ME42 Machine Design Project 1 Report

Taissa Gladkova, Casey Owen, Elliot Pavlovich

Executive summary

Both structures that we tested were designed to be able to bear a load of 5 lbs while only deflecting 0.1 inches inches in the y direction. Our first structure was designed to be overly conservative. We did not try to make it light, we wanted it to bend and deflect the specified amount, while maintaining its structural integrity.We did this by creating a simple C-bracket. We took a bit more of a risk with the second structure. For this one, we tried to make it as light as possible and created different unique design ideas.

Prior to testing the designs on the Instron machine, we did some finite element analyses (FEA) using SolidWorks for both structures. The FEA for our C-bracket was very close to our Castigliano calculations. Given it was slightly off as our Castigliano calculations did not account for the holes in the material. From Castigliano, we predicted that the deflection would be 0.099 inches while the FEA gave us a deflection value of 0.1013 inch. The final structure with filleted corners had a predicted deflection of 0.09656 inches. The simplified analyses were above the experimental deflection of the piece, 0.0969 inches, and the FEM for the final piece was slightly below.

For the second and more complex structure, we did not do a Castigliano analysis. Instead we made different designs and tested them using SolidWorks FEA. The FEA for our final design gave us a predicted value of 0.09402 inches, which was much less than our experimental value of 0.1320 inches. After the testing we figured out that the possible reasoning for the discrepancy between the FEA and the experimental deflection was most likely due to the fact that SolidWorks FEA does not take into account buckling and therefore yielding (caused by buckling). We assume that our piece buckled due to it having long slender supports.

Though our FEA and experimental deflections were different, an FEA is always a good starting point when one is designing a piece that must withstand high strain.

Introduction

The objective of this assignment was to learn how to apply Castigliano's method to real designs as well as learn how to use SolidWorks' FEA program. Our goal was to design a brace that deflected exactly 0.1 inches when 5 pounds of force was added to a specific point on the structure. We did all of our analyses before testing to see how accurately our analyses were able to predict the real deflection of the structure.

The structure was designed and then laser cut out of clear acrylic and tested on an Instron machine.

Approach

In our design of Structure #1 (Fig 1) we started with the base C- shape given in Phase I of the project and adjusted the width of the bars and changed the filet size until we achieved the desired deflection.

In designing Structure #2 (Fig 2), we used a highly iterative process of creating a base shape resembling a curved triangle (originally with more supports inside the triangle which were cut out pretty early on in the design process to reduce weight) and adjusting the width of the top bars until they were as small as can be within the allowable thickness of 0.1 inch. Further, we adjusted the width of the filets and size of the hole cut-outs in the bottom bar to reduce weight and achieve desired deflection.

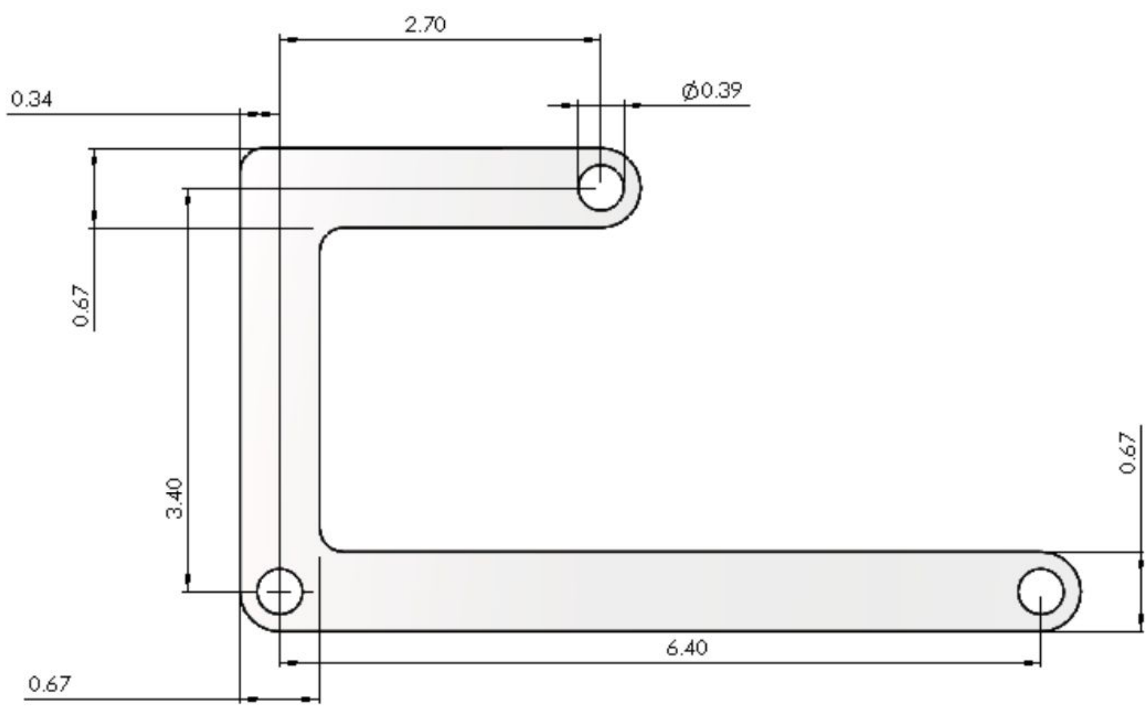


Figure 1: Structure #1 drawing with dimensions (in).

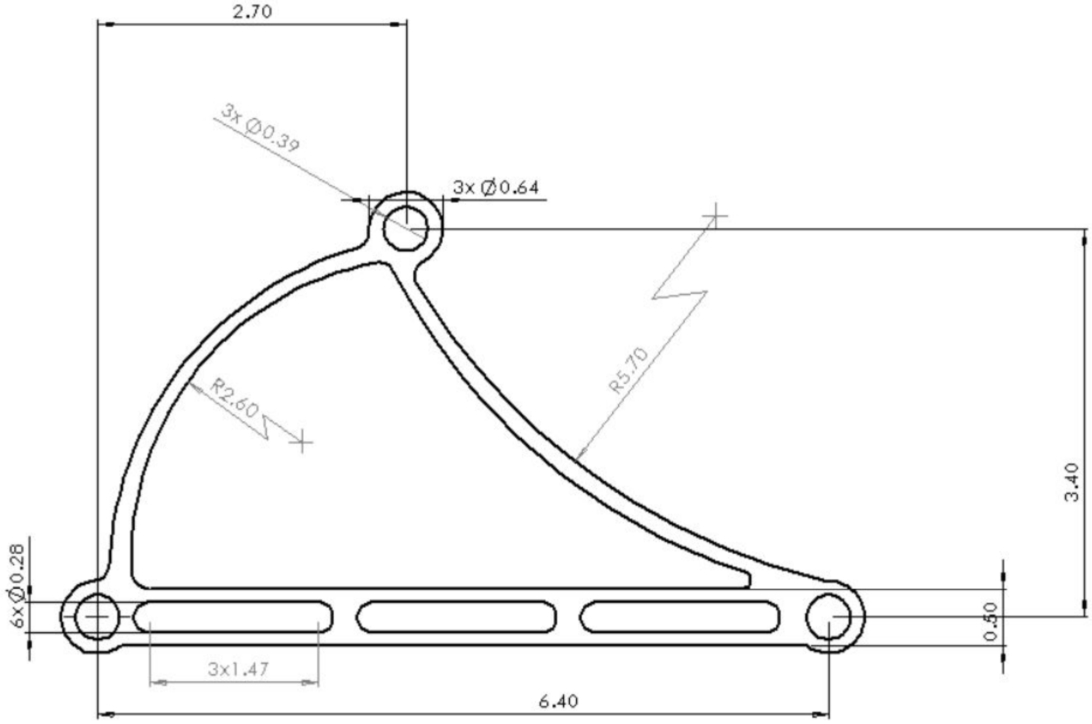


Figure 2: Structure #2 drawing with dimensions (in).

Castigliano Analysis

$F := 5\ lbf$ Properties of acrylic: $E := 379000\ psi$

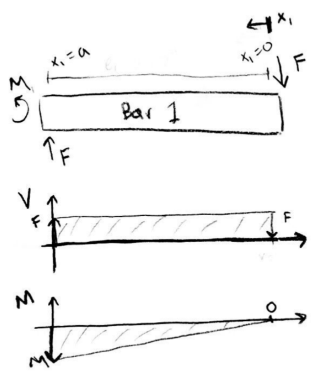
$a := 2.7\ in$ $S_y := 6527\ psi$

$b := 3.4\ in$

$c := 6.4\ in$

$t := 0.25\ in$

Approach: Free Body Diagrams



$\Sigma F := 0$

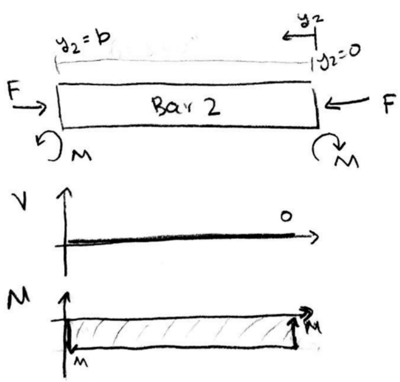
$\Sigma M := 0$

Reaction at Bar 2 = F

M = Fa

$M_1(F, x) := -F \cdot x$

$\frac{d}{dF} M_1(F, x) \rightarrow -x$



$\Sigma F := 0$

$\Sigma M := 0$

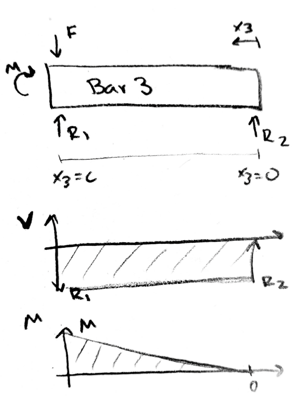
Force F is axial and therefore there is no shear

Compression load = F_2

M = Fa (constant)

$F_2(y, F) := F$ $M_2(F, y) := -F \cdot a$

$\frac{d}{dF} F_2(y, F) \rightarrow 1$ $\frac{d}{dF} M_2(F, y) \rightarrow -2.7 \cdot in$



$\Sigma F := 0$

$\Sigma M := 0$

$R_2 := F \cdot \frac{a}{c} = 2.109\ lbf$

$R_1 := F - R_2 = 2.891\ lbf$

at R1, M = R2 * c = F * a

$M_3(F, x) := F \cdot \frac{a}{c} \cdot x$

$\frac{d}{dF} M_3(F, x) \rightarrow 0.421875 \cdot x$

Analysis

The analysis of the total deflection of the structure at the point where the load is applied is evaluated using Castigliano's Theorem. The two types of loading conditions that need to be considered are bending and compression. Castigliano's equations for these are:

$$\delta_i = \frac{\partial U}{\partial F_i} = \int \frac{1}{EI}\left(M\frac{\partial M}{\partial F_i}\right)dx \qquad \text{(bending)}$$

$$\delta_i = \frac{\partial U}{\partial F_i} = \int \frac{1}{AE}\left(F\frac{\partial F}{\partial F_i}\right)dx \qquad \text{(compression)}$$

$F := 5\ lbf \qquad a := 2.7\ in \qquad b := 3.4\ in \qquad c := 6.4\ in \qquad t := 0.25\ in$

Untapered Bar 1: $h_1 := 0.67\ in$

Bar 1 has length *a* and has a rectangular cross section with height *h1* and thickness *t,* with moment of inertia given by :

$$I_1 := \frac{t \cdot {h_1}^3}{12} = 0.006\ in^4$$

The moment in bar 1 as a function of *x* is given by:

$$M_1(F, x) := -F \cdot x$$

And so, using this moment of inertia and moment in Castigliano's we get a deflection due to bending in bar 1 of:

$$\delta_{1bending} := \int_{0\ in}^{a} \frac{1}{E \cdot I_1} M_1(F,x) \cdot \left(\frac{d}{dF} M_1(F,x)\right) dx = 0.014\ in$$

Bar 2: $h_2 := 0.67\ in$

Bar 2 has length *b* and has a rectangular cross section with height *h2 = h1* and thickness *t,* with moment of inertia given by :

$$I_2 := I_1 = 0.006\ in^4$$

The cross-sectional area of bar 2 is given by:

$$A_2 := t \cdot h_2 = 0.168\ in^2$$

The moment in bar 2 as a function of *y* is given by:

$$M_2(F, y) := -F \cdot a$$

And so, using this moment of inertia and moment in Castigliano's we get a deflection due to bending in bar 2 of:

$$\delta_{2bending} := \int_{0\,in}^{b} \frac{1}{E \cdot I_2} M_2(F,y) \cdot \left(\frac{d}{dF} M_2(F,y)\right) dy = 0.052\ in$$

Further, in bar 2 we need to consider the deflection due to compression, because this bar--unlike the others--has compressive loads, and before calculation it is unkown if this value is negligible. The deflection due to the bending in bar 2 is given by:

$$\delta_{2compression} := \int_{0\,in}^{b} \frac{1}{A_2 \cdot E} F_2(y,F) \left(\frac{d}{dF} F_2(y,F)\right) dy = \left(2.678 \cdot 10^{-4}\right)\ in$$

After calculating this deflection, it is meaningful to note that compared to the deflection due to bending, the deflection caused by the compressive forces is negligible.

Bar 3: $h_3 := 0.67\ in$

Bar 2 has length *c* and has a rectangular cross section with height *h3 = h2 = h1* and thickness *t,* with moment of inertia given by :

$$I_3 := \frac{t \cdot h_3^{\,3}}{12}$$

The moment in bar 3 as a function of *x* is given by:

$$M_3(F,x) := F \cdot \frac{a}{c} \cdot x$$

And so, using this moment of inertia and moment in Castigliano's we get a deflection due to bending in bar 3 of:

$$\delta_{3bending} := \int_{0\,in}^{c} \frac{1}{E \cdot I_3} M_3(F,x) \cdot \left(\frac{d}{dF} M_3(F,x)\right) dx = 0.033\ in$$

Total Deflection

Using superposition, the total deflection (when bar 1 is untapered) of the structure at the point where the force is applied is a sum of all the deflections calculated above:

$$\delta_{total} := \delta_{1bending} + \delta_{2bending} + \delta_{2compression} + \delta_{3bending} = 0.099\ in$$

Analysis: Structure #1

First, in order to check the agreement of Castigliano's and the FEM analysis, we modeled the shape used in the simplified Castigliano equations (no filets, as seen in Fig 3). This structure is based off the original structure given in Phase I, however we adjusted the width of the bars until Castigliano's resulted in a value close to the desired deflection of 0.1 inches. Castigliano's analysis predicted a deflection of 0.099 inches. We chose to probe a node at the center of the bar at the support location to represent the location that Castigliano's was predicting deflection at. The probe result showed a 0.1013 inch deflection. This shows that the FEM analysis is fairly precise with respect to Castigliano's, with an error of 2.3%, especially considering that the mesh could be finer and that the placement of the node could be moved to better approximate the location that Castigliano's is predicting. This means that the FEM analysis can work accurately as a substitute for Castigliano's equations.

Knowing this fact, we move on to create Structure #1 to be used for testing. Trusting that the FEM analysis is a good estimate for shapes too complex to analyze with Castigliano's, we added filets to the original structure (final shape seen in Fig 1) to reduce the stress concentrators at the corners. After running the analysis on this structure and probing at the same node used in the previous structure, this yielded a vertical deflection of 0.09656 inch (Seen in Fig 4), close to the desired deflection of 0.1inch.

Next, we checked to make sure there was an appropriate safety factor for the stresses. As seen in Figure 5, under a Von Mises analysis, the maximum stress shown was 1166 psi, and the yield strength of the acrylic is 6527 psi, giving a safety factor of about 5.5. The max stresses are seen inside the C shape at the corners where the 3 bars meet each other. This is expected, and can be reduced by increasing the radius of the filets to reduce stress concentrators, however this would make the structure stronger and therefore it would not deflect as much as we want it to. Therefore, because the FEM of Structure #1 predicted a deflection close to the desired deflection and did not show plastic deformation with a safety factor of about 5.5, we decided that this was a good margin and settled on the final version of Structure #1.

For the mesh of the structure, we kept the SolidWorks default medium mesh because of the fairly simple geometry and as seen in Figure 6, the given mesh seems to do a good job of modeling the structure. Additionally, we tried adjusting the mesh to see if it did in fact affect the analysis, and the results did not change drastically so we settled on the medium mesh.

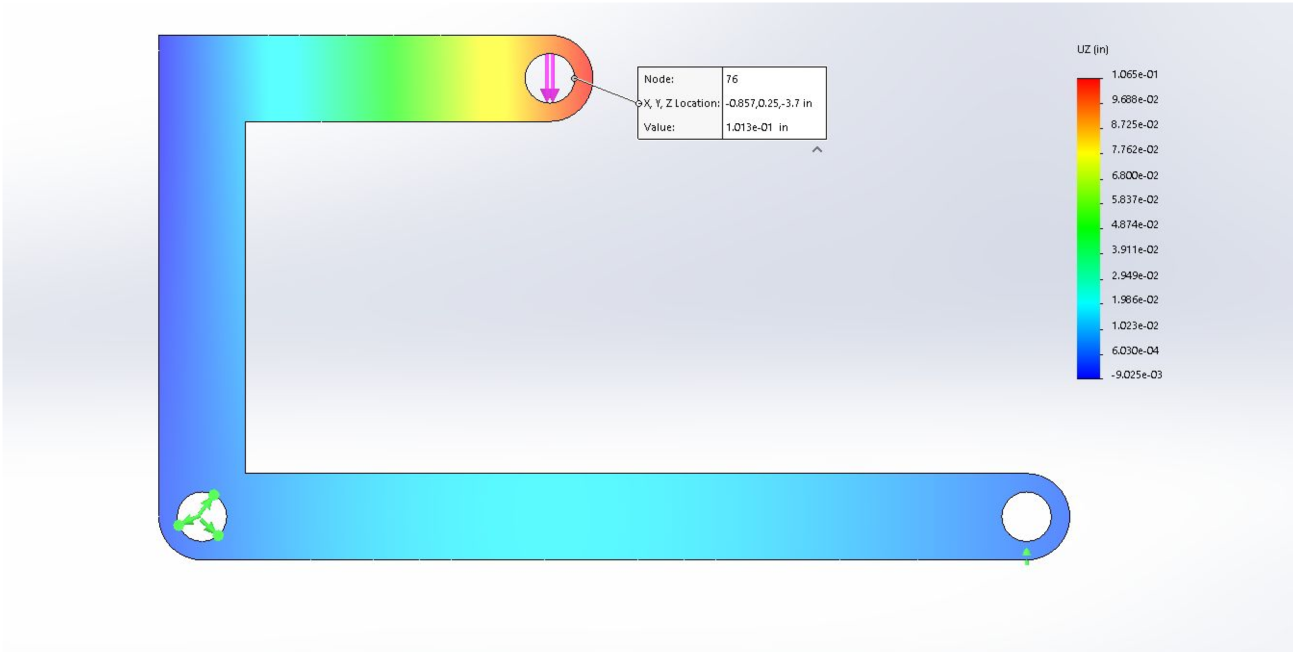


Figure 3: Deflection of Structure #1 with no filets (to compare to Castigliano). Probe at node 76 (X,Y,Z: -0.857,0.25,-3.7 in) value is 1.013e-01 in.

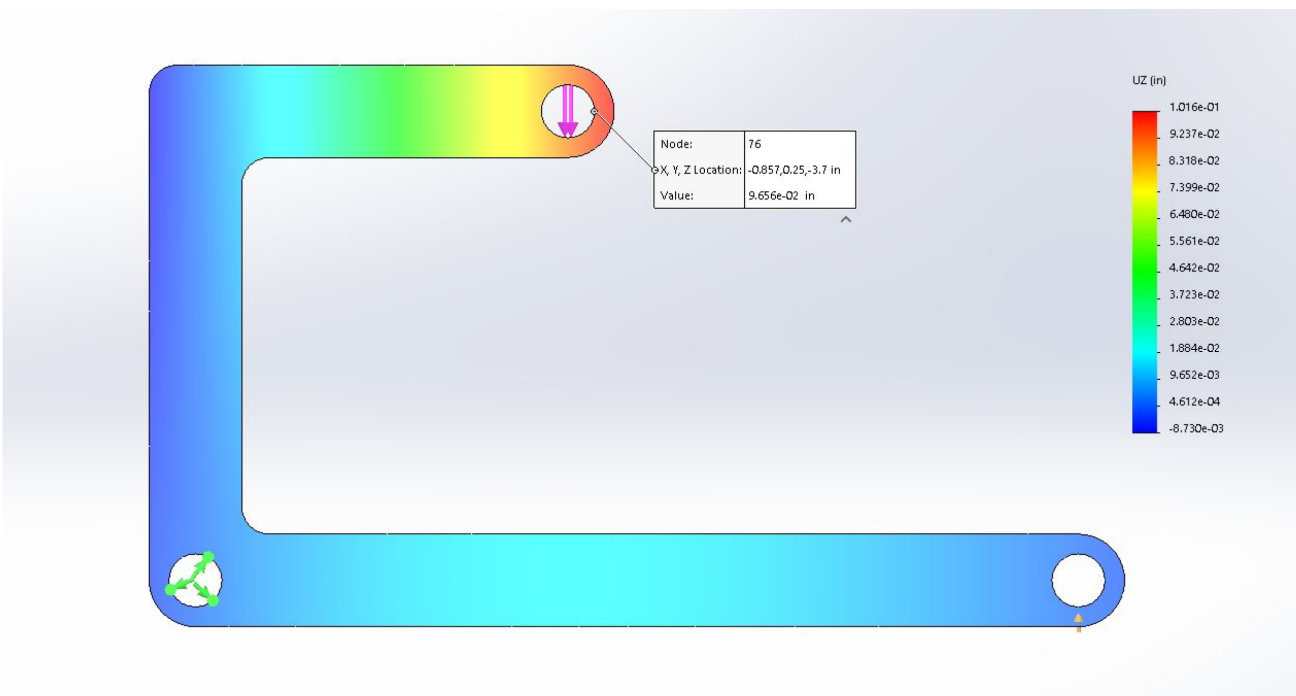


Figure 4: Deflection of Structure #1 with filets (final version used for testing). Probe at node 76 (X,Y,Z: -0.857,0.25,-3.7 in) value is 9.656e-02in.

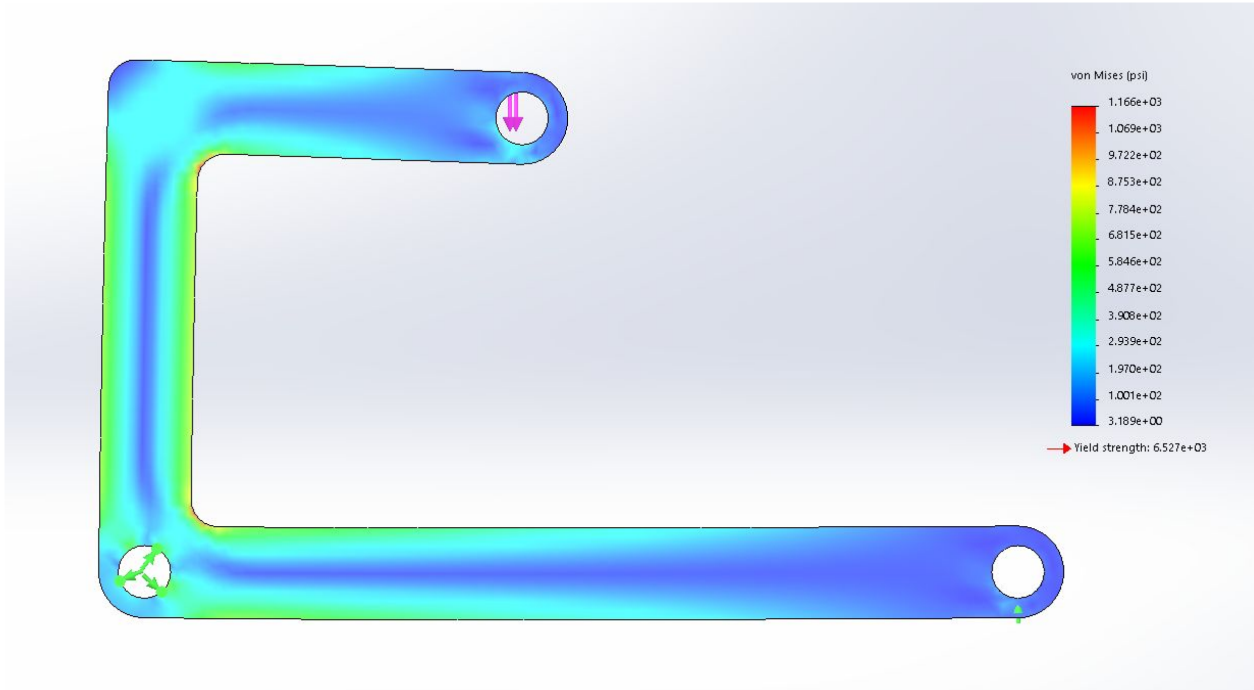


Figure 5: Stresses in Structure #1 with filets. Yield Strength is 6.527e+03 psi, max stress achieved under Von Mises analysis is 1.166e+03 psi.

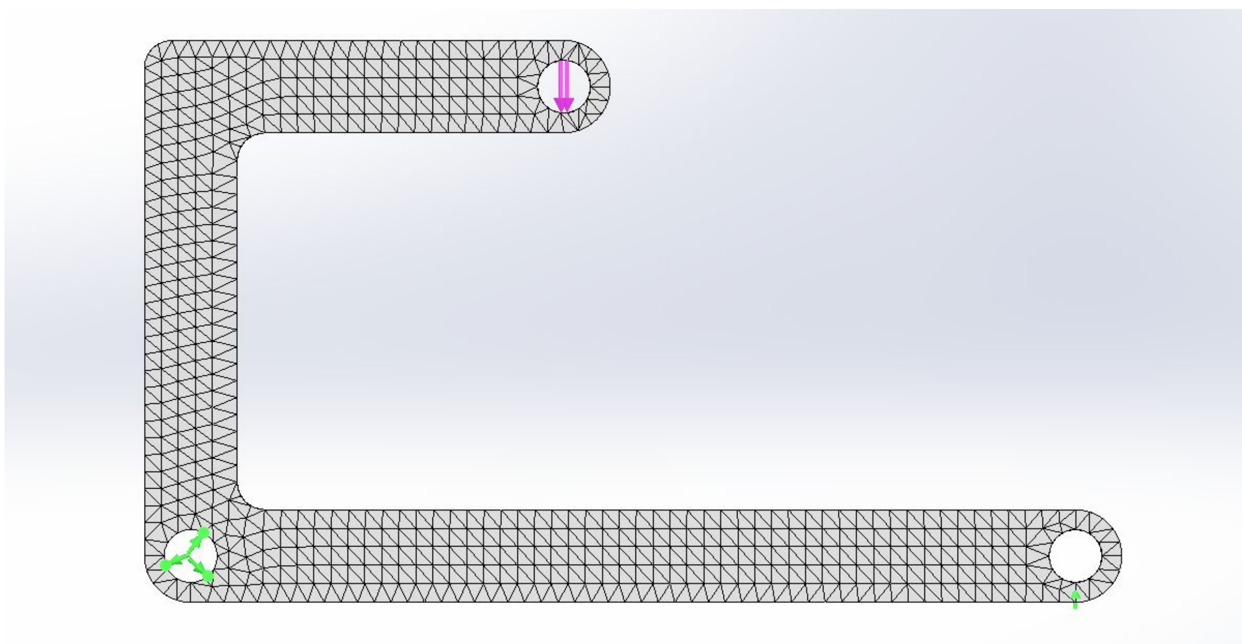


Figure 6: Structure #1 mesh.

Analysis: Structure #2

The main goal of designing the second structure was to take a less conservative approach in order to reduce its weight as much as possible. To do this, a shape similar to a triangle was created with thin and curved edges. This design choice was made because triangles are well known for their ability to withstand large loads due to their prevalence in trusses, and the curved geometry was implemented to avoid buckling. The long, thin members would be very susceptible to buckling and the basic static simulation in Solidworks does not model this well.

Once the basic geometry was decided upon, iterative tests of the displacement were run and material was added or removed as necessary until a displacement value of about .1 inches was achieved. The same node as before was chosen to probe and a displacement of .09402 inches was found. Since the maximum displacement was shown as .1036 in (Fig 7), slightly undershooting the goal displacement was deemed acceptable due to the high variance in expected results. This assumption of high variance was proved correct when this structure overshot the goal displacement with an average spring constant of about 40.6 (Fig 2).

There was no Castigliano's method to apply to this structure given the complicated geometry.

When analyzing the predicted yield of the structure, mesh controls of much finer mesh were applied to the structure at the sharpest corners and the highest stress concentrations: mainly the inside corners of the triangle (Fig 9). The highest stress present in the structure was found to be 3147 psi whereas the yield strength of the acrylic was listed as 6527 psi, a safety factor of about 2 (Fig 8).

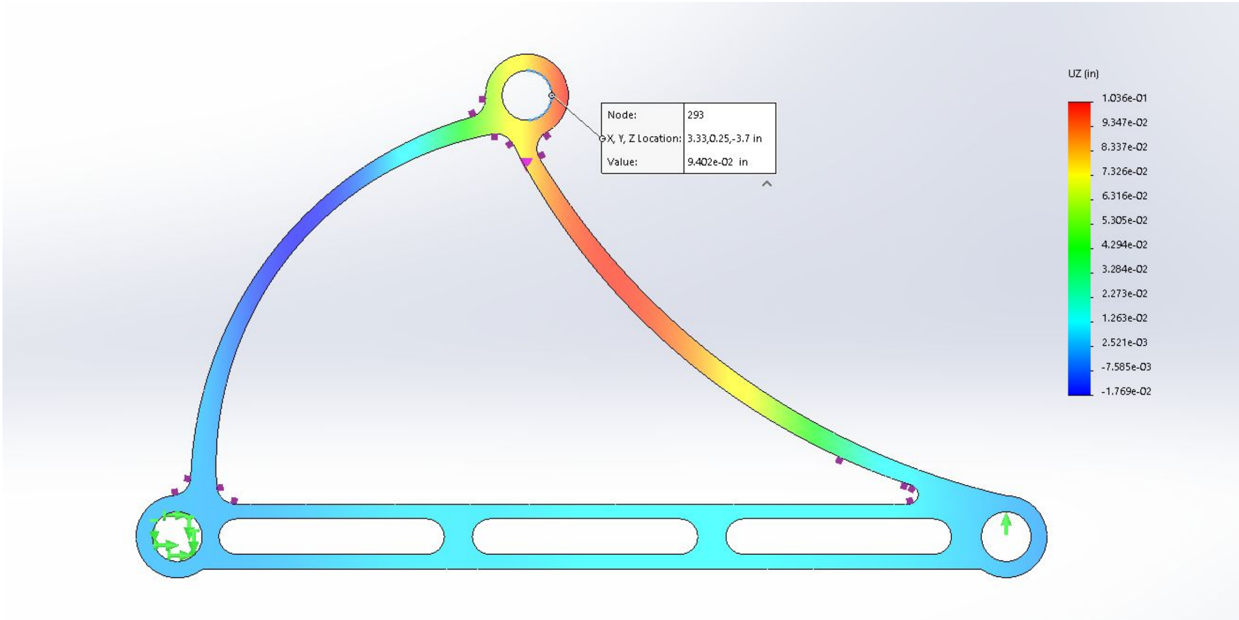


Figure 7: Deflection of Structure #2. Probe at node 293 (X,Y,Z: 3.33,0.25,-3.7 in) value is 9.402e-02in.

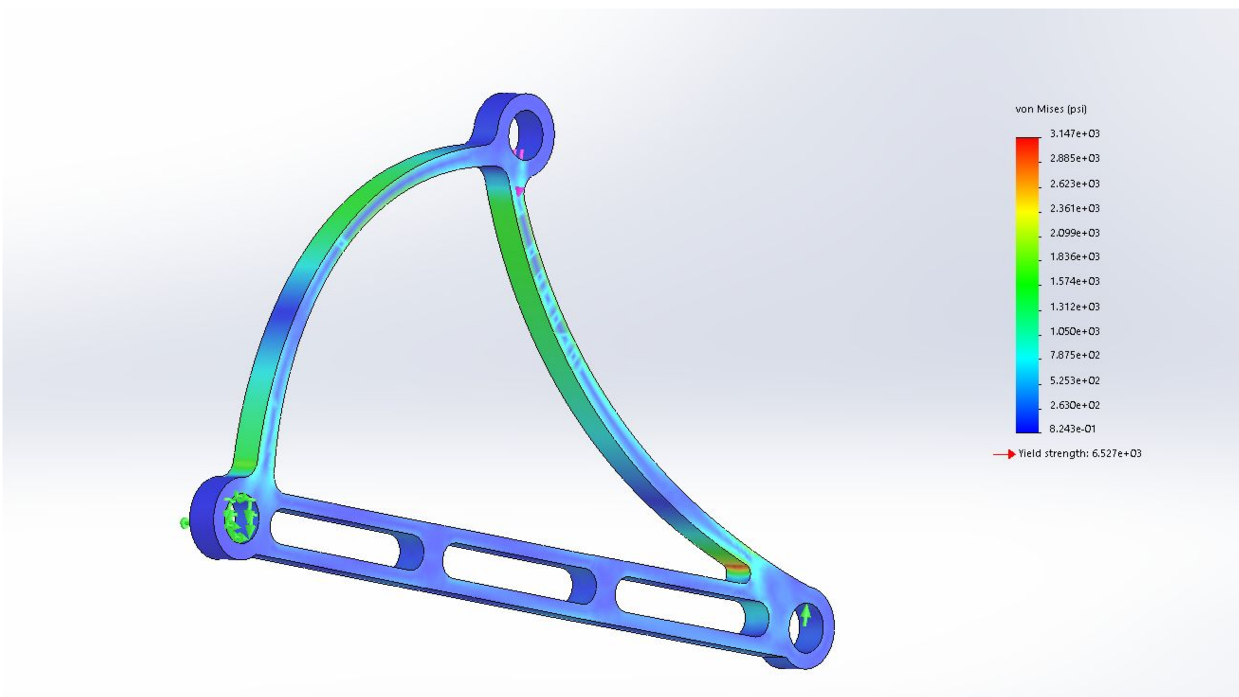


Figure 8: Stresses in Structure #2. Yield Strength is 6.527e+03 psi, max stress achieved under Von Mises analysis is 3.147e+03 psi.

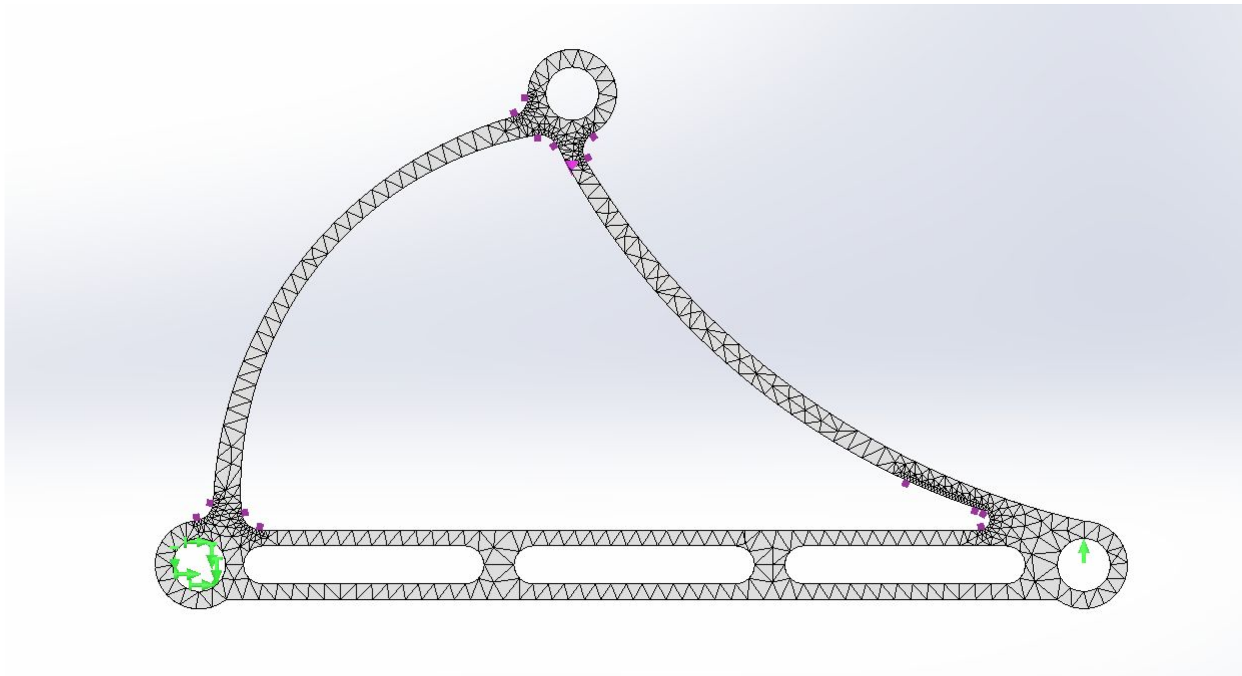
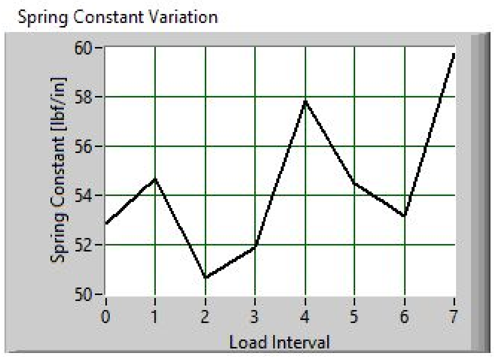


Figure 9: Structure #2 mesh.

Testing Results

Our more conservative structure (Structure #1) did not yield as weight was added to the structure. According to the labview analysis provided by Professor Liesk our average spring constant across the middle of the loading was 54.4 psi. This was only slightly above the ideal value of 50 psi, supporting the claim that our structure did not result in plastic deformation. The zoomed in graph of load vs spring constant for the middle of the loading cycle (0.5 lb to 4.5 lb) also shows that the deformation was elastic (Fig 10). Though it may not seem it, because of the autoscaling of the axes, the graph is linear, meaning that the deformation was not plastic. Meaning, the range of the plot goes from 50 to 60 lbf/in, while the plot of the second structure ranged from 32.5 to 50, so although at first glance appears more linear, the range is bigger so it actually varies much more than the first structure.

Our less conservative structure (Structure #2) on the other hand did end up deforming plastically when a load of 5 lb was added. Its spring constant was 40 psi which was far below the ideal value. The middle loading vs spring constant graph (Fig 11) for this structure was not as linear as the more conservative piece showing that it ended up yielding and deforming plastically.



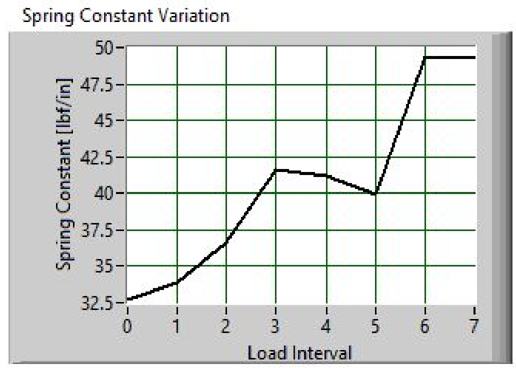


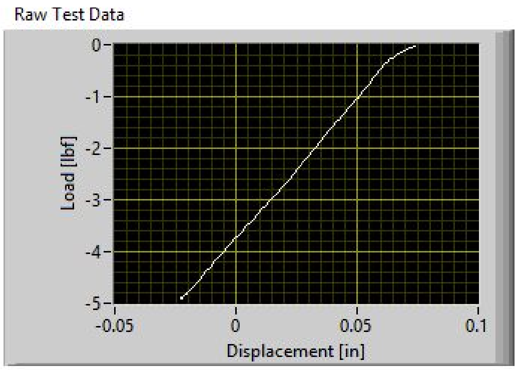
Figure 10 (left): Zoomed in graph showing the spring constant of Structure #1 when load was between 0.5 and 4.5 pounds.
Figure 11 (right): Zoomed in graph showing the spring constant of Structure #2 when load was between 0.5 and 4.5 pounds. All spring constants below 50 lbf/in showing that it was plastically deforming.

As one can see from the graphs (Fig 12 and Fig 13), both structures deflected close to the desired amount of 0.1 inches. The initial measurements from the extensometer was not exactly zero, but after accounting for the offset, it was determined that Structure #1 deflected to a max of 0.0969 inches while Structure #2 deflected 0.1320 inches.

One curious thing that we noticed when combing through the data was that for Structure #1 the Instron machine loaded the pieces to 4.92 lbs, but then began unloading. One would expect that when the machine began unloading, the deflection of the acrylic piece would lessen. However, this was not the case. The piece continued to deflect getting to a max of 0.0969 inches when unloaded to 4.88 lbs after being loaded to 4.92 lbs.

Structure #2 on the other hand was continuously loaded to 4.970 lbs, and achieved the max deflection of 0.1320 inches at the max load.

The main observable difference between the conservative structure and non conservative structure is that the non conservative structure's stress strain curve is slightly concave while the stress strain curve for the conservative structure is linear. The bowing in of Structure #2's stress-strain graph shows how it plastically deformed while being tested. It started linear, and then the slope changed once it yielded and entered the plastic deformation region.



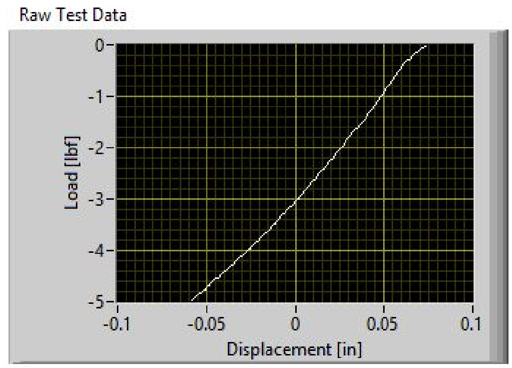


Figure 12 (left): Stress-strain graph for Structure #1 when loaded from 0 to 5 lbs
Figure 13 (right): Stress-strain graph for Structure #2 when loaded from 0 to 5 lbs

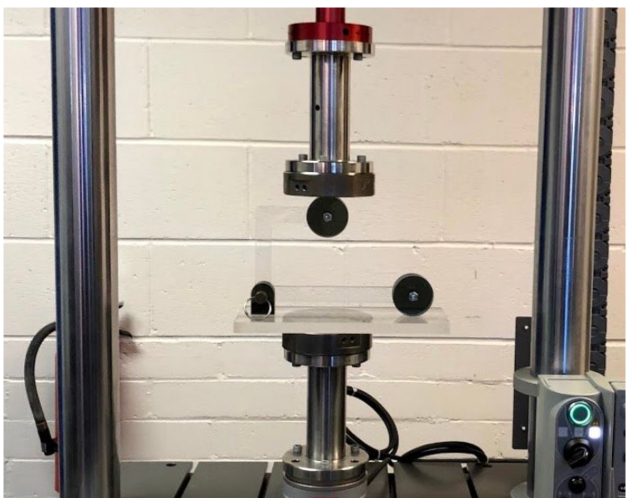



Figure 14 (left): Structure #1 in Instron Machine ready for testing
Figure 15 (right): Structure #2 in Instron Machine ready for testing

Discussion

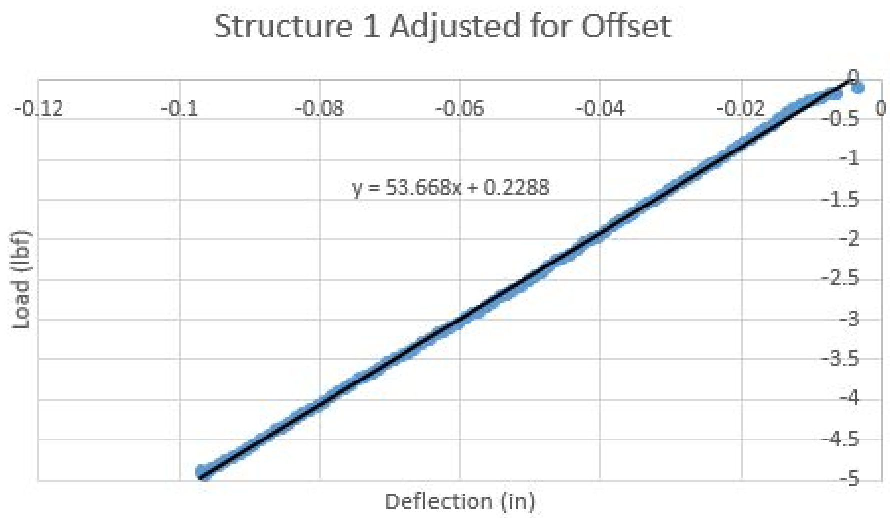


Figure 16: Structure #1 with zeroed start point and a linear interpolation of the data Data adjusted for offset by subtracting the initial deflection from every point

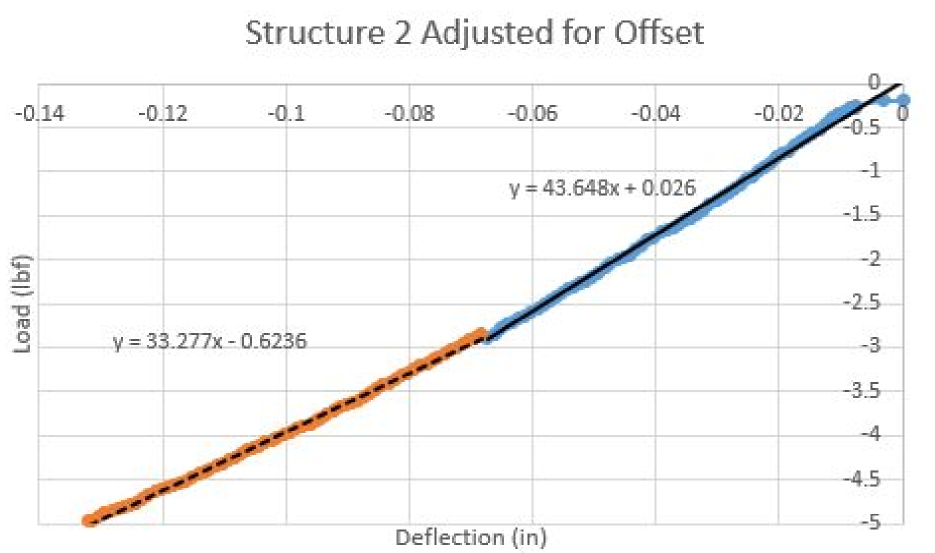


Figure 17: Structure #2 with zeroed start point and a piecewise linear interpolation Data adjusted for offset by subtracting the initial deflection from every point

The second structure underwent yielding, especially during the region of the higher force being applied, making this plot non-linear. It can be approximated as two linear sections of the plot, both before (blue) and after (orange) yielding. By inspection of the break in linearity, the yielding occurred just before 3 lbf of force was applied, and thus represents the break of the two linear fits. The orange line is not linear, but a linear trendline was fitted to determine the rough relationship between the slopes of the regions. It is clear from this that the structure was weaker after yielding (spring constant ~33 lbf/in) than before (spring constant ~43 lbf).

The average spring constant given by Professor Liesk's analysis for Structure #1 (54.4034 lbf/in) differed slightly from our analytical results (53.668 lbf/in, Fig 16) because the given value was calculated neglecting each of the first and last 0.5 lbf applied. Similarly, the calculated average spring constant for Structure # 2 (38.835 lbf/in, not shown) deviated from the given (40.5687 lbf/in) for the same reason.

For the first structure, the results from the testing agree with the Castigliano's calculations as well as the FEM analysis. Castigliano's predicted a deflection of 0.099 inches, while FEM predicted 0.095616 inches, and the results show a max deflection of .0969 inches. However, since the test did not reach exactly 5 lbs of load, we did a linear interpolation using the slope given in Fig 16 (53.668 lbf/in) and a reference point of the max load and max deflection (.0968 inch, 4.9256 lbf). This gave us a deflection of .0983 inches when loaded to exactly 5 lbs. This interpolation is likely accurate because it is not interpolating out a significant distance, so results are likely consistent with what was shows in the test. The FEM analysis shows a 2.7% error in modeling the results of the test (compared to the linearly interpolated results for 5 lbs).

Notice that for both Professor Leisk's results of 54.4034 lbf/in, and our analytical results of 53.668 lbf/in, both of these spring constants are above the ideal value of 50 lbf/in. This means that our structure was slightly too stiff and did not deflect enough.

The second structure, on the other hand, deflected too much. Castigliano's analysis was not performed for this structure given its complex geometry, but the FEM analysis predicted 0.0942 inch. The maximum deflection shown in the Instron test was 4.970 lbs, and achieved the max deflection of 0.1320 inches at the max load of 4.970 lbs. We then performed another linear interpolation at 5 lbs using the slope of 43.648 lbf/in from the linear section of the graph (Fig 17) a reference point in the initial section of the graph (blue point in Fig 17) (0.01554 in, 0.5157 lbf). This gave us 0.11828 inches as a prediction for deflection of the structure at 5 lbs of load if the structure did not yield and continued to deflect linearly like in the first section of the results. Here, the FEM analysis had an error of 28.6% , with slight variation due to the fact that the deflection is not given at exactly 5 lbs.

As stated in the results section this high error is due to the fact that the structure yielded, deforming plastically which allowed for more deflection than was originally accounted for. It is likely that it yielded because of buckling which is not accounted for in the static test runs in SolidWorks, one must run a separate analysis to accurately predict this behavior. It would be interesting to inspect the unloading data to see yielding, unfortunately we do not have this data. This would more accurately show if the structure started to plastically deform as the difference in deflection from when it was loading initially as to when it was being unloading would be greater if the piece had gone through plastic deformation.

Even interpolated data showed higher deflection (0.11828 inches) than FEM (0.0942 inch). It's closer than actual results (0.1320 inches), but still far from the FEM prediction. This may be due to error in selecting the point where the plot stops being linear; selecting a point at a lower load, effectively reducing the portion of the points that we are saying behave linearly, would result in a steeper slope of the blue section (since at smaller loads the line appears steeper), and a more accurate linear interpolation of a smaller deflection.

In the design for Structure 2, in order to get the weight of the structure down we sacrificed the stability of the supports, making them about as thin as the specifications allowed. However, the results show that the supports were *too* thin. The slenderness of the supports led to them buckling, so meaning we were to aggressive in attempting to reduce the weight. The supports should have been designed slightly thicker in order to not buckle and deform plastically. Along with modifications to other parts of the structure, this would increase the deflection to 0.1 inches while maintaining this integrity.