UNIVERSITY OF COLORADO - BOULDER

ASEN 3112 - 011

LAB 2 | GROUP 20

Finite Element Model and ANSYS Analysis

Author:
Bailis, FINN[a]

Author:
Bell, GARRETT[b]

Author:
Krebs, CHRISTOPHER[c]

Author:
Link, NATALIE[d]

Author:
MacFarland, ROBERT[e]

Author:
MacPherson, COLE[f]

Professors:
Alireza DOOSTAN
Kurt MAUTE

[a]SID: 108544972
[b]SID: 106929528
[c]SID: 108981667
[d]SID: 109005124
[e]SID: 109072567
[f]SID: 108521329

November 2, 2020

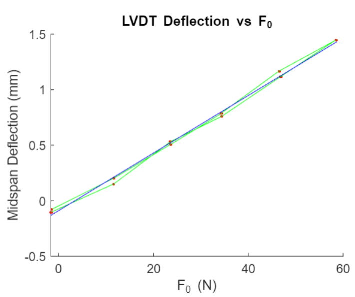
College of Engineering & Applied Science
UNIVERSITY OF COLORADO BOULDER

In the world of engineering, experimental analysis is not always possible and may not be the most efficient. Instead, computing software is often used to develop and test a model. For structural analysis of complex material configurations, programs such as ANSYS can model and compute the stresses and strains of a system. In this lab, the group delves into this type of artificial structural analysis and its inaccuracies compared to experimental results. In particular, the finite element method (FEM) is put to the test to see if it's a sufficient model for a real-world experiment. By performing an ANSYS simulation and comparing it to a real test, we found that the simulation predicted an 18% greater deflection than the actual experiment. This indicates that while not exact, FEM is a very useful tool for approximating the performances of complex structures that can't be measured exactly.

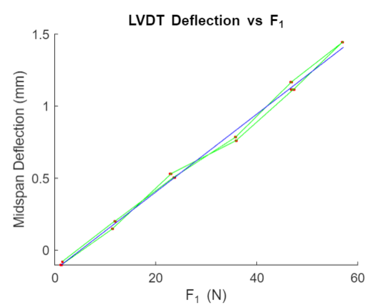
I. Results

A. Question 1: Experimental Results

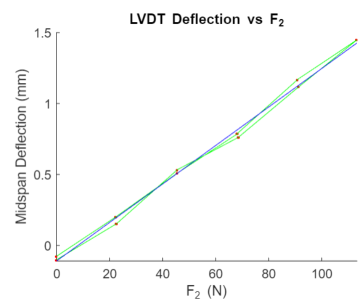
To demonstrate the linearity of the experimental results, the internal forces and displacements have been plotted in several different ways. The first three plots set the three internal forces as the independent variables and the measured deflection as the dependent. Figure 2 shows the deflection versus the force F_{3D}, and both of these, respectively, versus the external load case. Finally, Figure 3 shows each of the three internal forces plotted against the external load. All measured data points are in red, and the lines of best fit are shown in blue.



((a)) Experimental Results: LVDT vs F_0

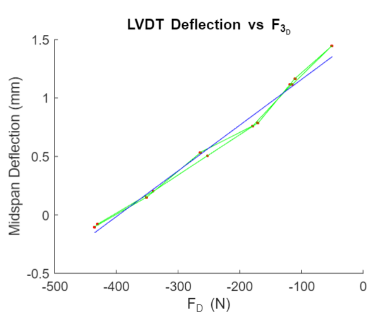


((b)) Experimental Results: LVDT vs F_1

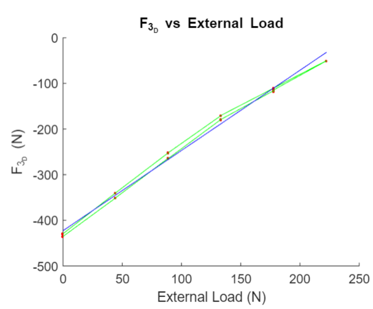


((c)) Experimental Results: LVDT vs F_2

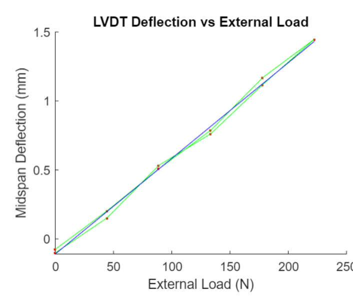
Fig. 1 Experimental Displacements vs Internal Forces



((a)) Experimental Results: LVDT vs F_{3D}

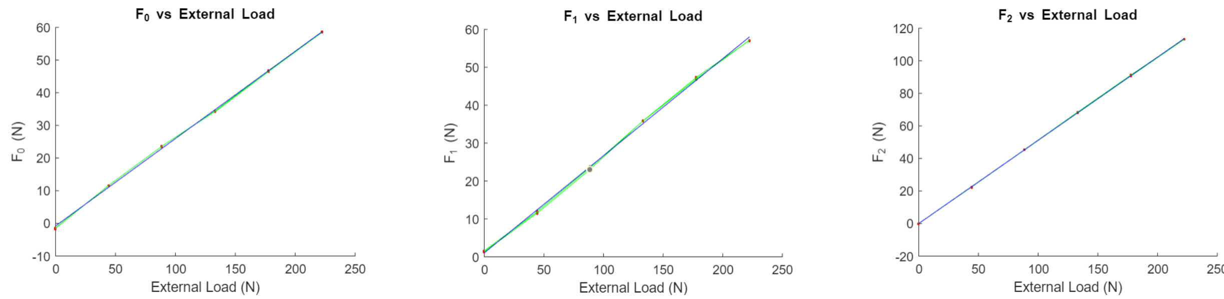


((b)) Experimental Results: F_{3D} vs External Load



((c)) Experimental Results: LVDT vs External Load

Fig. 2 Experimental Reaction Forces and Displacement



((a)) Experimental Results: F_0 vs External Load

((b)) Experimental Results: F_1 vs External Load

((c)) Experimental Results: F_2 vs External Load

Fig. 3 Experimental Internal Forces

For each of the five variables plotted against the external load, the slope and uncertainty of the best fit line are as follows.

Variable	Slope	Uncertainty
F_0	0.2674	0.4515 N
F_1	0.2561	0.5808 N
F_2	0.5111	0.1790 N
F_{3D}	1.757	8.970 N
LVDT	0.0069 mm/N	0.0243 mm

B. Question 2: Analytical and FEM Results

While using ANSYS is a very easy way to find displacements and internal and external forces, some assumptions were made when making the model during the computation of the model. These assumptions include perfect joints, connection with the load cell, friction, manufacturing errors, and uniform axial stress distribution through the bars. The joints are considered to be perfect in the ANSYS simulation whereas, in real life, it is likely that the joints could have separated partially leaving open space and effectively reducing the overall stiffness of the truss as a whole. Further, a similar thing could be viewed when looking at the connection with the load cell. It is possible that the connection, like with the joints, could be imperfect and have some amount of free-play associated with them and this would reduce the stiffness of the truss. Friction could play a role in the inaccuracy of the ANSYS calculations as well. Friction from the load cell could be so great that the roller support essentially acts as a pinned support and create differences between ANSYS and the actual test data. Manufacturing defects of the bars could also play a part in the inaccuracies of the ANSYS model. During the manufacturing process, the bars could have been made of slightly different lengths and had slightly different cross-sectional areas which would affect the actual system. The cross-sectional area differences could create larger or smaller rigidity, and the differing lengths can create areas of extra free play which reduces the rigidity. Lastly, the model assumes uniform axial stress distributions, and the joints are considered perfect pinned joints. This means that the bars have no bending moments and therefore do not bend. Additionally, the bars are considered to not move about the joints. All of the assumptions listed above are reasons why the ANSYS model is likely to differ from the actual test data. These assumptions show how it is important to consider these assumptions to understand the limitations of certain modeling software, thus you can appropriately choose software that will balance efficiency and accuracy depending on the project.

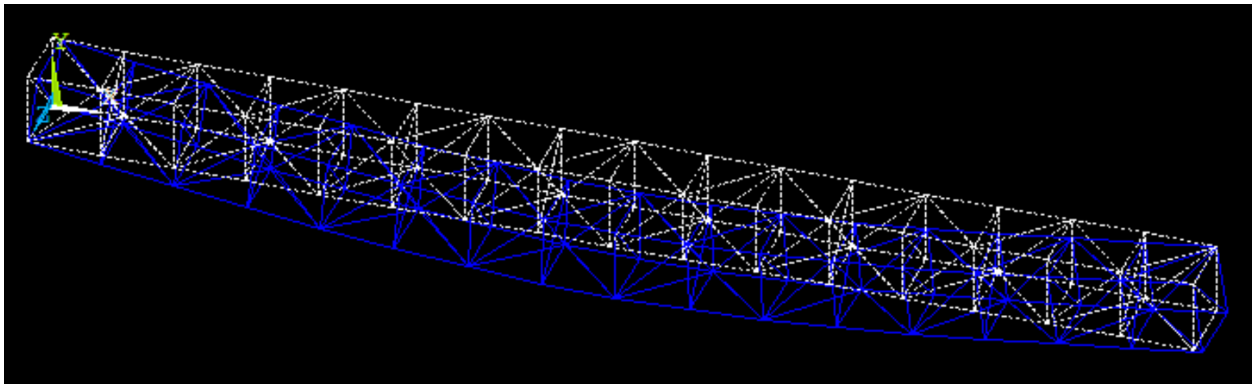


Fig. 4 Deflection of Truss in ANSYS (Exaggerated)

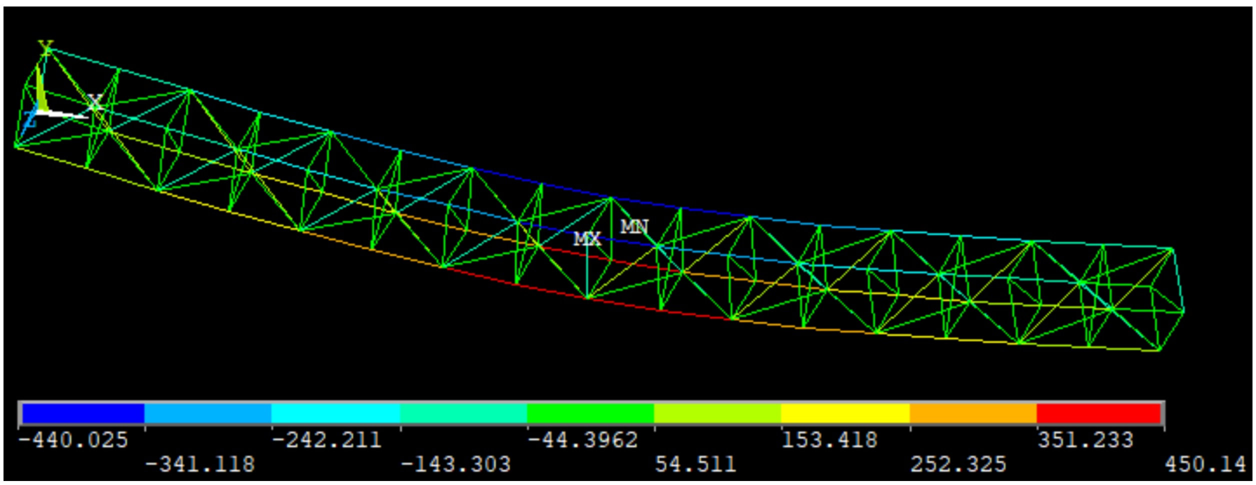


Fig. 5 Deflection of Truss in ANSYS (Exaggerated)

The maximum deflection was 1.8607 mm at the center of the beam. This deflection can be seen in figure fig:deflec ANSYS, which displays the original beam in white and the deflected beam in blue. It is clear that the middle of the truss, where the load was placed, displaces the greatest. Further, it is important to note that figure fig:deflec ANSYS is not an accurate depiction of the deflections as the software program, ANSYS, exaggerates deflection to make them more visible to the user. The minimum and maximum internal forces were found to be -440.025 N and 450.14 N, respectively. At node 1 and node 68 the reaction force was 55.562 N and at node 17 and 52 the reaction force was 55.638 N. Figure fig:forces ANSYS visually show the internal forces of the truss under load by color coating the bars based on their calculated stresses. When viewing this figure, it is seen that the beams in the middle of the truss have the most stress within them. Additionally, it is seen that the top bars are experiencing compression whereas the bottom bars are experiencing tension. When comparing the ANSYS results with the experiment, it is seen that the ANSYS model was a relatively accurate model considering all the assumptions that are used to simplify the analysis within ANSYS. The maximum deflection of the experiment was found to be 1.5494 mm. This is a nearly 25% difference between the ANSYS model and the experiment. It is believed that the most likely source of error between these two models is the lack of friction within the ANSYS model. Friction on the joints and the roller support are certainly going to affect the system as a whole and decrease the total deflection as the friction creates a reaction force in the X direction at the roller support which should, theoretically, be zero. When the effects of friction were analyzed in ANSYS, the roller was modeled as a pinned support to simulate infinite friction. This resulted in a decrease in the maximum deflection to 1.1809 mm. This shows that friction is the likely culprit for the discrepancies between the two, as the experimental deflection is in between the maximum displacements with maximum friction and with zero friction. Further, the experimental deflection is closer to the zero friction deflection which makes sense as the actual friction of the system is closer to zero than it is to infinity.

To check the results of the Finite Element Method (FEM) from ANSYS, an equivalent beam model is used to ensure that the FEM calculations were taken correctly. The truss is modeled as a beam, with the area moment inertia taken by the parallel axis theorem about the neutral axis. This parallel axis computation only considers the four longeron-strut cross-sections, with their moments of inertia about their bending axis ignored. This equivalent beam model also does not take the diagonal bar members into account. The calculation of the area moment of inertia of the beam is given by Fig.fig:EquilBeam and equations 1 and 2. From this, the beam is considered with a fixed support and roller support on either end, with 50 lb (222.4 N) point force P acting downwards at half the length of the beam. One could then compute reaction forces at either support and take cuts on either end of the P to derive the moment diagram of the beam. To determine the deflection, the external work due to point force P is set equal to the internal strain energy of the beam, and this relationship is given by Eq.3. Note that the elastic modulus of 6061-T6 aluminum alloy is 69-70 GPa, so 69.5 was used in the calculation, which is the same as used in the ANSYS model. This yields a maximum downward deflection of 1.7249 mm at half the center of the beam. The maximum internal shear force of the beam is then found by taking the derivative of the internal bending moment, which yields a shear force with a magnitude of 111.2 N. Note that this value isn't fully representative of a maximum internal force of a truss, for internal bar forces are axial. These values are also given in Table.tab:deflec and forces. The displacement of the equivalent beam model has a 7.29% discrepancy from the ANSYS model. This is mainly due to the moment of inertia calculation of this model. The diagonal elements are ignored, which are numerous throughout the truss design. This model also assumes that the moment of inertia is constant throughout the beam, which is not true in this truss design. Also, bars do not act in the same manner as beams. For finite element analysis, forces in bars are modeled axially, which is not modeled the same for beams, where shear forces and bending moments are considered. Other sources of discrepancy are that the equivalent beam model assumes the force is a perfect point load, with ideal fixed support and roller support. Further, this beam model follows one of the Euler-Bernoulli beam assumptions that axial stress and strain vary linearly across the thickness of the beam, which isn't true in the actual truss design.

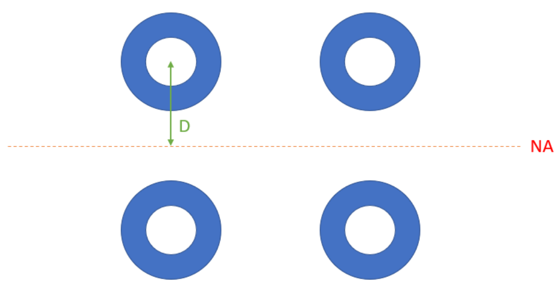


Fig. 6 Equivalent Beam Cross Section

$$I_{beam} = 4 * A_{member} * D^2 \tag{1}$$

$$A_{member} = \pi(R_e^2 - R_i^2) \tag{2}$$

$$-\frac{1}{2}Pv_{x=L/2} = \frac{1}{2}\int_0^L \frac{M_x^2}{EI}\,dx \tag{3}$$

C. Question 3: Uncertainty Analysis

After reading the experimental data and the ANSYS data, it was seen that the ANSYS model had a displacement that was 18% larger than the experimental data. The experimental model had a displacement of 1.5494 mm and the ANSYS model had a displacement of 1.8607 mm. Table tab:deflec and forces displays the maximum displacements and maximum internal forces for all of the models used. It can be seen that the maximum deflection occurs when the young's modulus decreases, this is important to note as it was later used to compute the necessary factor of safety.

Further, it can be seen that the maximum internal force was found during the ANSYS model and that this value differs from the experimental data. This discrepancy could be due to ANSYS assuming uniform axial stress distribution within the beams. This would cause the internal forces that were calculated by ANSYS to be greater than expected because all of the energy goes into the bar. In reality, the bar will likely experience internal bending moments and that will consume a portion of the energy that went into the bar resulting in a lower internal force.

	Maximum Downward Displacement	Maximum Internal Force
Experiment	1.5494 mm	385.49 N
Equivalent Beam	1.7249 mm	111.2 N
ANSYS	1.8607 mm	450.14 N
ANSYS (Infinite Friction)	1.1809 mm	450.14 N
ANSYS (Young's Modulus Decrease)	2.5864 mm	450.14 N

Table 1 The maximum displacements and internal forces from all five models

Many things could lead to discrepancies between the ANSYS model and the actual test. Most of which were discussed in section I.B of this report. Two of those possible sources of errors were picked to further analyze their effects on the system. These two factors were friction and a differing young's modulus. The effects of friction were chosen to be analyzed as there certainly was friction within the systems during testing. To enact this change within ANSYS, the friction at the roller was considered to be infinite and act as a pinned support. Additionally, the young's modulus was changed as many potential errors affected the effective stiffness of the stiffness. This change was enacted relatively easily as the young's modulus was altered to act as a decline or increase in the stiffness of the system. When analyzing these two changes, it was found that reducing the young's modulus by 28% to 50 GPa resulted in a maximum displacement downward of 2.5864 mm. This is nearly a 40% increase in the displacement from the original model. Including friction in the model resulted in the maximum displacement being 1.1809 mm. This is a 36.5% decrease in displacement from the original model. With the change in the young's modulus creating the largest increase in the deflection, it is used to find the safety factor for the truss. Assuming the maximum variation of the young's modulus is 28%, the deflection found that the young's modulus can be used to solve for the factor of safety. Taking the original ANSYS deflection, 1.8607 mm, and the maximum deflection with the decreased young's modulus, 2.5864 mm, the factor of safety can be found by dividing the change in young's modulus deflection by the experimental deflection. When this calculation is computed, the factor of safety was found to be 1.39. It was considered to use the experimental data instead of the original ANSYS deflection to calculate the factor of safety but in the end, this method was deemed to be an incorrect analysis as the ANSYS and experimental systems have many differences in their models. The factor of safety calculated was deemed reasonable as many other aerospace projects have factors of safeties of around 1.5.

II. Conclusion

In this lab, we created a Finite Element Method simulation of a truss structure and compared its load-displacement characteristics to the measured characteristics of an actual structure using a provided data set. However, this required several assumptions to be made, including uniform ideal behavior of the joints and bars themselves, zero frictional forces on an end support of the truss to avoid simulating unnecessary axial stresses, and fixed locations of the bars around the joints. These assumptions led to an 18% difference between the displacement of the actual truss, which was 1.5494 mm, and the ANSYS model, which had a 1.8607 mm displacement. This is significant enough that experimental data is still valuable, but small enough to make the Finite Element Method a very useful tool to predict the magnitude of displacement a truss will experience.

References

[1] Department of Aerospace Engineering Sciences. (2020, Fall). ASEN 3113 lab 2 description. Boulder, Colorado: University of Colorado at Boulder. Retrieved from class website at `https://canvas.colorado.edu/courses/65628/files/23672956?module_item_id=2328281`

[2] Department of Aerospace Engineering Sciences. (2020, Fall). ASEN 3113 lab 2 description. Boulder, Colorado: University of Colorado at Boulder. Retrieved from class website at `https://canvas.colorado.edu/courses/65628/files/23672960?module_item_id=2328346`

Appendix A: Team Member Contributions

Name	Plan	Model	Experiment	Results	Code	Report	C factor
Garrett Bell	1	1	0	1	0	1	100
Finn Bailis	1	1	0	1	0	1	100
Christopher Krebs	1	1	0	1	1	1	100
Robert MacFarland	0	0	0	0	0	1	100
Natalie Link	1	0	0	1	1	1	100
Cole MacPherson	1	1	0	1	0	1	100

Appendix B: MATLAB Code

```matlab
%% Natalie Link
% Christopher Krebs, Robert Macfarland, Cole MacPherson, Finley Bailis
% ASEN 3112
% Lab 2 Experimental Data and Plots
% Created 10/9/20

clear
clc
close all

% Read in data
data = readtable('ASEN3112_Truss_FA2020');

% Extract Columns
load_case = data.Var1(:);
load_case = load_case .* 4.44822;

F0 = data.Var2(:);
F0 = F0 .* 4.44822; % convert lbf to N

F1 = data.Var3(:);
F1 = F1 .* 4.44822;

F2 = data.Var4(:);
F2 = F2 .* 4.44822;

F3D = data.Var5(:);
F3D = F3D .* 4.44822;

LVDT = data.Var6(:);
LVDT = LVDT .* 25.4; % in to mm

% Plot load cases
test0 = polyfit(F0,LVDT,1);
line0 = (test0(1) * F0) + test0(2);

figure
hold on
plot(F0,LVDT,'r.');
plot(F0,LVDT,'g')
plot(F0,line0,'b')
xlabel('F_0 (N)')
ylabel('Midspan Deflection (mm)')
title('LVDT Deflection vs F_0')


test1 = polyfit(F1,LVDT,1);
line1 = (test1(1) * F1) + test1(2);

figure
hold on
plot(F1,LVDT,'r.');
plot(F1,LVDT,'g')
plot(F1,line1,'b')
xlabel('F_1 (N)')
ylabel('Midspan Deflection (mm)')
title('LVDT Deflection vs F_1')


test2 = polyfit(F2,LVDT,1);
line2 = (test2(1) * F2) + test2(2);

figure
hold on
plot(F2,LVDT,'r.');
```

```
plot(F2,LVDT,'g')
plot(F2,line2,'b')
xlabel('F_2 (N)')
ylabel('Midspan Deflection (mm)')
title('LVDT Deflection vs F_2')


test3 = polyfit(F3D,LVDT,1);
line3 = (test3(1) * F3D) + test3(2);

figure
hold on
plot(F3D,LVDT,'r.');
plot(F3D,LVDT,'g')
plot(F3D,line3,'b')
xlabel('F_3_D (N)')
ylabel('Midspan Deflection (mm)')
title('LVDT Deflection vs F_{3_D}')


ext0 = polyfit(load_case,F0,1);
line_ext0 = (ext0(1) * load_case) + ext0(2);

figure
hold on
plot(load_case,F0,'r.')
plot(load_case,F0,'g')
plot(load_case,line_ext0,'b')
xlabel('External Load (N)')
ylabel('F_0 (N)')
title('F_0 vs External Load')


ext1 = polyfit(load_case,F1,1);
line_ext1 = (ext1(1) * load_case) + ext1(2);

figure
hold on
plot(load_case,F1,'r.')
plot(load_case,F1,'g')
plot(load_case,line_ext1,'b')
xlabel('External Load (N)')
ylabel('F_1 (N)')
title('F_1 vs External Load')


ext2 = polyfit(load_case,F2,1);
line_ext2 = (ext2(1) * load_case) + ext2(2);

figure
hold on
plot(load_case,F2,'r.')
plot(load_case,F2,'g')
plot(load_case,line_ext2,'b')
xlabel('External Load (N)')
ylabel('F_2 (N)')
title('F_2 vs External Load')


ext3 = polyfit(load_case,F3D,1);
line_ext3 = (ext3(1) * load_case) + ext3(2);

figure
hold on
plot(load_case,F3D,'r.')
plot(load_case,F3D,'g')
plot(load_case,line_ext3,'b')
xlabel('External Load (N)')
```

```
ylabel('F_{3_D} (N)')
title('F_{3_D} vs External Load')


ext_disp = polyfit(load_case,LVDT,1);
line_ext_disp = (ext_disp(1) * load_case) + ext_disp(2);

figure
hold on
plot(load_case,LVDT,'r.')
plot(load_case,LVDT,'g')
plot(load_case,line_ext_disp,'b')
xlabel('External Load (N)')
ylabel('Midspan Deflection (mm)')
title('LVDT Deflection vs External Load')
```