Duke University

ME321 – Spring 2025

Pump Support Final Report

Cameron Galibois, Jay Parmar, and Chandler Wimmer (Group 8)
Due April 23, 2025

Contents

1 Introduction

The team was approached and tasked with designing a pump support structure that could enable the pump to be set at different heights. There will be a centrifugal pump head mounted to the end of the platform. The dimensions for the pump were pre-determined as well as its location relative to the bottom and front of the wall. This pump is driven by a motor via a belt and pulley system, and this action will generate forces and torques on our platform. In order to counteract these forces, as well as generally uphold the platform, support arms were manufactured. These arms were connected to the platform and wall mount through the use of clevis pins. These pins were selected given their ease of use and availability. Two key objectives of the project were to minimize both the weight and deflection of the platform, which go hand in hand as heavier platforms automatically cause more deflection.

With the outline of the system in place, the team began the design process by deriving expressions for the forces, moments, and torques in each unique component. These forces were then substituted into equations to find the appropriate stress in each component, wether it be axial, shear, bending, or torsional. At this stage, the team began iterative design by changing cross sectional geometries in order to minimize the weight of the system while still avoiding failure with a suitable safety factor. A crucial step in this step was selecting the material of each different component, as the material properties play a significant role in safety factor calculations.

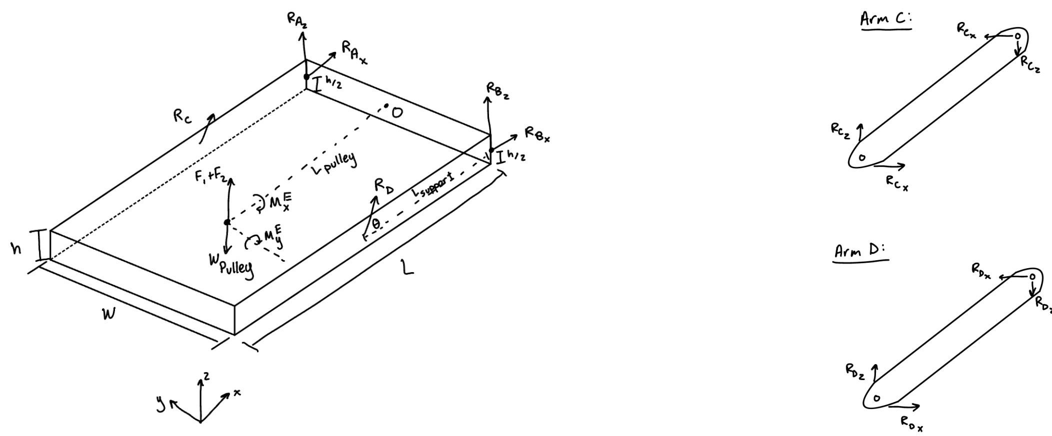
Once the dimensions, materials, and overall assembly details were finalized, the team took to Solidworks to model their work and perform finite element analysis. The first step was to create the assembly and ensuing drawings to document the connections and dimensions of the design. These drawings were used during the manufacturing process as a quick reference to ensure accuracy while machining. After the drawings and assembly were created, the structure underwent finite element analysis to simulate the loads that each part would experience in practice. The analysis provided an insight into the stresses and deflections in each component, which reinforced safety factor calculations and highlighted regions for potential weight reduction.

Once the FEA confirmed the safeness of the team's decision, the machining process began. The majority of the team's work was conducted using either a bandsaw to extract pieces of stock from a larger body or a manual mill to drill holes and create slots in the stock. These actions were carried out in the Pratt Student Shop with the use of the aforementioned drawings. After each component was created, the assembly was built and tested. Each of the steps described in this section is explored more in depth in the following sections, with the steps appearing in the report chronologically.

2 Equilibrium Calculations

2.1 Free Body Diagrams

The team was interested in finding the stresses at different critical locations throughout the system, so the first step in this process was to calculate the forces, moments, and reactions active in the assembly. In order to do so, the following free body diagrams for the arms and platform were utilized:



(a) Free Body Diagram for Platform

(b) Free Body Diagram for Support Arms

Figure 1: Free Body Diagrams of the Platform and Support Arms

2.2 Force, Moment, and Reaction Calculations

With these free body diagrams developed, the team created a system of equations that could be used to solve for the forces and moments of interest. The equations can be derived as follows:

$$\Sigma F_z = 0 = T_1 + T_2 - W_{pump} - S_1 * sin(\theta) - S_2 * sin(\theta) - W_{beam} + Wall_z \tag{1}$$

$$\Sigma F_x = 0 = S_1 * cos(\theta) + S_2 cos(\theta) + Wall_{x1} + Wall_{x2} \tag{2}$$

$$\Sigma(M_o)_y = 0 = M_G + (T_1 + T_2) * L_{pumptowall} - (S_1 * sin(\theta) + S_2 * sin(\theta)) * L_{support} - W_{beam} * \frac{L_{beam}}{2} \tag{3}$$

$$\Sigma(T_o)_x = 0 = T_G - (S_1 * sin(\theta) + \frac{Wall_z}{2}) * \frac{width_{beam}}{2} + (S_2 * sin(\theta) + \frac{Wall_z}{2}) * \frac{width_{beam}}{2} \tag{4}$$

$$\Sigma(M_o)_z = 0 = (S_1 * cos(\theta) - S_2 * cos(\theta) + Wall_{x1} - Wall_{x2}) * \frac{width_{beam}}{2} \tag{5}$$

The values that these variables represent are tabulated below.

Variable	Value
T_1	Torque 1
T_2	Torque 2
W_{pump}	Pump weight
S_1	Force in arm C
S_2	Force in arm D
θ	Angle between arm and platform surface
W_{beam}	Weight of the platform
$Wall_z$	Reaction force at wall in z-direction
$Wall_{x1}$	Reaction force at A in x-direction
$Wall_{x2}$	Reaction force at B in x-direction
M_G	Moment from the pump
$L_{pump\ to\ wall}$	Distance from the pump to the wall
$L_{support}$	Length from the wall to the support arm connection to platform
L_{beam}	Length of the platform
T_G	Torque from the pimp
$width_{beam}$	Width of the platform

Table 1: Variables and corresponding values from equilibrium equations

The components in the x and z directions for the support arms could also be calculated by multiplying the total force by the appropriate trigonometric function. A python script carrying out these computations can be found in the appendix (note there are numerical values assigned to variables to perform the operations).

2.3 Defined Calculations

Additionally, the team was tasked with calculating the fluid power and mechanical power. These values can be found using the following equations:

$$P_{fluid} = \frac{\rho g Q_{ft} h}{550} = 2.27HP \tag{6}$$

$$P_{mech} = \frac{P_f luid}{\eta_p} = 3.50HP \tag{7}$$

Where P_{fluid} is the fluid power, ρ is the density of water in $\frac{slugs}{ft^3}$, Q_{ft} is the flow rate in $\frac{ft^3}{sec}$, h is the final height, P_{fluid} is the fluid power, and η_p is the pump efficiency. The calculations are carried out more elaborately in a python script found in the appendix.

2.4 V-M Diagrams

The team also sought to develop shear and moment diagrams to help them understand the critical locations in their assembly. The general shape of the diagrams can be found in the figure below.

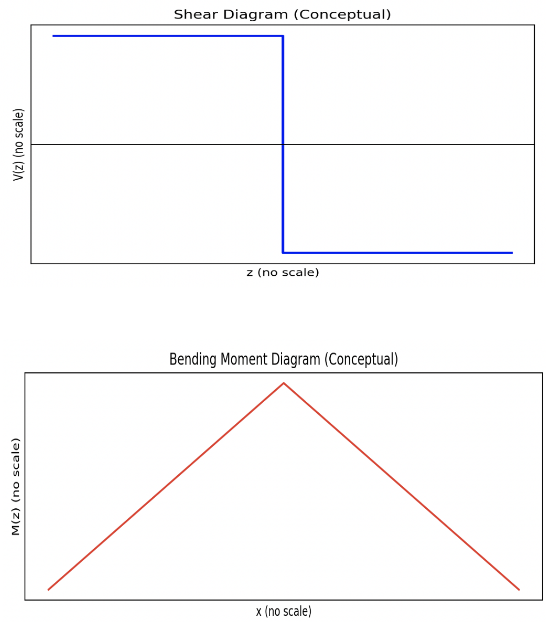


Figure 2: V-M Diagrams

While these pictures have no numerical values, their shape helps the team understand what part of the platform is under the greatest moment. The sharp change in the shear experienced by the platform can be attributed to the location of the support arms, and their affect can be seen in the moment diagram as well. To the left of the apex in the shear diagram, the tensions from the pump are the only forces in the z-direction. After the support forces are applied, the shear is brought back to 0 by the reaction forces at the wall.

3 Stress Analysis

3.1 Stress Type Identification

Based on the forces and moments found in section 2, the team proceeded by finding the stresses in each of the necessary components. Given the axial load applied to the support arms, the appropriate stress in these parts were compressive forces. This stress immediately alerted the team that they should be cautious of buckling and its catastrophic failure, so they designed the arms with this in mind. Next, the team proceeded with the platform, which was concluded to be experiencing a bending moment, torsion, transverse shear stress, and minor axial forces. The pins were determined to have shear stress, which the team was cautious of as the pins would have a relatively small cross sectional area.

3.2 Stress Concentration Factors

The stress concentrations in the assembly were found at the holes for the pins in the arms and platform. Since these holes were transverse, the key dimensions regarding the stress concentration factor were the diameter of the hole and the width of the part. For the arms, the forces were known to be axial, so the stress concentration factor could be approximated using the stress concentration table in the appendix. The holes in the platform were along the neutral axis of the part, so the bending moment at this location would not cause a concerning amount of stress.

3.3 Potential Critical Locations

The potential critical locations in the team's assembly were at the stress concentration factors, maximum bending moments, and pins. For the stress concentration locations, the compressive stress in the arms was amplified by the hole. A depiction of the hole in the arm can be seen in the figure below:

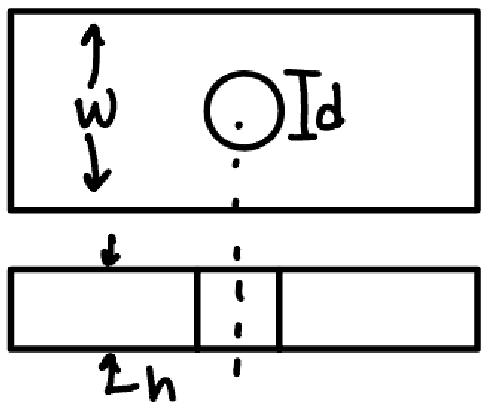


Figure 3: Transverse Holes in Arm

Another potential critical location is at the maximum bending moment in the platform. As previously mentioned, this occurs at the location where the support arms are joined to the platform. The bending moment and torsional shear stress are both maximized at the surface of the platform, so there would be another critical stress location here. Lastly, the pins are a critical stress location. Although they are only experiencing a shear force, the small cross sectional area of the parts causes a large shear stress.

3.4 Nominal Stress Expressions

The team wanted to express each of the stresses symbolically, so the following set of expressions was derived:

$$Arms: \tag{8}$$

$$\sigma_o = \frac{F}{A} = \frac{F}{h(w-d)} \tag{9}$$

$$Pins: \tag{10}$$

$$\tau_{max} = \frac{4}{3}\frac{V}{A} = \frac{4}{3}\frac{V}{\pi r^2} \tag{11}$$

$$Platform: \tag{12}$$

$$\sigma_b = \frac{M_{max} y_{max}}{I} = \frac{M_{max}\frac{h}{2}}{\frac{1}{12}bd^3} \tag{13}$$

$$\tau_{torsion} = \frac{Tr}{J} = \frac{(T_1 - T_2)R_{pulley}\frac{h}{2}}{J} \tag{14}$$

$$\tau_{max} = \frac{3}{2}\frac{V}{A} = \frac{3}{2}\frac{V}{h_{platform} * w_{platform}} \tag{15}$$

3.5 Maximum Shear and Principal Stresses

After these expressions were developed, the maximum shear stress and principal stresses were symbolically calculated using a Mohr's Circle and stress concentration factors. The work can be seen in the figure below.

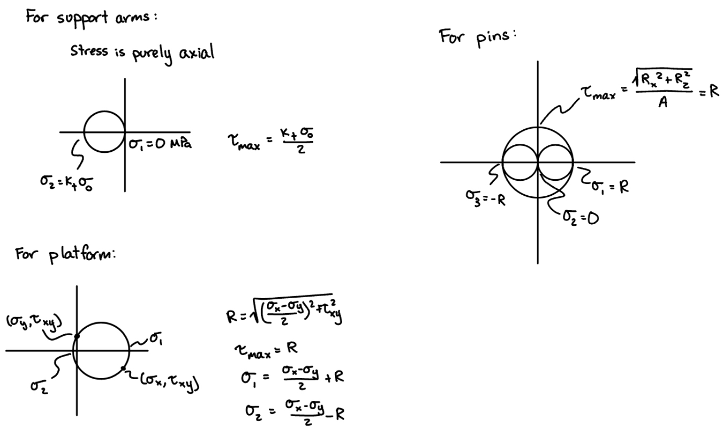


Figure 4: Mohr's Circles and Principal Stresses

4 Failure Determination

4.1 Factor of Safety

Using the stresses calculated in the previous section, the team wanted to find the safety factors for their design. The first iteration of this was done symbolically using the max shear stress theory and distrotion energy theory, as the team was only using ductile materials. From the max shear stress theory, the safety factor was calculated as

$$\tau_{max} = \frac{S_y}{2n} \tag{16}$$

$$n = \frac{S_y}{2\tau_{max}} \tag{17}$$

where τ_{max} is the maximum shear found using the Mohr's Circles above. For distortion energy theory, the Von Mises stress first needed to be calculated and then substituted into the proper expression. This work can be done as follows:

$$\sigma' = \frac{1}{\sqrt{2}}[(\sigma_x - \sigma_y)^2 + (\sigma_y - \sigma_z)^2 + (\sigma_x - \sigma_z)^2 + 6(\tau_{xy}^2 + \tau_{yz}^2 + \tau_{xz})]^{1/2} \tag{18}$$

$$n = \frac{S_y}{\sigma'} \tag{19}$$

4.2 Pump Deflection

Another key characteristic of the assembly is the maximum deflection in the pump platform. In order to approximate this deflection, the platform can be modeled as a cantilever beam experiencing intermediate loads. This maximum deflection can be calculated by using the relationship

$$M = EI\frac{d^2y}{dx^2} \tag{20}$$

This formula produces the maximum deflection as

$$y_{max} = \frac{F_1 a_1^2}{6EI}(a_1 - 3L) - \frac{F_2 a_2^2}{6EI}(a_2 - 3L) \tag{21}$$

The forces F_1 and F_2 belong to the pulley and support arms respectively, as do the dimensions a_1 and a_2. The dimensions in the formulas follow the outline in the image below.

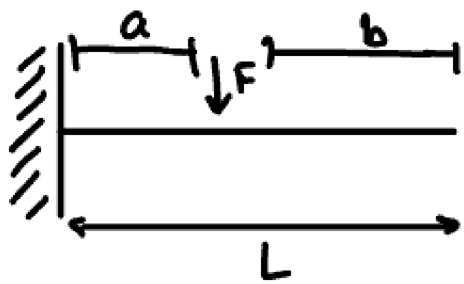


Figure 5: Intermediate Loads Dimensions

4.3 Critical Buckling Load

As mentioned above, the team was cautious of the compressive force in arms as they were liable to cause buckling. To check for this, the team calculated the critical buckling load using the Euler buckling formula for slender arms as

$$P_{cr} = \frac{\pi^2 EI}{L^2} \tag{22}$$

The team made sure to not make the arms too long to the point where the critical buckling load became less than the force that the arms were experiencing.

5 Design Iteration

5.1 Initial Geometry and Dimensions

All geometry dimensions and hole locations that depend solely on part shape are summarized in Table 2. No angled components are present in this design.

Table 2: Baseline GA2 Geometry

Component	Width (w)	Thickness (t)	Length (L)
Support Arm	1.00 in	0.145 in	18.00 in
Platform Plate	6.00 in	0.35 in	16.00 in
Clevis Pin Holes	Dia. $d = 0.25$ in,	centers 0.5 in from edges	1.25 in

5.2 Analytical FOS for Support Arms

Using the symbolic expressions from GA2, we substitute the baseline dimensions to verify compliance with the performance criteria. We begin by calculating the factor of safety for arm BF:

Arm dimensions: $t = 0.25'' = 6.35\,\text{mm}, \quad w = 0.50'' = 12.70\,\text{mm}, \quad d = 0.25'' = 6.35\,\text{mm}, \quad L = 23.8'' = 604.5\,\text{mm}.$ (23)

$$A_{BF} = (w - d)\,t = (12.70 - 6.35) \times 6.35 = 40.32\ \text{mm}^2 \tag{24}$$

$$\sigma_{\text{nom}} = \frac{F}{A_{BF}} = \frac{115\,\text{lbf} \times 4.448\,\text{N/lbf}}{40.32\,\text{mm}^2} \approx 12.68\ \text{MPa} \tag{25}$$

$$\sigma_{xx}^{BF} = K_t\,\sigma_{\text{nom}} = 4 \times 12.68 = 50.7\ \text{MPa} \tag{26}$$

$$\sigma_{\text{von}}^{BF} = \sqrt{\sigma_{xx}^2 - \sigma_{xx}\,\sigma_{yy} + \sigma_{yy}^2} \quad (\sigma_{yy} = 0) = 50.7\ \text{MPa} \tag{27}$$

$$S_y = 310\ \text{MPa} \quad \text{(yield of Al2024-T3)} \tag{28}$$

$$n_{BF} = \frac{S_y}{\sigma_{\text{von}}^{BF}} = \frac{310}{50.7} \approx 6.12 \tag{29}$$

This well exceeds the required factor of safety of 1.5.

5.3 Location 1: Analytical FOS for Platform

$$I_y = \frac{w}{12}\big(t^3 - d^3\big) = \frac{152.4}{12}\big(12.7^3 - 6.35^3\big) \approx 2.28 \times 10^4\,\mathrm{mm}^4, \tag{30}$$

$$c = \tfrac{t}{2} = 6.35\,\mathrm{mm}, \tag{31}$$

$$M = \big(115\,\mathrm{lbf} \times 4.448\,\tfrac{N}{\mathrm{lbf}}\big) \times 152.4\,\mathrm{mm} \approx 8.40 \times 10^4\,\mathrm{N\,mm}, \tag{32}$$

$$\sigma_{B0} = \frac{M\,c}{I_y} \approx \frac{8.40 \times 10^4 \cdot 6.35}{2.28 \times 10^4} = 23.4\,\mathrm{MPa}, \tag{33}$$

$$\sigma_B = K_b\,\sigma_{B0} = 2.1 \times 23.4 = 49.1\,\mathrm{MPa}, \tag{34}$$

$$A = (w - d)\,t = (152.4 - 6.35) \times 12.7 \approx 1.85 \times 10^3\,\mathrm{mm}^2, \tag{35}$$

$$\sigma_{\mathrm{ax,nom}} = \frac{F_x + F_z}{A} = \frac{218.1\,\mathrm{N}}{1.85 \times 10^3\,\mathrm{mm}^2} \approx 0.118\,\mathrm{MPa}, \tag{36}$$

$$\sigma_{\mathrm{ax}} = K_a\,\sigma_{\mathrm{ax,nom}} = 2.4 \times 0.118 = 0.28\,\mathrm{MPa}, \tag{37}$$

$$\sigma_x = \sigma_B + \sigma_{\mathrm{ax}} = 49.1 + 0.28 = 49.4\,\mathrm{MPa}, \qquad \sigma_y = 0, \quad \tau_{xy} = 0.47\,\mathrm{MPa}, \tag{38}$$

$$C = \frac{\sigma_x + \sigma_y}{2} = 24.7\,\mathrm{MPa}, \tag{39}$$

$$R = \sqrt{\Big(\tfrac{\sigma_x - \sigma_y}{2}\Big)^2 + \tau_{xy}^2} = 24.7\,\mathrm{MPa}, \tag{40}$$

$$\sigma_1 = C + R = 49.4\,\mathrm{MPa}, \tag{41}$$

$$\sigma_{\mathrm{von}} = \sqrt{\sigma_1^2 + \sigma_2^2 - \sigma_1\,\sigma_2} = 49.4\,\mathrm{MPa}, \tag{42}$$

$$S_y = 241\,\mathrm{MPa} \quad \text{(6061-T4)}, \tag{43}$$

$$n = \frac{S_y}{\sigma_{\mathrm{von}}} = \frac{241}{49.4} \approx 4.87. \tag{44}$$

This well exceeds the required factor of safety of 1.5.

5.4 Location 2: Analytical FOS for Platform

$$A_{(0,0;d/2)} = (w - d)\, t = (152.4 - 6.35) \times 12.7 = 1.856 \times 10^3 \text{ mm}^2, \tag{45}$$

$$\sigma_0 = \frac{F}{A_{(0,0;d/2)}} = \frac{218.08 \text{ N}}{1.856 \times 10^3 \text{ mm}^2} \approx 0.1175 \text{ MPa}, \tag{46}$$

$$\sigma_{\text{ax1}} = K_t\, \sigma_0 = 3.8 \times 0.1175 = 0.447 \text{ MPa}, \tag{47}$$

$$\tau_{1,\text{max}} = 0.22 \text{ MPa} \quad (\text{from } VQ/Itb, \text{ unchanged}), \tag{48}$$

$$C_1 = \frac{\sigma_{\text{ax1}}}{2} = 0.2235 \text{ MPa}, \tag{49}$$

$$R_1 = \sqrt{C_1^2 + \tau_{1,\text{max}}^2} = \sqrt{0.2235^2 + 0.22^2} = 0.3137 \text{ MPa}, \tag{50}$$

$$\sigma_1 = C_1 + R_1 = 0.5372 \text{ MPa}, \tag{51}$$

$$\sigma_2 = C_1 - R_1 = -0.0902 \text{ MPa}, \tag{52}$$

$$\sigma_{v,1} = \sqrt{\sigma_1^2 + \sigma_2^2 - \sigma_1\, \sigma_2} = 0.5876 \text{ MPa}, \tag{53}$$

$$S_y = 241 \text{ MPa} \quad (\text{yield of 6061-T4}), \tag{54}$$

$$n = \frac{S_y}{\sigma_{v,1}} = \frac{241}{0.5876} \approx 410. \tag{55}$$

This well exceeds the required factor of safety of 1.5.

5.5 Calculated Deflection for Platform

Assume the platform is a cantilevered beam with width $b = 12\,\text{in}$, thickness $h = 0.5\,\text{in}$, length $L = 18\,\text{in}$. Take $E = 29 \times 10^6$ psi for steel.

$$I = \frac{b\,h^3}{12} = \frac{12\,\text{in} \times (0.5\,\text{in})^3}{12} = 0.125 \text{ in}^4,$$

$$\delta_{\text{max}} = \frac{F\,L^3}{3\,E\,I} = \frac{37\,\text{lbf} \times (18\,\text{in})^3}{3 \times (29 \times 10^6\,\text{psi}) \times (0.125\,\text{in}^4)}$$
$$\approx 0.0198 \text{ in} = 0.50 \text{ mm},$$

which is below the 1 mm deflection limit.

5.6 Arm Buckling Calculations

The support arm is idealized as a prismatic column pinned at both ends, with length

$$L = 23.8'' = 604.5 \text{ mm}.$$

Euler's critical buckling load is

$$P_{\text{cr}} = \frac{\pi^2\, E\, I}{L^2},$$

where the second moment of area for our rectangular cross-section is

$$I = \frac{w\, t^3}{12},$$

with

$$w = 0.50'' = 12.7 \text{ mm}, \quad t = 0.25'' = 6.35 \text{ mm}, \quad E = 69 \times 10^3 \text{ N/mm}^2$$

(for Al 2024). Substituting,

$$I = \frac{12.7\,(6.35)^3}{12} \approx 2.71 \times 10^2 \text{ mm}^4,$$
$$P_{\text{cr}} = \frac{\pi^2\,(69 \times 10^3)\,(2.71 \times 10^2)}{(604.5)^2} \approx 5.05 \times 10^2 \text{ N}.$$

Each arm carries half the total vertical reaction, so the axial compressive load per arm is

$$P_{\text{ax}} = \frac{115\,\text{lbf} \times 4.448 \text{ N/lbf}}{2} \approx 255.8 \text{ N}.$$

Therefore the factor of safety against buckling is

$$n_{\text{buckling}} = \frac{P_{\text{cr}}}{P_{\text{ax}}} \approx \frac{505}{255.8} \approx 1.98.$$

This exceeds the required factor of safety of 1.5 by a small margin.

5.7 ANSYS Material Stress Plots

During our `Ansys Granta` material screening, we applied the following constraints and performance indices (Ashby material-selection charts) to identify the optimal alloy:

- **Yield-Strength Constraint.** To achieve a minimum factor of safety $n \geq 1.5$ under the maximum von Mises stress, we require
$$\sigma_y \;\geq\; n\,\sigma_{\text{vm,max}}.$$
This led us to select only materials with $\sigma_y 300\,\text{MPa}$ for the arms and $\sigma_y 240\,\text{MPa}$ for the platform.
- **Specific-Strength Index.** We plotted σ_y versus density ρ and maximized the ratio
$$M_1 = \frac{\sigma_y}{\rho},$$
so as to get the highest static strength per unit mass.
- **Specific-Stiffness Index.** To limit deflection of the cantilevered platform ($\delta_{\max} \leq 1\,\text{mm}$), we used
$$M_2 = \frac{E}{\rho}$$
(where E is Young's modulus), selecting materials that give maximum stiffness per unit mass.
- **Buckling Index.** For the slender support arms under compression, we considered Euler buckling and used
$$M_3 = \frac{\sqrt{E}}{\rho},$$
which maximizes the critical buckling load $P_{\text{cr}} \propto E\,I/L^2$ for minimum mass.
- **Corrosion-Resistance Filter.** We restricted candidates to aluminum alloys (2024-T3, 6061-T6) and stainless steels, excluding materials prone to rapid corrosion in wet environments.
- **Cost Constraint.** Finally, to stay under our \$150 raw-material budget, we filtered out any material whose cost per kilogram exceeded \$10 /kg.

These combined filters on the Granta/Ashby charts yielded aluminum 2024-T3 for the arms and aluminum 6061-T6 for the platform as the best trade-off between strength, stiffness, corrosion resistance, and cost."'

Figure 6 shows the ANSYS plot for relevant support arm materials graphed by price vs density. A plot for the platform is shown on Figure 7 and one for the pin is shown on Figure 8

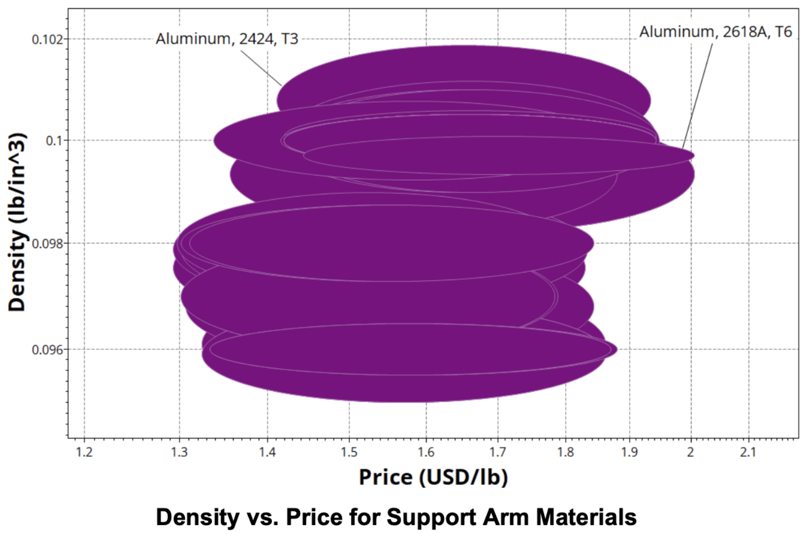


Figure 6: ANSYS Material Plot for Support Arm

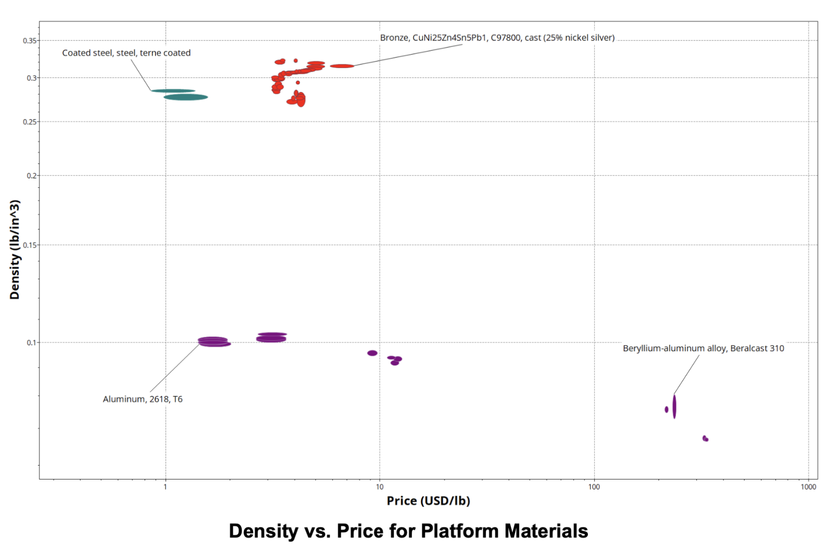


Figure 7: ANSYS Material Plot for Platform

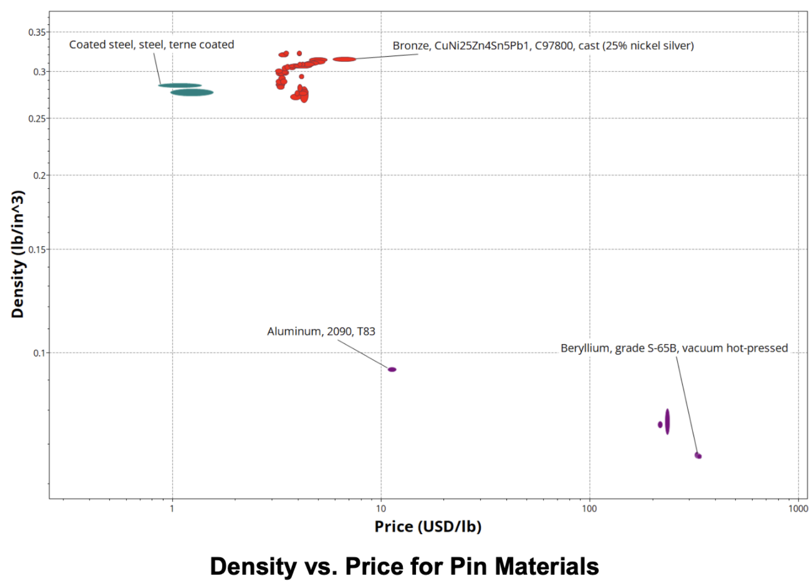


Figure 8: ANSYS Material Plot for Pins

5.8 Further Iterations and Adjustments

Based on the FEA and machining trials, two additional design tweaks were made:

1. Increased thickness from 0.145 in to 0.25 in to match standard stock and raise safety margin (FOS > 2.5).
2. Extended arm length from 18 in to 23.8 in to maximize counter-moment, with negligible change in deflection ($\delta \approx 0.025$ in).

Each change was validated analytically and with quick FEA checks; no further iterations were required.

5.9 Final Dimensions, Materials, and Cost

Table 3 lists the final design geometry and material choice. Table 4 shows the per-component and total cost estimates.

Table 3: Final Geometry and Materials

Component	Width	Thickness	Length	Material
Support Arm	0.50 in	0.25 in	23.80 in	Al 2024-T3
Platform I-Beams	1.00 in	0.50 in	17.50 in	Al 6061-T6
Central Plate	6.50 in	0.50 in	5.36 in	Al 6061-T6
Clevis Pins	Dia. 0.25 in		1.00 in	316SS

Table 4: Bill of Materials

Description	Vendor	Part #	Qty	Unit Price (USD)	Total (USD)
Flat-Head Quick-Release Pins 1/4”×1”	McMaster Carr	90156A507	1	4.91	4.91
3/8” Bolts Hex-Head Nuts & Bolts Kit	Amazon	B0DJP63Z67	1	14.99	14.99
2024 Aluminum Bar 0.25”×0.50”×24”	McMaster Carr	89215K413	2	20.44	40.88
6061 Aluminum Sheet 0.50”×6”×24”	McMaster Carr	8975K219	1	81.75	81.75
				Grand Total	**142.53**

6 Finite Element Validation

We used the full model assembly that was constructed in SolidWorks for our FEA, retaining all major components. Some simplifications were used for ease of setup and efficiency. The pins were modeled as simple cylinders, omitting certain geometric features that are present in the actual design. The pump assembly and the resulting upward force it generates were simplified by modeling the net effect as a single upward force applied evenly to the entire mounting platform. This way we consolidated both the pump’s weight and the lift it produces, and ignored the distributed nature of the force across the four individual mounting bolts.

Gravity was included as a force to account for the system’s own weight. Component interactions defined in the assembly were kept as connections in the FEA (as shown in Figure 8). Fixtures were applied to the pins to simulate support constraints. However, these were also simplified. Instead of modeling only the region of the pin that is constrained by the bracket, the entire pin was assumed to be fixed in the analysis. This assumption results in a slightly stiffer response than the physical system, however, it was considered acceptable for this analysis.

Figure 8 shows our finalized SolidWorks designs, with connections indicated, as used in our FEA study. Figure 9 shows our finalized SolidWorks designs with loads indicated in pink (pump) and red (gravity) along with static connection points indicated in blue. Figures 11-18 show FEA results on the entire system and the individual parts. Deflections are given in mm and stresses given in N/m^2.

Figure 19 is a convergence plot that shows how pin stress changes with increasing mesh density. We ran numerous simulations with increasingly finer mesh sizes and plotted the resulting stresses. This helps confirm that the results are somewhat stable and not overly dependent on mesh size. Some irregularity can be seen which is likely due to localized stress sensitivity or simplifications in the model. However, with more elements, the results do begin to converge.

The Following are questions we faced following our FEA analysis along with answers: **Using the material properties and stress values from the FEA simulations, which components are likely to be the first to fail? How does this result compare to what you expected from the selected "unique"components in GA2?**The component that is most likely to fail is the pins. According to the values seen in the raw data from the FEA, the pins have by far the highest stress. While the pins are made out of steel while other components are made of aluminum, the increase in Young's Modulus is not enough to offset the change in safety factor given the stresses in each component. Therefore, the pins have the lowest safety factor. This prediction coincides with our expectations from GA2 as we assumed the pins would have the highest stress due to their relatively small cross-sectional area. **Where is the highest stress location in each "unique" component of your design? How does this compare to what you expected in GA2?** The highest stress locations in each component of the design are at the points of stress concentration, which are primarily the fastener holes in the different parts. This is to be expected as the holes cause a decrease in cross sectional area as well as a sharp change in geometry. We predicted this result in GA2 as transverse holes are often followed by high stress concentrations. **What is the calculated deflection in the platform and where does it occur? How does this compare to GA2?**The max deformation in the platform is 0.17mm, and it occurs at the end furthest from the wall. There is a coupling of forces at the end of the platform between the pulley and the arms, which are causing the deflection to be at its maximum at that location. This deflection is relatively similar to our results in GA2. **If you had more time for design iteration this semester, would you make changes? Why or why not?** If we had more time for design iteration, we would definitely make changes in order to reduce the weight of our product. A common goal in design is to make a product as lightweight as possible, so iterative design would've highlighted areas with excessive material. Sections of our parts have vastly different stress values, so the regions of low stress can have some material reduced as their safety factor is very high.

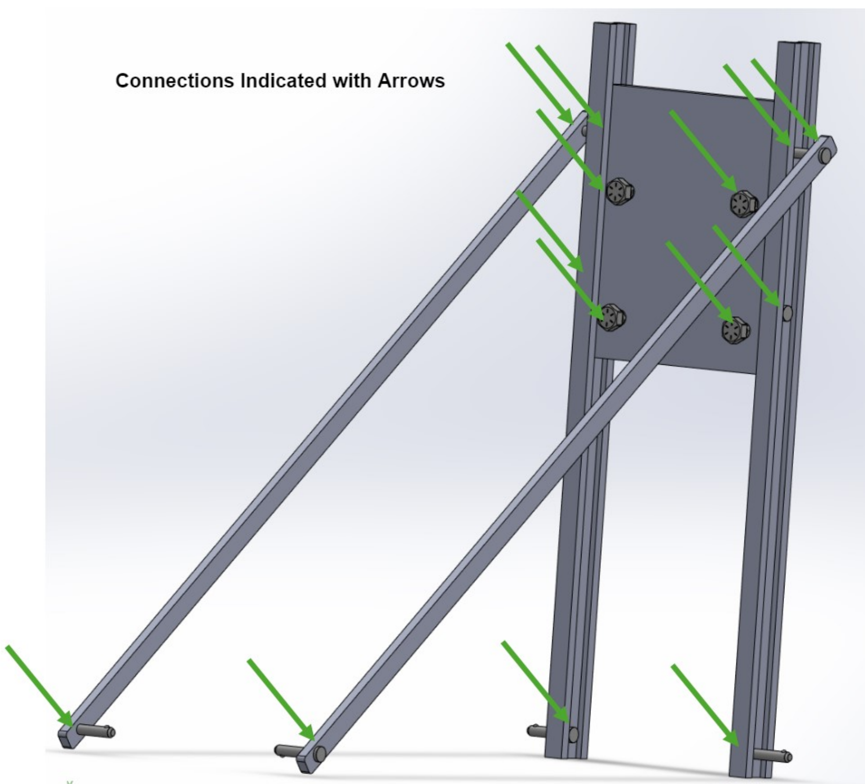


Figure 9: SolidWorks Assembly with Connections Indicated

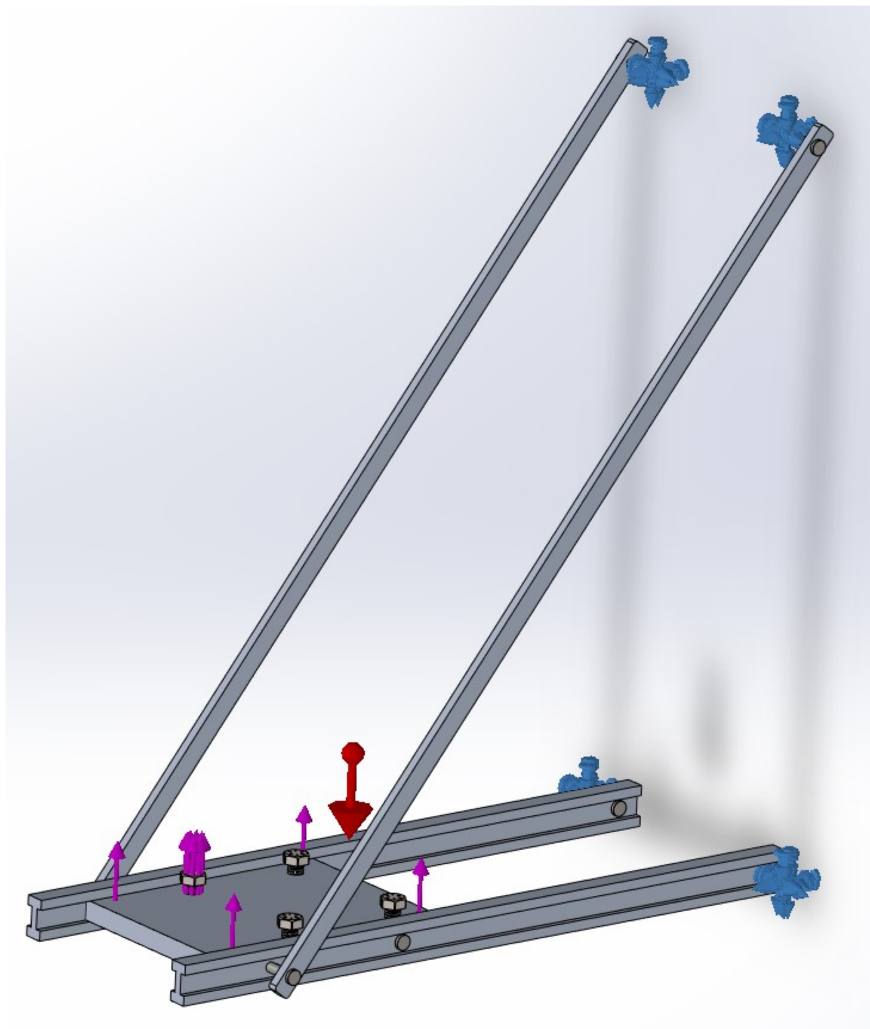


Figure 10: SolidWorks Assembly with Loads Indicated

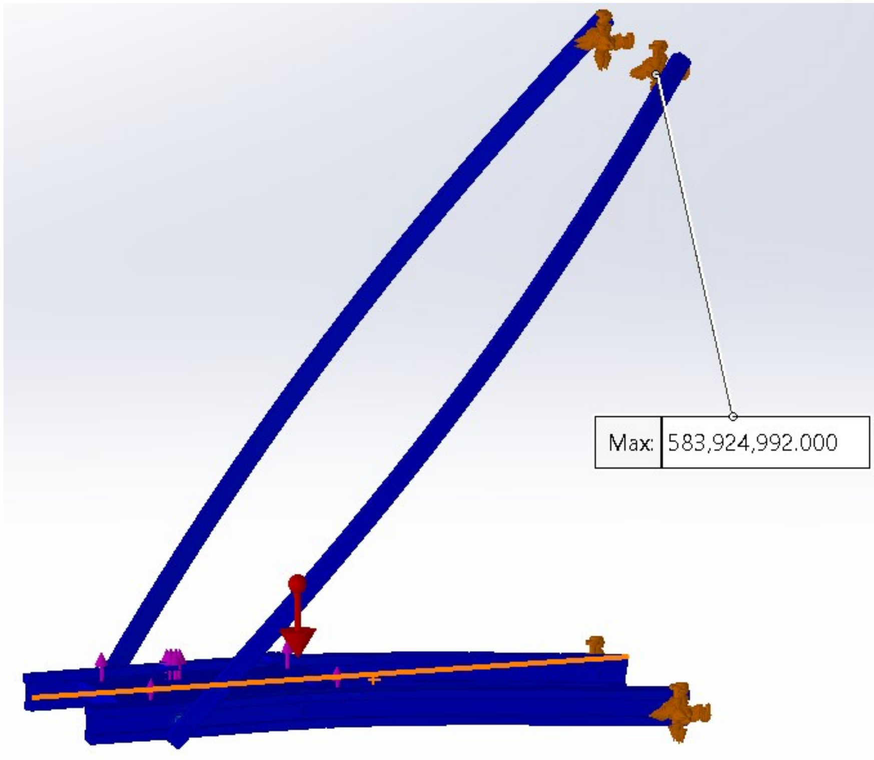


Figure 11: SolidWorks Simulation Result Showing Total Stress (Max)

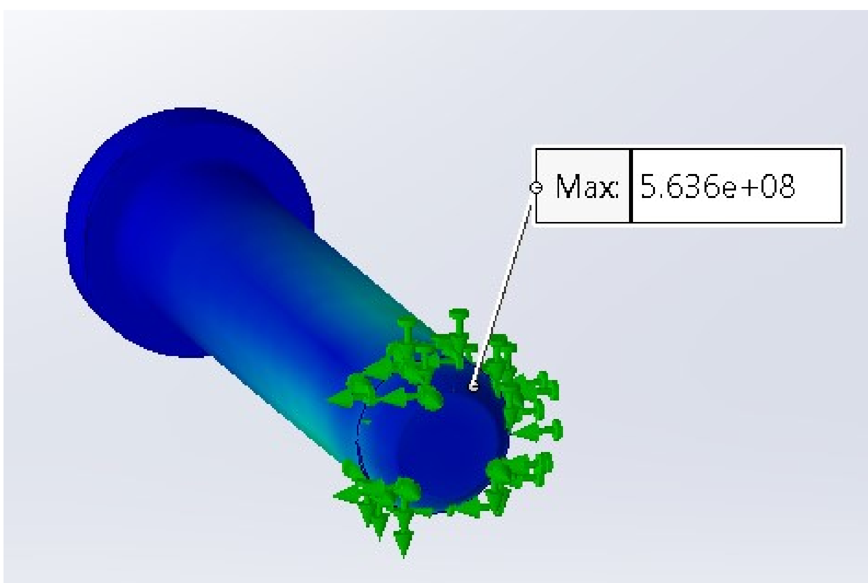


Figure 12: SolidWorks Simulation Result Showing Pin Stress (Max)

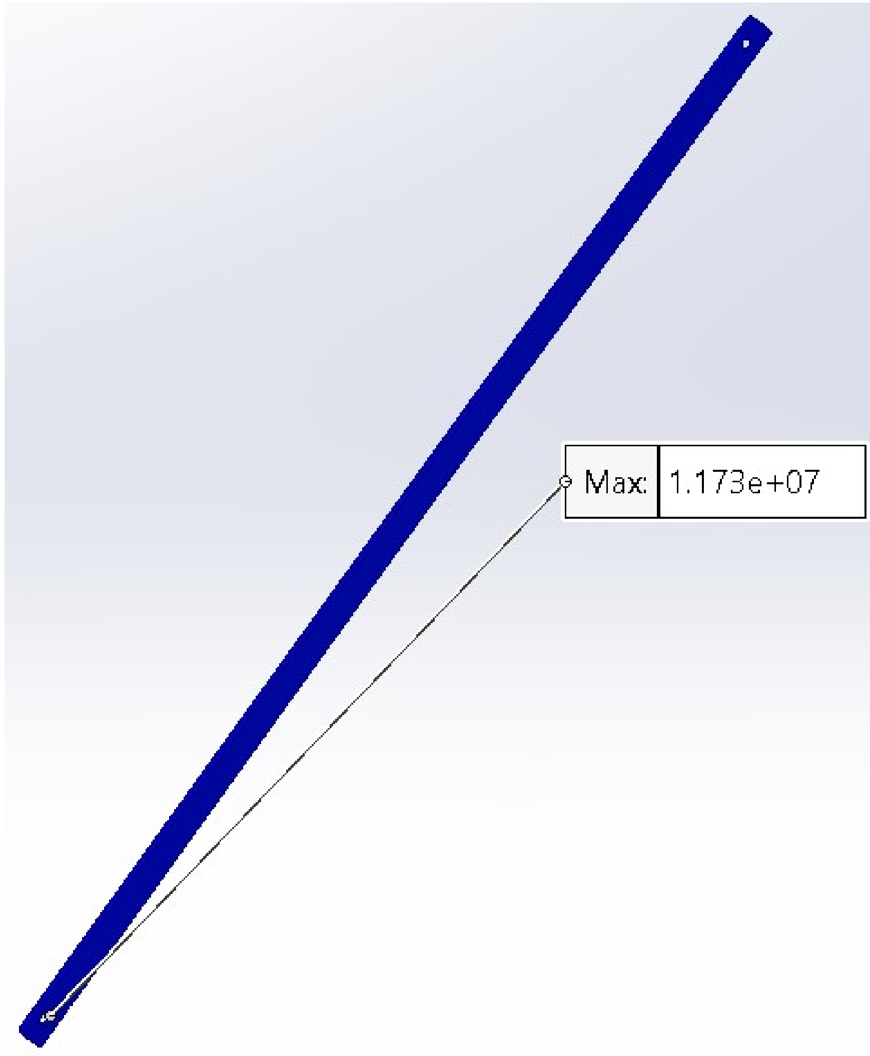


Figure 13: SolidWorks Simulation Result Showing Bar Stress (Max)

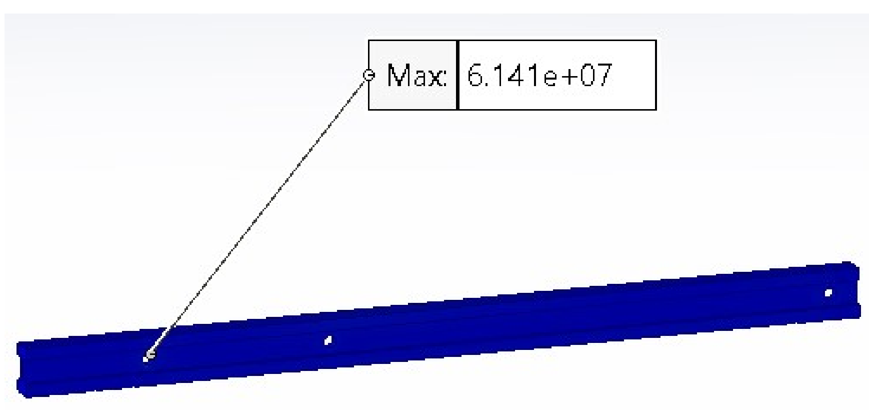


Figure 14: SolidWorks Simulation Result Showing I-Beam Stress (Max)

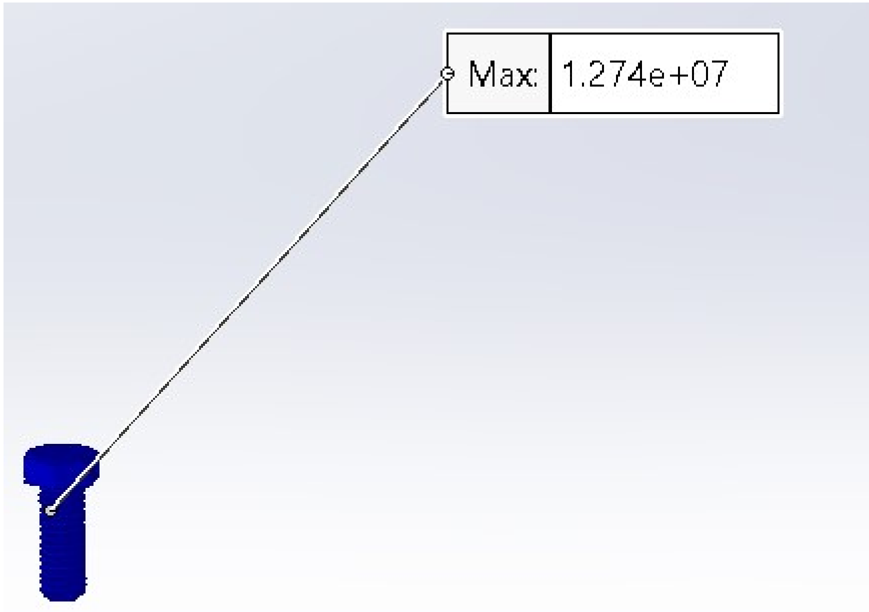


Figure 15: SolidWorks Simulation Result Showing Bolt Stress (Max)

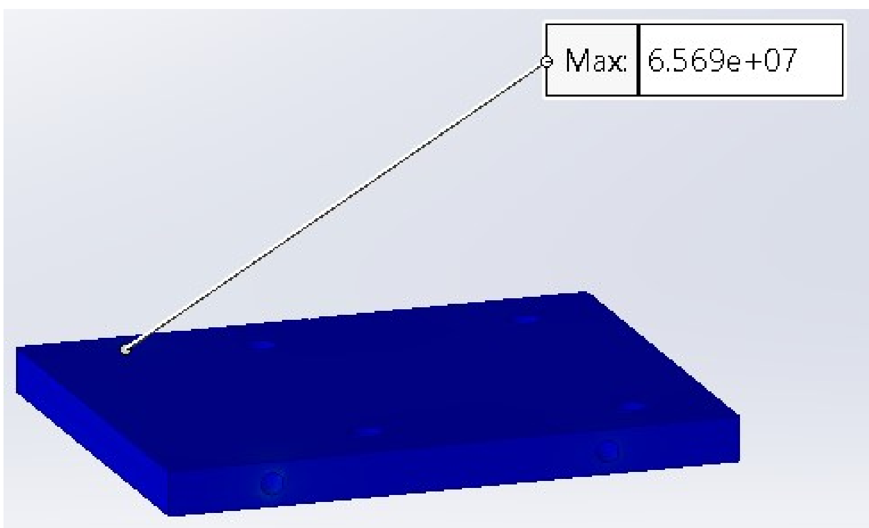


Figure 16: SolidWorks Simulation Result Showing Platform Stress (Max)

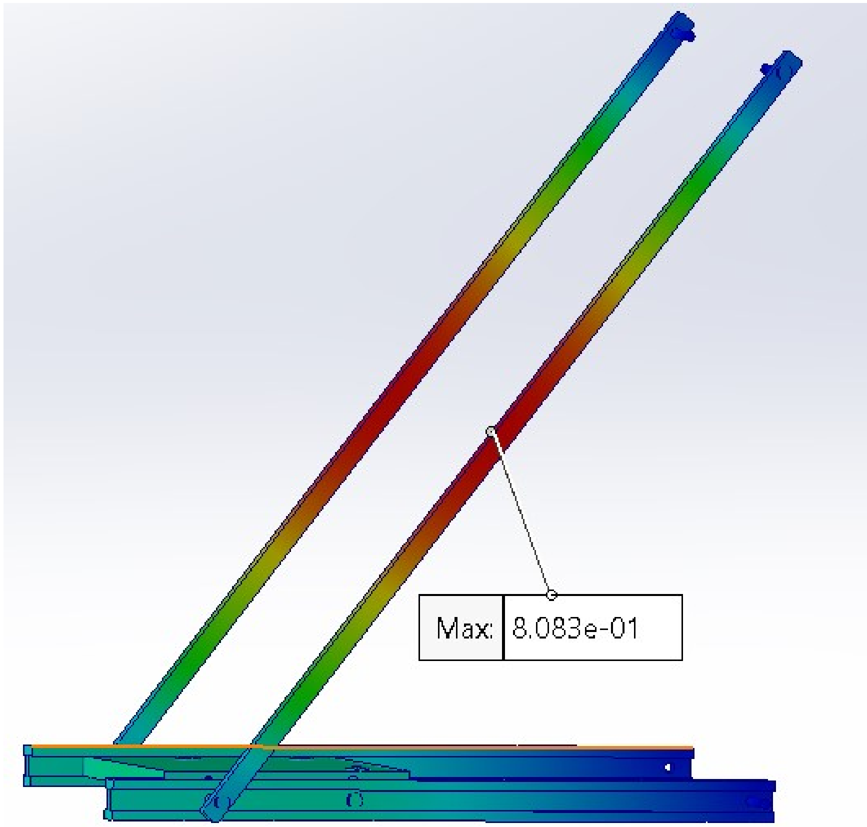


Figure 17: SolidWorks Simulation Result Showing Total Deflection (Max)

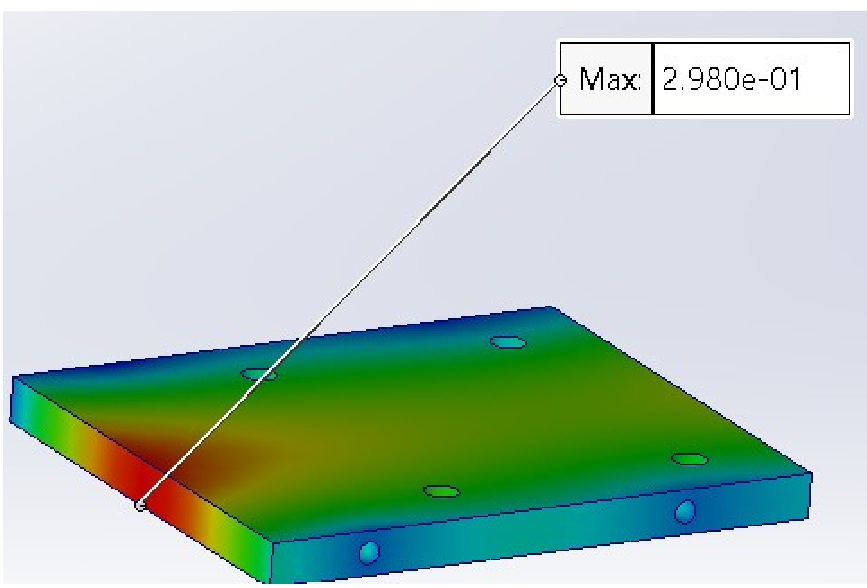


Figure 18: SolidWorks Simulation Result Showing Platform Deflection (Max)

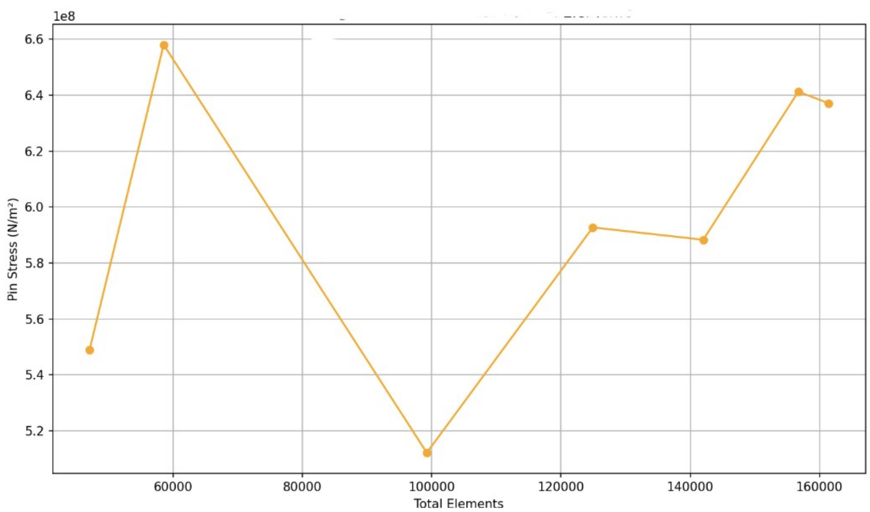


Figure 19: Convergence Plot of Pin Stress vs Total Elements

7 Design Changes

In this section, we describe the key modifications made to our original GA2 design, and provide the engineering rationale behind each change. Our GA2 submission served as the baseline for minimum material properties and a simple, hand-calculated cantilevered beam model. Subsequent iterations have been driven by real-world material availability, manufacturability concerns, factor-of-safety considerations, and targeted weight reduction.

7.1 Support Arm Geometry and Material Thickness

- **Baseline GA2 Design:** A straight aluminum beam, 18 inch long, with a calculated thickness of 0.145 inch (based on bending stress under a 115lbf upward load and allowable 2024-T3 yield strength).
- **Material Selection:** We retained *Aluminum 2024-T3* for its high strength-to-weight ratio and good fatigue properties.
- **Length Adjustment:** Extended arm length to 23.8 inch so that the pin support at the wall (top of the retaining structure) to the end of the platform is maximized. A longer lever arm increases the moment arm for counteracting vertical loads, thereby reducing deflection at the pump mounting point.
- **Thickness Increase to 0.25 inch:**
  - *Availability:* 0.25 inch plate stock is a standard size in raw material inventories, whereas 0.145 inch is not.
  - *Factor of Safety:* Raising thickness to 0.25 inch increases section modulus by over 70%, improving our factor of safety from roughly 1.5 in GA2 to over 2.5 under the same loading conditions.
  - *Weight Penalty:* The incremental added mass (0.2lb per arm) is negligible compared to the overall assembly weight.
- **Hole Placement:** Two 1/4 inch clearance holes drilled 0.5 inch inboard from each edge provide pin attachments while maintaining full cross-section at high-stress regions.

7.2 Platform Redesign for Weight Optimization

- **Original GA2 Platform:** A single solid rectangular plate (6×16×0.35 inch) serving both as structural support and pump mounting surface.
- **Three-Piece Assembly:** To reduce unnecessary material, we decomposed the platform into:
  1. *Two I-Beam Flanges:* Machined from 0.5 inch thick stock into cross-sectional I-beams of 1 inch (height) × 0.5 inch (web thickness) × 17.5 inch (length).
  2. *Central Mounting Plate:* A 6.5×5.36×0.5 inch plate that slides between the I-beams and carries the pump. Four 1/4 inch steel clevis pins at the corners ensure precise alignment.
- **Rationale for I-Beam Profile:**
  - *Material Efficiency:* I-beams remove surplus central material where bending stress is low, preserving flanges at extreme fibers where stress is highest.
  - *Stiffness-to-Weight Ratio:* The second moment of area for an I-section is approximately 20–30% higher than a solid rectangle of equal cross-sectional area, significantly reducing deflection under load.
  - *Manufacturability:* Although machining I-profiles from 0.5 inch plate is challenging, 0.5 inch stock is readily available; 0.3 inch stock was avoided due to limited supplier options and greater susceptibility to bending.
- **Pump Mount Interface:** By limiting the full-size mounting area to the localized central plate, we eliminate roughly half the plate volume compared to GA2, achieving a 30% weight reduction in the platform with negligible impact on stiffness (FEA predicts less than 5% increase in deflection under full load).
- **Assembly and Attachment Points:** The support arms attach at the same far two pin locations on the central platform plate, ensuring load transfer through high-stiffness elements and simplifying alignment during assembly.

These targeted modifications, grounded in practical considerations and validated by finite-element analysis, strike an optimal balance between structural integrity, manufacturability, and weight efficiency beyond our original GA2 submission.

8 CAD and Fabrication Drawings

The team documented their parts by creating drawings in Solidworks of the individual components as well as the overall assembly. These drawings can be found in the appendix along with a bill of materials and drawings of the fasteners used in the physical test. Physical pictures of the assembly can be found in the following section.

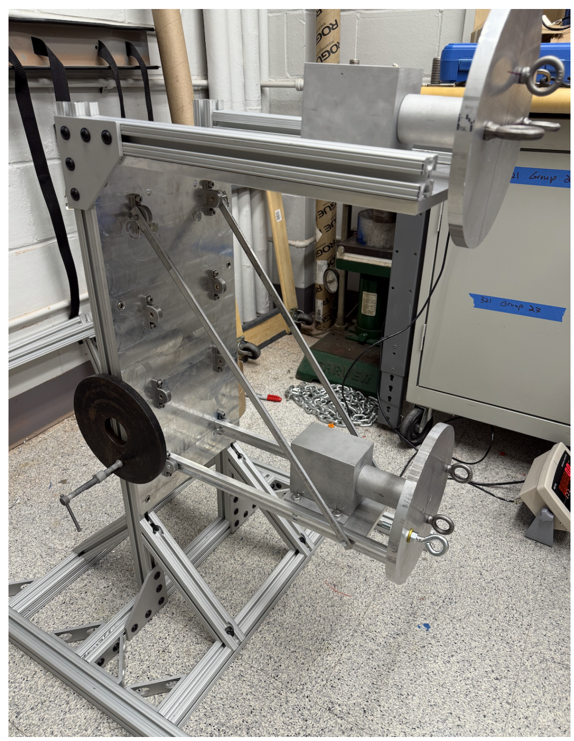
9 Physical Prototype Performance Results

The final prototype complied with all of the required properties. The prototype attached successfully to the test platform and assembled precisely to its original specifications. It maintained a length of less than 24 inches (measured at 19 inches) and had two support arms. All of the components, including pins and fasteners, weighed in at 3.8lbs.

Testing at 115 lbf of applied vertical force on the pump pulley yielded a measured deflection of 0.02 inches or 0.508mm. This amount of deflection satisfies the initial 1mm requirement. Calculating for the factor of safety by deflection by dividing the max deflection (1mm) by our actual deflection (0.508mm), we receive a tested factor of safety of roughly 2.

We did not observe any significant bending, buckling, or failures of any type. This design appeared to be sturdy, easy to assemble, and lightweight. The design reported very good efficiency in reducing deflection and weight properties relative to many alternative designs. The final assembly is depicted below in figure 20.

(a) Right Side Isometric View



(b) Left Side Isometric View2

Figure 20: Assembled final platform

10 Discussion

10.1 Design Performance Analysis

Our tested final prototype had successfully interfaced with the testing setup as intended. It was able to withstand the maximum applied vertical force of 115 lbf with minimal physical deformations, none of which were observable to the naked eye.

Comparing our calculated deflection values to the actual measured deflection, our preliminary calculations prove to be extremely accurate. The measured deflection of the platform was 0.02 inches or 0.508 mm–we calculated a deflection of 0.0198 inches or 0.503mm in Section 5.5. Our calculations had very slightly underestimated the deflection, but they were accurate to a degree that was more than acceptable. The SolidWorks FEA results had underestimated deflection even more, with an anticipated deflection of 0.33mm–our hand calculations proved to be much more accurate. SolidWorks' FEA almost always predicts a stiffer response than a hand-calculated cantilever because of how the model is constrained and discretized. In our case, the root of the platform was probably "built in" over an entire face rather than idealized as a line of support, which removes local compliance and artificially stiffens the part. The tested deflection under the max load more than exceeds the original 1mm requirement (FOS of 2) and the upated 3mm requirement (FOS of 6), indicating that the design is more than adequate. As predicted by our buckling factor of safety of around 1.9 in Section 5.6, we did not encounter any buckling in the arms despite significantly minimizing its cross-sectional area. Our weight of 3.8lbs was relatively lightweight, but there were some areas which this could be slightly optimized while still maintaining the required design properties and performances (Discussed in Section 10.3).

10.2 Design Challenges & Resolutions

Our design ideation was challenged by fast-approaching deadlines which often made the creation of the final prototype to be more arduous. At the time of placing our orders for our materials, we were still in the process of designing our platform. For this reason, we were left to work with whatever materials we had ordered after our final design was created. Upon submitting Group Assignment 2, we did not originally plan on using I-beams in our platform design, and our parts order was already placed. Since we were convicted to make a design that minimized weight and deflection characteristics, we committed to find a way to make I-Beams. We used our stock 1/2 inch aluminum sheet as the material of choice. We milled channels in the aluminum sheet, cut each arm out of the sheet using a vertical bandsaw, and milled the sides down to spec. This was a challenging and labor intensive process that proved to be rewarding. The final result was an I-Beam machined to a factory spec.

In our CAD drawing (see Section 8), there is a noticeable amount of space between the pins connecting the support arms to the platform. This presented a concern that the pin may not have a sufficient amount of room to support the platform or may introduce unnecessary stress. While FEA results indicated that it would not adversely impact our deflection, we wanted to address this as best as possible. Fortunately, the support arms were able to move inwards enough to allow the pins to fit flush with the platform, requiring no changes in our design.

10.3 Future Revisions

The only changes we may consider making in the future include properly sizing the holes in the I-beams for the pins, changing certain fasteners, and reducing the length of the I beams. The holes in the I-beams were milled out to 0.25 inches at the appropriate locations, but this ended up introducing some movement in the pin when inserted, despite the pin having a claimed diameter of 0.25 inches. To resolve this, we should have drilled each hole to a size slightly smaller than 0.25 inches and then reamed it to size. This would make the structure even more stable and less prone to movement.

Using pins to connect the pump mount to the I-beams proved to be convenient, but it made for a loose assembly of the platform. Given that there was only an applied vertical force, this was not a major concern. However, to ensure the consistent stability of the design upon assembly, the use of screws and threaded holes in the side of the platform may prove to be a more effective fastening method.

Threading the pump mount to accept the 3/8 inch screws, as opposed to using nuts, would allow for a small weight reduction while making the assembly process more straightforward. Fastening nuts would not need to be counter-held to fix the pump if it had threaded holes, reducing the time and effort required to assemble the support.

Overall, the design had exceeded our initial expectations and offered a great balance of low weight, reasonable manufacturing cost, short/easy assembly process, and optimal deflection characteristics under the maximum applied load.

11 Conclusion

11.1 Work Summary

Throughout the course, the team carried out a full design process for this pump platform system. It began by brainstorming the overall assembly and what parts would be designed. Once the overall outline was constructed, the internal forces, moments, and reactions were calculated to set a baseline of what size parts would be needed. With these values, the appropriate stresses were calculated to help identify critical stress locations. All of these values were found first symbolically and then numerically using python scripts. Then, the principal stresses and maximum shear stress for each part were calculated using a Mohr's Circle. With these values, the safety factors were determined using the max shear stress theory and distortion energy theory for ductile materials. These safety factors gave the team an idea of the rigidity of their model. The team also calculated different important quantities, such as the maximum deflection in the platform as well as the critical buckling load for the arms. Once all of these values were found symbolically, the team proceeded by picking a material, setting initial dimensions for their design, and iterating from there. The calculated deflection and factors of safety satisfied the requirements provided for the team, so the assembly proceeded to finite element analysis. The FEA reinforced these conclusions and asserted that the design would be successful, so the team documented their assembly and proceeded to manufacturing. The parts were largely constructed using a combination of a bandsaw and mill. The physical prototype then underwent testing and was able to withstand the required load with minimal deflection and a low total weight. Although the testing was a success, the team still brainstormed future revisions and improvements that could be made.

11.2 Engineering Principles

The project covered many bases of the engineering design process. As previously mentioned, safety factors were crucial in the design to ensure that each part would not fail during the test. Another equally important step toward this result was selecting the material for each part. The team settled upon aluminum given its machinability, low density, stiffness, and low price. An unexpected hurdle that the team would correct in the future would be the tolerances of the holes in each part for the pins. While the pins had a diameter of 0.5in, the drilled holes of 0.5in did not provide a perfect fit for the pins. Upon reflection, the team concluded that the connection could have been more properly toleranced if a smaller hole was drilled and then reamed to the proper diameter.

11.3 Lessons Learned

This design process showed the team what was required to fully create a part. The team was presented with a set of requirements and constraints and a lot of freedom outside of these. The project required diligent brainstorming and theorizing to design each part as well as possible. The parts also had to be designed to manufacture, as the team would be the ones creating each part. A key component of this choice was material selection, since the material needed to be machinable but also strong enough to withstand the applied load. In order to find the proper material, the team learned how to use the Ansys Granta EduPack. As mentioned before, the team learned new machining methods from both their successes and shortcomings. The process also revealed the importance, benefits, and limitations of finite elements analysis. While FEA is a powerful tool, it was used to reinforce the hand calculations that had already been completed. Overall, this project provided the team with an in depth view of what is required to go from a list of requirements to a full physical prototype undergoing testing.

12 Appendix

12.1 Stress Concentration Factors

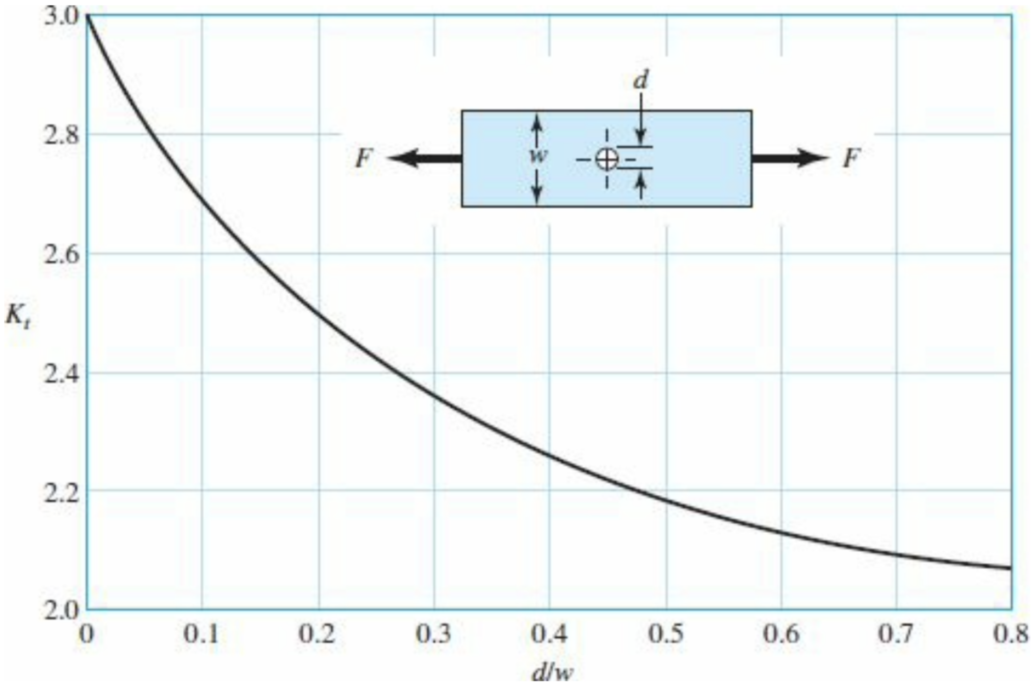


Figure 21: Stress Concentration Factor for Transverse Holes in Rectangular Cross Sections

12.2 CAD Drawings

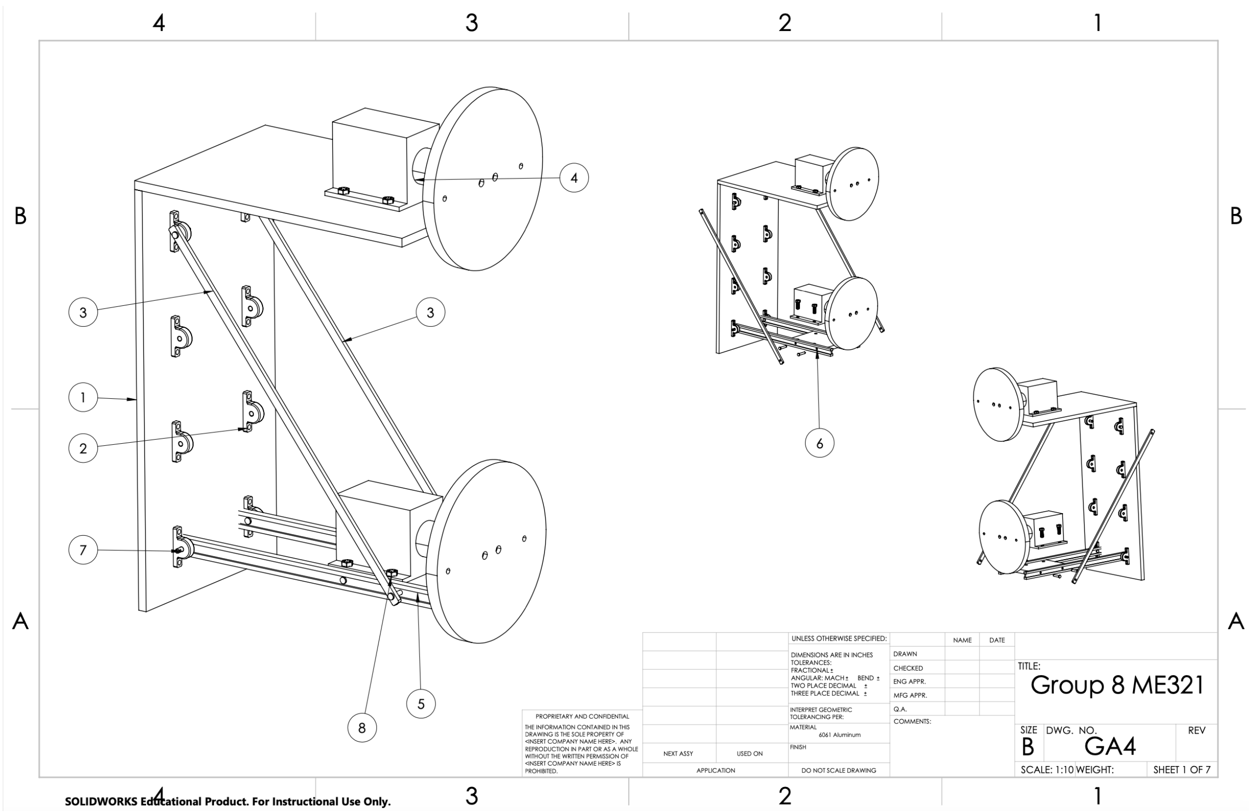


Figure 22: Isometric drawing

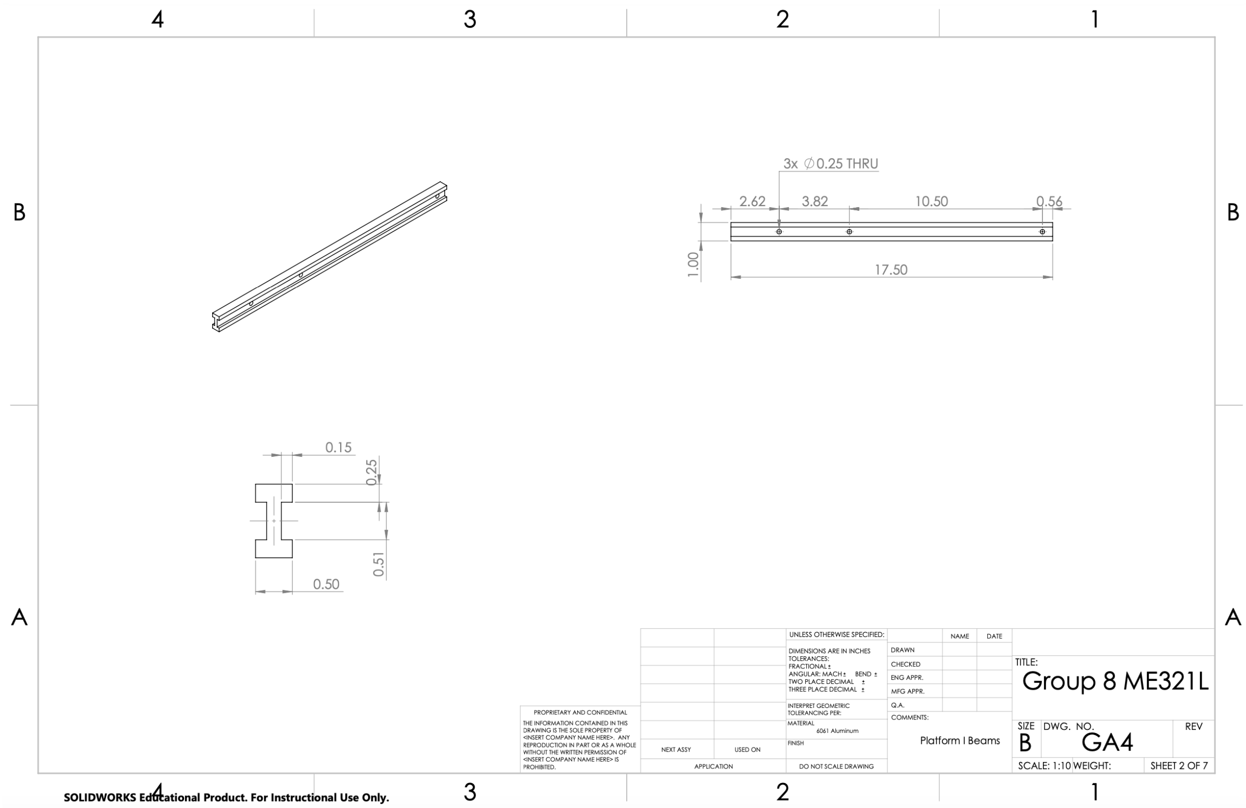


Figure 23: I-Beam drawing

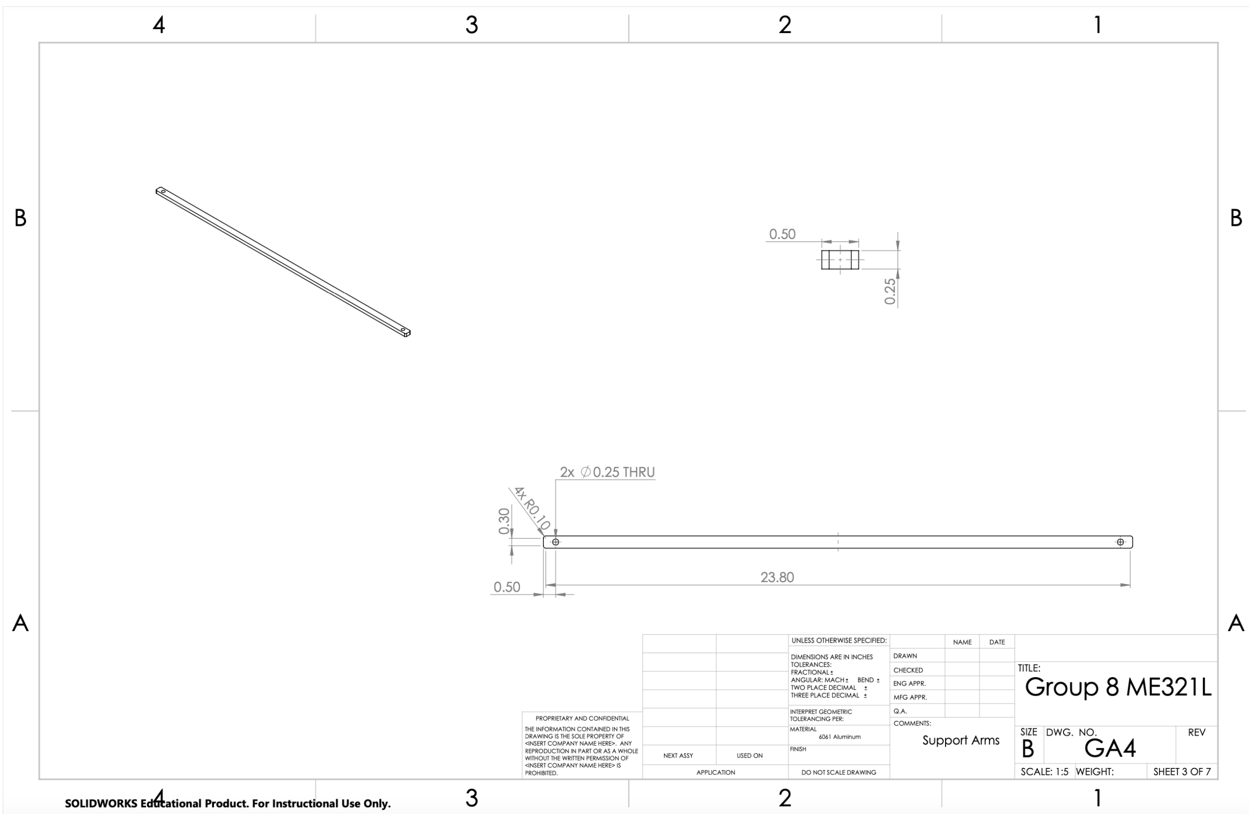


Figure 24: Support arm drawing

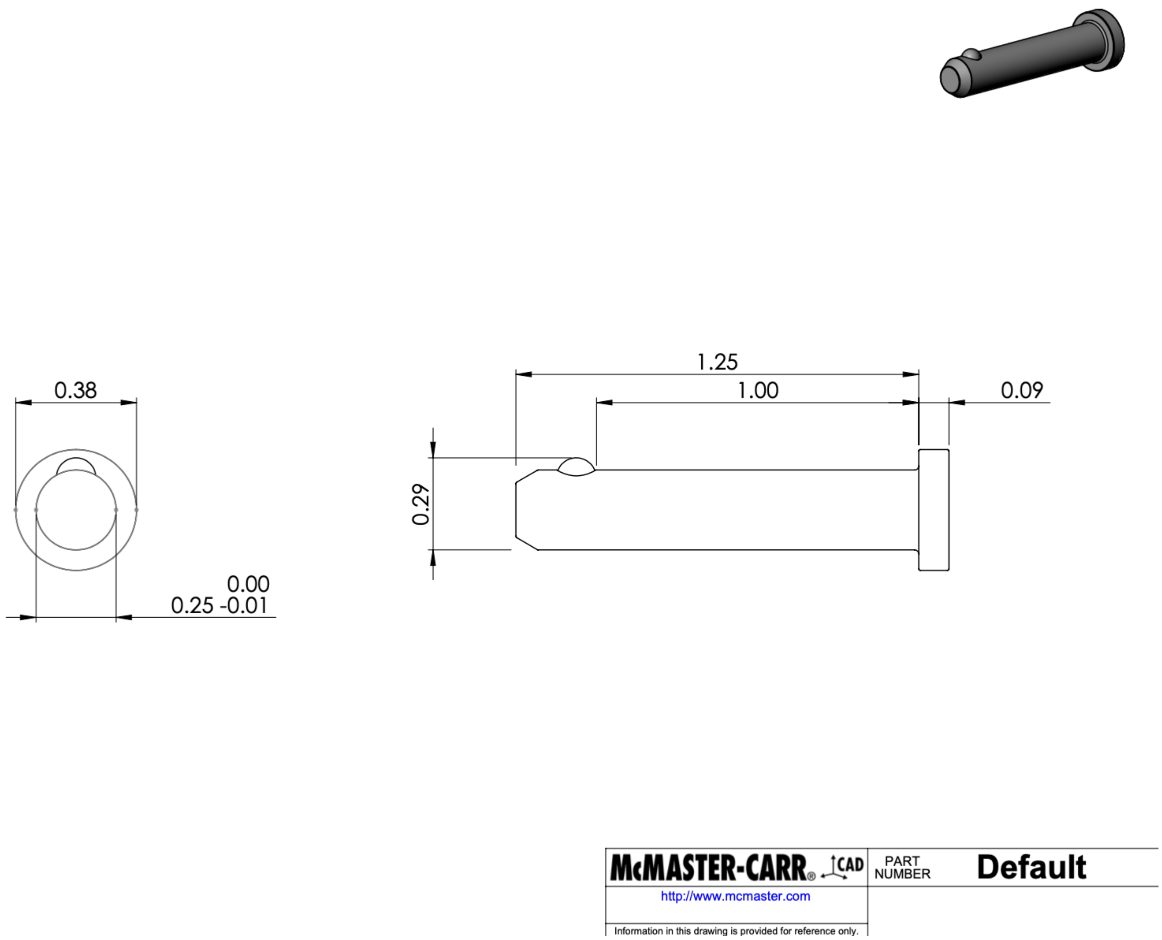


Figure 25: 1/4" pin drawing

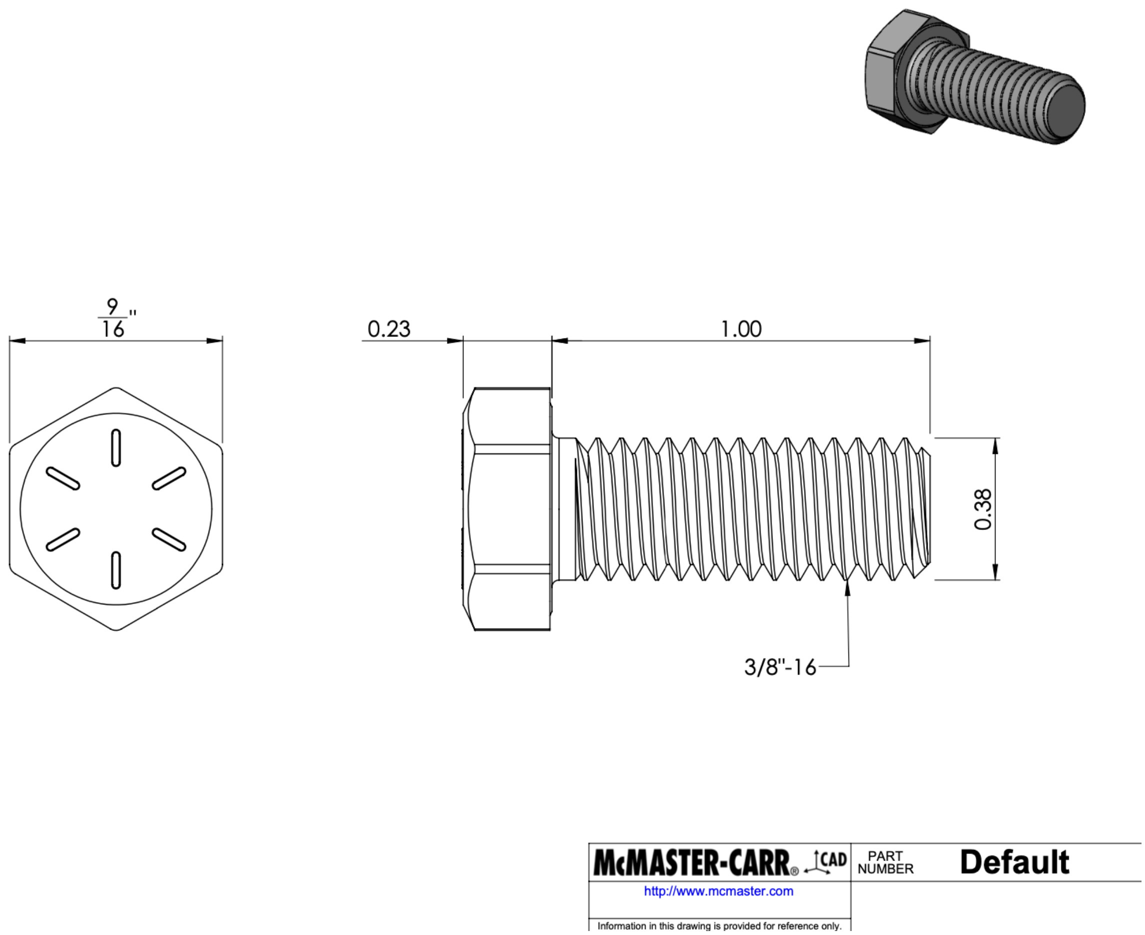


Figure 26: 3/8 screw drawing

ITEM NO.	PART NUMBER	DESCRIPTION	Material	QTY.
1	Mount Plate	Plate flsuh to wall to support frame	6061 Aluminum	1
2	Mounting Bracket	Brackets used to mount frame	6061 Alunimum	8
3	arm22.53design1	Support arms for platform	6061 Aluminum	2
4	Pump-Pulley (1)	Pump used in the system	6061 Aluminum	2
5	I-Beam	Arms used to connect platform to wall	6061 Aluminum	2
6	platform	Platform used to hold pulley	6061 Aluminum	1
7	90156A507	Flat-Head Quick-Release Pins	1004-1045 Carbon Steel	8
8	92620A624	Zinc Yellow-Chromate Plated Hex Head Screw	Steel	8

Figure 27: Bill of materials

12.3 Source Code

Listing 1: Python code to calculate forces, moments, reactions, and other requested values

```
import math
import numpy as np
from sympy import symbols, Eq, solve
import matplotlib.pyplot as plt
import itertools


def calculate_fluid_mechanical_power(Q_ft, rho, g, h, n_p):
    P_Fluid = (rho * g * Q_ft * h) / 550
    P_Mech = P_Fluid / n_p
    return P_Fluid, P_Mech

def calculate_pulley_tension(P_Mech, RPM, Pulley_rad, T1):
    T2 = ((P_Mech * 550) / ((RPM * 2 * np.pi / 60) * (Pulley_rad / 12))) + T1
    return T2

def calculate_moment_and_torque(T1, T2, L_Pulley_to_Wall, L_Pump_to_Wall):
    M_G = (T1 + T2) * (L_Pulley_to_Wall - L_Pump_to_Wall)
    T_G = (T2 - T1) * (Pulley_rad)
    return M_G, T_G

def solve_forces(T1, T2, W_beam, W_pump, theta, L_Pump_to_Wall, L_support, L_beam,
     Width_beam, M_G, T_G):
    S1, S2, Wall_X1, Wall_X2, Wall_Z = symbols('S1 S2 Wall_X1 Wall_X2 Wall_Z')
    eq1 = Eq(T1 + T2 - W_pump - S1 * math.sin(theta) - S2 * math.sin(theta) -
        W_beam + Wall_Z, 0)
    eq2 = Eq(S1 * math.cos(theta) + S2 * math.cos(theta) + (Wall_X1 + Wall_X2), 0)
    eq3 = Eq(M_G + ((T1+T2-W_pump) * L_Pump_to_Wall) - ((S1 * math.sin(theta) + S2
     * math.sin(theta)) * L_support) - (W_beam * (L_beam/2)), 0)
    eq4 = Eq(T_G - ((S1*np.sin(theta) + (Wall_Z/2)) * (Width_beam/2))  + ((S2*np.
        sin(theta) + (Wall_Z/2)) * (Width_beam/2)), 0)
    eq5 = Eq((S1*np.cos(theta) - S2*np.cos(theta) + Wall_X1 - Wall_X2)*0.5*
        Width_beam, 0)
    soln = solve((eq1, eq2, eq3, eq4, eq5), (S1, S2, Wall_X1, Wall_X2, Wall_Z))

    # print(soln)
    return soln

def calculate_connection_reactions(soln, theta):
    S1_ConnectionX = -soln[symbols('S1')] * math.cos(theta)
    S1_ConnectionZ = -soln[symbols('S1')] * math.sin(theta)
    S2_ConnectionX = -soln[symbols('S2')] * math.cos(theta)
    S2_ConnectionZ = -soln[symbols('S2')] * math.sin(theta)
    return S1_ConnectionX, S1_ConnectionZ, S2_ConnectionX, S2_ConnectionZ


# CONSTRAINTS
Q = 150  # Volume flow rate [GPM]
Q_ft = Q / 7.48 / 60  # Volume flow rate [ft^3/sec]
n_p = 0.65  # Pump Efficiency
h = 60  # Final Height [ft]
rho = 1.94  # slugs/ft3
g = 32.174  # Acceleration Due to Gravity
W_pump = 12.5  # Weight of Pump [lbf]
RPM = 1800  # RPM of Motor
```

```
L_Pump_to_Wall = 9.56 + 2.5  # Distance from Pump to Wall [in]
L_Pulley_to_Wall = 18.56  # Distance from Pulley to Wall [in]

# FREE VARIABLES
L_mounting = 6  # Distance between mounting studs [in]
L_support = 5  # Placement [in]
L_beam = 18  # Placement [in]
Width_beam = 6  # [in]
Thick_beam = 0.5  # [in]
density_beam = 0.1  # [lb/in^3]

# FREE VARIABLES
L_mounting = 6  # Distance between mounting studs [in]
L_support = 2.00  # Placement [in]
L_beam = 14.56  # Placement [in]
Width_beam = 6.00  # [in]
Thick_beam = 0.38  # [in]

"""
# UNCOMMENT FOR USER INPUT
L_mounting = float(input(print("Enter Distance between mounting studs [in]: ")))
L_support = float(input(print("Enter Distance between support attachment and wall
    [in]: ")))
L_beam = float(input(print("Enter Length of Beam [in]: ")))
Width_beam = float(input(print("Enter Width of Beam [in]: ")))
Thick_beam = float(input(print("Enter Thickness of Beam [in]: ")))
density_beam = float(input(print("Enter Density of Beam [lb/in^3]: ")))
"""

# Calculated Values
W_beam = density_beam * L_beam * Width_beam * Thick_beam  # Weight of Beam [lbf]
theta = np.arctan(L_mounting / L_support)

P_Fluid, P_Mech = calculate_fluid_mechanical_power(Q_ft, rho, g, h, n_p)
print(f"Fluid Power: {P_Fluid:.6f} hp \nMechanical Power: {P_Mech:.6f} hp")

T1 = 44.9618  # Pulley Tension 1 [lbf]
Pulley_rad = 10 / 2  # [in]
T2 = calculate_pulley_tension(P_Mech, RPM, Pulley_rad, T1)
print(f"Pulley Tension 1: {T1:.6f} lbf")
print(f"Pulley Tension 2: {T2:.6f} lbf")

M_G, T_G = calculate_moment_and_torque(T1, T2, L_Pulley_to_Wall, L_Pump_to_Wall)
print(f"Moment at Point G: {M_G:.6f} lbf-in")
print(f"Torque at Point G: {T_G:.6f} lbf-in")

soln = solve_forces(T1, T2, W_beam, W_pump, theta, L_Pump_to_Wall, L_support,
    L_beam, Width_beam, M_G, T_G)
print(f"Force S1: {soln[symbols('S1')]:.6f} lbf")
print(f"Force S2: {soln[symbols('S2')]:.6f} lbf")
print(f"Force Wall_X1: {soln[symbols('Wall_X1')]:.6f} lbf")
print(f"Force Wall_X2: {soln[symbols('Wall_X2')]:.6f} lbf")
print(f"Force Wall_Z: {soln[symbols('Wall_Z')]:.6f} lbf")

S1_ConnectionX, S1_ConnectionZ, S2_ConnectionX, S2_ConnectionZ =
    calculate_connection_reactions(soln, theta)
print(f"Force S1 Wall Connection X: {S1_ConnectionX:.6f} lbf")
print(f"Force S1 Wall Connection Z: {S1_ConnectionZ:.6f} lbf")
print(f"Force S2 Wall Connection X: {S2_ConnectionX:.6f} lbf")
```

```
print(f"Force␣S2␣Wall␣Connection␣Z:␣{S2_ConnectionZ:.6f}␣lbf")




\begin{lstlisting}[caption={Python code for bending, deflection, buckling, and
    factor of safety calculations}]
import math
#Arm FOS
print("Arm␣Calculations")
arm_w = 0.5
arm_t = 0.145
arm_l = 18
k_arm = 2.5
arm_E = 1.04*10**7
sy_arm = 45000
hw = 0.5/arm_w
dw = 0.25/arm_w
print(f"hw␣{hw}␣dw␣{dw}")
A_arm = (arm_w-0.25)*arm_t
print(f"Area:␣{A_arm}")
sigma_arm_axial = -246.58/A_arm #-1096N
print(f"Sigma_D_arm:␣{sigma_arm_axial}")
sigma_arm_xx = k_arm*sigma_arm_axial
print(f"Sigma_axial_arm:␣{sigma_arm_xx}")
mohrs_arm_center = sigma_arm_xx/2
mohrs_arm_radius = math.sqrt((0.5*sigma_arm_xx) ** 2)
sigma_arm_1 = mohrs_arm_center + mohrs_arm_radius
print(f"Sigma_principal_arm_1:␣{sigma_arm_1}")
sigma_arm_2 = mohrs_arm_center - mohrs_arm_radius
print(f"Sigma_principal_arm_2:␣{sigma_arm_2}")
sigma_von_arm = math.sqrt(sigma_arm_1 ** 2 + sigma_arm_2 ** 2 -
sigma_arm_1*sigma_arm_2)
print(f"sigma_von_arm:␣{sigma_von_arm}")
print(f"Arm␣FOS:␣S_y␣/␣{sigma_von_arm}")
sf = sy_arm / sigma_von_arm
print(f"Arm␣FOS:␣{sf}")
print()
print("-------------New␣Case------------")
print()
#Platform FOS Case 1
print("Platform␣Case␣1␣Calculations")
p_w = 6 #width (mm)
p_t = 0.35 #thickness (mm)
p_l = 16 #length (mm)
k_p = 2.1
alpha = 0.299
sy_p = 45000
E = 7*10**10
F1 = 25
F2 = 448.9435
a1 = 1.5
a2 = p_l - 3.75 - a1
dh = 0.25/p_w
dw = 0.25/p_t
print(f"dh␣{dh}␣dw␣{dw}")
I_y = p_w/12 * (p_t**3 - 0.25**3)
print(f"Moment␣of␣Inertia␣(y):␣{I_y}")
sigma_b_0 = 742*(p_t/2)/I_y
```

```
print(f"Sigma_b_1: {sigma_b_0}")
sigma_b = sigma_b_0 * k_p
print(f"Sigma_b: {sigma_b}")
#axial calculatios:
A_p = (p_t-0.25) * p_w
print(f"Platform axial area: {A_p}")
sigma_d = 318.08/A_p
print(f"Sigma_D: {sigma_d}")
sigma_axial = k_p * sigma_d
print(f"Sigma_axial: {sigma_axial}")
#torsion
t_max = 2296.5/(alpha * p_w * p_t**2)
print(f"T_max: {t_max}")
#total
sigma_x = sigma_b + sigma_axial
c = sigma_x / 2
r = math.sqrt((sigma_x/2)**2+t_max ** 2)
print(f"Mohr's radius: {r}")
sigma1 = c+r
sigma2 = c-r
sigma_von = math.sqrt(sigma1**2 + sigma2**2 -sigma1*sigma2)
print(f"Sigma_von: {sigma_von}")
print(f"Platform Case 1 (bottom face along hole axis) FOS: S_y / {sigma_von}")
sf = sy_p / sigma_von
print(f"Platform Case 1 FOS: {sf}")
print()
print("-------------New Case------------")
print()
#platform case 2 calculations 1
print("Platform case 2 (cross section where pin holes c/d lies) calculations")
print("for (0, 0, +-d/2)")
k_p = 3.8
sigma_axial = k_p * sigma_d
print(f"Sigma_axial: {sigma_axial}")
#transverse
q = p_w/2*(((0.25+p_t)/4)**2-(0.25/2)**2)
transverse= 318.08*q/(I_y * p_w)
sigma_x = sigma_axial
r = math.sqrt((sigma_x/2)**2+transverse**2)
c = sigma_x / 2
sigma_1 = c + r
sigma_2 = c - r
sigma_von = math.sqrt(sigma1**2 + sigma2**2 -sigma1*sigma2)
print(f"Sigma_von: {sigma_von}")
print(f"Platform Case 2 FOS: S_y / {sigma_von}")
sf = sy_p / sigma_von
print(f"Platform Case 1 FOS: {sf}")
#platform case 2 calculations 2
print()
print("Platform case 2 calculations II")
print("for (+-d/2, 0, 0)")
k_p = 3.8
sigma_d = (174.37+143.71)/(p_t*p_w)
sigma_axial = k_p * sigma_d
print(f"Sigma_axial: {sigma_axial}")
#transverse
transverse= 4*318.08/(3*p_t*p_w)
sigma_x = sigma_axial
r = math.sqrt((sigma_x/2)**2+transverse**2)
```

```
c = sigma_x / 2
sigma_1 = c + r
sigma_2 = c - r
sigma_von = math.sqrt(sigma1**2 + sigma2**2 -sigma1*sigma2)
print(f"Sigma_von: {sigma_von}")
print(f"Platform Case 2 FOS (+- d/2, 0, 0): S_y / {sigma_von}")
print()
print("-------------New Case------------")
print()
print("Platform case 3 (at pinhole EF) calculations")
tmax = 4*318.08/(3*p_w*p_t)
print(f"tmax: {tmax}")
k = 3
torsion = 0.47*k
print(f"Torsion: {torsion}")
sigma_vm = math.sqrt(3*(torsion+tmax**2))
print(f"Platform Case 3 FOS: S_y / {sigma_vm}")
sf = sy_p / sigma_vm
print(f"Platform Case 1 FOS: {sf}")
print()
print("-------------New Case------------")
print()
#deflection
print("Platform Deflection Calculations")
y_max = F1*a1**2 * (a1 - 3*p_l) / (6*E*I_y) - F2*a2**2 * (a2 - 3*p_l)/(6*E*I_y)
print(f"deflection max: {y_max}")
fs = 0.01 / y_max
print(f"deflection FOS {fs}")
print()
print("-------------New Case------------")
print()
print("Arm Buckling Calculations")
print(f"{I_y}")
arm_I = (arm_t)**3*(arm_w)/12
P_cr = math.pi**2 * arm_E * arm_I / ((arm_l)**2)
print(f"P_cr: {P_cr}")
FOS = P_cr / F1
\FloatBarrier
```