Computer-Aided Engineering - 6928

Project 3: FEA Bracket

Professor: Dr. DeSilva

Group Members

Andrew Noseworthy | 201628864

Rowan Meaney | 20162860

Steven Kennedy | 201550530

Erik Silveira Mello | 201618063

M16

Date Submitted: November 21, 2019

Abstract

It is critical to be able to design, analyze and test engineering ideas prior to their implementation. As engineers it is of the highest priority to not be content with the first concept that is generated but to conduct multiple iterations to ensure the best design is reached. There were two main goals of this bracket; the first being to have the lightest weight possible, and secondly must be able to support a minimum of 100lbf prior to yield and also ensure it breaks prior to 200 lbf. This report outlines the process M16 went through from ideation to analysis to real life testing.

Table of Contents

List of Figures

List of Tables

1.0 Introduction

Using rapid prototyping techniques the goal of the project was to design and 3D print a bracket that could support a load of at least 100 lbf and fail prior to 200 lbf (failure being past ultimate stress failure to a fracture).

Ultimate stress is at the point in which the part will just barely fracture. This means the part has to undergo plastic deformation → elastic deformation → ultimate failure → fracture. A graph of isotropic materials undergoing this process is seen in Figure 1.

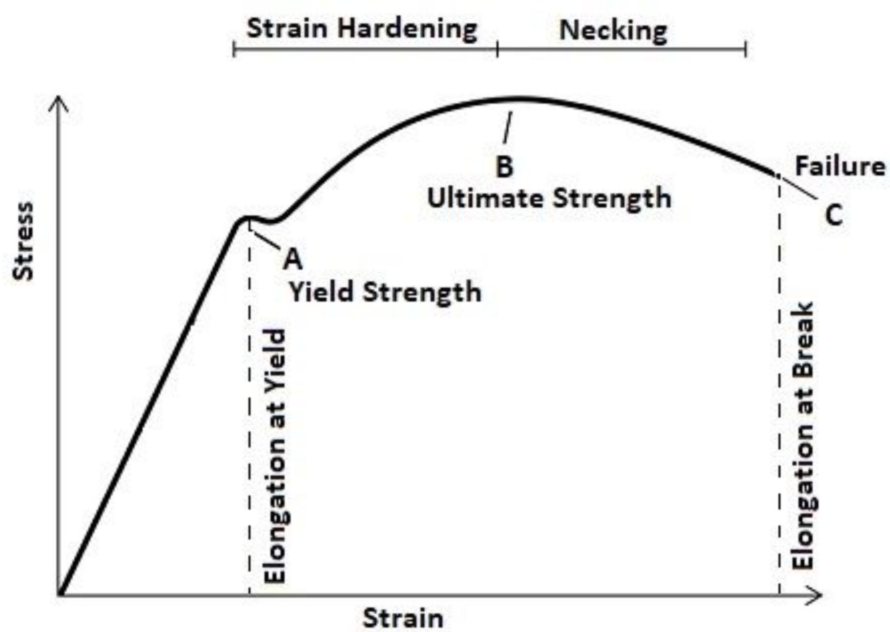


Figure 1: Stress vs Strain plot of isotropic materials [1].

To ensure the design of the bracket was sufficient a series of analysis studies were conducted include, a static finite element analysis study to locate the high stress concentrations and deflection along with a buckling study to see if the part would buckle before it failed.

Following the design and analysis we sent it to get 3D printed using ABS P430. After getting it back following the print we did a quality assurance check to ensure it matched up to the 3D model. Lastly it was tested where the force and deflection were measured with a load cell. The bracket successfully fractured between 100 lbf - 200 lbf however there was buckling noticed just prior to the crack.

2.0 Methodology

2.1 Hand Calculations

2.1.1 Cantilever Beam Study

Prior to starting the design it was critical we determine some general design parameters to design our first bracket. Using a basic cantilever beam study we were able to determine a good starting point for our bracket concepts.

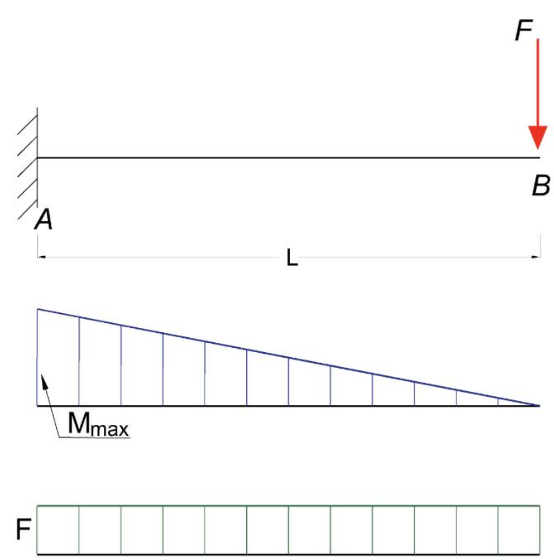


Figure 2: Cantilever beam study parameters with shear and bending moment diagrams [2].

Before beginning iterations of the bracket design, some hand calculations were completed on a simple cantilever beam to find dimensions of height and width for the design of the bracket. To begin, the moment acting on the bracket was calculated as follows;

$M = F * l = 100 * 4.91 = 491\ lb * in$

Using the yield strength of the material (ABS P430) and a safety factor of 2 we were able to find the maximum allowable stress on the bracket. Those calculations are as follows;

$$\frac{Sy}{n} = \sigma = \frac{26*10^6}{2} = 13000000\ Pa = 1885.49\ psi$$

The same was also conducted for the transverse shear stress on the beam;

$$\frac{Ssy}{n} = \tau = \frac{0.577*26*10^6}{2},\ \tau = 7501000\ Pa = 1087.928\ psi$$

Now with values for Bending and Transverse shear stresses, our heights were able to be computed with an inputted width. For the width we chose to iterate on a 1/16 base increment starting at half an inch, and solving for our height. The equations can be rearranged as follows to solve for height.

$$\sigma = \frac{M*c}{I} = \frac{M*\frac{h}{2}}{\frac{1}{12}*w*h^3} \qquad \tau = \frac{3*V}{2*A} = \frac{3*F}{2*w*h}$$

such that,

$$h^2 = \frac{6*M}{\sigma*w} \qquad h = \frac{3*F}{2*w*\tau}$$

Using these two equations, values for widths can be used as an input and height will be returned. As an example of the iteration, a width of 9/16 in will be inputted for w.

$$h^2 = \frac{6*491}{1885.49*\frac{9}{16}} = 2.777 \quad , \quad h = 1.666\ in$$

$$h = \frac{3*100}{2*\frac{9}{16}*1087.928} = 0.254 \quad , \quad h = 0.245\ in$$

From the above results we can clearly see that the bending stress will create a limit on the allowable height of the beam. Therefore for the iterations comparing width and height the focus will be placed on the equations for bending stress.

Using these equations we can iterate width vs height and produce a graph to help choose an optimal width and height for the design. The graph seen below in figure 3 displays the iterations of width vs height.

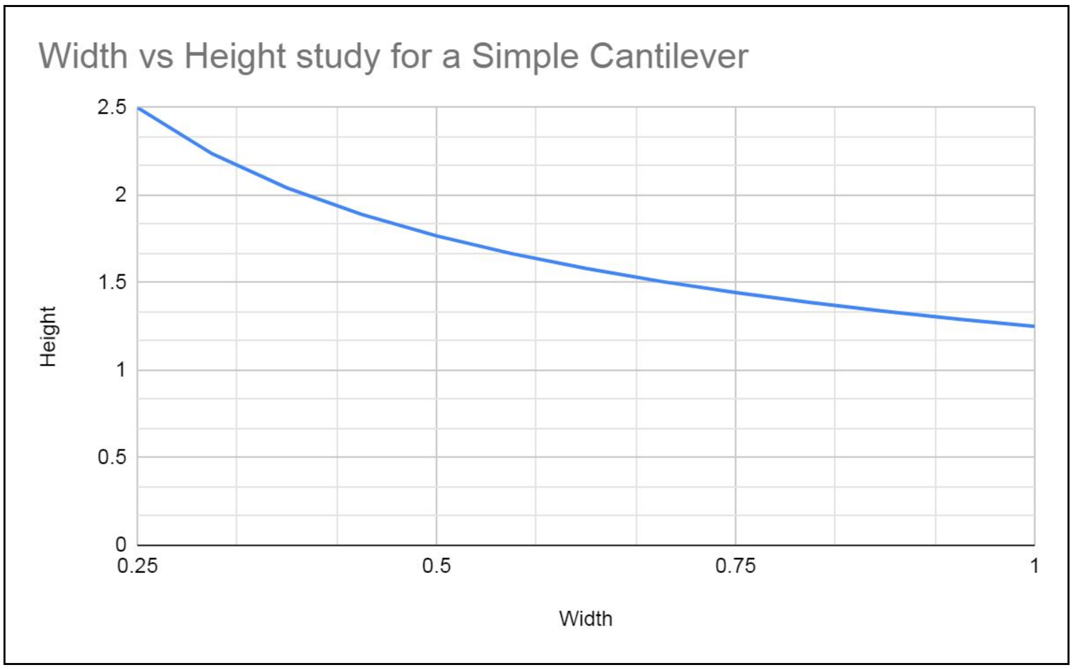


Figure 3: Width vs Height comparison for a simple cantilever with the same initial properties of the bracket.

After the iterations were complete, it can be clearly seen that as the width varies the height varies at a greater value. By observing these results it would make sense for the group to choose a value that compensates for both height and width to minimize mass and cost while not making the dimension unrealistic. For this reason the group decided to go with a width of 9/16 in and the resultant height of 11/16 in (although this is changed later to accomodate for strength of the bracket).

2.1.2 Bearing Stress from Bolts

The same force and material yield strength will be used to calculate the bearing stress experienced by the bracket. For this the critical dimensions necessary to calculate the bearing stress will be taken from the design iterations completed during bracket design. These values were as follows;

$$t\ (bracket\ on\ wall) = 0.4375\ in$$
$$d\ (bolt\ hole) = 0.2\ in$$

To find the experienced bearing stress the bearing area for a loosely fitted bolt will first be considered;

$$A_{bearing} = \frac{\pi}{4} * t * d = \frac{\pi}{4} * 0.4375 * 0.2 = 0.06872234\ in^2$$

Now the bearing stress can be calculated by using the 100 lb force on the end of the bracket.

$$\sigma_{bearing} = \frac{F}{A_{bearing}} = \frac{100}{0.06872234} = 1455.130908\ psi$$

This value of stress is also the von mises effective stress used to calculate the respective safety factor, hence;

$$n = \frac{Sy}{\sigma_{bearing}} = \frac{3771}{1445.13091} = 2.59152$$

Since this safety factor is well above 1, failure due to bearing stress is not of worry for the current bracket design iterations.

2.1.3 Tearout Stress from Bolts

As for the bearing stress calculations, the tearout stress calculations will also use the same force and yield strength. The critical dimensions used in each iteration of the bracket will also be utilized to calculate the safety factors for the tearout stress. The critical dimensions are listed as follows;

$$t\ (bracket\ on\ wall) = 0.4375\ in$$

$$d\ (bolt\ hole) = 0.2\ in$$

$$h\ (center\ of\ top\ hole\ to\ top\ of\ bracket) = 0.28125\ in$$

To find the tearout stress of the bracket first the tearout area must be calculated. The area can be calculated as follows;

$$A_{tearout} = 2 * t * (h - \tfrac{d}{2}) = 2 * 0.4375 * (0.28125 - 0.1) = 0.158594\ in^2$$

Now the tearout stress can be calculated by considering the 100 lb force on the end of the bracket;

$$\tau_{tearout} = \frac{F}{A_{tearout}} = \frac{100}{0.158594} = 630.5419\ psi$$

The von mises effective stress can be computed as follows;

$$\sigma' = \sqrt{3 * \tau_{tearout}^{\ 2}} = \sqrt{3 * (630.5419)^2} = 1092.13056\ psi$$

And now the safety factor can be calculated;

$$n = \frac{Sy}{\sigma'} = \frac{3771}{1092.13056} = 3.4529$$

Since this safety factor is well above 1, failure due to tearout stress is not of worry for the current bracket design iterations.

2.2 Material Research

Based off the following data given from the handout a custom material was created in Solidworks (SW). This custom material was then used within the mass estimates as well as the results from the finite element studies.

ARGYLE MATERIALS — ABS P430 METRIC

TEST	TEST METHOD	ARGYLE ABS B31	OEM P430
Tensile Strength	ASTM D638	36 Mpa	36 Mpa
Tensile Modulus	ASTM D 638	2,395 Mpa	2,400 Mpa
Flexural Strength	ASTM D790	60Mpa	61 Mpa
Flexural Modulus	ASTM D290	2,322 Mpa	2,300 Mpa
Nothed IZOD Impact (23c)	ASTM D256	137 J/m	139 J/m
Heat Deflection Temp	D 648	96° C	96° C
Elocation at Break	AST 638	4%	4%

Figure 4: Material properties of ABS P430 plastic which was used to fabricate the bracket [3].

In addition to the material that was used, the infill and printing method was crucial to the strength of the bracket. The following figure is a graph showing the strengths of the same material being printed in different orientations and with different infills.

Figure 5: A plot showing the combination of minimal infill to an increasing infill to a fully solid infill with different printing orientations [4].

The first conclusion that can be easily taken from this graph is that a fully solid infill makes the most sense as it has the highest breaking point regardless of the printing orientation. Secondly, the orientation of the print can make dramatic differences in the strength of the material especially during Filament Extrusion Deposition (FDM) style printing where one layer in added at a time.

Since one layer is added at a time there is a very high risk for creating a part that is not completely homogenous throughout which can lead to localized stress concentrations meaning an unexpected failure that wouldn't normally be noticed in SW due to the assumption the SW FEA solver uses of a “perfect” material.

The FDM printing method can be seen below:

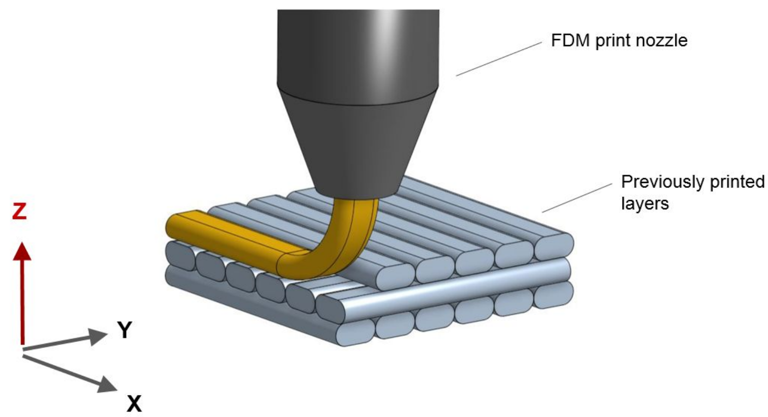


Figure 6: Diagram showing how filament deposition printing works [5].

Based off this layering construction of additive manufacturing additional research was done on how layer orientation affects strength. The following figure highlights the approximate load case. Where the bending of the beam essentially creates a lot of tension in the top of the beam.

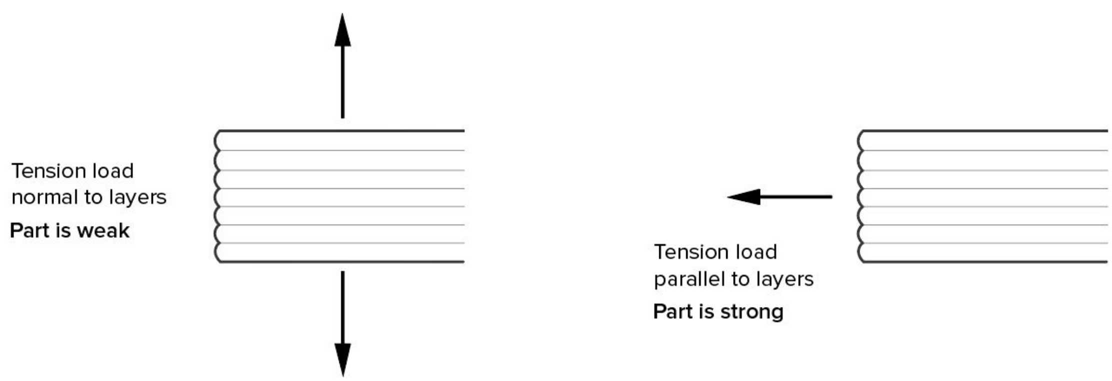


Figure 7: Part orientation during printing correlation to strength under load [6].

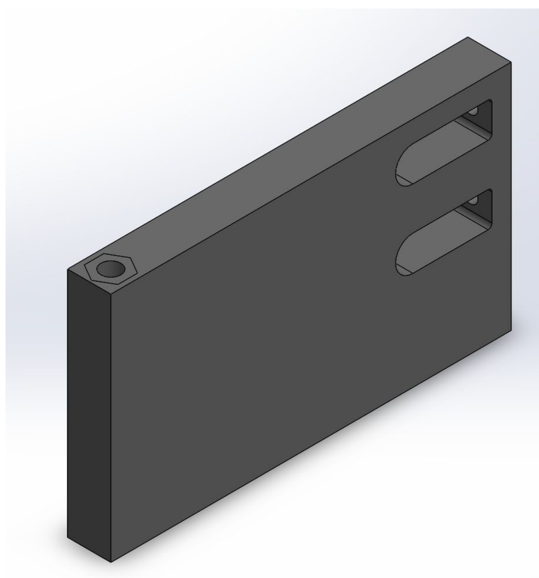
With this information it was critical that we designed our part so it could be printed to have the above layer construction to ensure the results we were getting in SW FEA were as close to a uniform material and the layer construction wasn't going to under rate our part significantly.

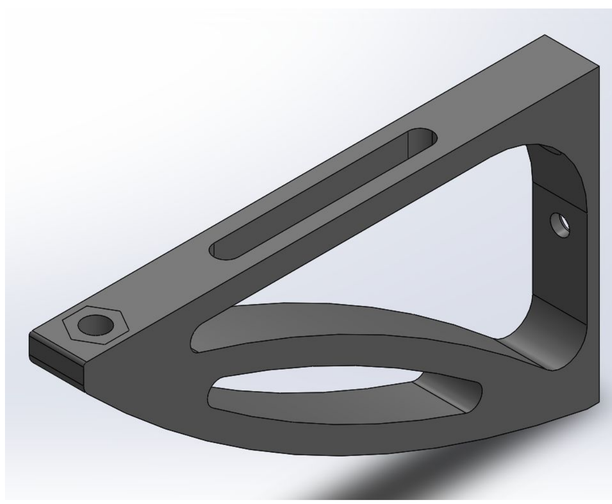
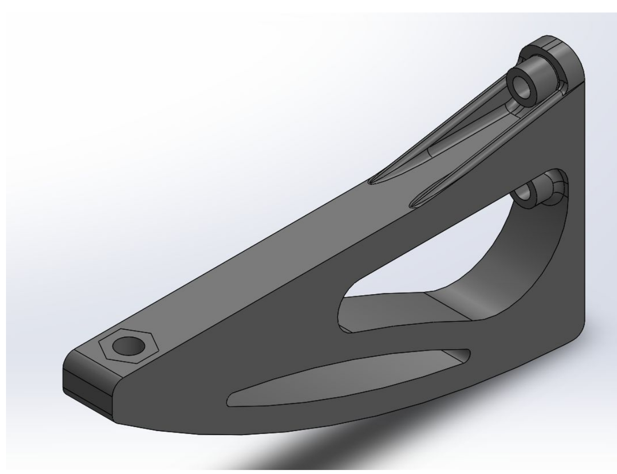
2.3 Design Iterations

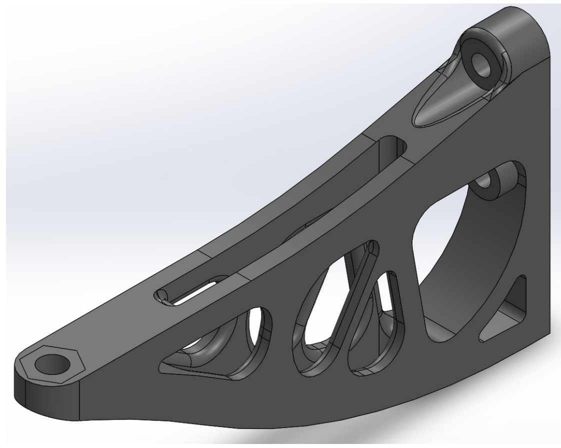
The goal of the interactions were to design both a lightweight and strong bracket. During this multiple types of bracket configurations were used where we test the beams in tension versus compression styles of loading.

The following table are images of the brackets along with the critical properties used to make the decision of which bracket to proceed with. All brackets were run with 100lb load

Table 1: Part iterations with critical values.

<table>
<tr><th>Iteration #</th><th>Part Image</th><th colspan="2">Critical Values</th></tr>
<tr><td rowspan="2">1</td><td rowspan="2"></td><td>Mass</td><td>133.51 grams</td></tr>
<tr><td>FOS</td><td>3.64</td></tr>
</table>

2		Mass	17.41 grams
		FOS	0.19
3		Mass	41.75 grams
		FOS	0.8
4		Mass	52.85 grams
		FOS	0.95

5		Mass	36.74 grams
		FOS	1.01

2.4 FEA Process:

2.4.1 Rapid FEA Testing

When conducting FEA a big time dependance is on the details of your mesh as the software needs to analyze more points throughout the part. In order to reduce this during iterations we lowered the mesh quality to a medium between coarse and fine. Even though the results will vary with a coarser mesh it was satisfactory to do this until we reach a bracket we were content with. The actual setup did not vary for normal FEA testing.

The process to set up the FEA can be seen in section 2.4.2.

2.4.2 Final FEA Testing

The setup of the study can be broken into 6 main aspects. Each of these aspects will be a heading below where the details of each process is highlighted.

2.4.2.1 Geometry

- The part is simply made in solidworks
- Designed under the criteria specified in the handout
  - Bolt spacing

- Distance to point load from wall
- Etc

2.4.2.2 Material Properties

Using the material properties of ABS P430 seen in figure 4 a new custom material was created to have the same properties since this specific material did not exist in the Solidworks database. The table below contains the solidworks labelled inputs and their corresponding values.

Property	Value	Units
Elastic Modulus	2322000000	N/m^2
Poisson's Ratio	0.394	N/A
Shear Modulus	2395000000	N/m^2
Mass Density	1020	kg/m^3
Tensile Strength	36000000	N/m^2
Compressive Strength		N/m^2
Yield Strength	26000000	N/m^2

Figure 8: Solidworks material inputs.

2.4.2.3 Study Creation

Since the load is applied gradually it can be considered a static study.

Note: The buckling study will be rediscussed as the process below varies for a buckling study.

1. Click New Study
2. Study Type Selection
    a. Static

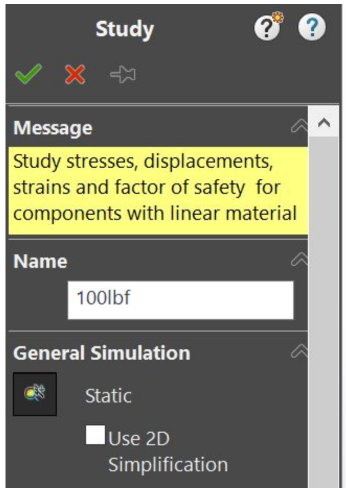


Figure 9: Solidworks screenshot to set up the type of study (static).

2.4.2.4 Operating Environment

Fixtures:

Since the bracket will rest inside the bolt holes it makes the most sense to fix the inside the bolt holes.

1. Select Fixtures
2. Select Fixed Geometry
3. Select the inside face of the bolt holes

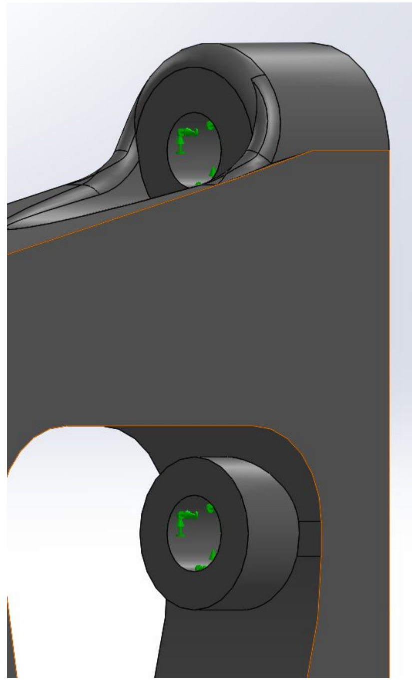


Figure 10: Fixtures on part inside bolt holes.

Virtual Wall:

Since the back face of the bracket will rest against a flat plate we can treat this as a virtual wall. We cannot treat this back plate as a fixture as that is not what is actually happening during the loading period.

1. Click contact set
2. Select Type: Virtual Wall
3. Select the face in which the virtual wall will act on
4. Select the plane that corresponds to this face
    a. This was the front plane in our study seen in the figure below.

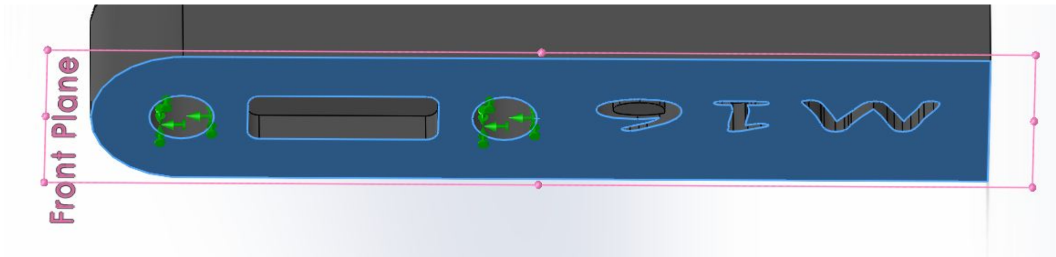


Figure 11: Virtual wall setup.

Loads

Since the load is applied off of a nut it is more accurate to model the location of the force as a polygonal split line surface.

1. Create a polygonal split line feature in the model tab
2. Select Loads
3. Select Force
4. Select the split line feature for the face at which the load is applied
5. Adjust the load value to desired input
    a. 100 lbf

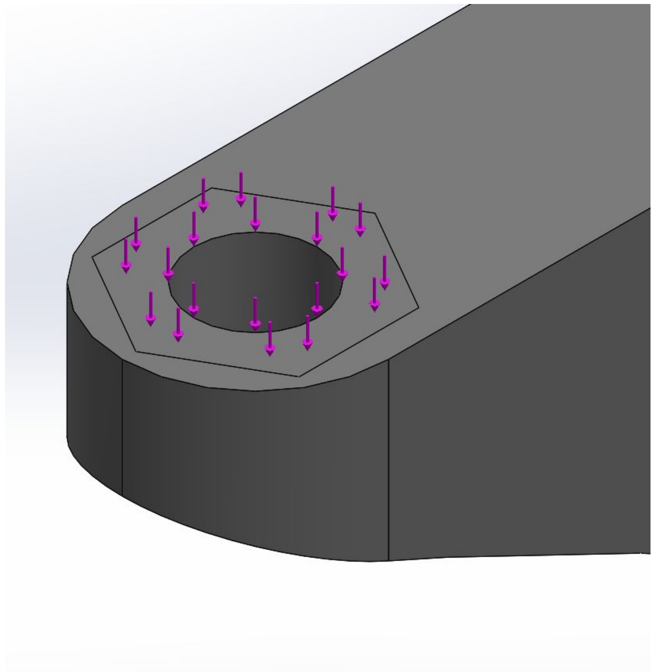


Figure 12: Loads applied to a split line feature that is the same size as a ¼-20 Nut.

2.4.2.5 Processing (Meshing the Model)

During the finalized study of the bracket a fine mesh was used to complete a detailed pass on the model prior to running a h-adaptive study which is much more time intensive. This study will be a good indicator of the stresses and stress concentrations we can expect from our bracket during loading.

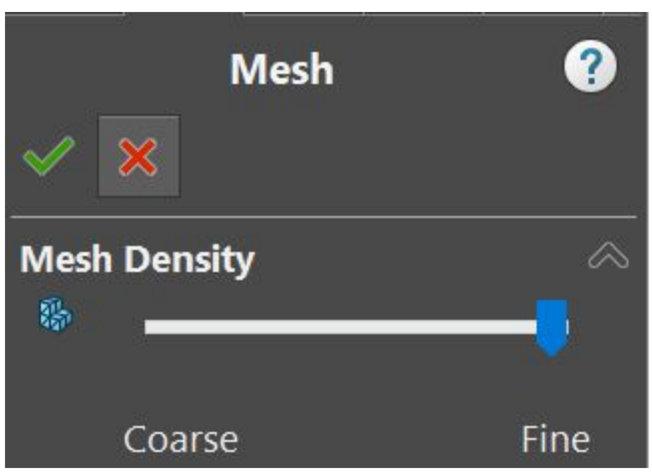


Figure 13: Mesh density throughout the part.

2.4.2.6 Results

There are a variety of features with solidworks that allows us to easily spot the stress locations and displacements of the part. Using the von mises stress factor of safety plots and specifying the von mises type to ensure it was as accurate as possible.

Per any static study the plots it automatically generates stress, strain and displacement plots. To generate a Factor of Safety plot:

1. Right click on results
2. Select Define Factor of Safety Plot
3. Specify FOS type
    a. Von mises.

In order for us to determine the stresses on our bracket we used a tool called iso clipping. This tool allows you to take the stress plot and select a value of stress at which you want SW to display the stress regions on the graph. We did this by selecting the yield strength of our material to see the points in which the part would begin to yield. The locations at which the part will yield will likely lead to an ultimate fracture the more load that is applied.

2.4.3 H-Adaptive Testing

H-adaptive studies were set up the exact same way as a normal study except the convergence type during the solving process of the study and the details of the mesh itself. In order to make the changes from a normal mesh simply;

1. Right click the study
2. Select h-adaptive mesh

3. Set parameters of the mesh to match our desired target values seen in the image below:

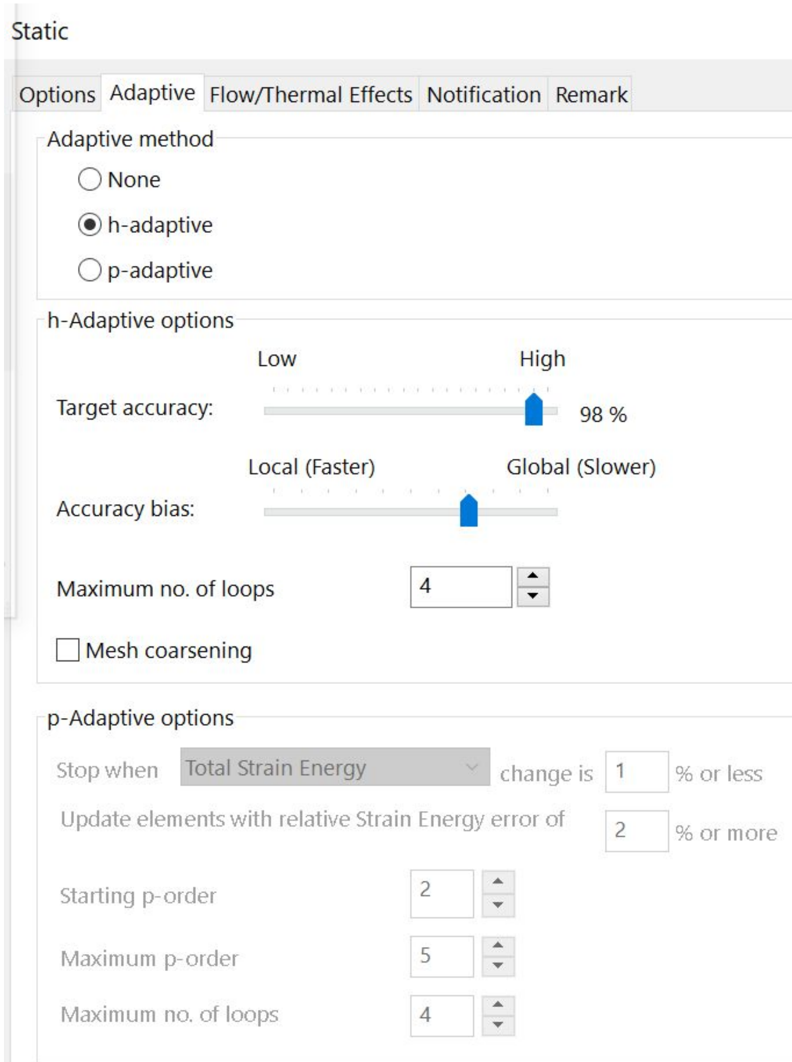


Figure 14: H-adaptive setup of the mesh.

2.4.4 Buckling

The first two steps outlined in section 2.4.2 will hold true in the buckling study since the material and geometry are identical.

2.4.4.1 Study Creation

1. Click New Study
2. Study Type Selection
    b. Buckling

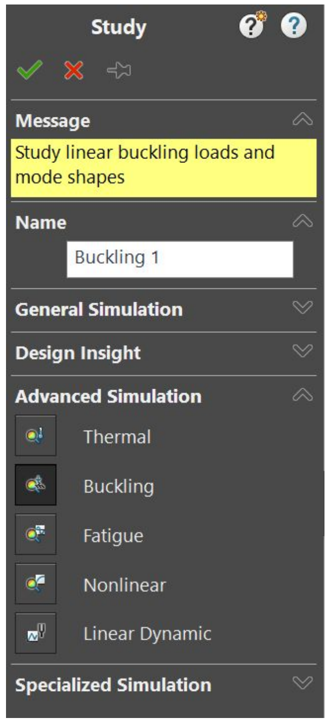


Figure 15: Solidworks screenshot to set up the type of study (buckling).

Following the selection of the study you proceed to apply the load and fixtures as you would for a static FEA study without the wall contact and you run the study with whatever mesh fineness you want. As per usual the more dense (fine) your mesh is the more accurate the results will be.

2.5 Printing Specifications

As per figure 7 printing load orientation it makes the most sense to design the part in order for the force to be applied parallel to the layers. This will ensure the part is as strong as possible and therefore closely match to solidworks.

This also then easily ties into how the part should be designed so this can be as simple as possible. The following figure highlights the print orientation and the direction in which the layers will be added during the printing process.

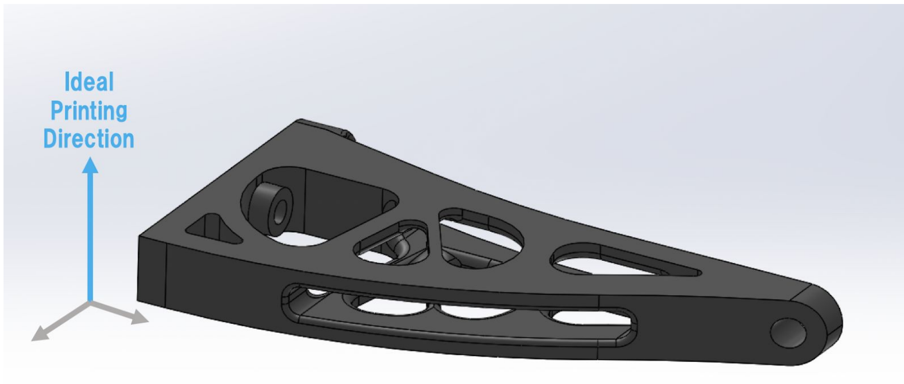


Figure 16: Identifying optimal printing direction of the bracket.

2.6 Experimental Testing Procedure

Once in the lab with our bracket the following procedure was followed to test our bracket under load:

1. Insert #10-32 bolts into the bracket
2. Place the bracket with the bolt holes into the mounted beam.
3. Place nuts on the end of the bolts
4. Hand tighten bolts
5. Use needle nose pliers to hold the nuts in place
6. One at a time using an allen key to tighten the bolts till relatively torqued.
7. Slightly push on the bracket with your hand to ensure the bracket is in full contact with the wall
8. Line up load cell in place in the designed hole
9. Ensure data is recording in real time
10. Begin pumping the cylinder to reach 20 lbf increments
11. At each 20 lbf increments wait ~5s to ensure the load will settle on the bracket and will not become a dynamic load.
12. Continue pumping until the bracket fractures.

3.0 Results

3.1 FEA Results

All results are completed with 100lbf loading applied to the tip of the bracket.

3.1.1 Static Study

3.1.1.1 Normal Mesh

Stress:

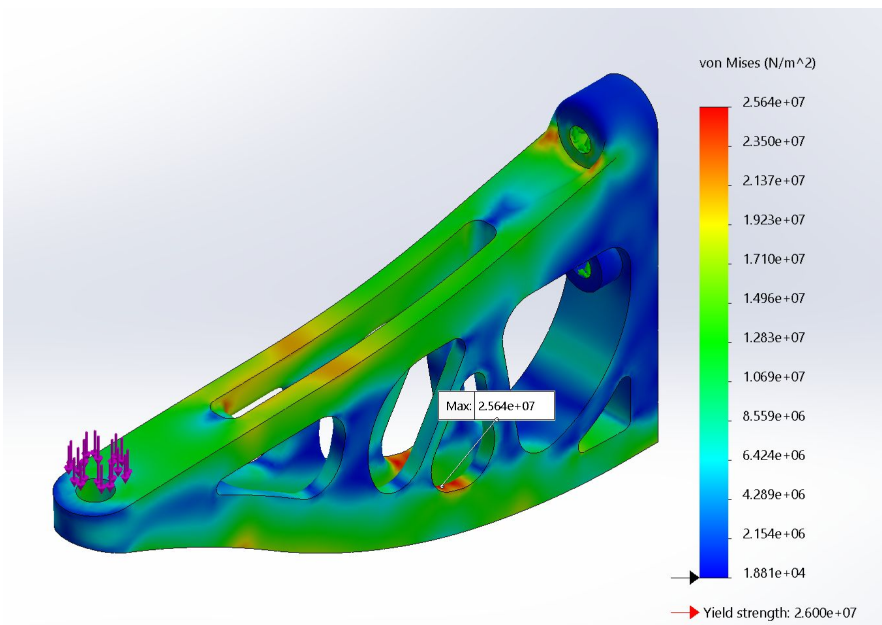


Figure 17: Stress plot of the bracket using non-adaptive mesh. Max Stress value of 2.564e7 Pa.

Displacement:

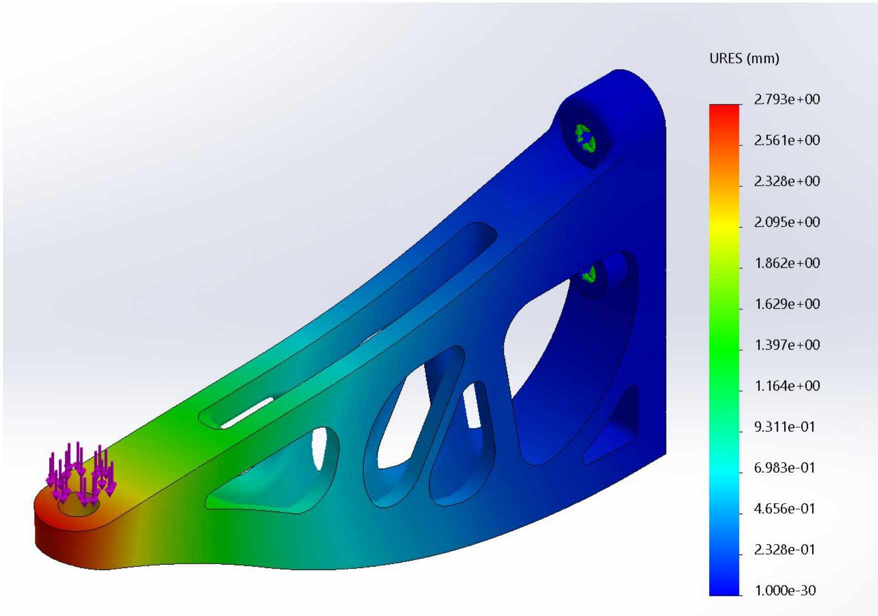


Figure 18: Displacement plot of the bracket using non-adaptive mesh. Max displacement of 2.79 mm.

FOS:

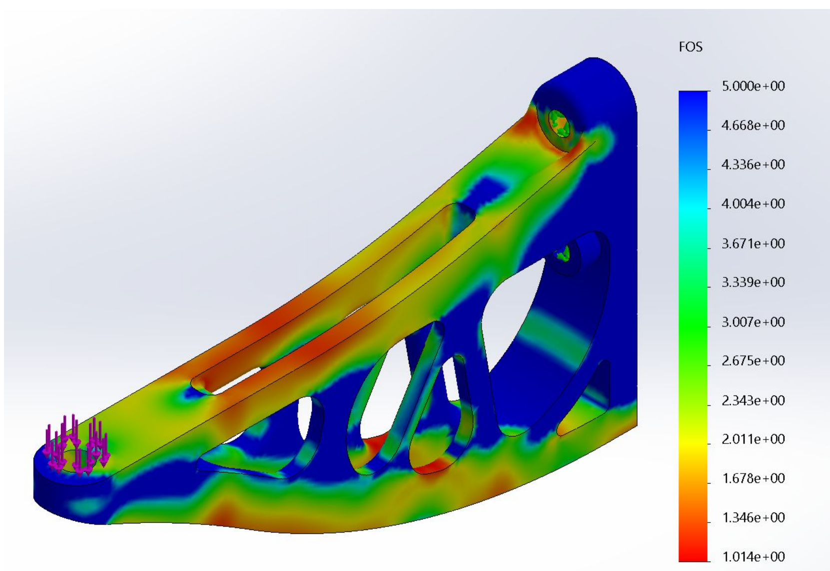


Figure 19: Factor of Safety plot of the bracket using non-adaptive mesh. Minimum FOS of 1.014.

3.1.1.2 H-Adaptive Mesh

Stress:

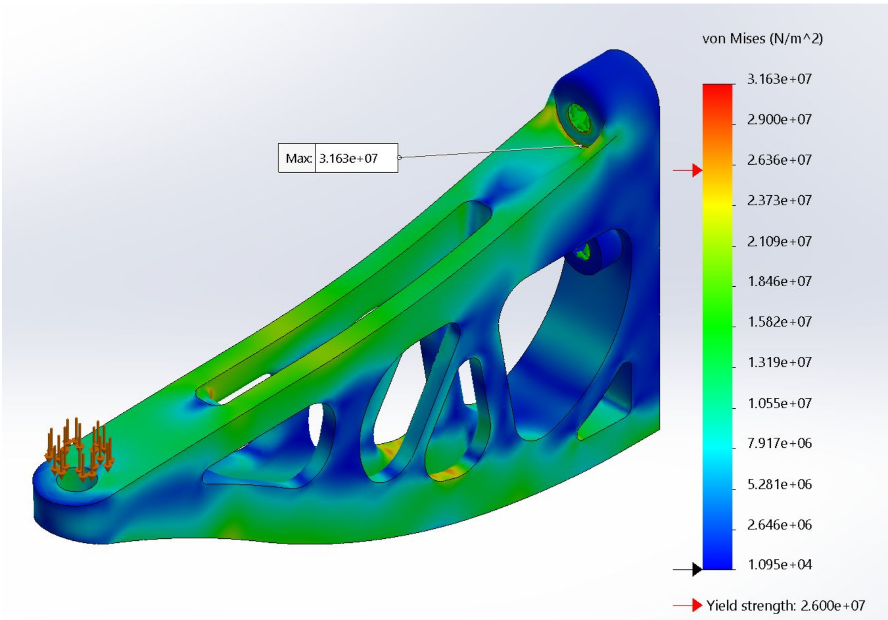


Figure 20: Stress plot of the bracket using h-adaptive mesh. Max stress of 3.163e7.

Displacement:

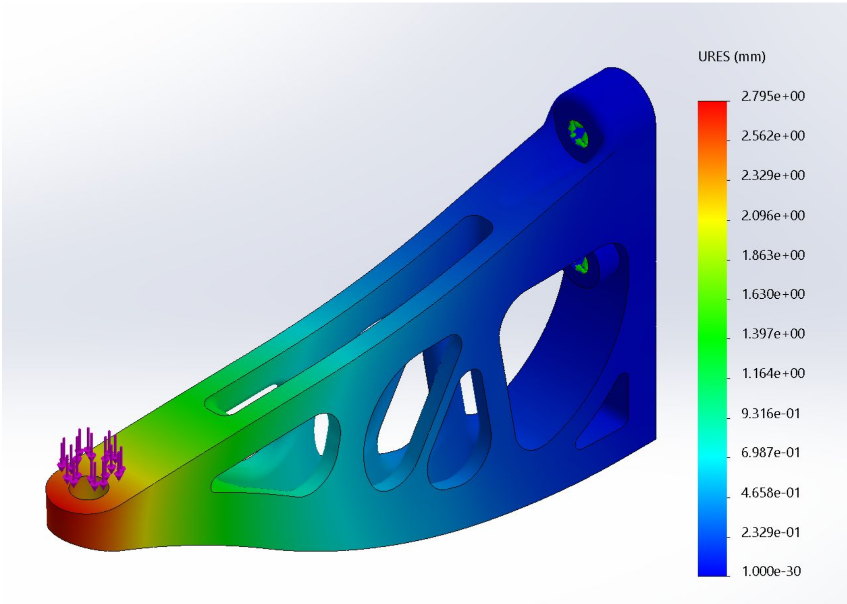


Figure 21: Displacement plot of the bracket using h-adaptive mesh. Max displacement of 2.795 mm.

FOS:

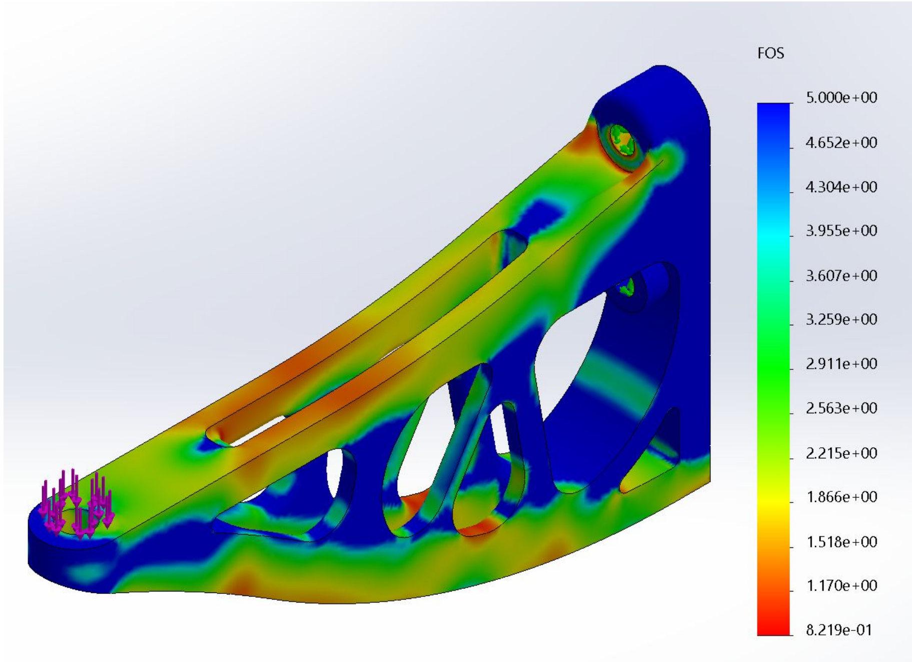


Figure 22: Factor of Safety plot of the bracket using h-adaptive mesh. Minimum factor of safety of 0.8219.

3.1.2 Adaptive vs Non Adaptive Plot

The main differences between the h-adaptive plot vs non adapting meshes is that the h-adaptive identifies more local yielding in the part vs the non adapting mesh. Initially when looking at the plot it seemed worrisome due to a lower safety factor. However, upon inspection it just creates more details within the local yielding. This was deemed as a lower concern and reinforcing these areas was not required.

3.1.3 Buckling Study

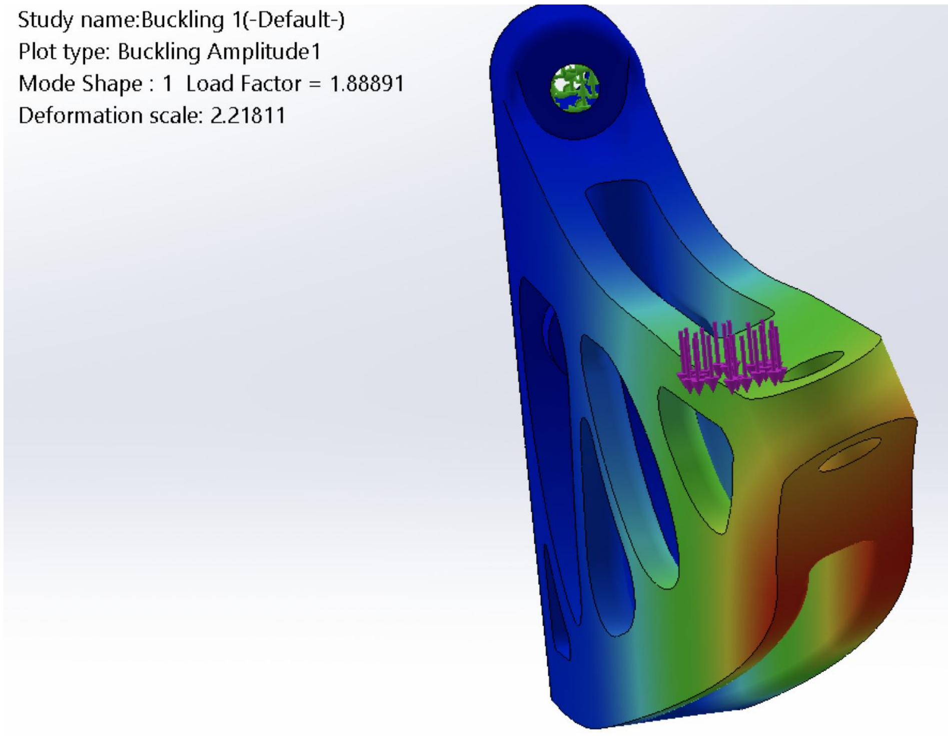


Figure 23: Buckling plot indicating buckling will occur at 188.9 lbf.

3.2 Predicted Failure

After completing analysis on the stress concentrations produced by Solidworks FEA it is clear that there are some areas of localized stress concentrations. After discussion with the group it was considered that fracture would occur when the stress concentrations grew to a magnitude that would induce fracturing. It was concluded that fracturing of the bracket would occur well before the bracket would fail due to buckling. This is due to the load factor provided from FEA, and the group anticipated that buckling would occur at around 190 lbf. Under the assumptions implied, buckling of the bracket should not have been concerned, hence leading us to believe the bracket would fail due to fracture rather than failure due to buckling.

3.3 Testing

3.3.1 Testing Setup

In order to test the strength and fracture mode of our bracket the group moved to the strength testing lab to use an apparatus. The testing apparatus can be seen in figure XYZ below with the groups specimen attached to the C-section beam.

Figure 24: Test apparatus with bracket installed.

Once the bracket was secured to the beam using bolts and nuts the hydraulic jack fitting was moved down to the bracket and secured in place to the hole at the tip of the bracket. This completes the setup for testing of the bracket. The hydraulic press was initiated and controlled by the use of lever located to the side of the press itself and can be seen in Figure 25 below.

Figure 25: Lever device used to control the actuation of the hydraulic press.

The results of the test were recorded on a computer nearby with sensors attached to the testing apparatus. This data was sent to the group at a later date. The computer recording data can be seen in Figure 26 below.

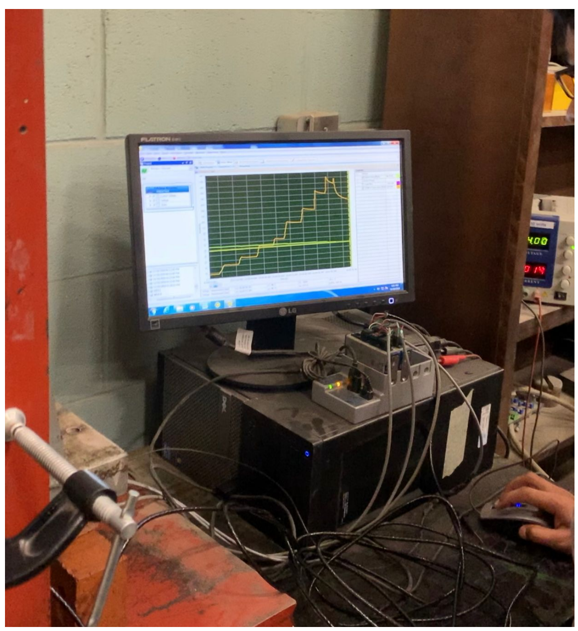


Figure 26: Computer responsible for recording testing data.

3.3.2 Testing Results

The following are plots highlighting the loading period of the bracket test:

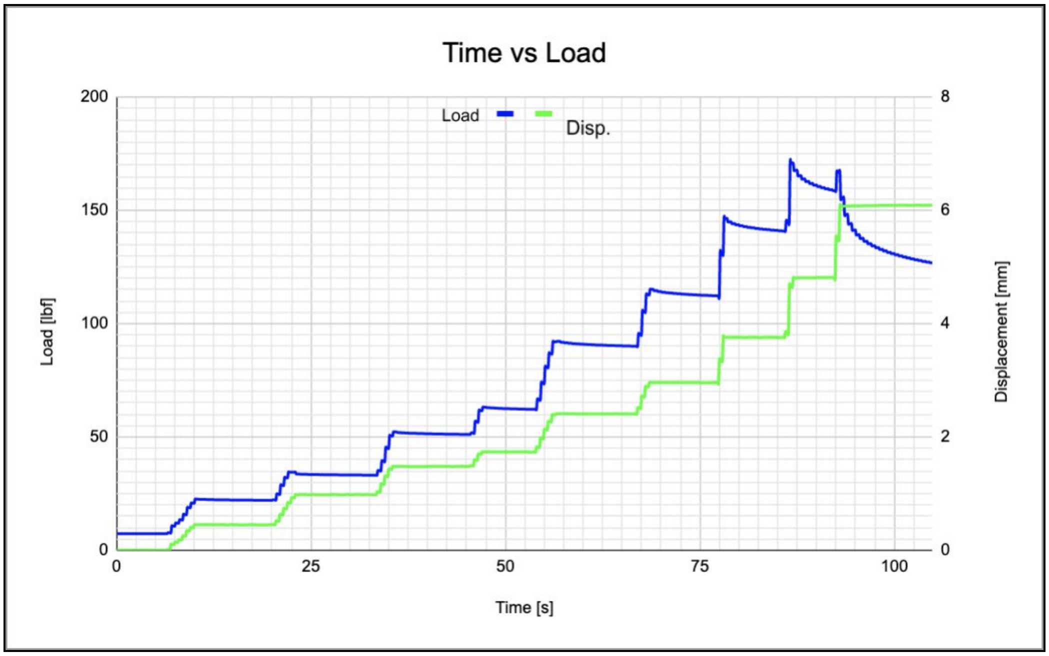


Figure 27: Load and Displacement of the bracket during testing

Figure 28: Displacement vs Load of the bracket during testing

Around 170 lbf the bracket had yet to fracture and unfortunately went into buckling causing the bracket itself to weaken and not be able to bear near the same load. As we went to increase the load back to ~165 lbf the bracket eventually snapped simultaneously in 4 different locations and is plastically deformed in the buckling shape seen in figure 23. A picture of the broken bracket can be seen below in Figure 29.

Figure 29: The broken bracket post testing procedures.

In any ideal situation a bracket should never have a concern of buckling as when something buckles in drastically decreases its ability to hold load and therefore will fail if the load is maintained or increased.

Table 2: Summary of Bracket Testing

Failure Mode	Buckling with shattering fracture
Failure Load	170 lbf
Assembly Time	59.11 seconds

4.0 Bracket Improvements

From design and FEA we did not attempt to try and fracture our bracket in a specific location as we anticipated one of the stress concentrations to turn into a fracture. In the future it would be optimal to design a distinct location that would result in the bracket to fail and a desired load. In addition ensuring the safety factor on buckling was much higher than the stresses to completely ensure there is no chance of buckling.

References

[1]"Strength at Break Tensile", *Omnexus.specialchem.com*, 2019. [Online]. Available: https://omnexus.specialchem.com/polymer-properties/properties/strength-at-break-tensile. [Accessed: 28- Nov- 2019]

[2]"Cantilever Beams - Moments and Deflections", *Engineeringtoolbox.com*, 2019. [Online]. Available: https://www.engineeringtoolbox.com/cantilever-beams-d_1848.html. [Accessed: 28- Nov- 2019]

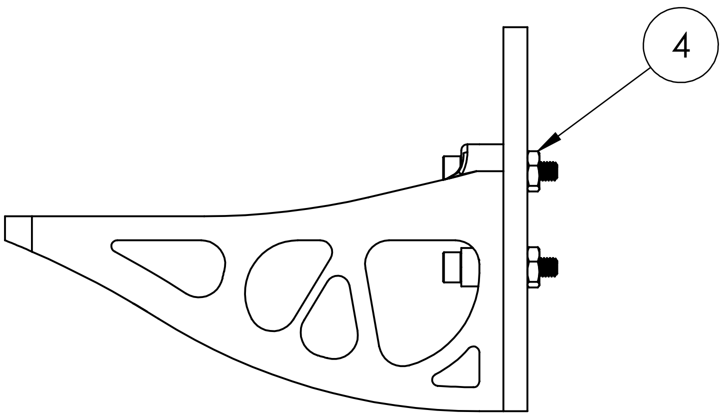
[3]*Cdn3.bigcommerce.com*, 2019. [Online]. Available: https://cdn3.bigcommerce.com/s-2zb8s6li/product_images/uploaded_images/p430m.gif. [Accessed: 28- Nov- 2019]

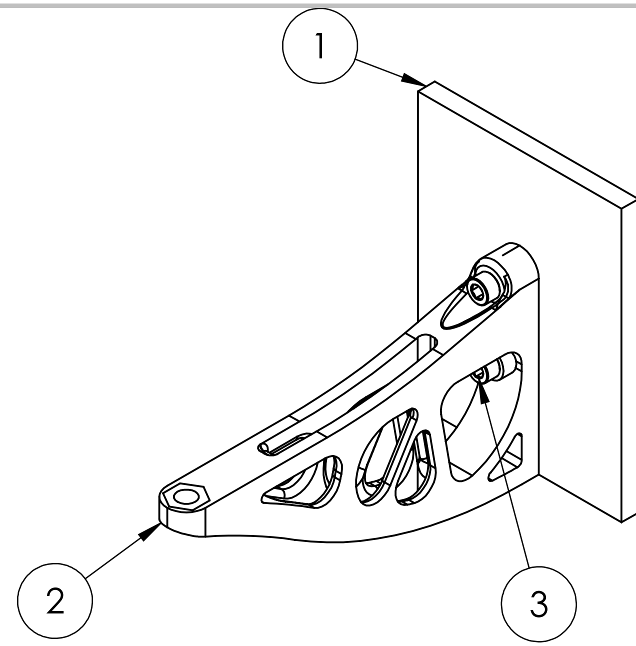
[4]"3D Print Orientation and the Effect on Part Strength | Hardware, Software & Product Development | Sparx Engineering", *Sparxeng.com*, 2019. [Online]. Available: https://www.sparxeng.com/blog/mechanical/3d-print-orientation-effect-part-strength. [Accessed: 28- Nov- 2019]

[5]"How to 3D Print Jigs & Fixtures, Part 2: Designing for Print", grabcad.com, 2019. [Online]. Available:https://grabcad.com/tutorials/how-to-3d-print-jigs-fixtures-part-2-designing-for-print. [Accessed: 28- Nov- 2019]

[6]"How does part orientation affect a 3D print", *3dhubs.com*, 2019. [Online]. Available: https://www.3dhubs.com/knowledge-base/how-does-part-orientation-affect-3d-print/ [Accessed: 28- Nov- 2019]

Appendix A - Drawing Package





ITEM NO.	PART NUMBER	QTY.
1	BackPlate	1
2	Bracket	1
3	Bolt	2
4	Nut	2

NEXT ASSY	USED ON
APPLICATION	

UNLESS OTHERWISE SPECIFIED:

DIMENSIONS ARE IN INCHES
TOLERANCES:
FRACTIONAL ±
ANGULAR: MACH ± BEND ±
TWO PLACE DECIMAL ±
THREE PLACE DECIMAL ±

INTERPRET GEOMETRIC TOLERANCING PER:

MATERIAL

FINISH

DO NOT SCALE DRAWING

	NAME	DATE
DRAWN		
CHECKED		
ENG APPR.		
MFG APPR.		
Q.A.		

COMMENTS:

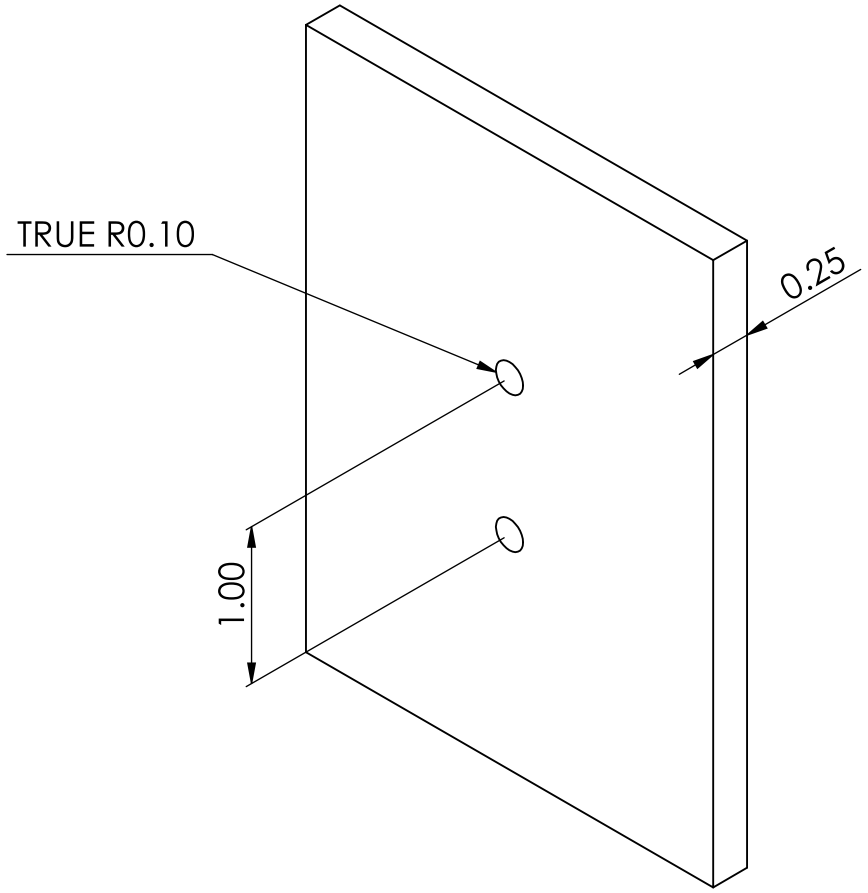
M16

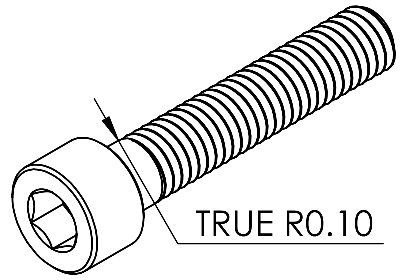
TITLE:

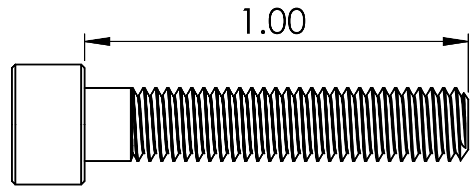
Assembly Drawing

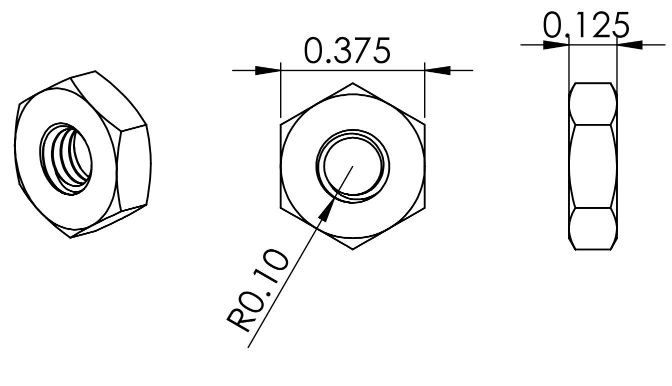
SIZE **A** DWG. NO. DRW 1 REV

SCALE: 1:1 SHEET 1 OF 3



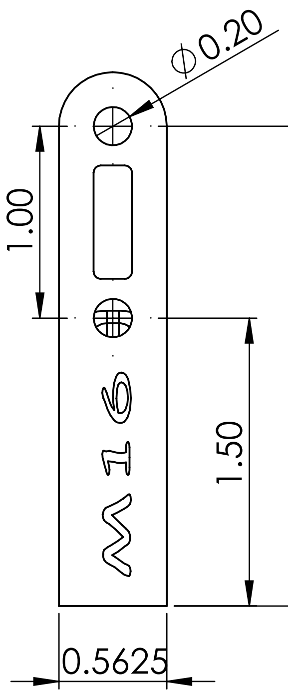


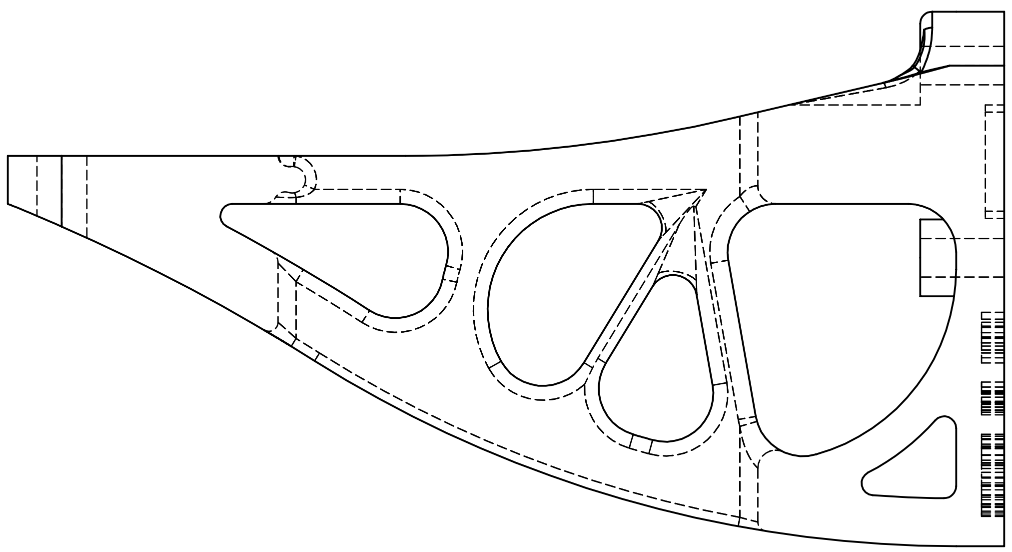


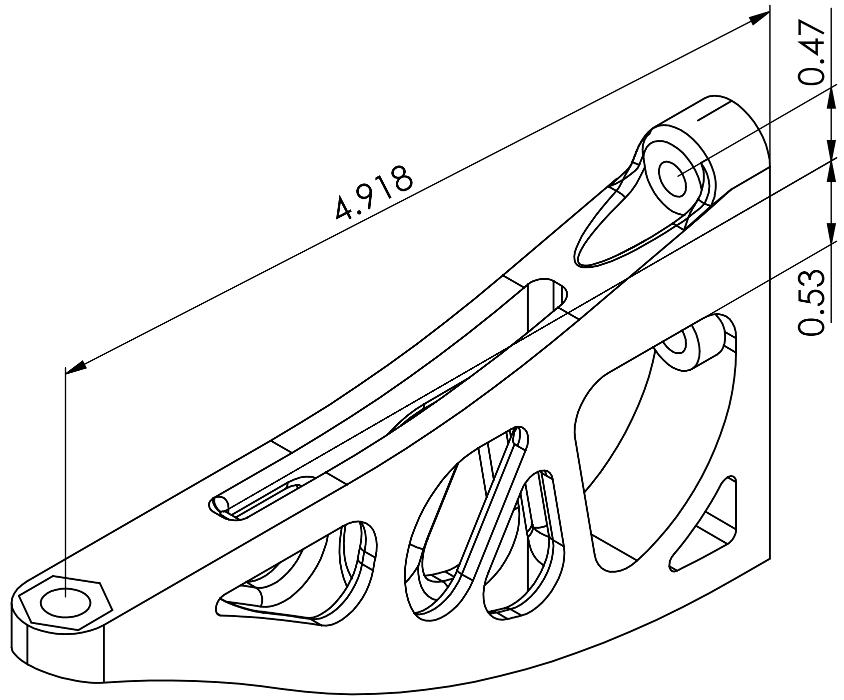


UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH: TOLERANCES: LINEAR: ANGULAR:	FINISH:	DEBURR AND BREAK SHARP EDGES	DO NOT SCALE DRAWING	REVISION

	NAME	SIGNATURE	DATE		
DRAWN					TITLE: Nut, Bolt and Plate
CHK'D					
APPV'D					
MFG					
Q.A				MATERIAL:	DWG NO. DRW 2 — A3
				WEIGHT:	SCALE:1:1 — SHEET 2 OF 3







UNLESS OTHERWISE SPECIFIED: DIMENSIONS ARE IN MILLIMETERS SURFACE FINISH: TOLERANCES: LINEAR: ANGULAR:		FINISH:		DEBURR AND BREAK SHARP EDGES		DO NOT SCALE DRAWING	REVISION
	NAME	SIGNATURE	DATE			TITLE: Bracket	
DRAWN							
CHK'D							
APPV'D							
MFG							
Q.A				MATERIAL:		DWG NO. DRW 3	A3
				WEIGHT:		SCALE:1:1	SHEET 3 OF 3