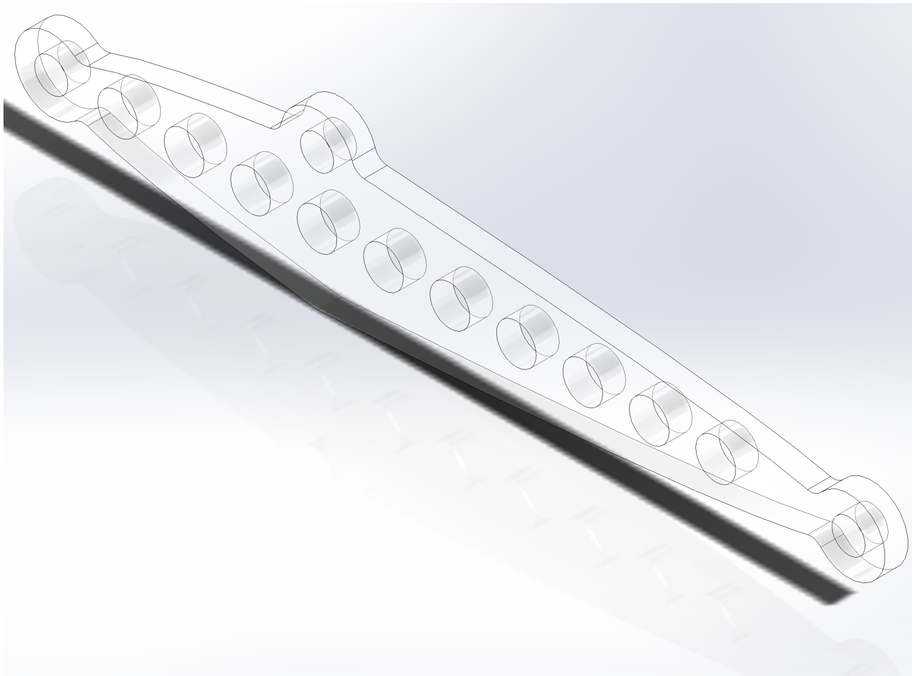
Project 1 Report Addendum

Oliver Zhang (ozz), Clement Wong (clementw)

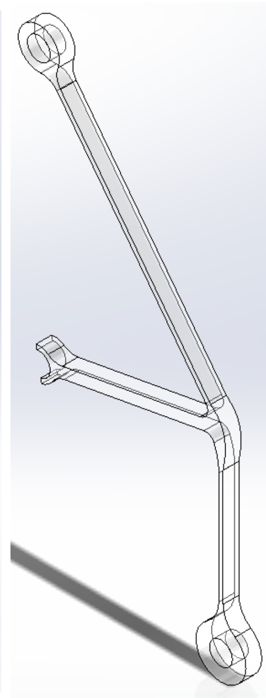
October 9, 2017

1 Summary

a. Isometric views -



Original



Final

b. Reasoning -
The original part was entirely under bending stress, which increases with distance from the supporting pegs. Adding holes only minimally reduces mass, and the real issue lies in the loading mode. We decided to overhaul our original design and try to minimize bending force. We took inspiration from the optimal design from test day. Tensile and compressive loading is uniform and more optimal, so the new design has all parts almost entirely under tension or compression, resulting in uniform stresses that use material efficiently. The angle of each member is optimized to create the lowest mass.

c. Mass estimation - 3.08 g
Using Solidworks mass properties volume is $2.59\,\text{cm}^3$.
From acrylic phyiscal properties specific gravity is 1.19.

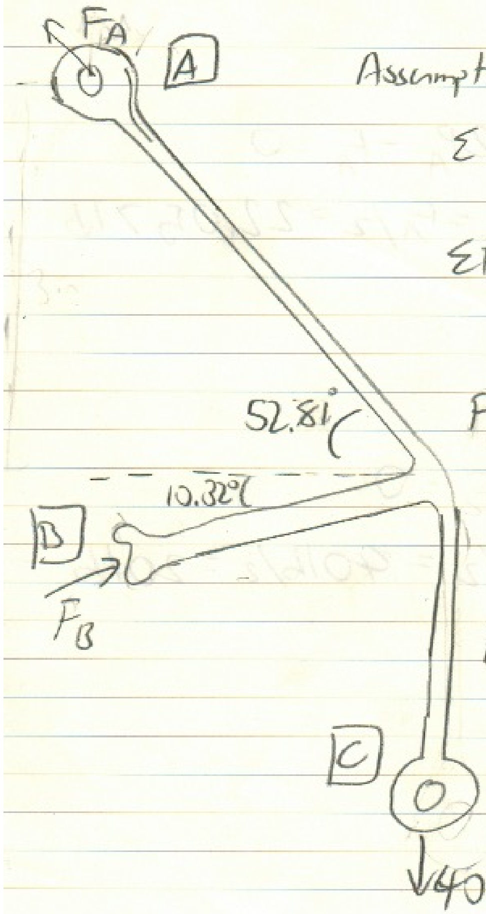
$$m = (\rho)(V)(SG) = (1\,\text{g/cm}^3)(2.59\,\text{cm}^3)(1.19) = 3.08\,\text{g}$$

d. Factor of safety estimation - 2.3
Calculated on page 14.

e. Failure mode prediction - The part will fail due to bending stresses where the three members meet, on the upper right.

2 Revised Free Body Diagrams

FBD (Entire Part)



Assumption: body is 3 two force members

$$\Sigma F_y = F_A \sin(52.81°) + F_B \sin(10.32°) - 40\,lb = 0$$

$$\Sigma F_x = -F_A \cos(52.81°) + F_B \cos(10.32°) = 0$$

$$F_A = F_B \cos(10.32°)/\cos(52.81°)$$

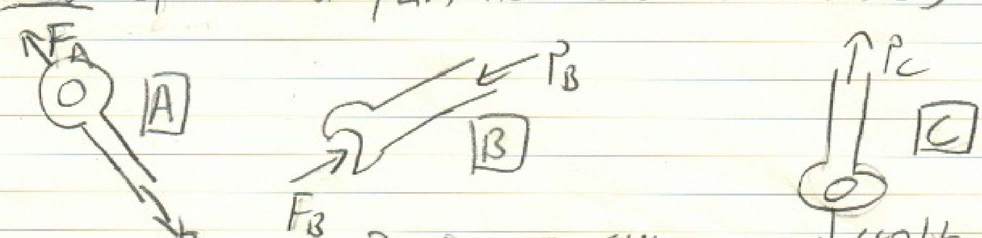
$$\frac{F_B \cos(10.32°)\sin(52.81°)}{\cos(52.81°)} + F_B \sin(10.32°) = 40\,lb$$

$$\boxed{F_B = 27.1056\,lb}$$

$$F_A = (27.1056\,lb)\frac{\cos(10.32°)}{\cos(52.81°)}$$

$$\boxed{F_A = 44.1141\,lb}$$

FBD (portion of part near each attachment)



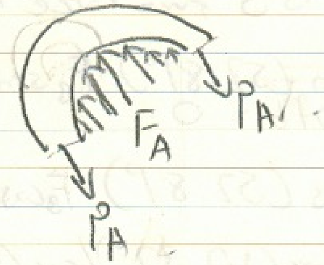
$P_B = F_B = 27.1056\,lb$

$P_A = F_A = 44.1141\,lb$

$P_C = 40\,lb$

FBD (Holes)

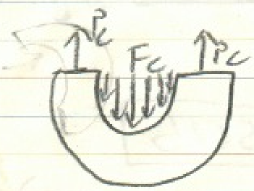
A



$\Sigma F = 2P_A - F_A = 0$

$P_A = F_A/2 = 22.057 lb$

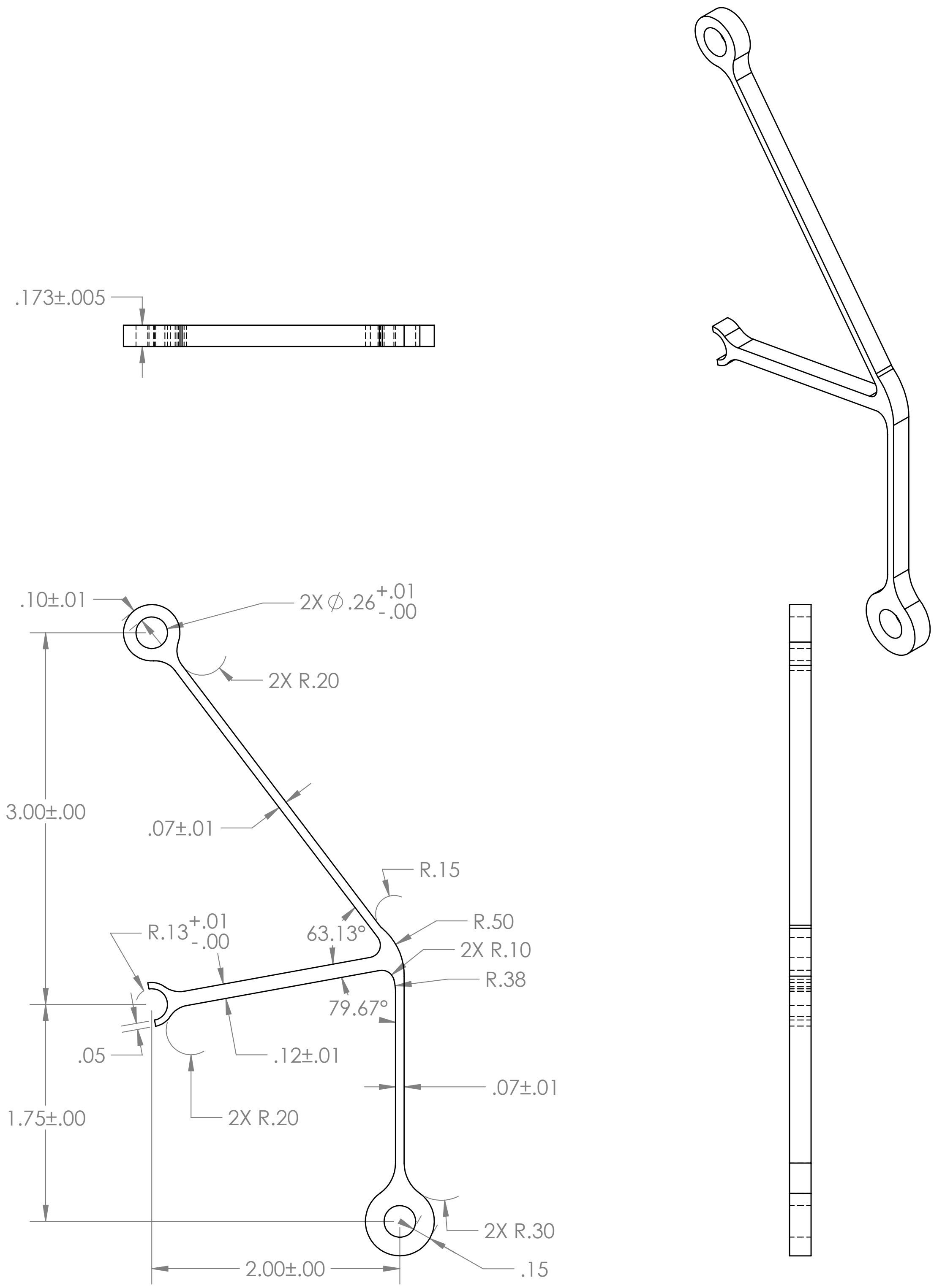
C



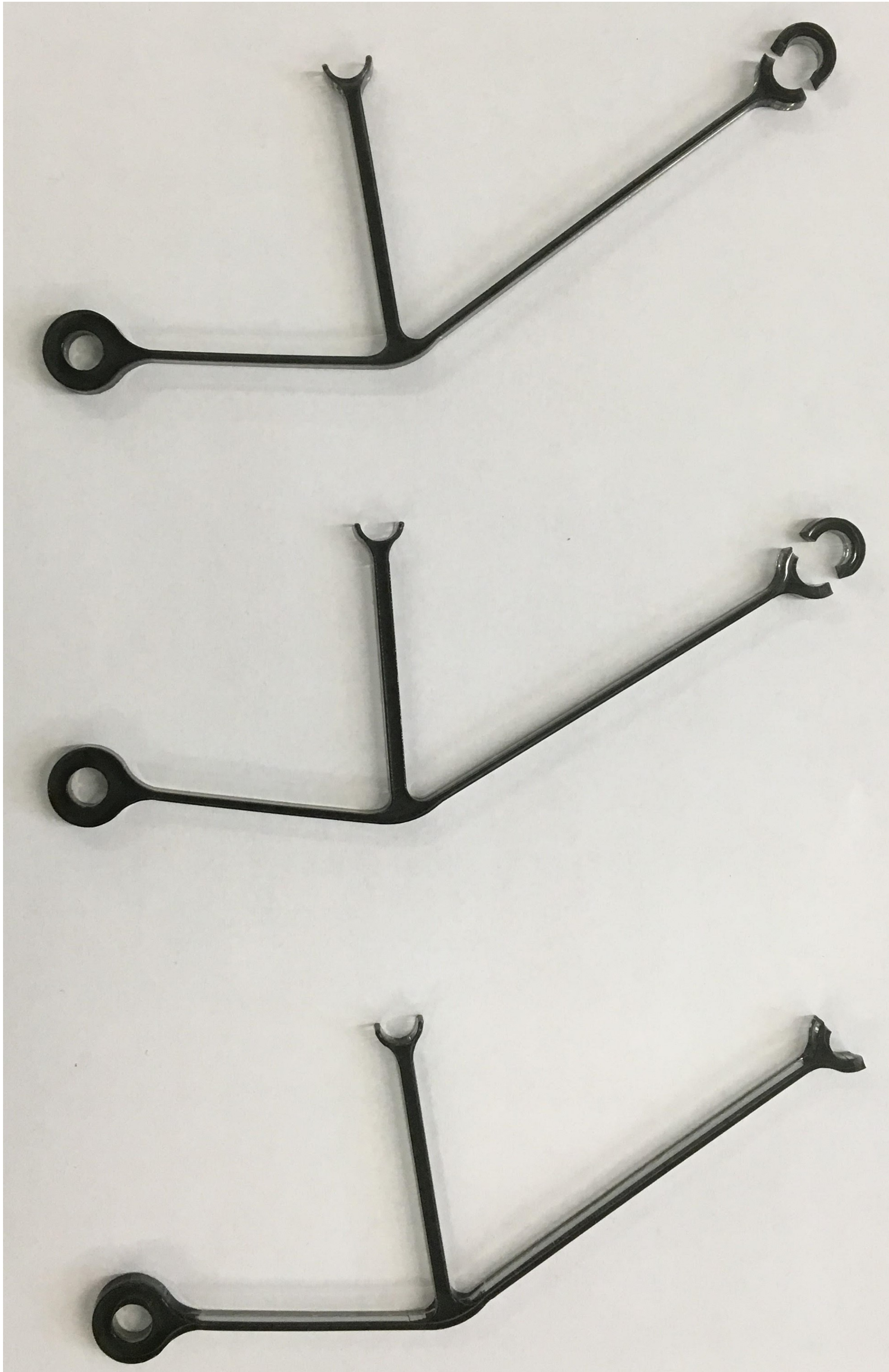
$\Sigma F = 2P_c - F_c = 0$

$P_c = F_c/2 = 40 lb/2 = 20 lb$

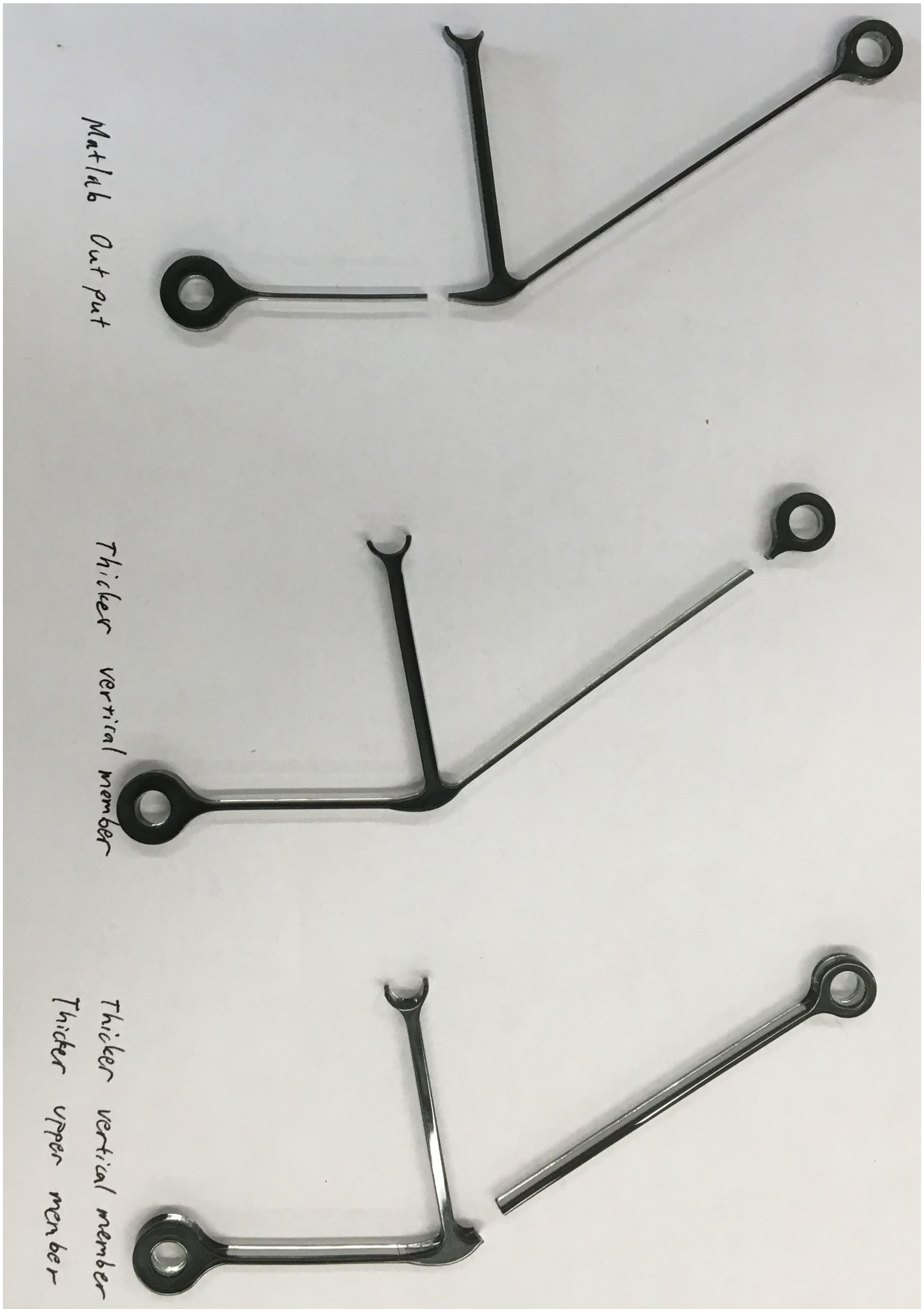
3 Engineering Drawing



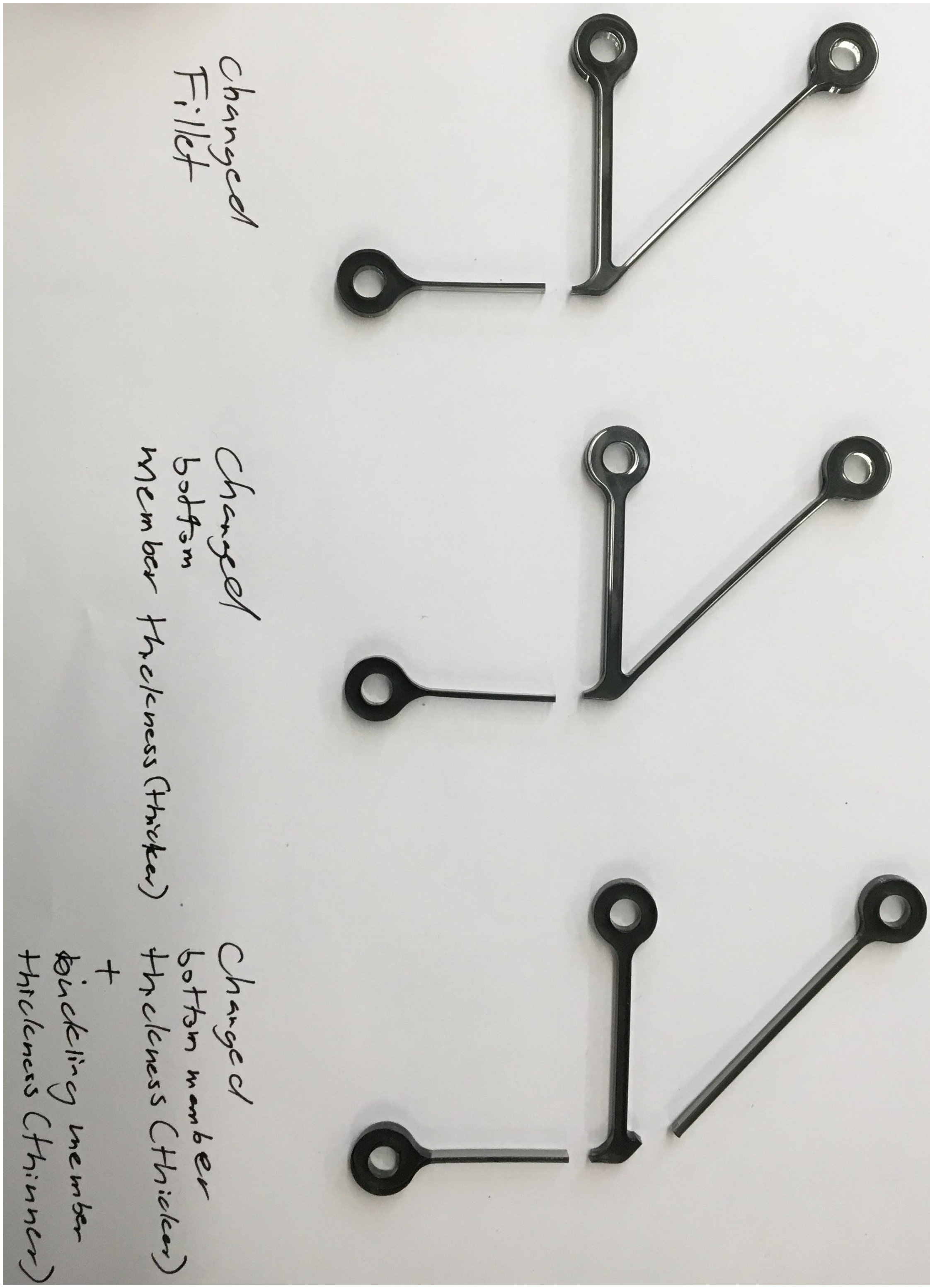
4 Photographs of Prototypes



Testing three cuts of the final design. All failed at the top peg.



Testing how modifying the dimensions will affect failure mode. The right most part failed due to error during loading.



Testing earlier horizontal compressive member design. The failure modes agreed with the FEA analysis very well.

Screenshot of a slow motion video of an early design where we forgot to consider buckling. The part failed due to the compressive member being too thin to resist buckling.

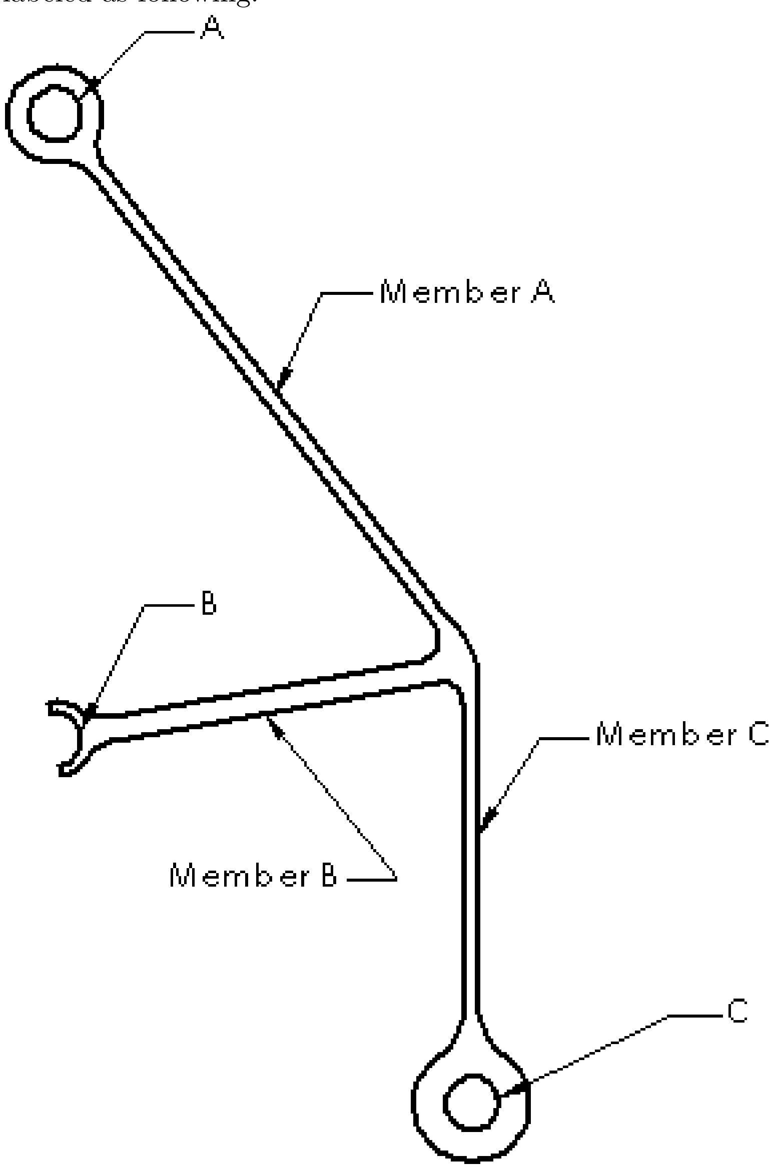
5 Supporting Materials

5.1 Stress Analysis

Because our final design was so drastically different from our original design here we will include various analysis we performed to come to this result.

5.1.1 Hand Analysis

We performed hand analysis to get stress equations for each member. They are typeset below for ease of viewing. Scans of hand written notes are in the Handwriting Appendix. The members are labeled as following:



Member A is approximated as a two-force member under tension. Since it is only under an axial load, we can assume bending stress is zero. Thus the stress equation is:

$$\sigma_A = \frac{P_A}{w_A t}$$

Where w_A is the width of the member and t the thickness of the material.

Member C is also two-force, under a single vertical load, also in tension. Thus the stress equation is:

$$\sigma_C = \frac{P_C}{w_C t}$$

Member B is also two-force, but because it is under compression, buckling needs to be considered. The compressive stress equation is:

$$\sigma_B = \frac{P_B}{w_B t}$$

The buckling equation is:

$$P_{cr} = \frac{\pi^2 EI}{L_B} = \frac{\pi^2 E \frac{1}{12} t {w_B}^3}{L_B}$$

Plugging in our actual material properties, we find that width needs to be much thicker to avoid buckling than to sustain compressive stress, so buckling is the dominant failure mode in member B.

5.1.2 Inverse Analysis

The system cannot be solved explicitly with one variable because there are too many unknowns. However, using Matlab, an optimal solution can be found.

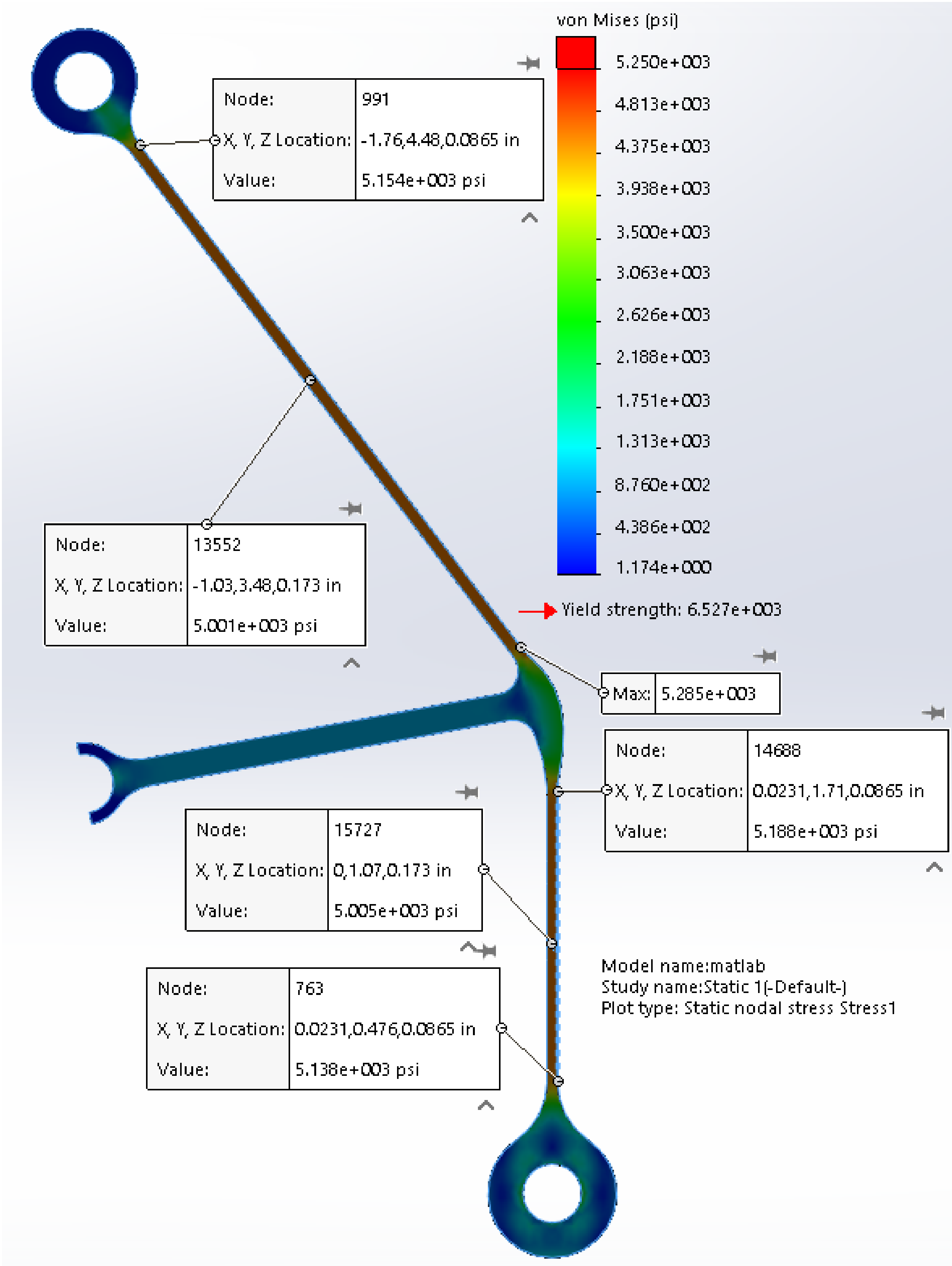
In the code on the next page, the top most peg is defined as A, the middle peg defined as B, and the bottom peg defined as C. We are trying to optimize two values, the separation between the pegs, named h, and the length of member C above peg B, named x. These dimensions will define the angles of the members and thus their loading. Using the dominant stress equations in each member, the ideal dimensions can be found.

We chose a factor of safety of 2 because we did not have a part to test for our main report. We want to guarantee we can get full score on nominal part performance, so we did not go for a competitive FoS.

The final results were $h = 3\,\text{in}$ and $x = 0.3644\,\text{in}$.

```
%Variables
hs = 0.25:0.25:3.25; %Distance between support pegs A and B
x = linspace(0,3.25,1000); %Length of vertical member above peg B
V = zeros(numel(hs),numel(x)); %Set up result matrix

%Parameters
FoS = 2; %Different FoS give different optimal angles.

t = 0.173; %Thickness of material
P = 40; %Load at peg C
Sy = 10000; %Yield stress
S = Sy / FoS; %Nominal stress
D = 2; %Separation between peg A/B and peg C
y = 1.75; %length of vertical member below peg B
E = 480000; %Elastic modulus

for i = 1:numel(hs)
    h = hs(i);

    %Angles
    theta = atan((h-x)./D); %Angle from the horizontal of member A (clockwise)
    phi = atan(x./D); %Angle from the horizontal of member B (counter clockwise)

    %Member C equations (tensile stress)
    Ac = P/S; %Cross sectional area
    Wc = Ac/t; %Width of member C
    Lc = y + x; %Length of member C
    Vc = Lc .* (Wc * t); %Volume of member C

    %Member B equations (buckling load factor)
    Fb = P./(sin(phi) + cos(phi).*tan(theta));
    Pcrit = Fb .* FoS; %Critical force desired
    Lb = sqrt(x.^2 + D.^2);
    Wb = (Pcrit .* (Lb./(pi^2*E/12*t))).^(1/3);
    Vb = Lb .* Wb .* t;

    %Member A equations (tensile stress)
    Fa = Fb .* cos(phi) ./ cos(theta);
    Aa = Fa./S;
    Wa = Aa/t;
    La = sqrt(D^2 + (h-x).^2);
    Va = La .* Wa .* t;
    V(i,:) = Vc + Va + Vb;
end

[value, index] = min(V(:));
[row, col] = ind2sub(size(V), index);
[h_min, x_min] = [hs(row), x(col)]; %Values were used to find ideal widths
```

5.1.3 FEA Analysis

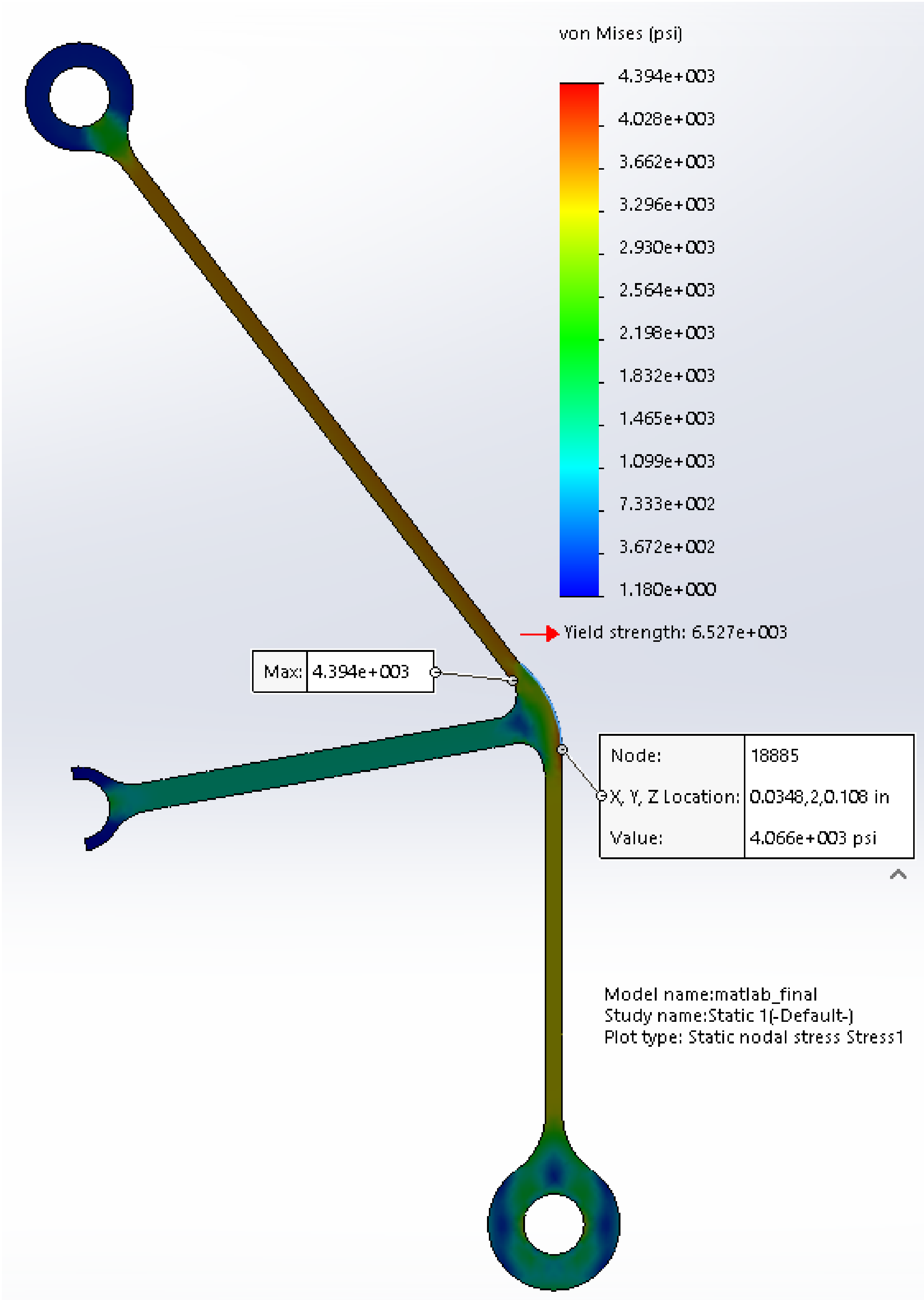


FEA analysis agreed quite well with our hand analysis. The stress in member A and C are both 5000 psi, which is half of the yield stress 10000 psi, giving a factor of safety of 2 as predicted. The FEA buckling study was not able to correctly model buckling.

However, our simple model does not consider bending effects at the joint where all three members meet, and at the ends where the member meets the piece surrounding the peg. Where the three members meet, stresses go up to 5285 psi, lowering the FoS to 1.89. At the ends, stresses go up to 5150 psi.

To combat these stresses, we had added fillets at the ends of each member, and also increased the amount of material where all three members meet. To be even more certain we will pass the nominal performance testing, we added even more material to all the members, resulting in the final design. The FEA analysis is included on the next page.

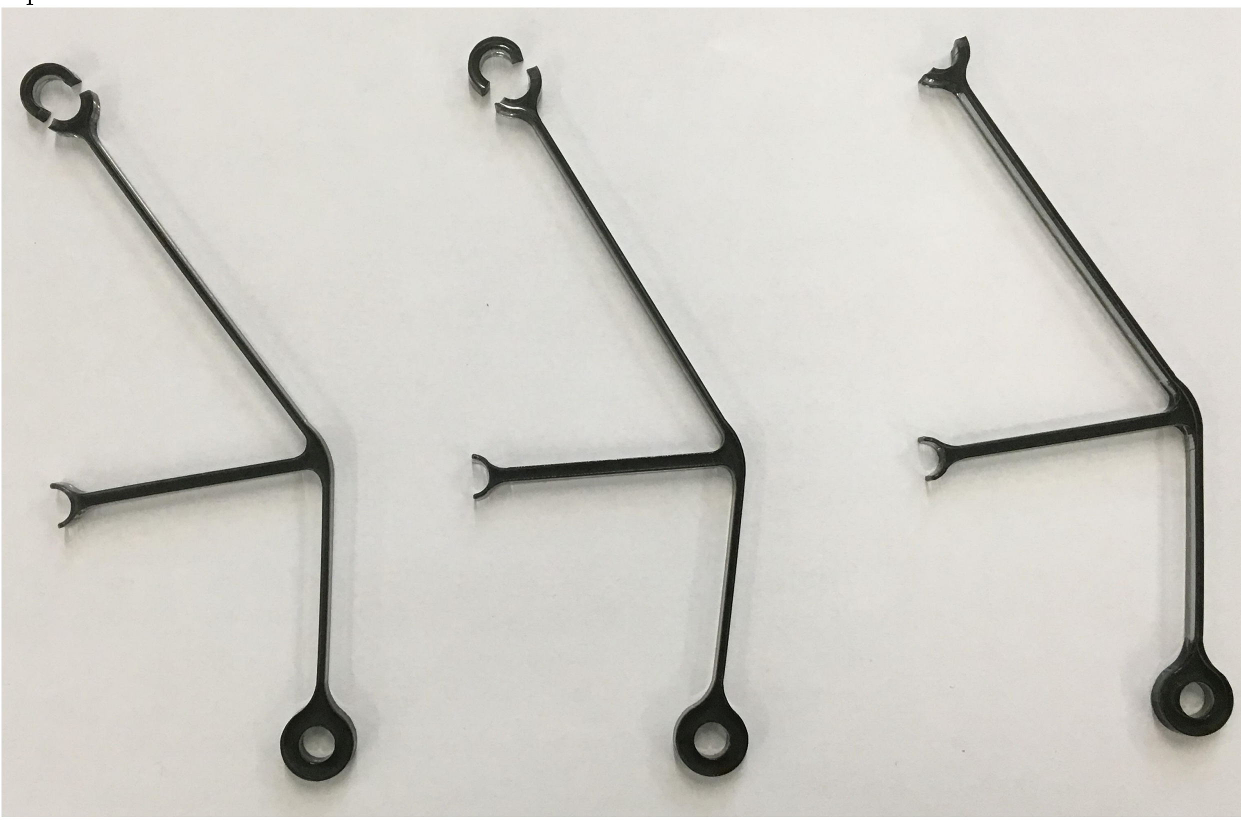
The max stress is still in the same location, so we expect the failure mode to be the same. Using this max stress, the factor of safety is 2.3.


von Mises (psi)
4.394e+003
4.028e+003
3.662e+003
3.296e+003
2.930e+003
2.564e+003
2.198e+003
1.832e+003
1.465e+003
1.099e+003
7.333e+002
3.672e+002
1.180e+000
Yield strength: 6.527e+003
Max: 4.394e+003
Node: 18885
X, Y, Z Location: 0.0348,2,0.108 in
Value: 4.066e+003 psi
Model name:matlab_final
Study name:Static 1(-Default-)
Plot type: Static nodal stress Stress1

5.1.4 Failure Analysis

Since the FEA analysis could better model bending stresses, it was used to predict failure. The stresses were highest where the three members meet, so we predicted that the material will fail there.

We cut out the same piece many times, to ensure we can consistently predict failure and pass nominal performance tests. These photos were also included in the photo section earlier in the report.

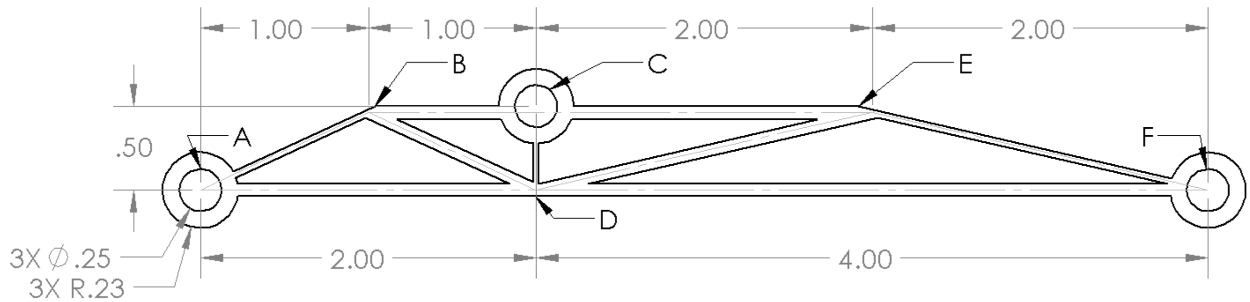


In the actual testing, all parts failed at peg A due to contact stresses rather than where we predicted. We believe this is due to us not over-sizing the hole, and the press fit caused very high contact stresses that caused the part to fail. Since Solidworks static study does not model this at all, it was not reflected in our FEA analysis. For our final design, we enlarged the hole by .01 inches, to ensure contact stresses will remain insignificant as predicted.

5.2 Rejected designs

We considered various other designs that were determined to be suboptimal. Each subsection below will describe the design and why we decided to reject it.

5.2.1 Truss design



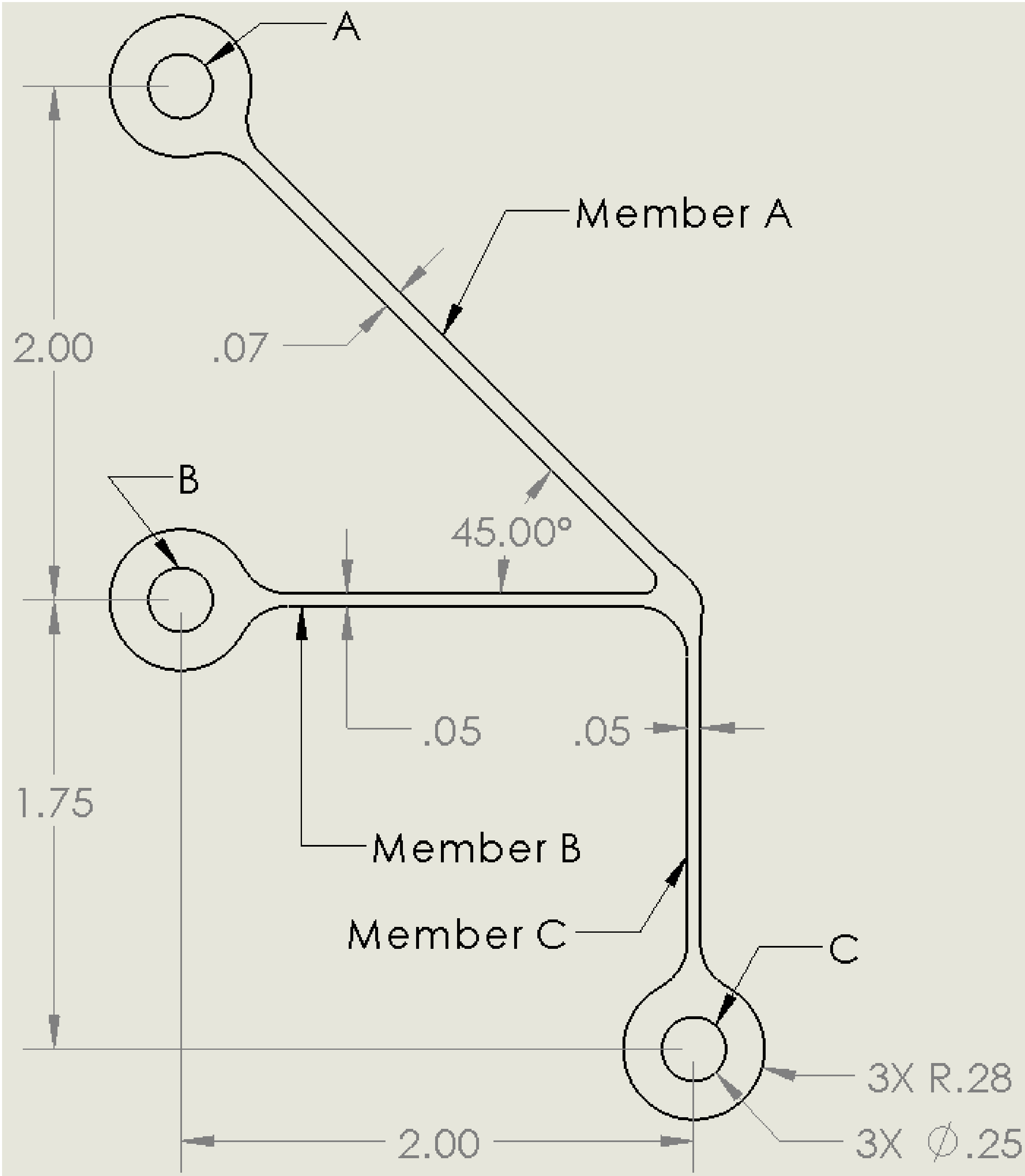
This is an attempt to improve our original simply supported beam design. We were inspired by the best design's use of two force members. All members in a truss would be under either tension or compression, solving the issue of large bending stresses. Using this assumption, all internal loads can be solved. The scanned hand analysis is shown in the Handwritten Appendix.

Solving all the forces we found these forces:

$$F_{AB} = 59.62\,\text{lb} \quad \text{(compression)}$$
$$F_{AD} = 53.332\,\text{lb} \quad \text{(tension)}$$
$$F_{BC} = 106.65\,\text{lb} \quad \text{(compression)}$$
$$F_{BD} = 59.62\,\text{lb} \quad \text{(tension)}$$
$$F_{CD} = 40\,\text{lb} \quad \text{(compression)}$$
$$F_{CE} = 106.65\,\text{lb} \quad \text{(compression)}$$
$$F_{DE} = 54.957\,\text{lb} \quad \text{(compression)}$$
$$F_{DF} = 53.332\,\text{lb} \quad \text{(tension)}$$
$$F_{EF} = 54.957\,\text{lb} \quad \text{(tension)}$$

In the end we didn't pick this design because almost every member was longer and under more load than the final design. Half the members are also in compression, requiring more thickness to combat buckling. The task of bridging a 6 inch gap is just far more difficult than dealing with just a 2 inch separation, so we decided to overhaul our design.

5.2.2 The horizontal compression member design



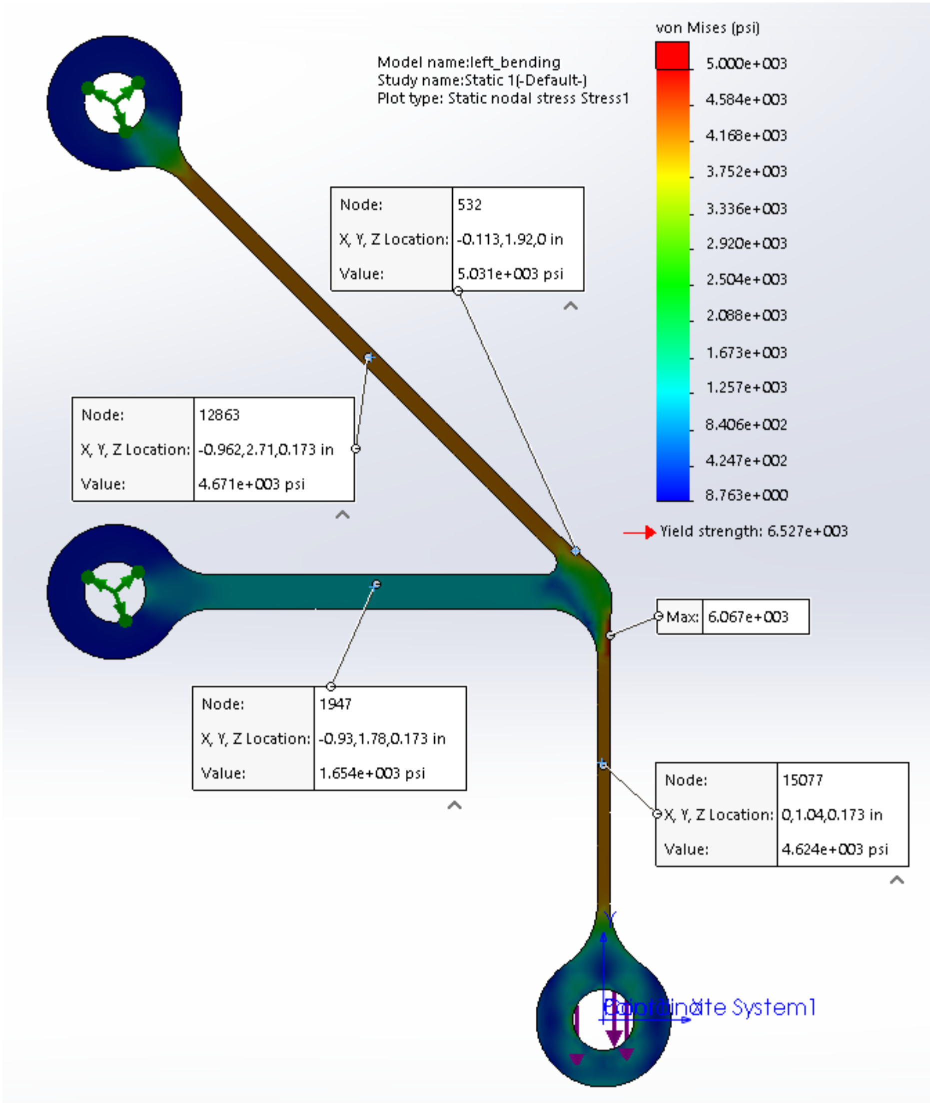
This is the design we considered before the final design. We were inspired by the lowest mass bracket on test day, but we wanted to see if there were any possible improvements to it. Since member B is in compression to balance the moment, we thought it would be most efficient at horizontal, where its length is shortest. The bracket is analyzed by approximating every part as two-force. We optimized angle theta to minimize mass:

$$
\begin{aligned}
&F = \frac{P}{\sin\theta}, L = \frac{S_x}{\cos\theta}, A = \frac{F}{\sigma} \\
&m = \rho L A = \rho L \frac{F}{\sigma} = \rho \frac{S_x}{\cos\theta} \frac{P}{\sigma \sin\theta} \\
&m \propto \frac{1}{\cos\theta \sin\theta}
\end{aligned}
$$

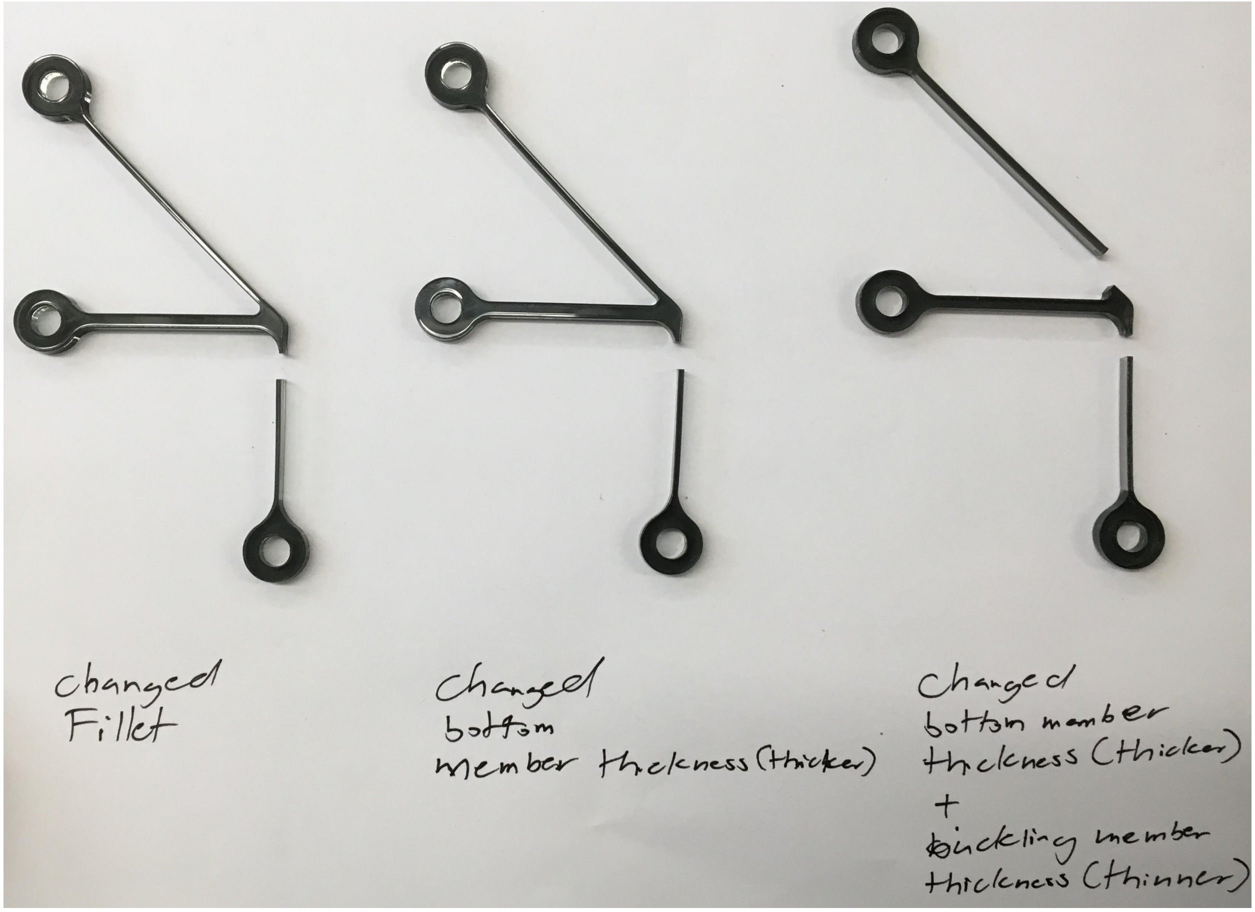
Which gives $\theta = 45^\circ$ as the ideal angle θ.

We then performed inverse analysis to determine the appropriate widths. Since the stress equations and calculations are very similar to our final design, we will not repeat them here.

The FEA agrees with our analysis and our two-force member assumption. The peak stresses were at the outside edge where the three members meet, suggesting failure there. The FEA is shown on the next page.


von Mises (psi)
5.000e+003
4.584e+003
4.168e+003
3.752e+003
3.336e+003
2.920e+003
2.504e+003
2.088e+003
1.673e+003
1.257e+003
8.406e+002
4.247e+002
8.763e+000
Yield strength: 6.527e+003
Model name:left_bending
Study name:Static 1(-Default-)
Plot type: Static nodal stress Stress1
Node: 532
X, Y, Z Location: -0.113,1.92,0 in
Value: 5.031e+003 psi
Node: 12863
X, Y, Z Location: -0.962,2.71,0.173 in
Value: 4.671e+003 psi
Max: 6.067e+003
Node: 1947
X, Y, Z Location: -0.93,1.78,0.173 in
Value: 1.654e+003 psi
Node: 15077
X, Y, Z Location: 0,1.04,0.173 in
Value: 4.624e+003 psi

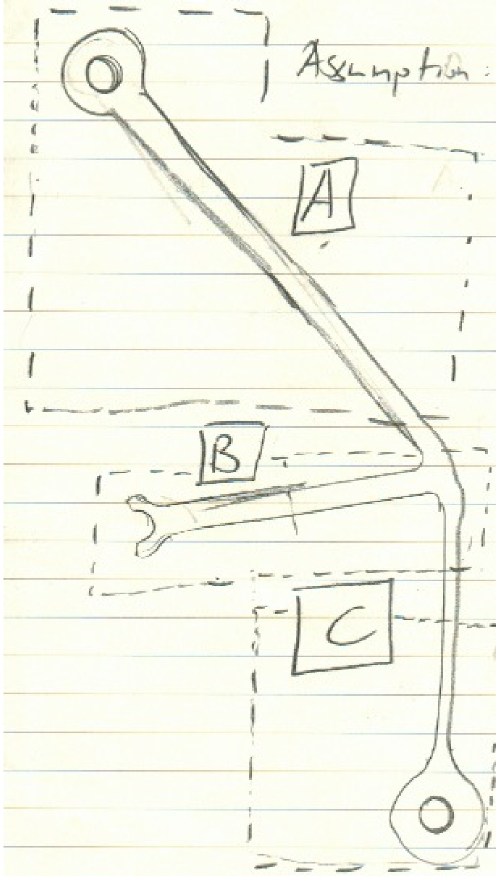
The actual testing reflected the max stress locations. Every part failed in the predicted location:



Note that the right-most bracket failed in the upper location first, since if had failed on the lower point first, there will no longer be a load on the upper parts of the bracket.

In the end, we decided that it was arbitrary to make the compressive member horizontal. Instead, an angled member could provide a force in the y-direction and perhaps reduce overall stress. This is when we got the idea to use Matlab to generate optimal dimensions.

5.3 Handwritten Appendix



Assumption: this part is comprised of 3 two-force members

Member A: two-force member under tensile stress

$$\sigma_A = \frac{P_A}{wt}$$

where w = width of A
t = thickness (.173 in)

Member B: two force member under compressive stress

P_B P_B

Since in compression, buckling stress is the important stress to consider

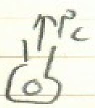
$$P_{cr} = \frac{\pi^2 EI}{\ell_b} = P_B \times F.O.S$$

$$P_B = \frac{\pi^2 E(\frac{1}{12} t w^3)}{\ell_b (F.O.S)}$$

$$\sigma_B = \frac{P_B}{wt} = \frac{\pi^2 E(\frac{1}{12} t w^3)}{\ell_b w t (F.O.S)}$$

$$\sigma_B = \frac{\pi^2 (E)(\frac{1}{12} w^2)}{\ell_b (F.O.S)}$$

Member C: two force member under tensile stress

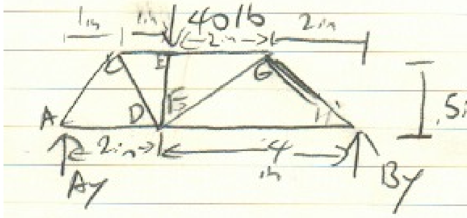


$$\sigma_C = \frac{P_C}{w_c t}$$

where w_c = width of C
t = thickness (.173 in)

Truss Design FBD

Assumption: all parts are two force members



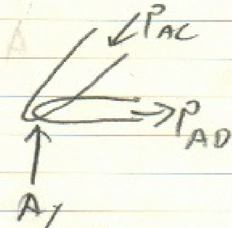
$\Sigma F_y = A_y + B_y - 40\,lbs = 0$

$A_y + B_y = 40\,lb$

$\Sigma M_A = B_y(6\,in) - 40\,lb(2\,in) = 0$

$B_y = 13.33\,lb$

$A_y = 26.66\,lb$

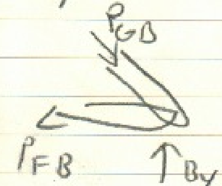


$\Sigma F_y = A_y - P_{AC}\left(\frac{.5}{\sqrt{1^2+.5^2}}\right) = 0$

$P_{AC} = 59.621\,lbs$ compression

$\Sigma F_x = P_{AD} - P_{AC}\left(\frac{1}{\sqrt{1^2+.5^2}}\right) = 0$

$P_{AD} = 53.332\,lb$ tension

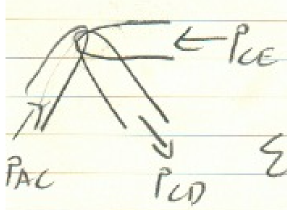


$\Sigma F_y = -P_{GB}\left(\frac{.5}{\sqrt{2^2+.5^2}}\right) + B_y = 0$

$P_{GB} = 54.9746\,lb$ compression

$\Sigma F_x = -P_{FB} + P_{GB}\left(\frac{2}{\sqrt{2^2+.5^2}}\right)$

$P_{FB} = 53.332\,lb$ tension

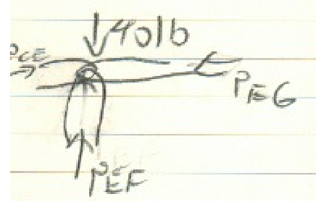


$\Sigma F_y = P_{AC}\left(\frac{.5}{\sqrt{1^2+.5^2}}\right) - P_{CD}\left(\frac{.5}{\sqrt{1^2+.5^2}}\right) = 0$

$P_{AC} = P_{CD} \Rightarrow P_{CD} = 59.621\,lb$ tension

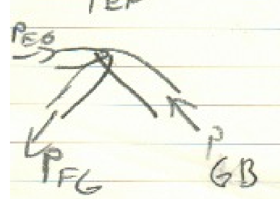
$\Sigma F_x = P_{CE} = P_{AC}\left(\frac{1\,in}{\sqrt{.5^2+1\,in^2}}\right) + P_{CD}\left(\frac{1\,in}{\sqrt{.5\,in^2+(1\,in)^2}}\right)$

$P_{CE} = 106.65\,lbs$ compression



$P_{EG} = P_{CE} = 106.65\,lb$ compression

$P_{EF} = 40\,lb$ compression



$\Sigma F_x = +P_{EG} - P_{GB}\left(\frac{2\,in}{\sqrt{.5^2+2\,in^2}}\right) - P_{FG}\left(\frac{2\,in}{\sqrt{(.5\,in)^2+(2\,in)^2}}\right)$

$P_{FG} = 54.957\,lb$ tension