Deflection Project Final Report

Roy Hidalgo, Blair Horning, Andrew Voss, and Dorothy Waskow

Department of Mechanical Engineering

ME41: Engineering Design II

Douglas Matson

November 17, 2021

Executive summary

Method of Determining Deflection	Part 1 Deflection (in)
Castigliano's Theorem	.476
Finite Element Method (original design)	.490
Instron Universal Testing (original design)	.363
Finite Element Method (redo design)	.482
Instron Universal Testing (redo design)	.344

Table 1. Deflection values from all Methods (note: For the Instron Universal testing method the deflection is calculated up until yielding occurred)

Introduction

This project was an application of our knowledge learned in Engineering Design (ME41). We used both an analytical method (strain energy analysis, Castigliano's Theorem) and a numerical method (Finite Element Analysis in SolidWorks).

Initially, we tested our knowledge with an already existing structure. We were given a structure with which to apply Castigliano's Theorem (strain energy analysis) and to test in SolidWorks with a Finite Element Analysis. This gave us the chance to solidify the basic analysis methods before designing our own structure. We compared those results to each other to determine the accuracy of each method. We were looking to confirm that the methods were of the same magnitude and gave similar results, not the same results.

In the second phase of this project we designed our own structure. For this part of the project, we were attempting to make an aluminium structure that had a linear elastic deflection of 0.5" under a vertical load of 200 lbf. The structure had to fit into a 4"x8" area and had designated locations for the holes for the force and simple supports. This gave us the opportunity to apply the analytical methods previously discussed. We did a Finite Element Analysis of our piece in SolidWorks as well as performing strain energy analysis via Castigliano's Method. Our piece was then tested on an Instron device. The analysys, numerical, and analytical results of this project are shown below.

Approach

We ran a Finite Element Analysis on our designed part to validate deflection and yielding prior to cutting with a water jet. The study was set up such that the two lower holes were pinned and the upper right hole had a vertical load of 200 lbf applied downward. The results of the study were then analyzed to determine if our design would meet the desired criteria. Based on results from the FEA study, we adjusted the dimensions of our part to improve performance (i.e. get as close to a spring constant of 400 lbf/in as possible).

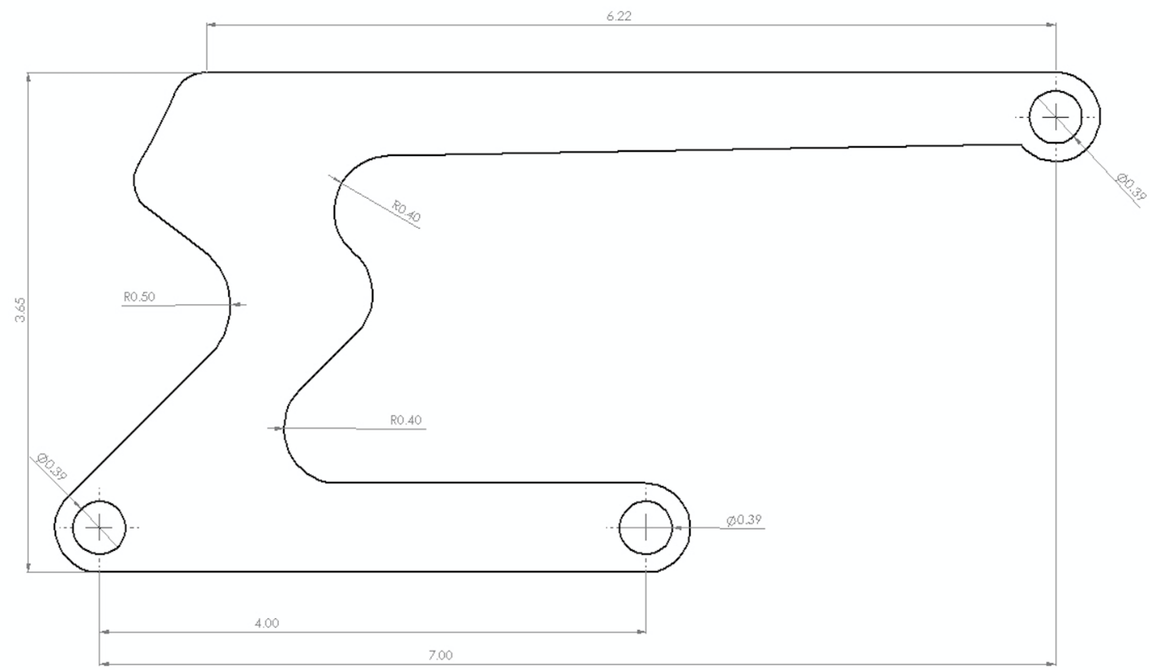

Figure 1: Original design of part. Dimensions are in inches.

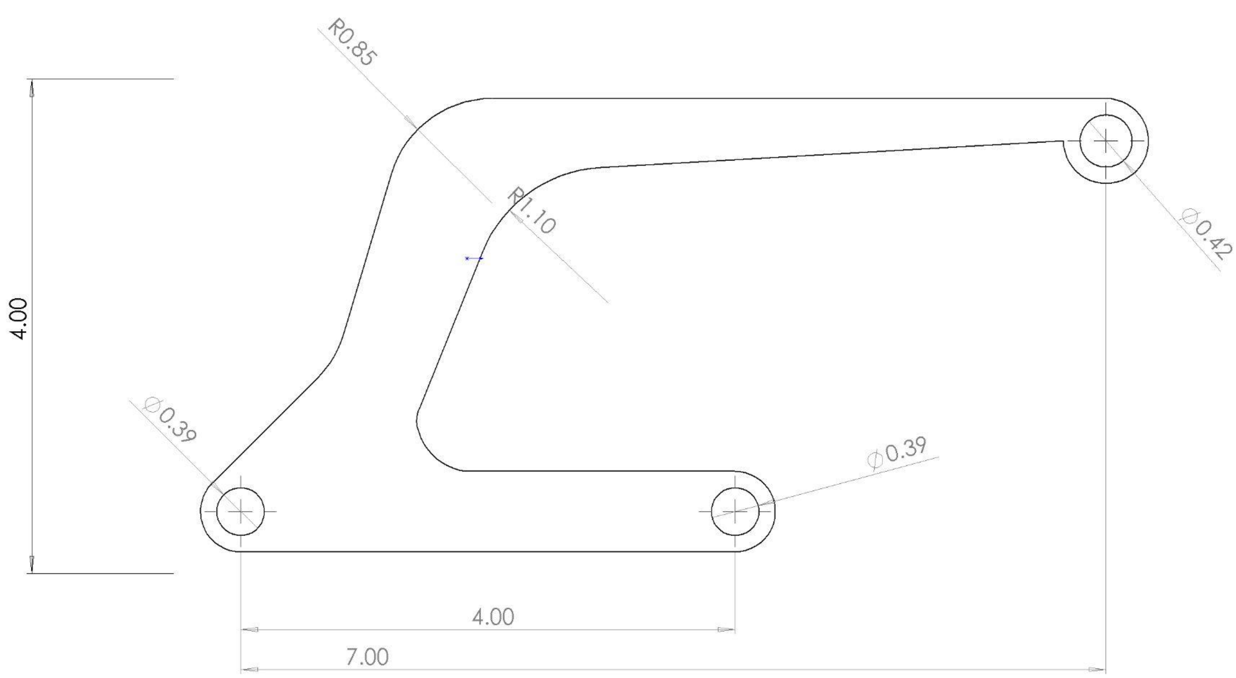

Figure 2: Resubmit of redesigned part. Dimensions are in inches.

The FEA analysis was cross-checked by analytically predicting the deflection of the part with Castigliano's Theorem using a simplified geometry. Doing this allowed us to determine if the results from the FEA study made sense and gave us a quick way to check design changes. Simplifying the geometry saved time and allowed us to rely on the FEA study (which, because the solutions are determined numerically, is generally much more accurate than simplified analytical solutions).

After satisfactorily designing a piece that maximized elastic deformation for the smallest possible mass, the part was water cut from a four by eight piece of aluminum. The manufactured part was pinned in an Instron machine by its two lower holes and force was slowly applied to a cylindrical piece pinned through the upper right hole. Force was applied until the machine either exerted 200 lbf or the test part yielded significantly.

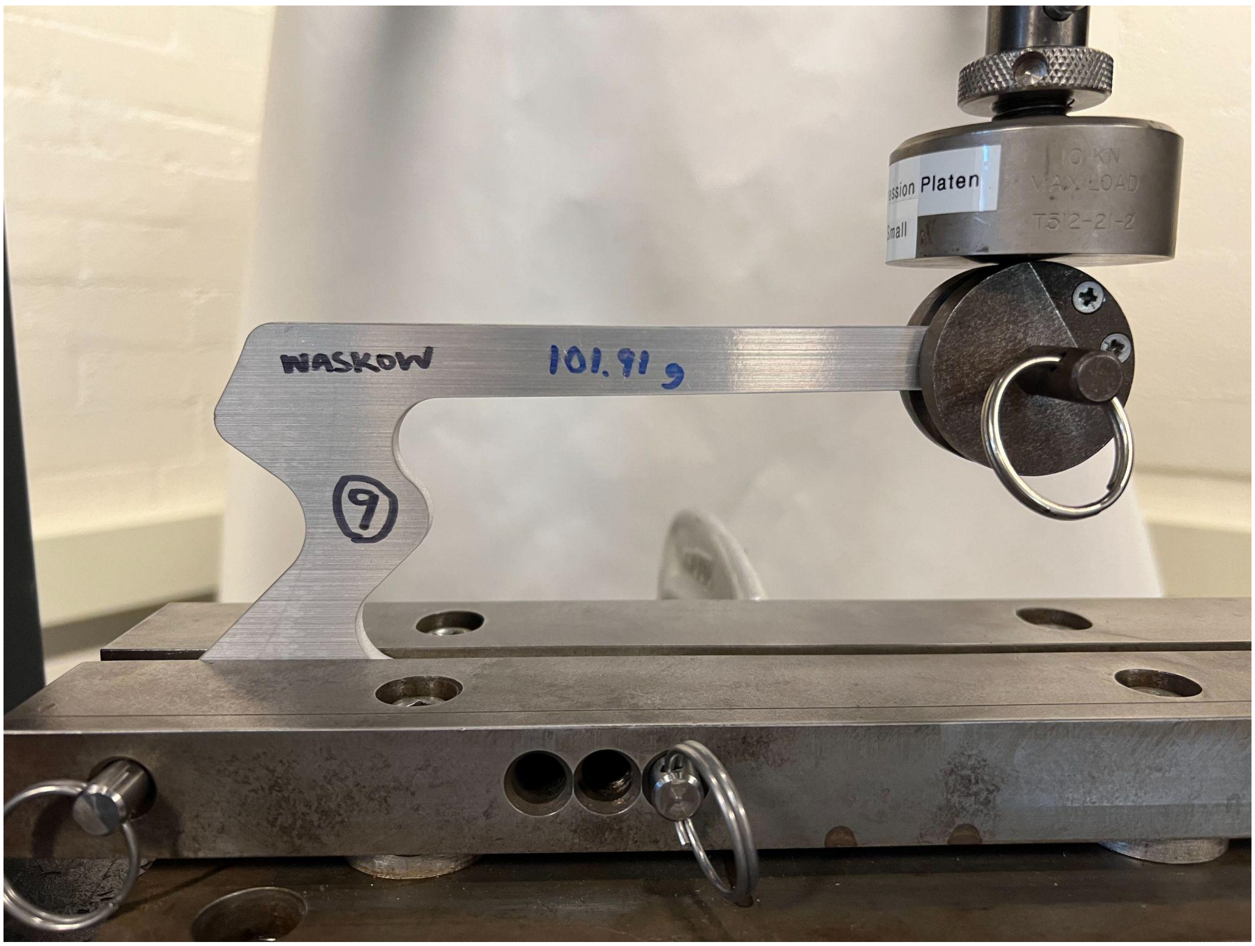

Figure 3: Instron Machine testing setup for first design. Notice the two pins that hold the part which allow rotation (and thus different deflection than if they were clamped.)

Because we did poorly on our first design (significant yielding was observed--more than we predicted in our FEA analysis), we redesigned our piece and ran the deflection analysis with the Instron machine again.

Analysis

This section details the expected outcome of the testing. First, it will cover the SolidWorks analysis then the Castigliano's Theorem. For this analysis, just the first structure will be examined, not the redo structure.

Solidworks Finite Element Analysis

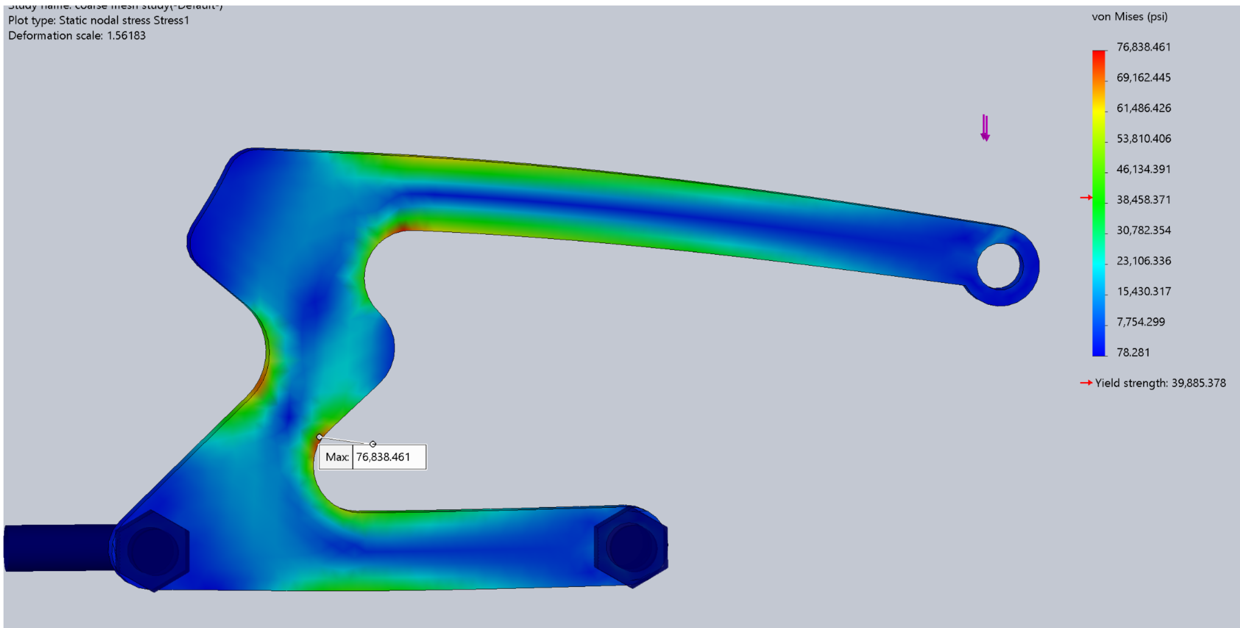

Figure 4: FEA result of von Mises stress. Notice how the maximum stress exceeds the yield strength, which suggests yielding will occur.

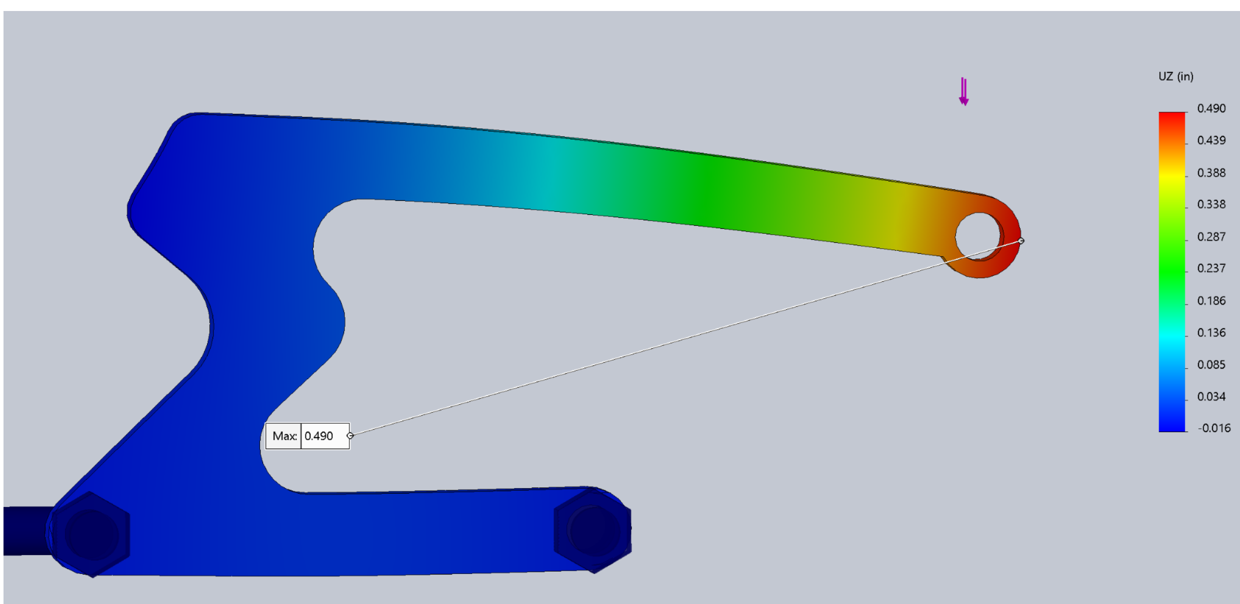

Figure 5: The FEA prediction of maximum deflection. This value is likely lower than what we will see in testing because the FEA also predicts yielding.

The Solidworks FEA analysis of our final design demonstrated that it would deflect 0.490 inches. However, this datum was overshadowed by the fact that the analysis also predicted that our design would yield. This suggests that the deflection analysis of the FEA does not take into account yielding when calculating deflection (which can be seen in the testing results). To get accurate deflection data, one should then be sure that the part they are analyzing does not experience stresses that exceed the yield strength of the material.

Mesh Analysis

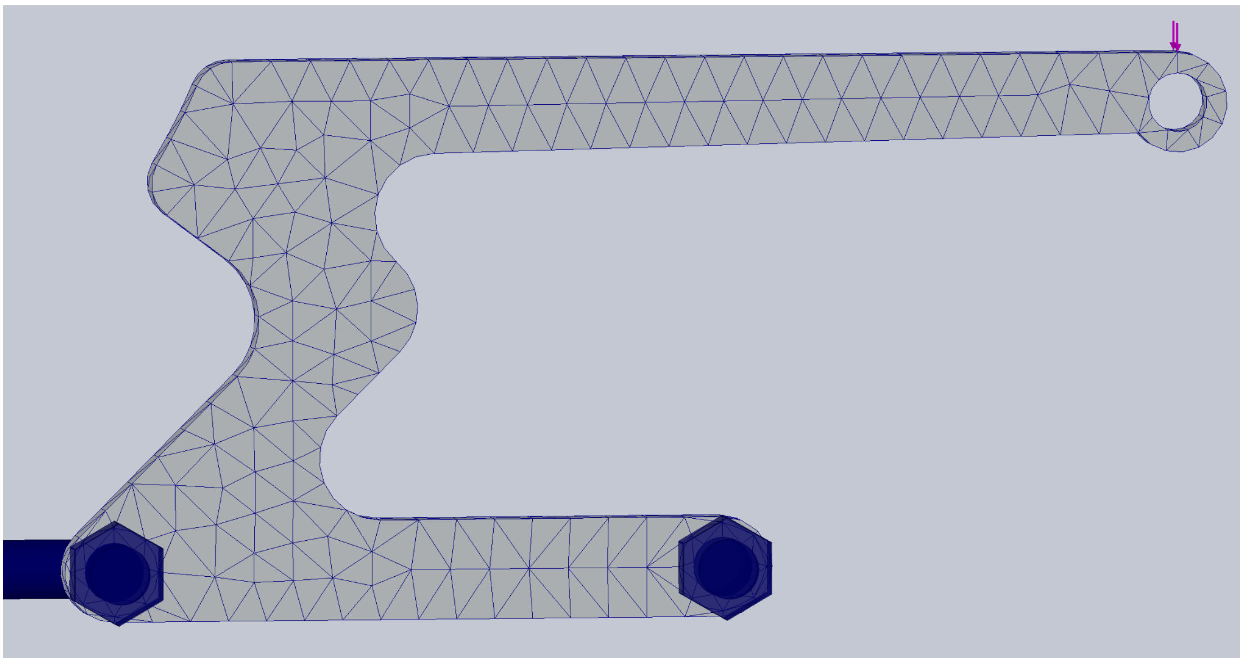

Figure 6: Coarser Mesh

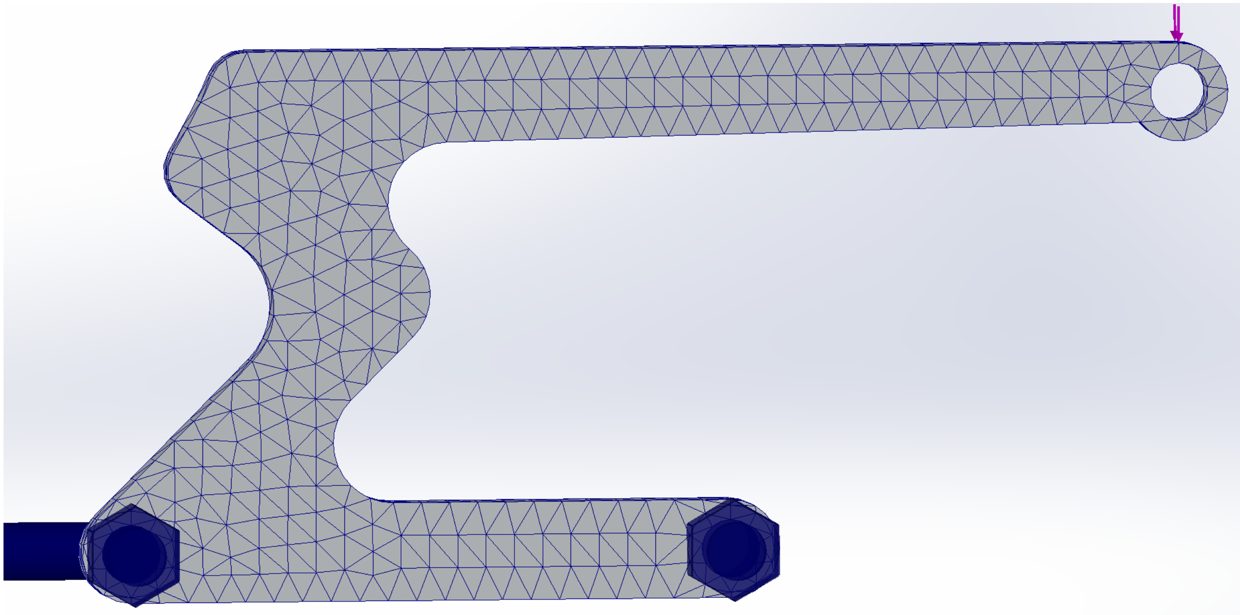

Figure 7: Coarse Mesh

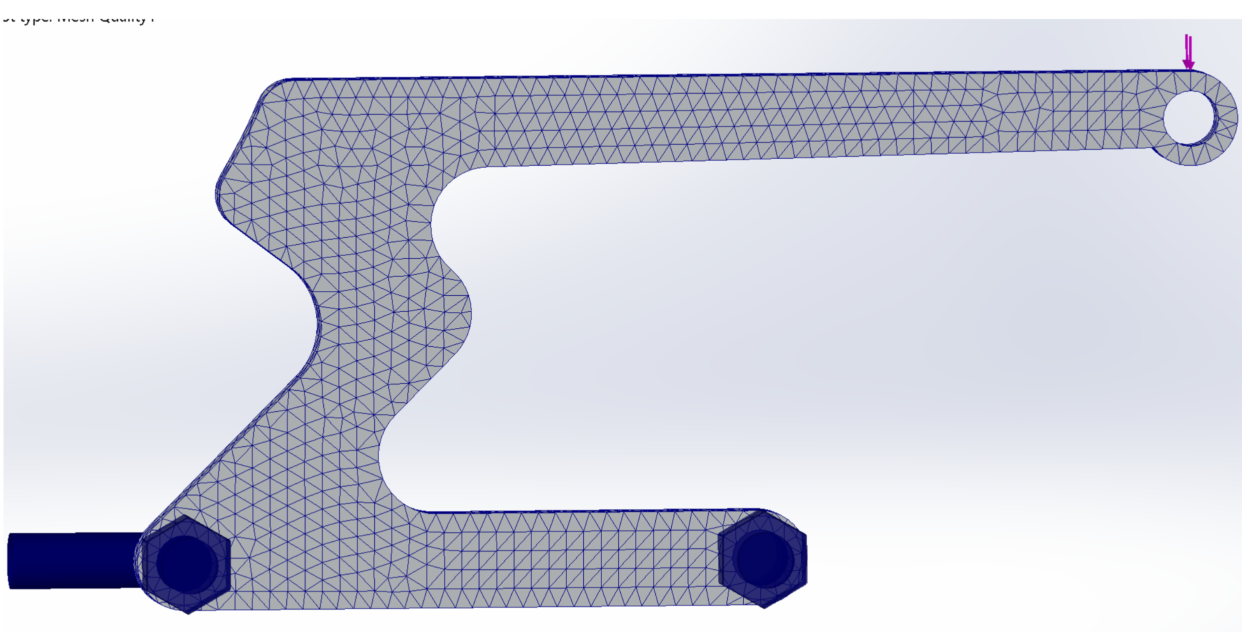

Figure 8: Fine Mesh

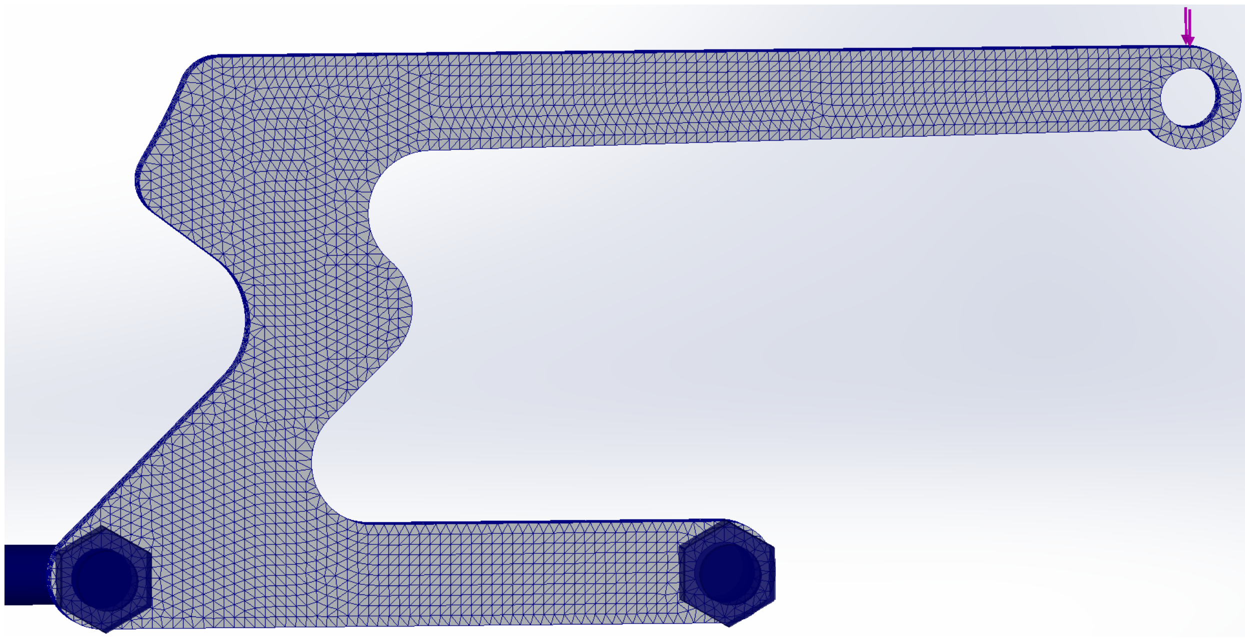

Figure 9: Finer Mesh

	Coarser Mesh	Coarse Mesh	Fine Mesh	Finer Mesh
Element Size (in)	0.265188	0.192261	0.119335	0.0662971
Total Elements	1313	3227	10774	56386
Total Nodes	2793	6028	18430	87885
Maximum Deflection (in)	0.488	0.490	0.490	0.489
Maximum Stress (von Mises, kpsi)	77.15	76.84	80.04	82.36

Table 2: Various mesh data

As one can see, increasing mesh size doesn't inherently improve the accuracy of the FEA model. Both the coarse and fine mesh give the same maximum deflection and even the coarser mesh varies from those values by only two-thousandths of an inch.

The most significant difference in results is the maximum stress, which basically increases with the number of nodes (and therefore also the number of elements). However, this discrepancy is unimportant since all meshes show that our design will experience yielding. Calculation time also increased significantly with finer meshes which demonstrated the cost of trying to get more accurate results with “better” meshes. The results we got from the coarsest mesh, which took about two minutes to run, are not significantly different from the results of the finer mesh, which took over an hour and a half to run.

Failure Load Prediction

We went back to the model and made an attempt to predict where we thought the model would fail, as we knew there were stresses on it and were unsure how long it would last. Since running the model at 200 lbf already indicated that the model was going to fail, we started this test at 190 lbf and went down by 10 lbf until the part did not fail.

Force (lbf)	Maximum Stress (psi)	Deflection (in)	Is stress < yield strength?
190	77,937	0.466	no
180	73,835	0.441	no
170	69,733	0.417	no
160	65,631	0.392	no
150	61,529	0.368	no
140	57,427	0.343	no
130	53,325	0.319	no
120	49, 223	0.294	no
110	45,121	0.270	no
100	41,019	0.245	no
90	36,917	0.221	yes

Table 3: This is a table of the maximum stress and deflection against the force applied to the model. The last column indicates if the part will fail based on the stress and yield strength.

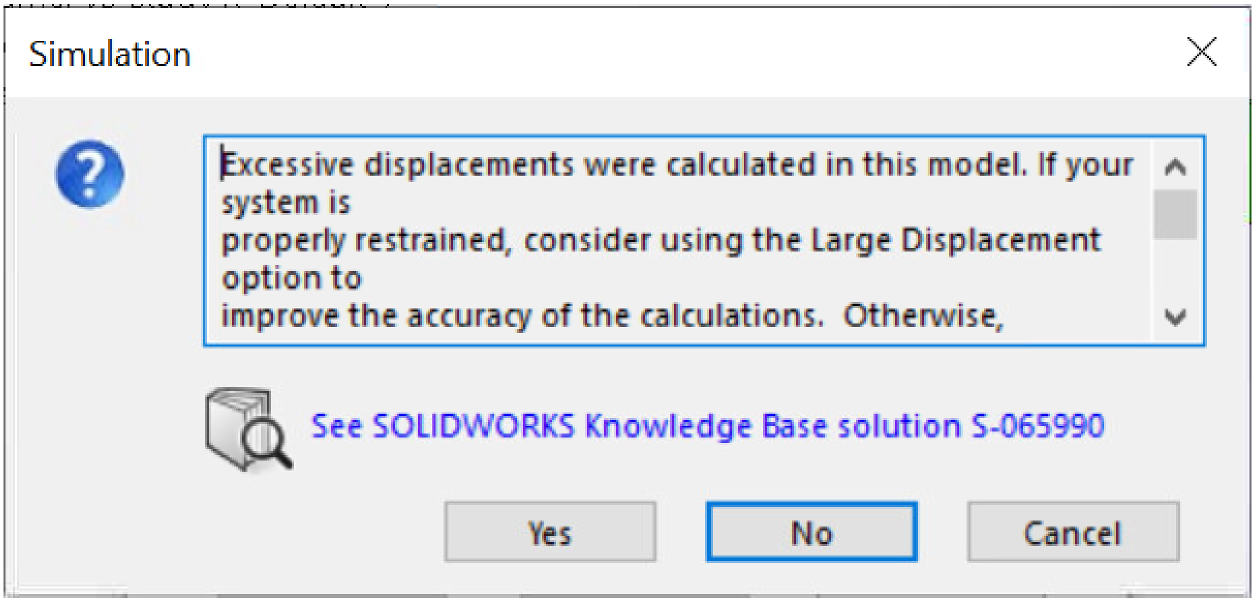

Figure 10: This is an image of the error message that appears when we run our SolidWorks model.

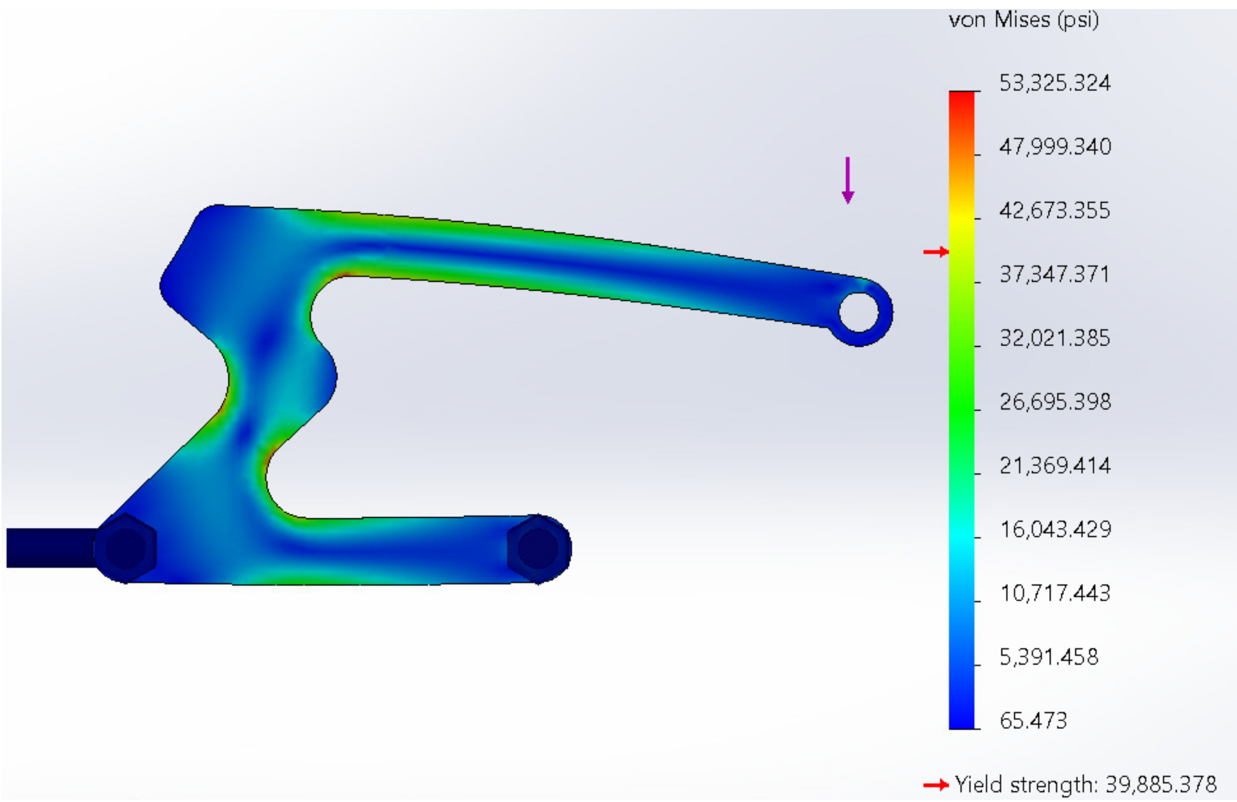

Figure 11: This is the stress model at 130 psi.

As seen in table 3, according to SolidWorks, our part will yield at 90 lbf. Everytime we ran the simulation at all of those amounts, there was the error shown in figure 10. We were wondering how large the stress concentration needs to be for the part to fail. As can be seen in figure 11, the stress concentrations at a force of 130 lbf and lower are very small. While they are greater than the yield strength, we predict this will not cause the part to fail due to the size of the concentration.

Castigliano's Theorem: Strain Energy Analysis

We choose to simplify our structure as a C beam, ignoring the non-linear vertical section. Doing so allowed us to do simple calculations that would verify that we had set up our FEA analysis correctly.

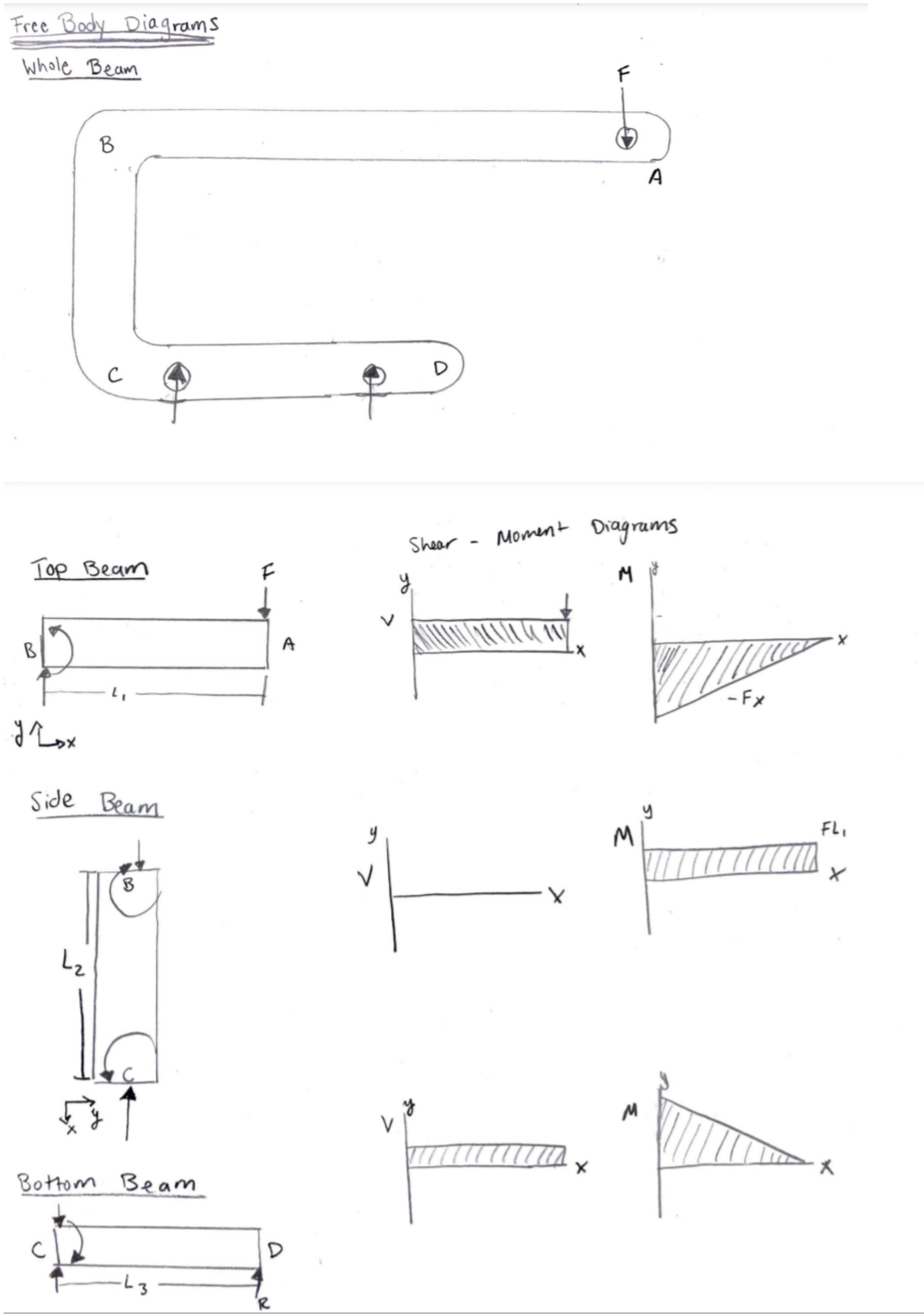

Figure 12: Free body diagrams of simplified structure

For beam 1 (Top Beam):

Moment equation found from graph.

$$\delta = \int_0^{L_1} \frac{M\frac{dM}{dF}}{EI}dx$$

$$= \int_0^{L_1} \frac{(-Fx)(-x)}{EI}dx$$

$$= \frac{F}{EI}\int_0^{L_1} x^2 dx$$

$$= \frac{F}{EI}\frac{x^3}{3}\Big|_0^{L_1} = \frac{F}{EI}\frac{L_1^{\ 3}}{3}$$

For beam 2 (Side Beam):

Moment equation found from graph.

$$\delta = \int_0^{L_2} \frac{M\frac{dM}{dF}}{EI}dx$$

$$= \frac{1}{EI}\int_0^{L_2} (FL_1)(L_1)dx$$

$$= \frac{FL_1^{\ 2}}{EI}x\Big|_0^{L_2} = \frac{FL_1^{\ 2}L_2}{EI}$$

For beam 3 (Bottom Beam):

Find the Moment Equation:

$$\sum M = RL_3 - FL_1 = 0$$

$$R = FL_1/L_3$$

Solve Castigliano's Theorem

$$\delta = \int_0^{L_3} \frac{M\frac{dM}{dF}}{EI}dx$$

$$= \int_0^{L_3} \frac{(\frac{FL_1}{L_3}x)(\frac{L_1}{L_3})x}{EI}dx$$

$$= \frac{F(\frac{L_1}{L_3})^2}{EI}\frac{x^3}{3}\Big|_0^{L_3} = \frac{FL_1^{\ 2}L_3}{3EI}$$

MATLAB Code

This is the MATLAB code that was used to solve Castigliano's equation.

```
% ME41 Project 1
% Castigliano's and Superposition of the C shape
% 11/17/21

clear
close all

%currently set up for medium lengths.
```

Top Beam

```
L1 = (5.17+7.01)/2; %inches
F = 200; %lbf
E = 10^7; %psi, modulus of elasticity
I = 0.005721;

deflectionTop = (F/(E*I))*((L1^3)/3);
```

Side Beam

```
L2 = (2.39+3.65)/2; %in

deflectionSide = (F*L1^2*L2)/(E*I);
```

Bottom Beam

```
L3 = (2.94+4.63)/2;

deflectionBottom = (F/(E*I*3))*(L1^2*L3);
```

Superposition

```
totalDeflection = deflectionTop + deflectionSide + deflectionBottom;
disp(totalDeflection)
```

```
    0.8183
```

Published with MATLAB® R2021a

Results

The results of the Castigliano's vary depending on where we take the measurements for the beams. Here are the results with varying lengths. These short and long lengths were all measured on our SolidWorks model. The short length is the length of the interior edge of the model. The long length is the length of the outer edge of our model. The middle length was calculated to be the midpoint between the short and long lengths.

	Top Beam Length (in)	**Vertical Beam Length (in)**	**Bottom Beam Length (in)**	**Deflection Result (in)**
Short	5.17	2.39	2.94	0.476
Middle	6.09	3.02	3.79	0.818
Long	7.01	3.65	4.63	1.294

Table 4: Measurements used in Castigliano calculations

From these results, the short length is clearly the best approximation for the beam as it is the closest to the SolidWorks. This will be discussed in the comparison.

Comparison

Our Castigliano analysis confirms what our FEA model predicts. While it is a simplified shape of what our actual part was, it is still quite accurate. The difference in predicted deflection is only off by 0.014 inches (less than three percent). Both of these predictions gave us deflections very close to the desired deflection of 0.5 inches. However, Solidworks importantly showed us predicted stress concentrations that would be subject to yielding. Castigliano assumes that the structure will not have internal forces that exceed the yield strength. While the Solidworks simulation likewise ignores forces that will induce yielding, it does show where there are stress concentrations that exceed the yield strength of the material in question.

Testing Results

The two designs we created, our original and redesign, were put through tests using an Instron machine to show the deflection of our designs under a load. The machine placed a vertical force of 200 lbf on our physical models. These models were made using high pressure waterjet-cutting to cut aluminum (Al 6061-T6511) into our designs. The aluminum had a modulus of elasticity of 10×10^{6} psi, with a yield strength of 46.50 ksi. After the testing was complete, the Instron machine produced data consisting of the designs yielding along with it’s calculated spring constant.

Design	Spring Constant (lbf/in)	Amount of Yielding	Load at Which Yielding Started
Original	424.84	Moderate	154.14 lbf
Redesign	436.213	Moderate	150.646 lbf

Table 5: Instron machine test results

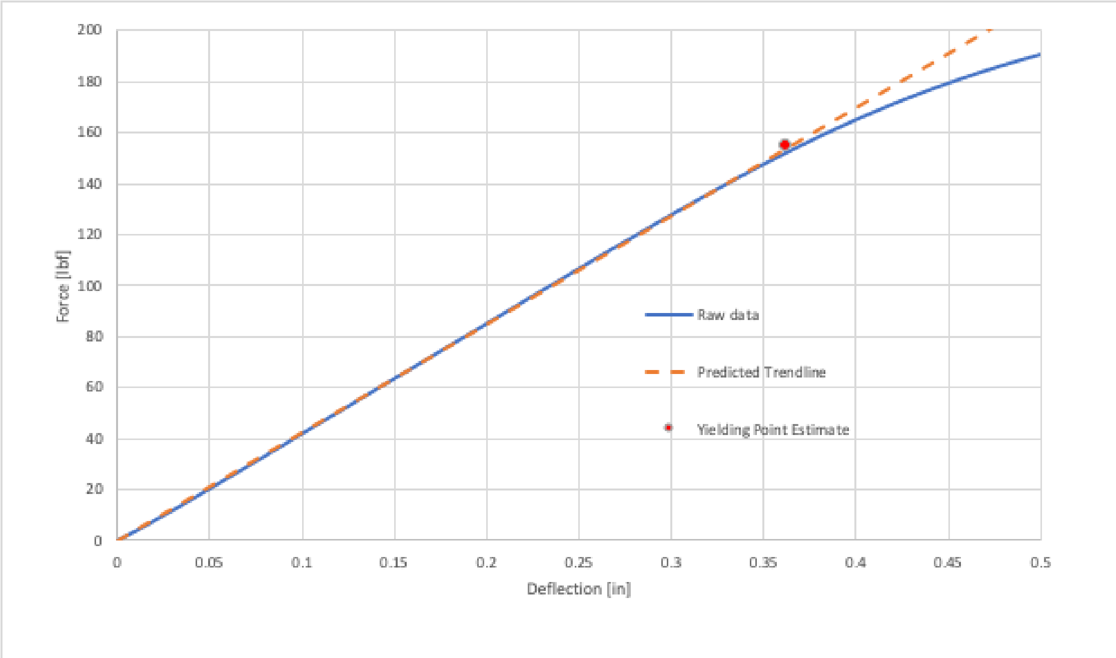

Figure 13: This plot illustrates the spring constant ratio of the compressive load against the compressive deflection that occured when our original design was in the Instron machine. The compressive load caused our original design to yield at about 154 lbf.

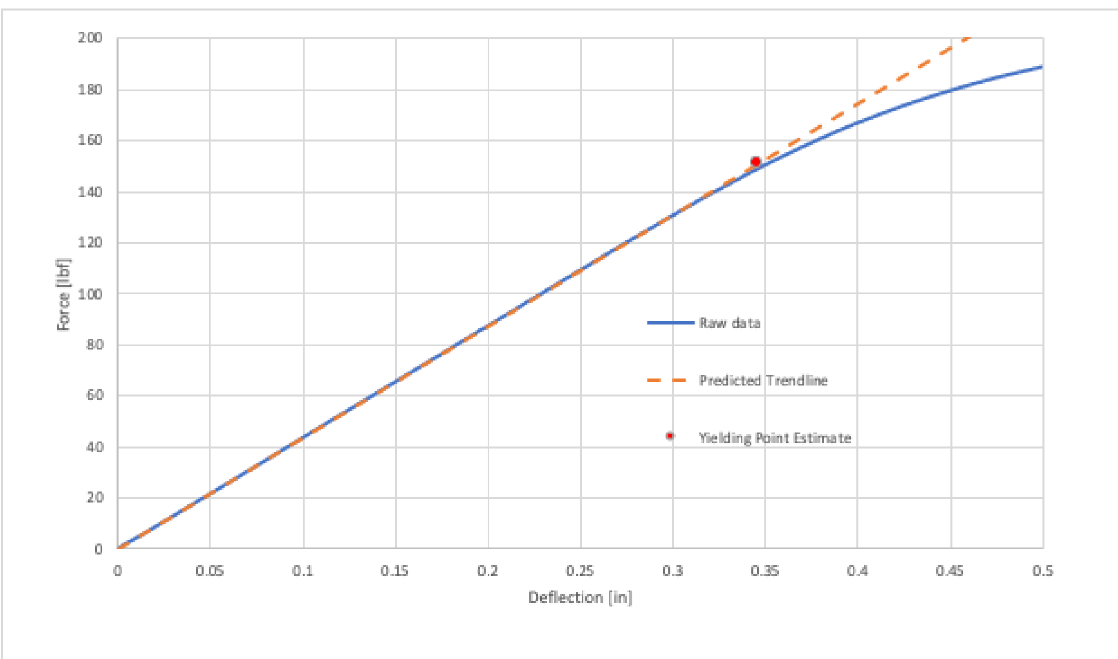

Figure 14: The plot above shows the spring constant ratio of the compressive load against the compressive deflection that occurred during the trial of our redo of our original design in the Instron machine. Yielding under the compressive load began at around 150 lbf.

Error Analysis

There was not a large difference between our original design model and the redo-design model we constructed after the first trial. As you can see on the figures about, we incorporated a curved aspect between the upper and lower pieces. We believed that having curvature on the portion of the model would allow for it to withstand more load and yield a larger amount. After the Instron trial, we took a different approach on the redo-design, where we instead had an angled, straight portion that connected the bottom segment to the top.

Our original design in the Instron machine produced a spring constant value of 424.84 lbf/in and yielded moderately at a compressive load value of 154.14 lbf. The second structure we put in the Instron machine produced a spring constant value 436.21 lbf/in and yielded moderately at a compressive load value of 150.64 lbf. The ideal values for these two numbers would be a spring constant of 400 lbf/in and a load value of 200 lbf. Using these, we were able to calculate the percent error of our designs.

$$\%_{error} = \left|\frac{analytical\ value - numerical\ value}{numerical\ value}\right| \times 100$$

Design	Spring Constant	Spring Constant Error	Compressive Load	Compressive Load Error
Original	424.84 lbf/in	6.2%	154.14 lbf	23%
Redesign	436.21 lbf/in	9.1%	150.64 lbf	24.7%

Table 6: Error results

Original design structure showed a 6.2% error for the spring constant value, and a 23% error for the compressive load at which it yielded. The redesign structure showed a 9.1% error in the spring constant value, and a 24.7% error for the compressive load at which it yielded. The error in both of the designs are similar, but overall the redesign of our original model performed worse.

Post-Testing SolidWorks Analysis

Fixtures and Load Conditions

After submitting both our parts with the same fixtures and force, it came to our attention that we had done both of those things incorrectly in the study. This is one factor that may have contributed to our incorrect model. The SolidWorks model was redone for the second structure. However, as can be seen in the figure below (Figure 13), the calculated displacement is still good, 0.482 inches. The original predicted deflection of the second structure was 0.494 inches. This is much closer to the desired value than the value that was tested, so this condition cannot be the only reason our model was not what we predicted it to be.

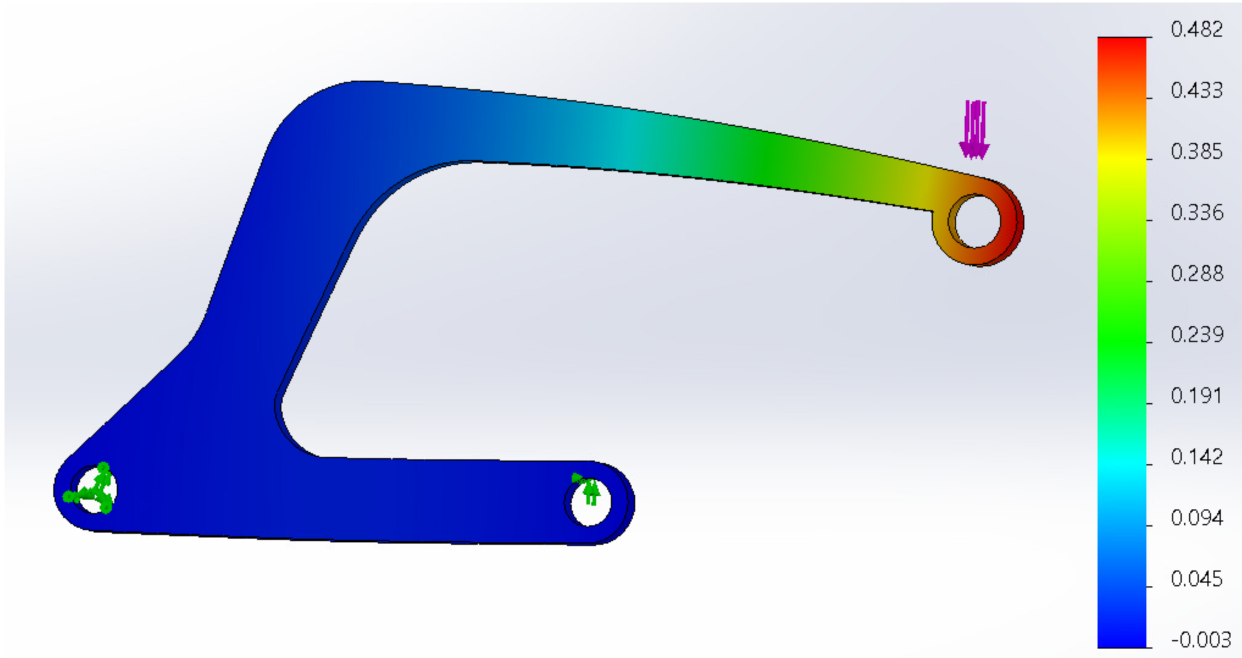

Figure 15: This image shows the displacement of the second structure with correct model conditions.

Discussion

Castigliano's analysis showed our original design would deflect .476 inches while the FEA predicted our model would deflect .480 in, and when tested the model deflected .363 inches before it began to yield. For our redesign, FEA predicted it would deflect .482 inches, and when Instron tested the redesign deflected .344 inches before yielding.

Instron testing demonstrated that the original design performed better than the redesign since the spring constant of the original was 424.84 lbf/in and began to yield at 154.14 lbf, while redesign had a spring constant of 436.21 lbf/in and began to yield at 150.64 lbf. In addition, the FEA method determined that the max load the original design could handle before it would experience yielding is 120 pounds.

From the mesh analysis it was determined the size of the mesh was not a significant factor in the accuracy of the FEA since all mesh sizes provided the same relative outcomes. Specifically the coarse and fine mesh yielded the same deflection results and values from coarser and finer mesh did not deviate enough for the results to be important.

This project dealt heavily with optimization for multiple things. We were trying to optimize for a desired deflection, without yielding, and minimize weight. This was a struggle for us as every time we made the structure thicker in areas of high stress, the deflection would go down. We eventually made the trade-off to have higher than preferred stress in order to have the desired deflection (according to the FEA analysis). This resulted in our structure yielding plastically before reaching the desired force.

The lesson from utilizing Solidworks FEA to predict deflection is that it will not do so accurately if yielding is also predicted. Another important takeaway from this process is that creating a mesh that one's computer can run in a timely manner is valuable. Fortunately, Solidworks typically does a good job of choosing a mesh that will complete the simulation quickly. It also seems, based on our mesh analysis, that one can choose a coarser mesh to save time with a minimal difference in the solution.

Deviations between our Castigliano, FEA, and Instron testing can be explained for a few reasons. The most significant is that the Solidworks and Castigliano calculations of the predicted deflection ignored the predicted yielding that occurred in parts of our design. While SolidWorks also calculates yielding, it does not handle yielding in the deflection calculation well. This describes why our part deflected much less when tested with the Instron machine than either the simulation or analytical approach predicted. Another factor that affected our results was the speed at which the test was performed. Faster application of force prevents heat that is generated via internal friction from dissipating quickly enough that the temperature of the part is held constant. This would lower the yield strength of our part in these hot spots and therefore increase the measured yielding. Whether this was a factor could be determined had we measured the temperature of the part before and after testing, but, as noted previously, the design itself caused the vast majority of the yielding. Other minor factors that could have caused discrepancies include: the testing setup not exactly aligning with the setup of the FEA analysis (see the Post-Testing SolidWorks Analysis section) and the FEA mesh not accurately lining up with stress concentrations. These likely do not account for the differences seen in our results.