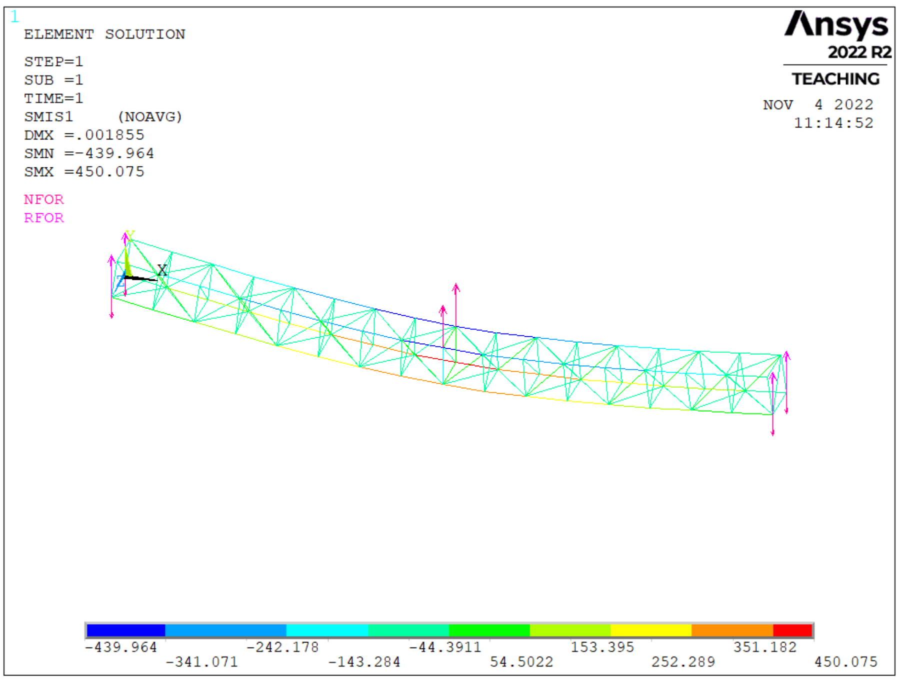


Figure 2: The Internal Forces from ANSYS

The plot depicting the undeformed versus deformed truss is also shown below.

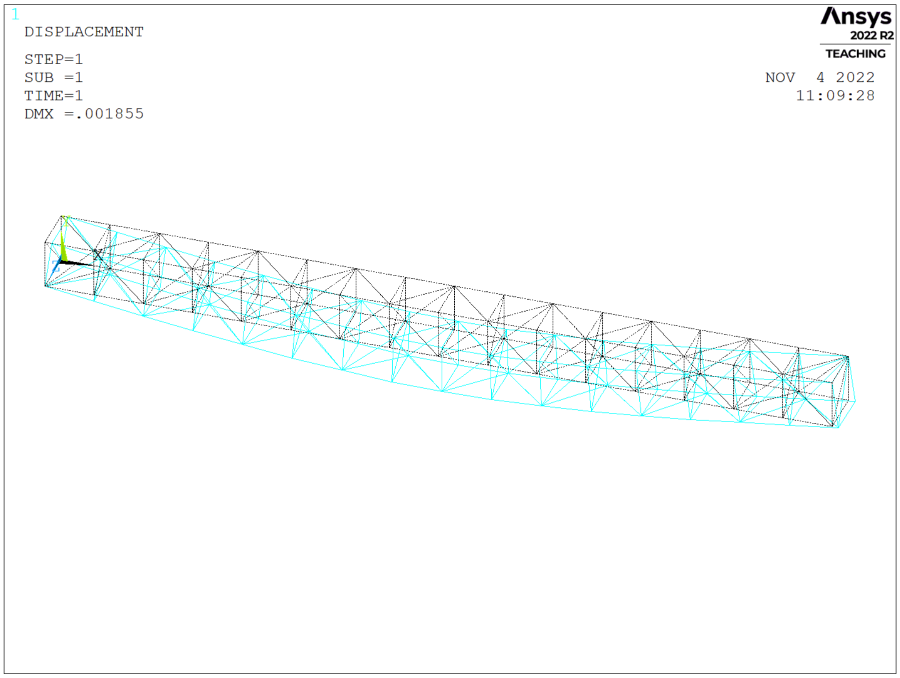


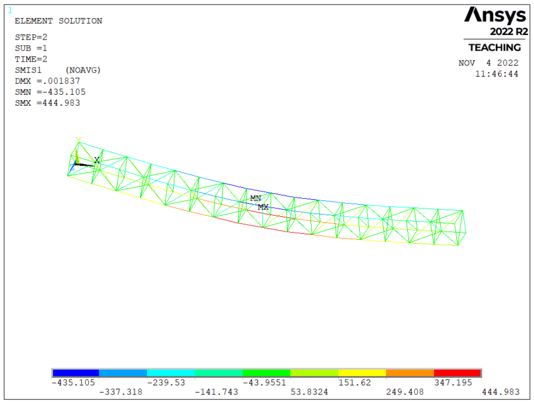
Figure 3: The Deformation from ANSYS

The equivalent beam model simplifies the entire truss structure into a single considered beam. To do this, a single second moment of inertia was calculated about the center of the truss, which was considered to be the "neutral axis." This was achieved with the parallel axis theorem. The entire derivation for the equivalent beam can be seen in Figure 6 in the Appendix.

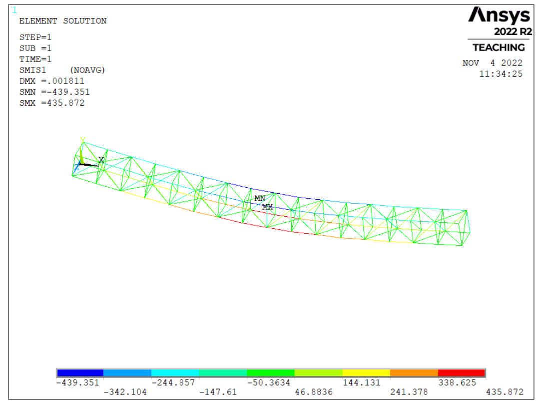
The beam calculation yielded a maximum displacement of -1.736 mm for the truss. The FEM analysis yielded a maximum displacement of -1.84 mm for the same truss. This is a difference of 0.104 mm or about a 6% error if the FEM is the value we expect from the calculation. This difference can be accounted in how the two values are calculated. The FEM takes into account each node and tube in the truss while the beam model is a simplistic representation of only four tubes spanning the length of the truss. The beam model does not account for joints or members that connect the main span truss sections together. Despite this, it is accurate within 6% because of the loading of the truss. It is not under torsion or other significant forces that the cross members are designed to resist against.

1.3 Question 3: Uncertainty Analysis

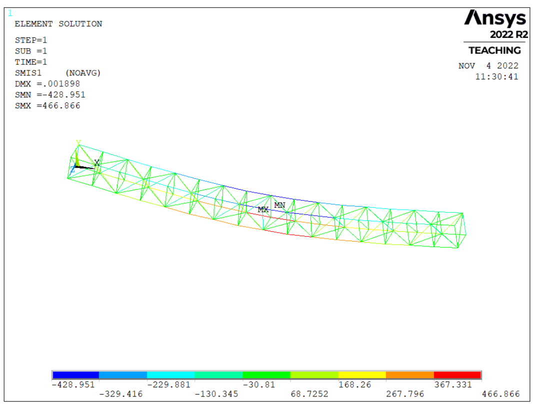
Following the analysis of the ANSYS and experimental data, we were able to determine the ANSYS model had a greater displacement when compared to the experimental data. The ANSYS model had a maximum displacement of 1.84 mm while the experimental model had a maximum displacement of 1.32 mm meaning the ANSYS displacement was 39% larger than the experimental data. This difference between the model and real-world truss means that the ANSYS model contained numerous simplifications. There is a correlation between the maximum deflection as it occurs when the Young's Modulus decreases. Not only that, due to the ANSYS model assumption of axial stress load distribution we can see that the maximum internal force of the ANSYS model differed from the experimental data. This is due to energy going into the bars, therefore the internal forces calculated through ANSYS result in a higher than expected value. Note in a real life application, the bar would result in a lower internal force as we would expect there to be internal bending moments which would withdraw some of the energy that goes in the bar. To explore the simplifications that the ANSYS model used, Figure 4 shows the internal forces throughout the truss, Figure 5 shows the deformations of the truss, and Table 2 shows the maximum displacements for each of the varied models being applied in this lab.



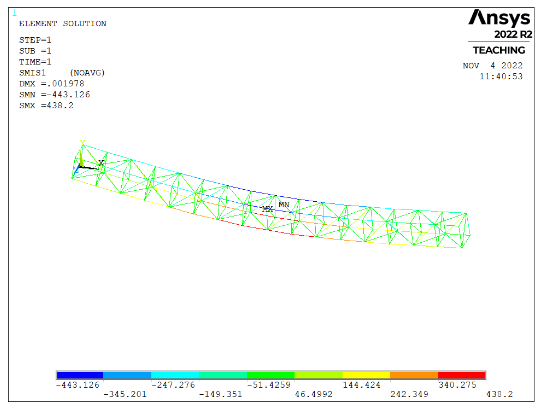
(a) Beam



(b) With Friction



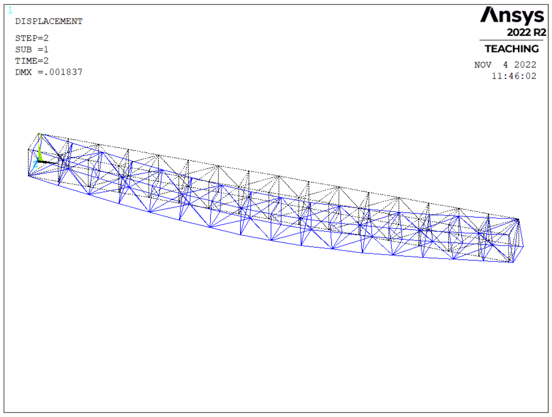
(c) Changing Inline Load Cell



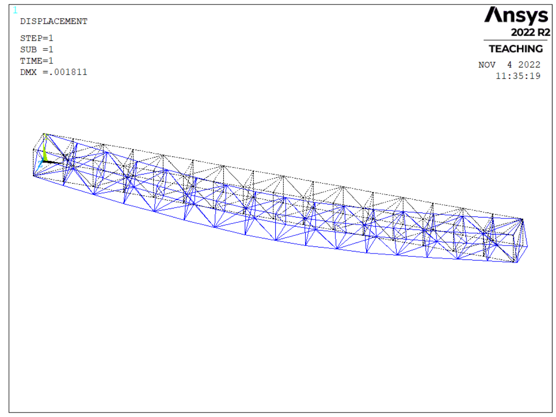
(d) Manufacturing Imperfection

Figure 4: The Internal Forces In Different Situation

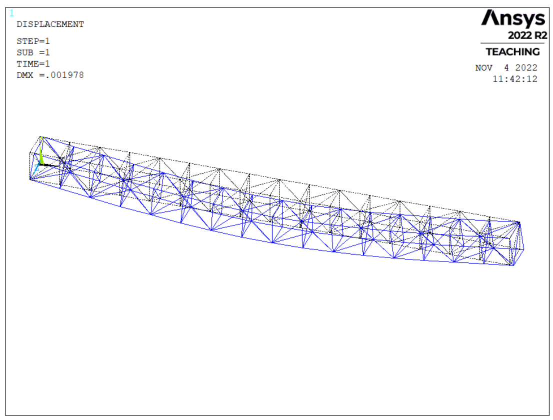
The Imperfect Joints model accounts for free-play within the truss, which reduces the overall stiffness in the structure. This was accomplished by decreasing the Young's Modulus in the ANSYS model. The Free-Play model looks at the strut that had the in-line load cell which causes a decrease in stiffness for that singular bar. The original ANSYS model assumed that there was no friction at the roller support, so the Friction model added this factor in by applying a small force opposing the direction of displacement at the support. The Manufacturing model looked at the error in the manufacturing of the struts, so random struts within the truss had different cross sectional areas. The final model analyzed the struts as beams to analyze the internal bending moments that the struts would feel.



(a) Beam



(b) With Friction



(c) Manufacturing Imperfection

Figure 5: The Deformation In Different Situation

	Maximum Downward Displacement [mm]
Experiment	1.32
Original ANSYS	1.84
Imperfect Joints	2.59
Free-Play in the In-Line Load Cell	1.85
Friction at Supported Joints	1.80
Manufacturing Imperfections	1.94
Acting as a Beam	1.83

Table 2: Maximum Displacements

The two main sources of error/varying parameters that were tested to further analyze the 16-Bay Truss were friction and the Young's Modulus. There is a clear show for friction occurring on the system when being experimented with. When Friction is applied to the model, we determined a maximum displacement of 1.80mm. When compared to the original model, this was a decrease in displacement by 2.17%. While this value didn't decrease the displacement drastically, it was the model that reflected the experimental data the closest. To further see the effects of friction on the system, a more accurate frictional force would have to be determined. To affect the overall stiffness of the system, the Young's Modulus was varied as many of the potential errors affected the stiffness. Lowering the Young's Modulus by 28.6% to 50 GPa resulted in a maximum downward displacement of 2.59 mm. When compared to the original model, there is an increase in displacement by 0.75 mm which is 40.7% higher. Overall the variation of the Young's Modulus resulted in an increase in deflection and this is used to calculate the factor of safety of the system. Using the Original ANSYS deflection of 1.84 mm along with the maximum deflection from the Imperfect Joints of 2.59 mm, the factor of safety can be calculated. After dividing the maximum deflection of the Imperfect Joints by the Original ANSYS Deflection, we receive a factor of safety of 1.407. When using the Experimental deflection of 1.32 mm, we determined a factor of safety of 1.962. In both cases, the factor of safety yielded approximately close to 1.7 which is common for most Aerospace applications.

2 Conclusions

Through this lab we were able to use a FEM (Finite Element Method) simulation to construct a 16-bay truss to better understand the load displacement of the structure by varying different parameters and seeing whether it is structurally stable enough for real life applications. Due to assumptions such as axial stress, load distribution, and imperfect joints being applied, we observed that the maximum internal force of the ANSYS model differed from the experimental data. This resulted in a difference of 39% between the displacements, but at the same time we were able to determine the truss held a factor of safety between 1.407 to 1.962 which is reasonable for Aerospace applications. Some improvements that could have been made to the lab to enhance the user's understanding of load distribution, displacements and other structural parameters would be testing on different sized trusses. This way we could see how the FEM would affect the magnitude of displacements across a broader range of structures while also being able to determine whether they are structurally stable. Overall this lab did a great job of aiding the user understand the significance of the Finite Element Method and how accurate the approximations can be.

3 Acknowledgments

3.1 Member Contributions

Member	Contribution
Nick Barbattini	100%
Jacob Greco	100%
Kate Kosmicki	100%
Rishab Pally	100%
Luis Romo	100%
Yuzhou Shen	100%

Signature of Approval: Nick Barbattini, Jacob Greco, Kate Kosmicki, Rishab Pally, Luis Romo, Yuzhou Shen

3.2 Member Contributions Notes

Nick Barbattini, Kate Kosmicki and Yuzhou Shen did an exceptional job modeling the Truss and depicting the displacements using the ANSYS application. Jacob Greco derived the initial equations needed to understand displacements and internal forces withing the truss. Rishab Pally did a great job with the Uncertainty Analysis on the system and the lab write up. Luis Romo provided the group with an analytical Matlab Analysis.

4 Appendix

4.1 Initial Derivations/Equations

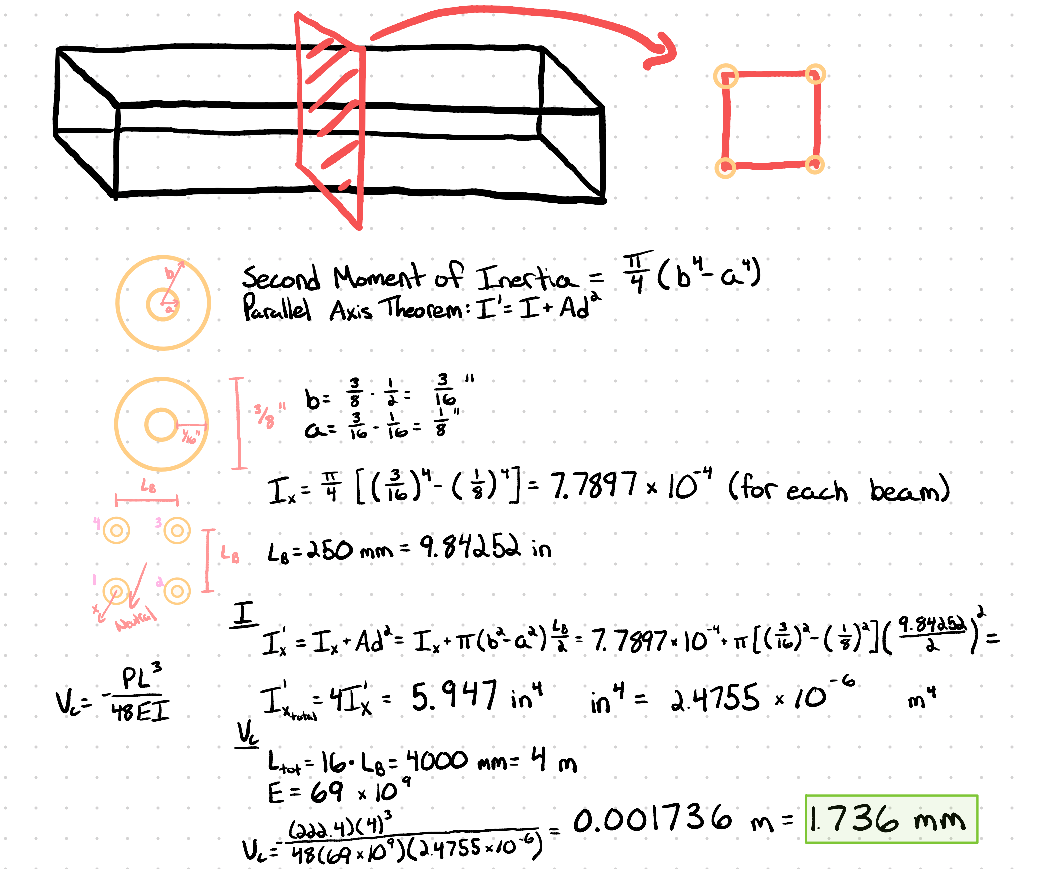


Figure 6: Beam Derivation

4.2 ANSYS Analysis

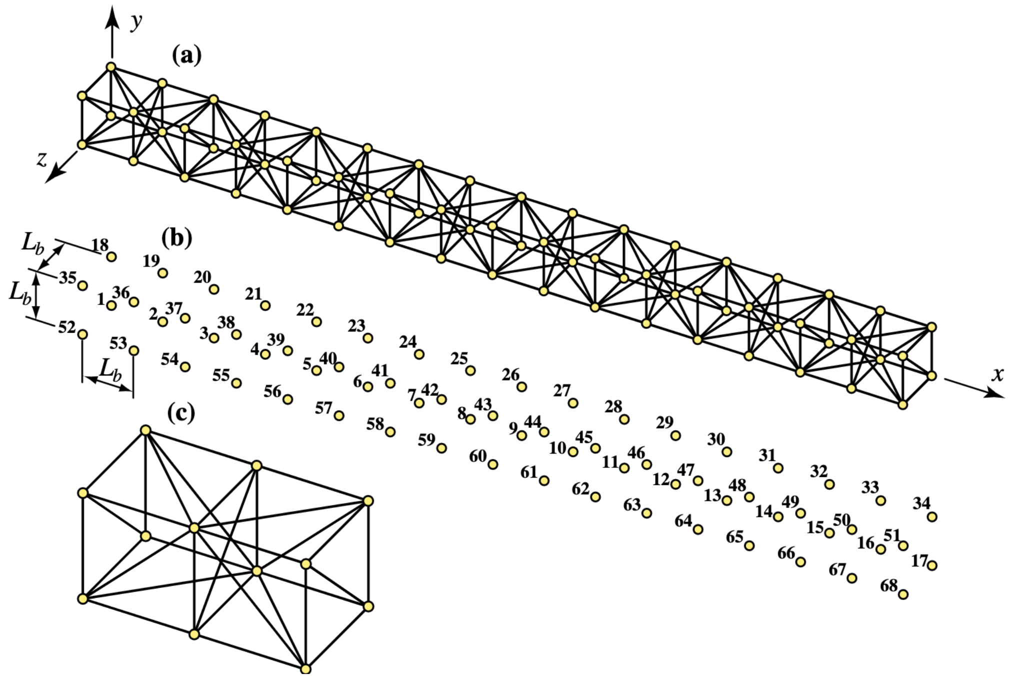


Figure 7: Truss Used For Modeling

4.3 Matlab Analysis

lab2.m

```
% 3112: Bay Truss Lab Data Analysis - Group 8
%Housekeeping
clc;
clear all;
close all;
%Main Code
[loadcase,F0,F1,F2,F3D,LVDT] = unPack('Test 08');

%Givens
L = 0.250; %[mm] joint to joint
d = 9.525; %[mm] bar diameter
t = 1.587; %[mm] bar thickness
E = 70;    %[GPa] Elastic modulus of 6061-T6 Al


%Zero Load Case
F0_0 = [F0(1:10);F0(106:115)];
F1_0 = [F1(1:10);F1(106:115)];
F2_0 = [F2(1:10);F2(106:115)];
F3D_0 = [F3D(1:10);F3D(106:115)];
LVDT_0 = [LVDT(1:10);LVDT(106:115)];

F0_0avg = mean(F0_0);
F1_0avg = mean(F1_0);
F2_0avg = mean(F2_0);
F3D_0avg = mean(F3D_0);
LVDT_0avg = mean(LVDT_0);
```

```
%10lbs Load Case
F0_10 = [F0(11:23);F0(96:105)];
F1_10 = [F1(11:23);F1(96:105)];
F2_10 = [F2(11:23);F2(96:105)];
F3D_10 = [F3D(11:23);F3D(96:105)];
LVDT_10 = [LVDT(11:23);LVDT(96:105)];

F0_10avg = mean(F0_10);
F1_10avg = mean(F1_10);
F2_10avg = mean(F2_10);
F3D_10avg = mean(F3D_10);
LVDT_10avg = mean(LVDT_10);

%20lbs Load Case
F0_20 = [F0(24:35);F0(86:95)];
F1_20 = [F1(24:35);F1(86:95)];
F2_20 = [F2(24:35);F2(86:95)];
F3D_20 = [F3D(24:35);F3D(86:95)];
LVDT_20 = [LVDT(24:35);LVDT(86:95)];

F0_20avg = mean(F0_20);
F1_20avg = mean(F1_20);
F2_20avg = mean(F2_20);
F3D_20avg = mean(F3D_20);
LVDT_20avg = mean(LVDT_20);

%30lbs Load Case
F0_30 = [F0(36:45);F0(76:85)];
F1_30 = [F1(36:45);F1(76:85)];
F2_30 = [F2(36:45);F2(76:85)];
F3D_30 = [F3D(36:45);F3D(76:85)];
LVDT_30 = [LVDT(36:45);LVDT(76:85)];

F0_30avg = mean(F0_30);
F1_30avg = mean(F1_30);
F2_30avg = mean(F2_30);
F3D_30avg = mean(F3D_30);
LVDT_30avg = mean(LVDT_30);

%40lbs Load Case
F0_40 = [F0(46:55);F0(66:75)];
F1_40 = [F1(46:55);F1(66:75)];
F2_40 = [F2(46:55);F2(66:75)];
F3D_40 = [F3D(46:55);F3D(66:75)];
LVDT_40 = [LVDT(46:55);LVDT(66:75)];

F0_40avg = mean(F0_40);
F1_40avg = mean(F1_40);
F2_40avg = mean(F2_40);
F3D_40avg = mean(F3D_40);
LVDT_40avg = mean(LVDT_40);

%50lbs Load Case
F0_50 = F0(56:65);
F1_50 = F1(56:65);
F2_50 = F2(56:65);
F3D_50 = F3D(56:65);
LVDT_50 = LVDT(56:65);

F0_50avg = mean(F0_50);
F1_50avg = mean(F1_50);
F2_50avg = mean(F2_50);
F3D_50avg = mean(F3D_50);
LVDT_50avg = mean(LVDT_50);


%Graphical Table
```

```
Cases = {'0lbs';'10lbs';'20lbs';'30lbs';'40lbs';'50lbs'};
F0new = [F0_0avg;F0_10avg;F0_20avg;F0_30avg;F0_40avg;F0_50avg];
F1new = [F1_0avg;F1_10avg;F1_20avg;F1_30avg;F1_40avg;F1_50avg];
F2new = [F2_0avg;F2_10avg;F2_20avg;F2_30avg;F2_40avg;F2_50avg];
F3Dnew = [F3D_0avg;F3D_10avg;F3D_20avg;F3D_30avg;F3D_40avg;F3D_50avg];
LVDTnew = [LVDT_0avg;LVDT_10avg;LVDT_20avg;LVDT_30avg;LVDT_40avg;LVDT_50avg];

T = table(F0new,F1new,F2new,F3Dnew,LVDTnew,'RowNames',Cases);

uit = ...
    uitable('Data',T{:,:},'ColumnName',T.Properties.VariableNames,'RowName',T.Properties.RowNames,'Units','No

%Question 1 Analysis
%F2 is the in-line loading cell and measures internal forces
%LVDT is the displacement
%all others are the reaction forces

%Perform Linear Regression First to Overlay Later

[coeffF0,errorF0] = polyfit(loadcase,F0,1);
F0fit = polyval(coeffF0,loadcase);

[coeffF1,errorF1] = polyfit(loadcase,F1,1);
F1fit = polyval(coeffF1,loadcase);

[coeffF2,errorF2] = polyfit(loadcase,F2,1);
F2fit = polyval(coeffF2,loadcase);

[coeffF3D,errorF3D] = polyfit(loadcase,F3D,1);
F3Dfit = polyval(coeffF3D,loadcase);

[coeffLVDT,errorLVDT] = polyfit(loadcase,LVDT,1);
LVDTfit = polyval(coeffLVDT,loadcase);


%Plotting

figure()
subplot(3,2,1);
scatter(loadcase,F0);
hold on
plot(loadcase,F0,'LineStyle','none');
plot(loadcase,F0fit,'r');
title('Reaction Forces at F_0 vs. Loading Case');
xlabel('Load Case [lbs]');
ylabel('Reaction Force [lbf]');
hold off

subplot(3,2,2);
scatter(loadcase,F1);
hold on
plot(loadcase,F1,'LineStyle','none');
plot(loadcase,F1fit,'b');
title('Reaction Forces at F_1 vs. Loading Case');
xlabel('Load Case [lbs]');
ylabel('Reaction Force [lbf]');
hold off

subplot(3,2,3);
scatter(loadcase,F2);
hold on
plot(loadcase,F2,'LineStyle','none');
plot(loadcase,F2fit,'g');
title('Internal Forces at F_2 vs. Loading Case');
xlabel('Load Case [lbs]');
ylabel('Internal Force [lbf]');
hold off

```

```
subplot(3,2,4);
scatter(loadcase,F3D);
hold on
plot(loadcase,F3D,'LineStyle','none');
plot(loadcase,F3Dfit,'m');
title('Reaction Forces at F_{3D} vs. Loading Case');
xlabel('Load Case [lbs]');
ylabel('Reaction Force [lbf]');
hold off

subplot(3,2,5);
scatter(loadcase,LVDT);
hold on
plot(loadcase,LVDT,'LineStyle','none');
plot(loadcase,LVDTfit,'c');
title('Displacement (LVDT) vs. Loading Case');
xlabel('Load Case [lbs]');
ylabel('Displacement [in]');
hold off



% Function to unpack the experimental data

function [loadcase, F0, F1, F2, F3D, LVDT] = unPack(filename)

data = readtable(filename);

loadcase = table2array(data(:,1));
F0 = table2array(data(:,2));
F1 = table2array(data(:,3));
F2 = table2array(data(:,4));
F3D = table2array(data(:,5));
LVDT = table2array(data(:,6));


end
```