Introduction:

For this project, we were tasked with designing an Aluminum structure, and learn how to apply the strain energy method and perform Finite Element Analysis on it. Finite Element Analysis in SolidWorks helps determine how a structure is going to behave under stress. Our main goal in the design process was to have our structure deflect the desired maximum deflection, with little to no yielding as possible. We were first required to design our structure and then use FEA analysis in SolidWorks Simulation to see if our desired values were achieved. Once the structure was perfected, it was then fabricated using a water jet cutter and tested on an Instron testing device.

Our design requirements for this project were that the structure needed to be designed on a 4” x 8” x 0.25” thick, extruded Al 6061-T6511 with a modulus of elasticity of 10 x 10^6 psi and yield strength of 46.50 ksi. This structure also required three support holes which allowed the pins to be inserted which held the structure onto the testing device. At the topmost pin in the corner of the aluminum structure, a load of 200 lbf was applied. From this location, the desired deflection of 0.5” was measured. With this desired 200 lbf load and the deflection of 0.5”, a 400 lbf/in spring constant was also targeted, which is the ratio between the force applied and the deflection occurred while experiencing elastic deformation. During testing, our structure was only allowed to experience elastic deformation and not plastic deformation since that would signify that yielding was present. Another requirement of the structure designed was that it had to be efficient, meaning, it had to achieve the desired values with the least amount of material possible.

Approach:

Our initial design for our geometry was a C-shaped cut out. After our initial Castigliano, it indicated that our max deflection was 0.6924" but we knew this would cause our structure to yield. So we had to make structural changes to the C-shape to reduce the yielding. To find the deflection we desired, we tapered our geometry. Using the new design and using Castigliano's method, we found our deflection to be 0.46". To assist with yielding, we kept the sidebar at a reasonable thickness. Additionally, we tapered the bottom bar to add additional deflection. After this design, we decided to cut out sections of the top bar to further increase the deflection experienced by the shape. We found this to be really helpful with small increments of deflection without the cost of excessive yielding.

One of the essential features we used in coming up with our geometry was the optimization feature in SolidWorks. Once we selected a geometry that resulted in values approximately equal to the target values, to pinpoint the perfect dimensions that increase the accuracy of our values we set up a 'Design Study' simulation where we assigned specific dimensions of our geometry as variables. These variables were restricted between upper limit max and lower limit min. We also assigned 'step values', which is the amount that the simulation will step from a selected optimization value to the next. In our setup, we input our step values in such a way that the optimization process will not exceed 38 iterations. This selection was made to minimize computation time.

The optimum optimization value as predicted was found within the iteration created as it can be seen below.

Variable View | Table View | Results View

37 of 38 scenarios ran successfully. Design Study Quality: High
Current scenario's results are interpolated (Right Click + Run to calculate accurate results for a scenario)

		Current	Initial	Optimal (6)	Scenario 1	Scenario 2	Scenario 3	Scenario 4	Scenario 5	Scenario 6
tb		0.51in	0.51in	0.5275in	0.49in	0.4975in	0.505in	0.5125in	0.52in	0.5275in
te		0.32176in	0.32176in	0.29in	0.29in	0.29in	0.29in	0.29in	0.29in	0.29in
Stress1	< 47 ksi	60.582 ksi	60.582 ksi	46.459 ksi	57.843 ksi	54.442 ksi	50.924 ksi	50.924 ksi	46.277 ksi	46.459 ksi
Displacement1	Is close to 0.5 in	0.5205in	0.5205in	0.49994in	0.52025in	0.51543in	0.51108in	0.51108in	0.50336in	0.49994in

Table 1. Simulation Optimization Data

The optimal value was found in the 6 iteration of our optimization function where dimensions that were set as variables attain the given values of 0.5275in and 0.29in. In this setup the maximum stress the function predicted was 46.459 ksi and a displacement of 0.5". Hence, with this data, we were set on a structure that we then analysed further through setting up our own simulation for. The structure's dimensions have been given below.

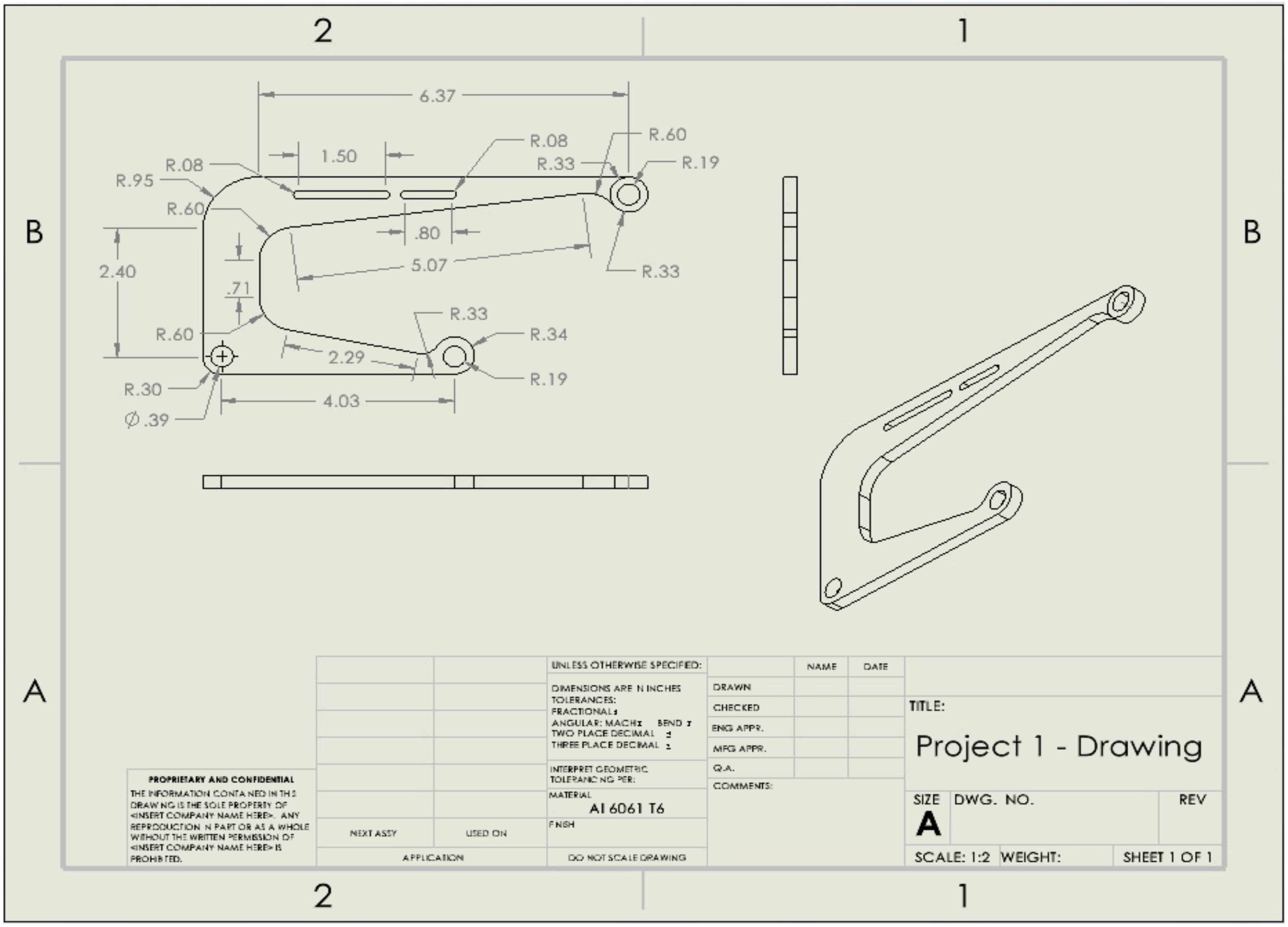

Figure 1: Engineering Drawing of our Structure

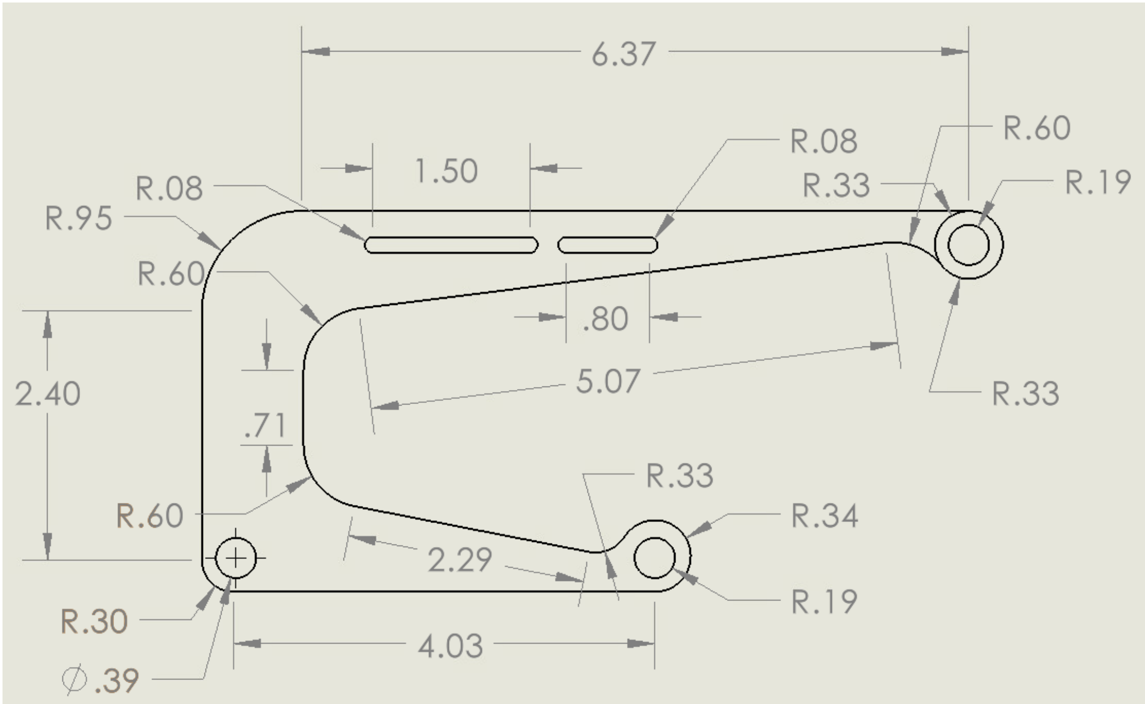

Figure 2: Zoomed in View of Dimensionalised Engineering Drawing of our Structure

Analysis:

Plots from SolidWorks Simulation

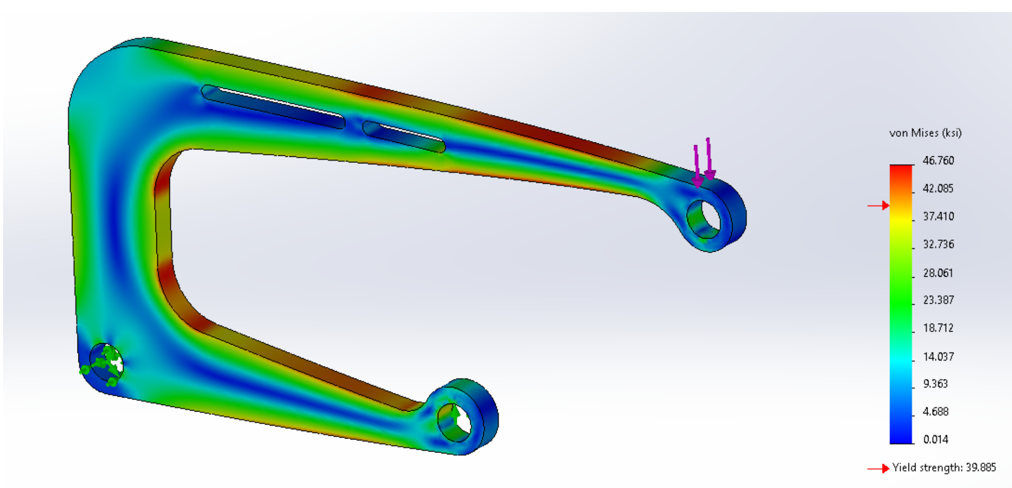

Figure 3: Color Contour Plot for Yield Prediction

Figure 3 above shows how the stress changes along with the structure. Following the color scheme, it is clear that the points of greatest stress are located at the curves connecting the middle section and the other sections, as well as at the very top of the structure. The plot also shows that the lowest stress is located right in the middle of each part of the structure, which is the location farthest away from the edges. The stress is also lowest at each of the holes, especially the location where the load is applied. This might be due to the fact that this location is pushed directly down, so at this location, it is not able to counteract the force down, so there is little or no stress. This plot also shows the impact the cutouts on the top section make. Above and below each cutout, the stress is slower, but in the gap in between each cutout, the stress is much higher.

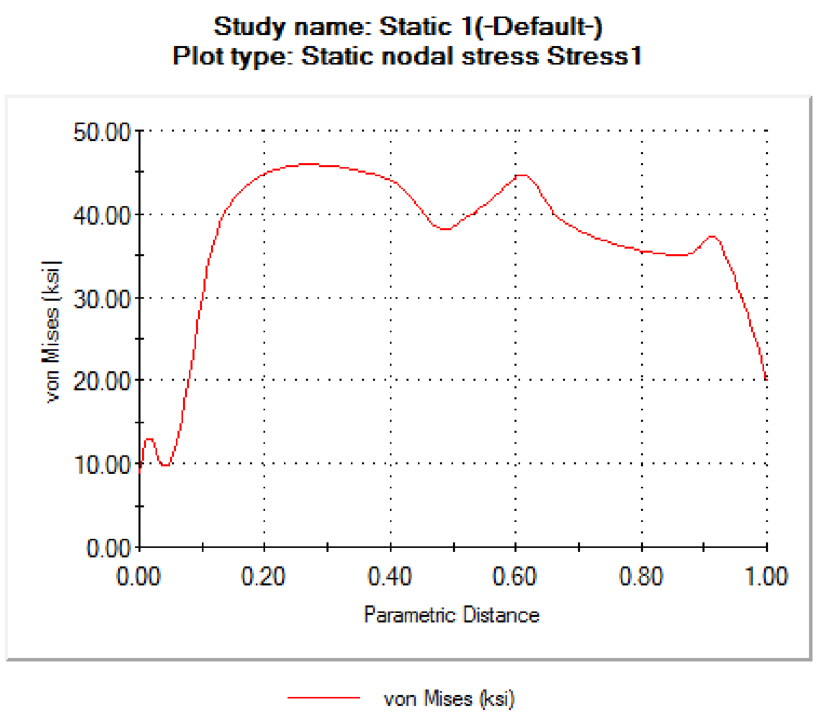

Figure 4: Graphical Representation of Yield Prediction

Figure 4 above is a graphical interpretation of how the stress changes along the top section, with the origin being the location at which the load is applied. We chose this particular section as the section of interest for creating a plot since it had the highest stresses, which is made clear by the darker red above and below the memeber. From the graph, we can see that there is a lot of stress happening almost immediately near where the load is applied, but that the stress decreases after a while, once the top section gets closer to the connection with the middle section and as the section gets thicker. This is a clear way to see how the tapered top is susceptible to yielding.

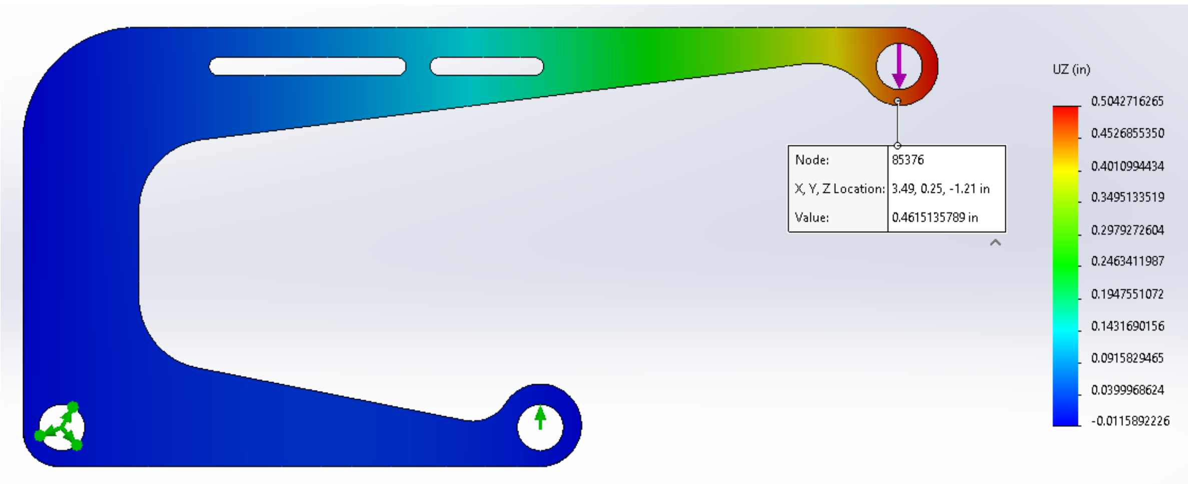

Figure 5: Color Contour Plot of Deflection Prediction

The figure above shows how the displacement changes along with the member. It is easy to tell that there was very little or no deflection at the middle and bottom members and that the most significant deflection occurs near the point where the load is applied. This makes sense as the top section is the one directly pushed down, while the middle and bottom sections are secured tightly by the fixtures at the bottom. The top section is also much longer than the bottom section and its thickness decreases closer to the loading point, which is an added factor for greater deflection. This deflection is lower on the top section where it connects with the middle section since the middle section adds extra support upwards to counteract the deflection.

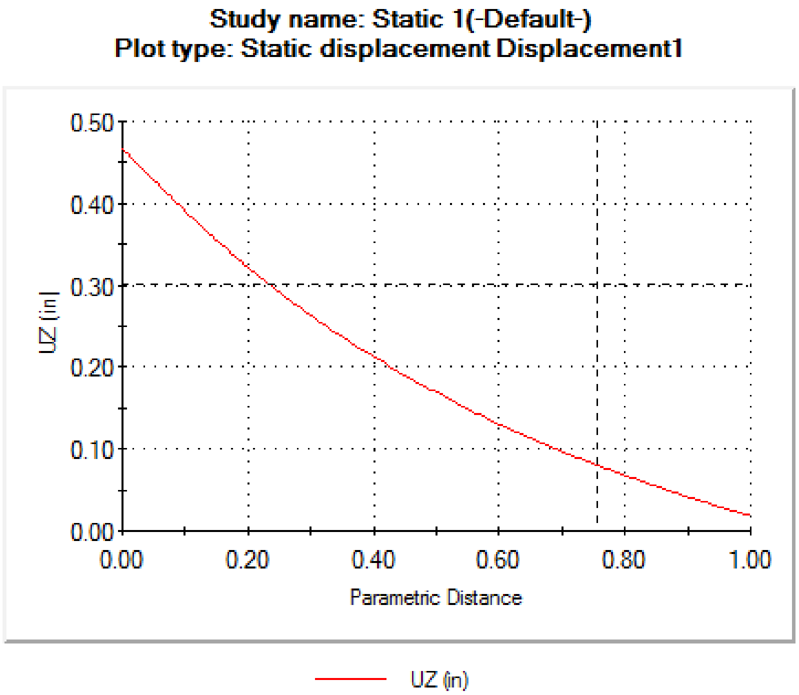

Figure 6: Graphical Representation of Deflection Prediction

Figure 6 above provides a numerical and graphical understanding of how the displacement varies along the top section. The origin once again is defined to be the point at which the load is applied. This graph makes it very easy to see how the origin point is the location with the most deflection, which makes sense given that it is the location immediately being "pushed down." As you travel farther away from the origin, the deflection decreases since the thickness of the structure increases, and the top section receives added support up closer to the middle section.

Mesh Analysis

In the initial phases of running simulations to determine the best structure that fits the requirements given, we used coarse mesh sizes to save time. After multiple iterations to our design, we were getting promising results on multiple simulations with structural geometry similar to our final one. Noticing these results, we decided to reduce our mesh size gradually to try to get more accurate results. As a result, we were able to run our final simulation with the finest mesh size available.

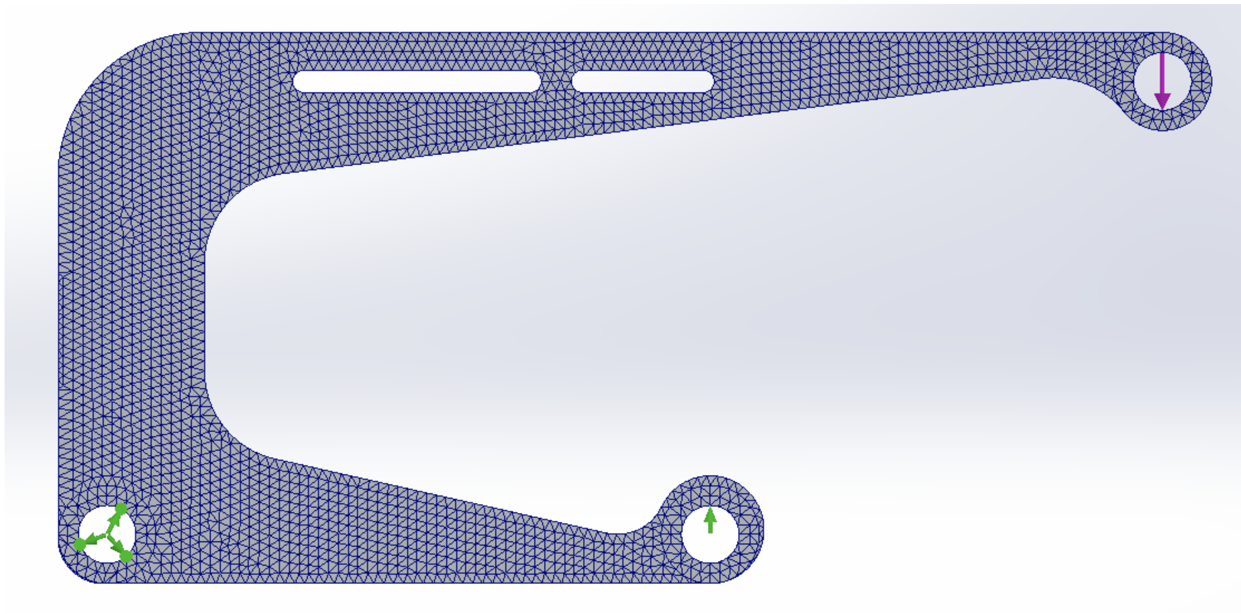

Figure 7: Fine Global Mesh used for analysis

To assess the convergence of our meshes and validate the results we were getting we generated an h-Adaptive convergence plot with an error percentage of 1%. An h-Adaptive meshing has the ability to adjust the size of the mesh automatically based on stress/strain concentration expected to be observed in specific areas. For example, in the setup of our structure, we are applying a 200lbf at a specific concentrated location. The h-adaptive mesh analysis tool meshes that region with a finer mesh as it is susceptible to experiencing high yielding due to it being a point of stress application.

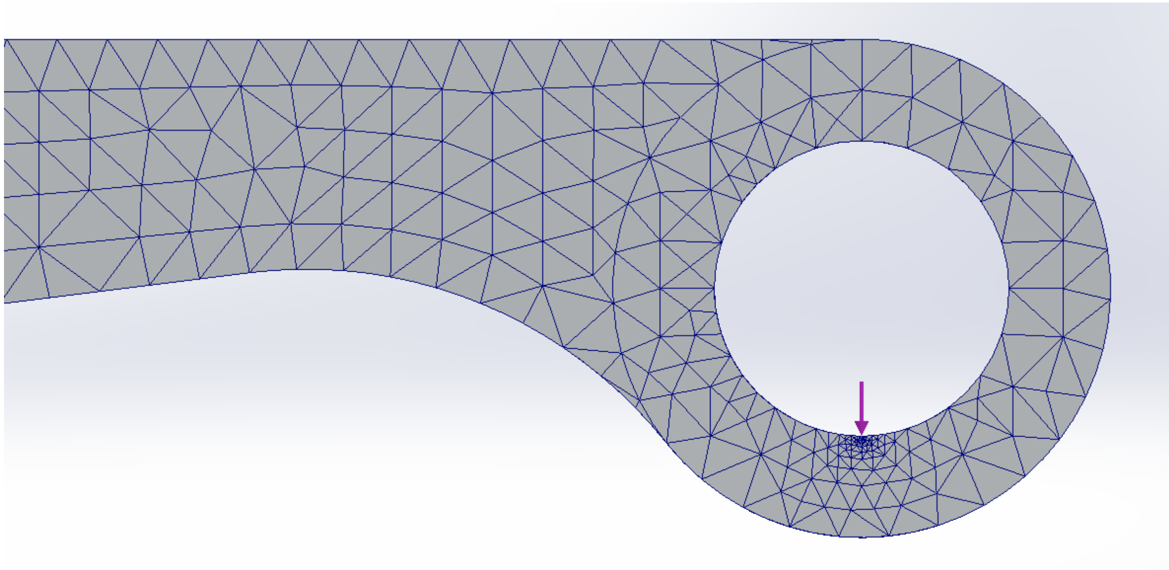

Figure 8: Concentrated Mesh at Application Point

We say an h-Adaptive convergence graph is an acceptable/accurate measure of convergence if the curve gets asymptotic as we increase the number of iterations. In multiple iterations (loops) the values are within the adjusted range of error, which is 1% for our case. Our h-Adaptive convergence graph shown below gets asymptotic as the number of loops increases showing an acceptable accuracy of the data we have obtained in our simulations.

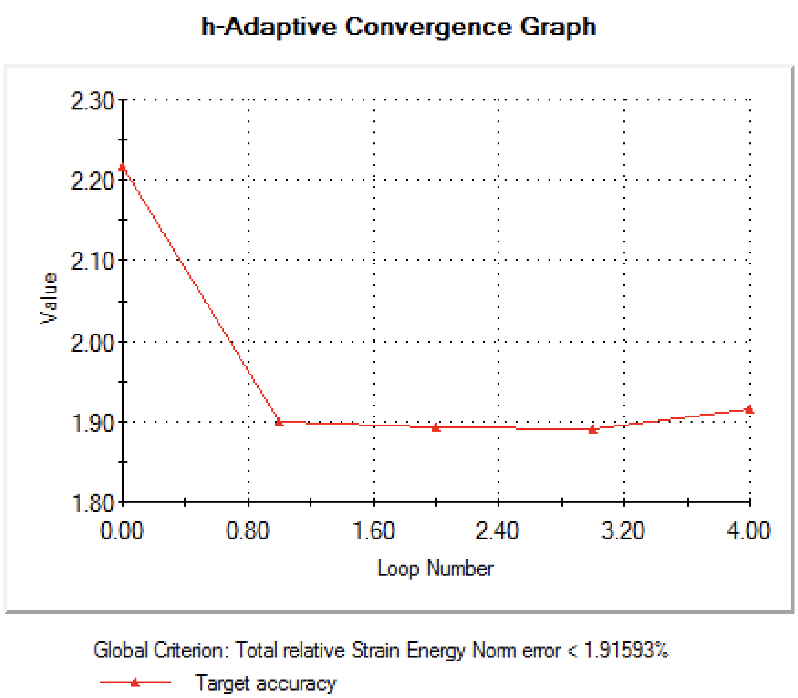

Figure 9: h-Adaptive Convergence Graph for Meshing

Castigliano Analysis for the Structure

[Insert]

Testing Results:

Our design for this project performed very well in the Instron Testing procedure. With the desired deflection of 0.5”, our structure had a total deflection of 0.5222” under a load of 200.1 lbf and with a weight of 97 grams. Our structure also did not yield under the applied load and only experienced elastic deformation. The graph below shows the deflection of the structure as the load applied increases. We can see from the graph below that the data obtained is very linear. From the graph, we can also see that our structure achieved a spring constant of 391.02 lbf/in when the goal was to have a spring constant as close as possible to 400 lbf/in. Due to our experiment focusing on elastic deformation, the target spring constant was 400 lbf/in due to the max load of 200 lbf being applied to the structure and the desired deflection of 0.5”. Our actual spring constant ended up slightly lower than the target due to our structure having a slightly higher deflection than expected. Initially, we increased the amount of deflection our structure experiences by adjusting the curves, the tapering, and the cut-outs. For further analysis, we could easily approach the target deflection and the target spring constant by adjusting these three factors. At the beginning of testing our structure, there is a small offset in our results. This is most likely due to issues with the rollers setup when the load was initially applied.

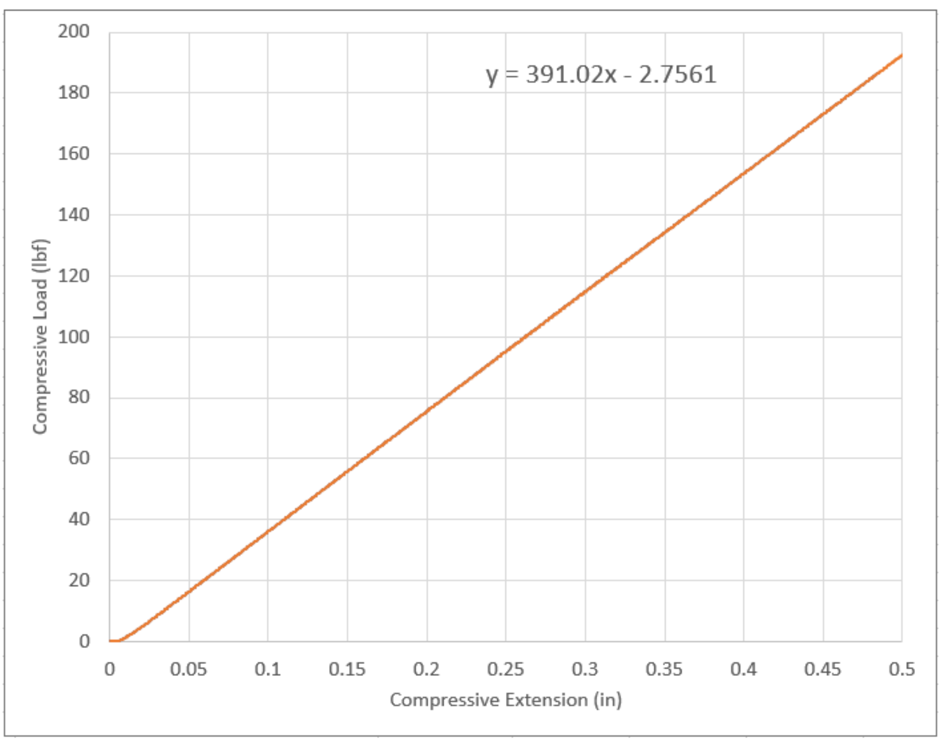

Figure 10: Graph of Applied Load and Deflection

Figure 11: Picture of Instron Load Testing

Figure 12: Designed Aluminum Structure

Summary of results

Our structure weighed in at 97 grams and performed quite closely to what we were expecting. The final amount of deflection for the 200 lbf vertical load applied on the structure was .5222 in, which was very close to our target of 0.5” deflection. From the slope of the line, it is clear that our overall spring constant is 391.02 lbf/in which is quite successful given that our target spring constant was 400 lbf/in. The test results also showed that no yielding occurred, which met our goal of achieving a set amount of deflection without suffering plastic yielding.

Discussion:

From our results in the previous section, we saw that we were quite close to the desired 0.5” deflection for a spring constant of 400 lbf/in, but not exactly there. In order to get closer to the target results, we could have modified our design by adding more of the material back from the sections we removed. In our final design, we had two small sections removed from the top member of our geometry to assist with the deflection without costing too much yielding. To modify our design we could add back some material to help stiffen the top member allowing for a larger spring constant by reducing the deflection. Another way in which we could have reduced the deflection without causing too much yielding would be to adjust the curves between the top and bottom members and the vertical member. This would have made our model slightly stiffer which would have reduced deflection. A trade off which we had to make for this design was the size and the thickness of the members. As we reduced size, yielding increased and weight decreased, but as we increased size, yielding decreased but weight increased.

The deviations in our predicted and observed results were quite small but were due to the complexity of our geometry. Due to the curves in our shape, completing a Castigliano that accounted for every curve would have been really difficult. So simplifications were made in order to get a close enough estimate for our deflection. As for our SolidWorks simulation, the Young's Modulus of the material is different from our real-life material with the material in SolidWorks being slightly stiffer. Due to this, we expected a larger deflection in our real life results than our simulated model.