MECH 3401– Mini Project Design 2

FEA Analysis

By: Jaskamal Saini (217974775) - jassu@my.yorku.ca

Herman Balagan (217136490) - herman04@my.yorku.ca

Nadary Hernandez (217532136) - nahr2300@my.yorku.ca

David Au (208590507) - davidau1@gmail.com

York University

Lassonde School of Engineering

Department of Mechanical Engineering

April 4, 2023

Meshing & Boundary Conditions

In terms of boundary conditions acting on our design the only consideration that was taken into account was gravity and rotational speed. As the wave oscillation was clearly stated to be about 1-2 cm with about a 2 hz oscillation and knowing that the design was sturdy and buoyant to withstand these conditions the simulations for the weather conditions were not shown. Area of interest with the greatest weight excretion and bending moment will be analysed to ensure that they meet the design specification. That being said, stress tests using displacement contours, shear stress contours, Factor of safety and von mises contours will be used to evaluate the reaction each component of interest has to its respective loud.

Simulations & Calculation Differences

Part 1: Solar Panel Arms

The first area of interest is the solar panel arms. These arms are relatively thick and provide support to the solar panel assembly to assure that these devices are adequately held up. With their orientation in this design they are pron to a bending moment which can cause mechanical defects over time To set up this simulation a fine mesh was used to achieve the most accurate results. In addition to this, a downwards force equivalent to the weight of the solar panels was applied onto the supports. As the area of interest was specifically these arms alone, the base plate was set as fixed to ensure no force was transferred to this section allowing for a finite element analysis to be conducted on the arm. For this analysis AISI 304 steel was used to gather appropriate material properties. In terms of results, it was found that the mechanism far exceeds the strength and tensile threshold making it very safe for design usage to take place. With a max displacement of under 1mm, this demonstrates how sturdy the design is to the gravitational force exerted on it by the solar panels. In terms of calculation comparison, the yield strength far exceeded that in which it was calculated. This was likely due to the material selected when simulation took place. For the calculations phase, plastic was used as it was assumed that this part would be 3D printed.

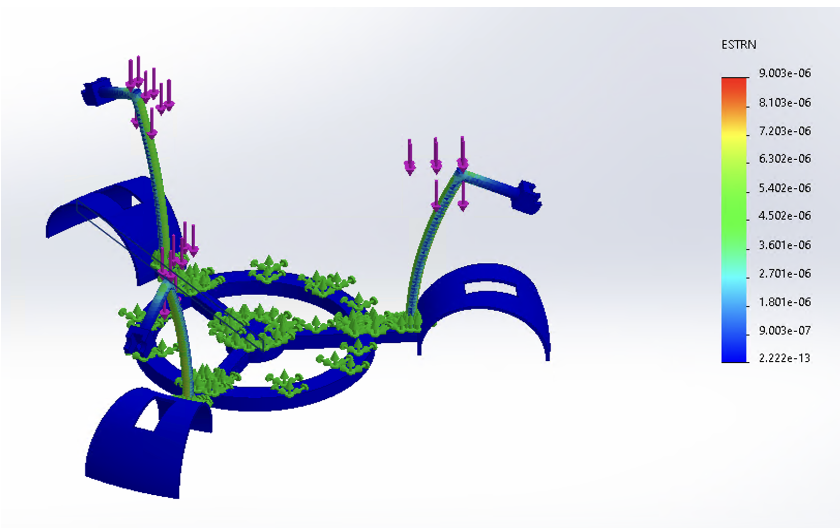


Figure 1: Tensile stress on solar panel arms

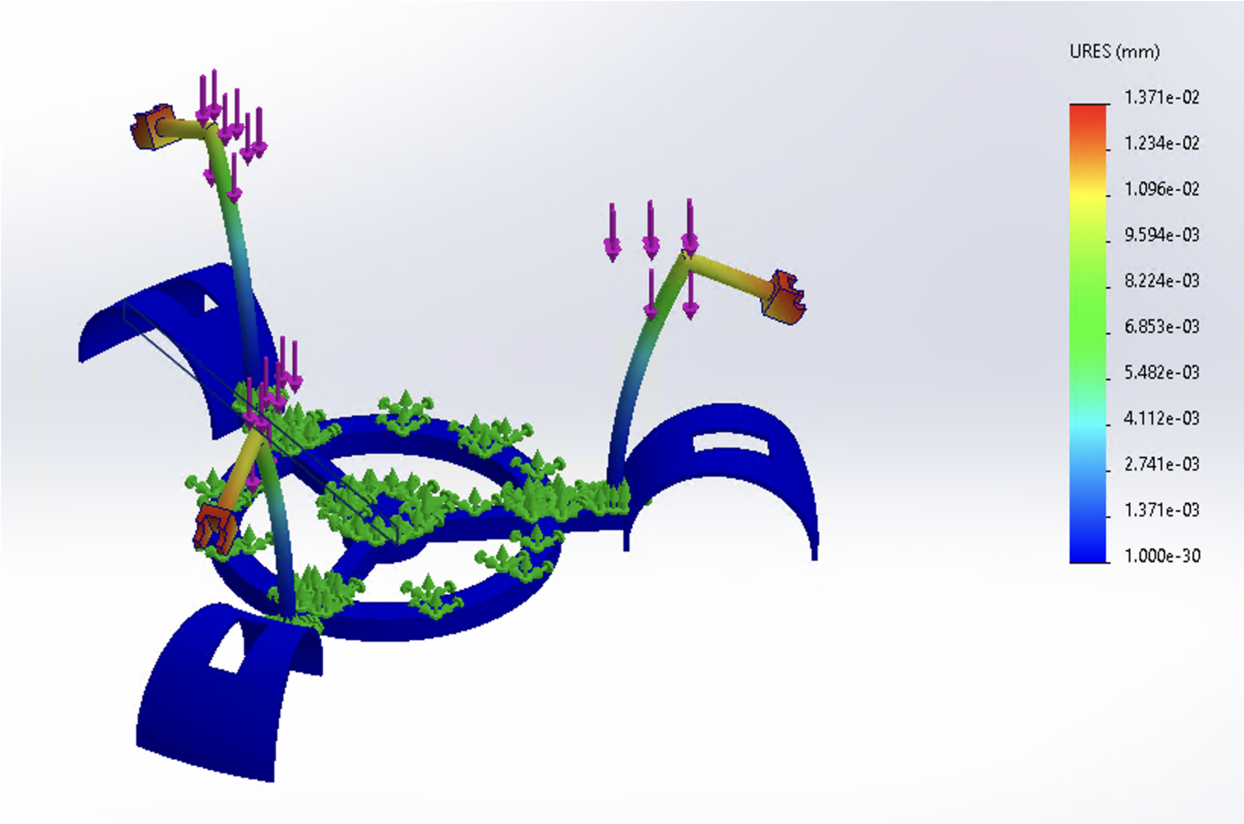


Figure 2: Yield stress on solar panel arms

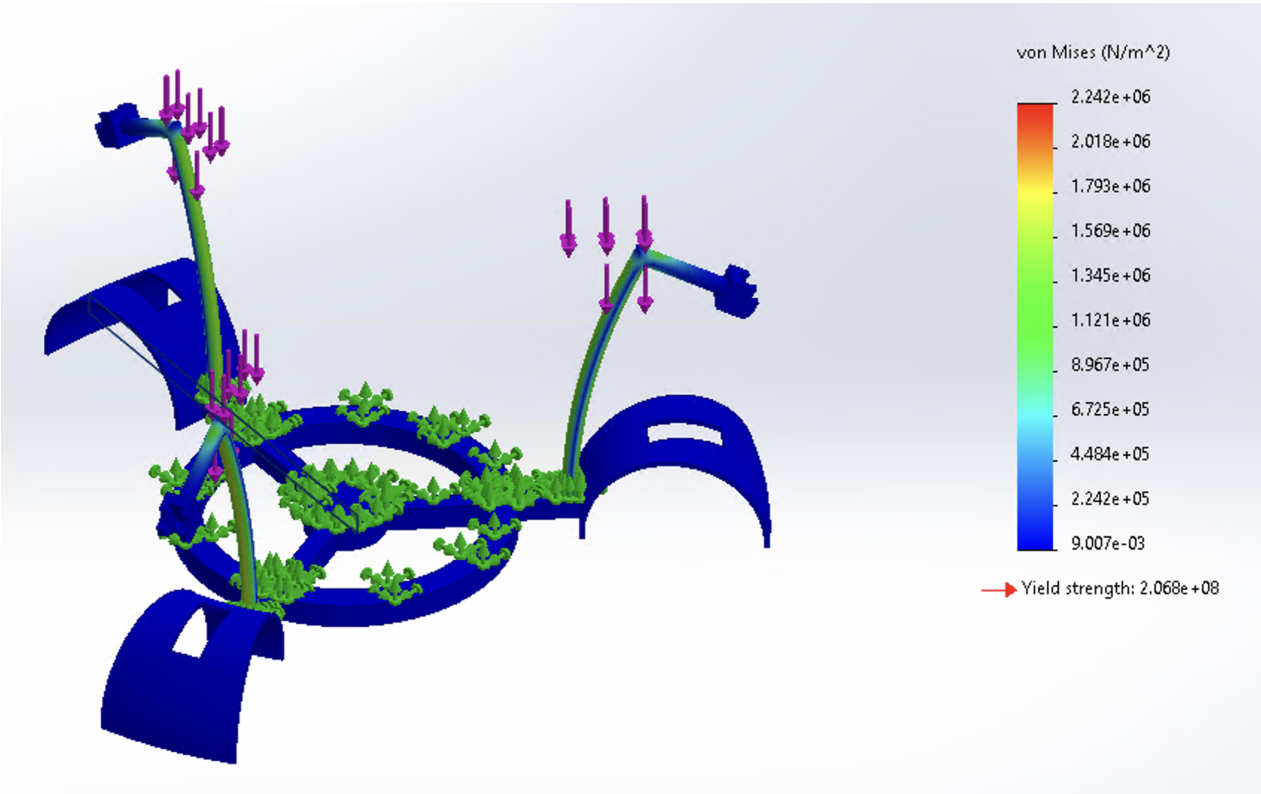
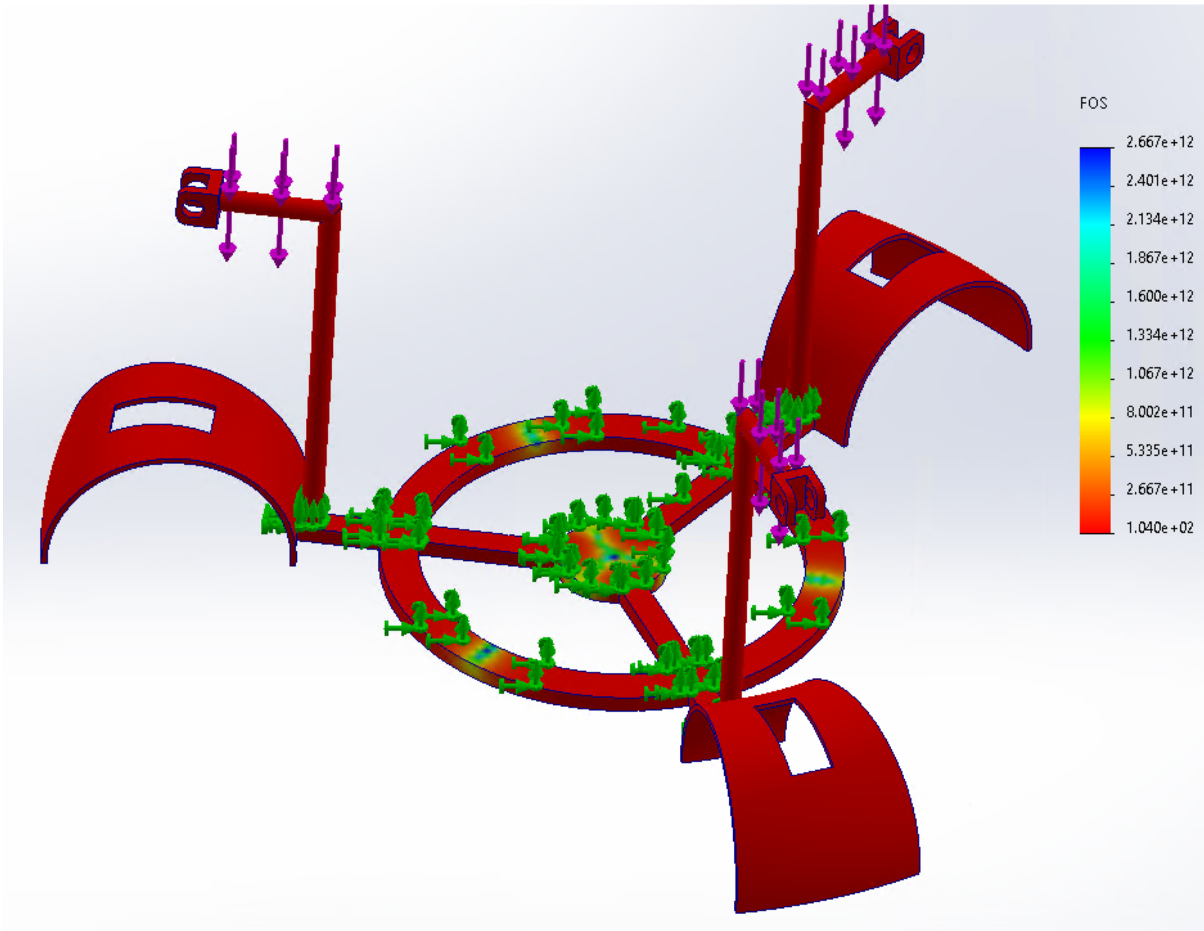


Figure 3: Von mises on solar panel arms



Part 2: Mirror Arm

The second area of interest is the mirror arm. These arms are relatively thin with a 2mm diameter and provide support to the mirror assembly to assure that these devices are adequately held up. With their orientation in this design they are pron to a bending moment which can cause mechanical defects over time To set up this simulation a fine mesh was used to achieve the most accurate results. In addition to this, a downwards force equivalent to the weight of the mirror was applied onto the supports. As the area of interest was specifically the upper portion of the arm, the lower base was set as fixed to ensure the forces would not just translate the entire body. For this analysis Brass was used to gather appropriate material properties. In terms of results, it was found that the mechanism far exceeds the strength and tensile threshold making it very safe for design usage to take place. With a max displacement of under 1mm, this demonstrates how sturdy the design is to the gravitational force exerted on it by the mirror. In terms of calculation comparison, the yield strength far exceeded that in which it was calculated. This was likely due to the material selected when simulation took place. For the calculations phase, plastic was used as it was assumed that this part would be 3D printed.

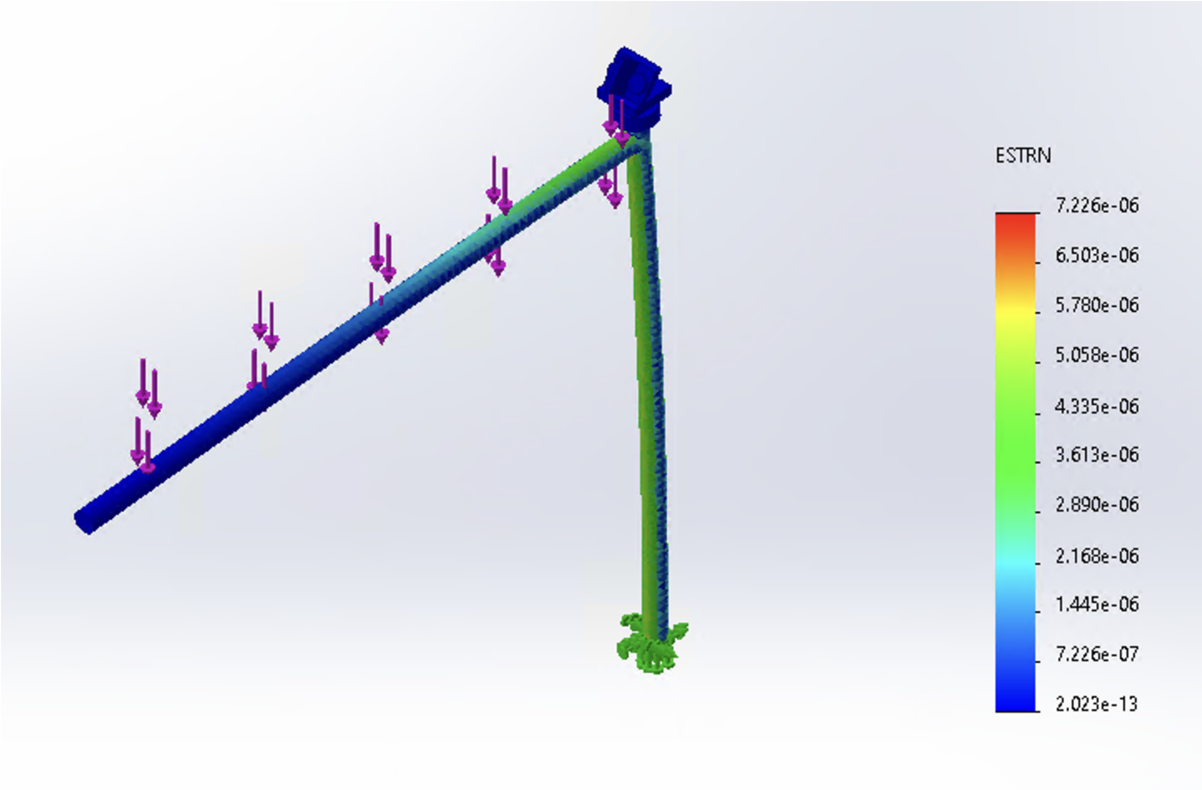


Figure 4: Tensile stress on mirror arm

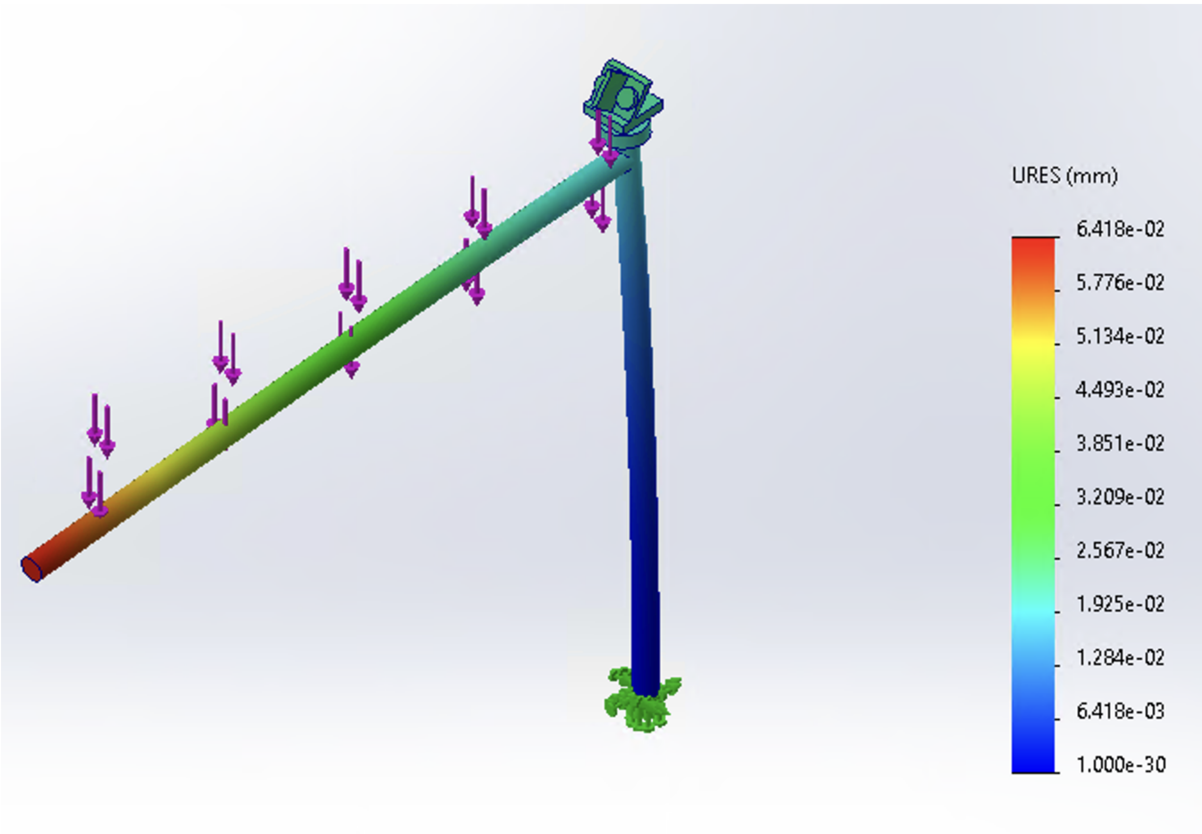


Figure 5: Yield stress on mirror arm

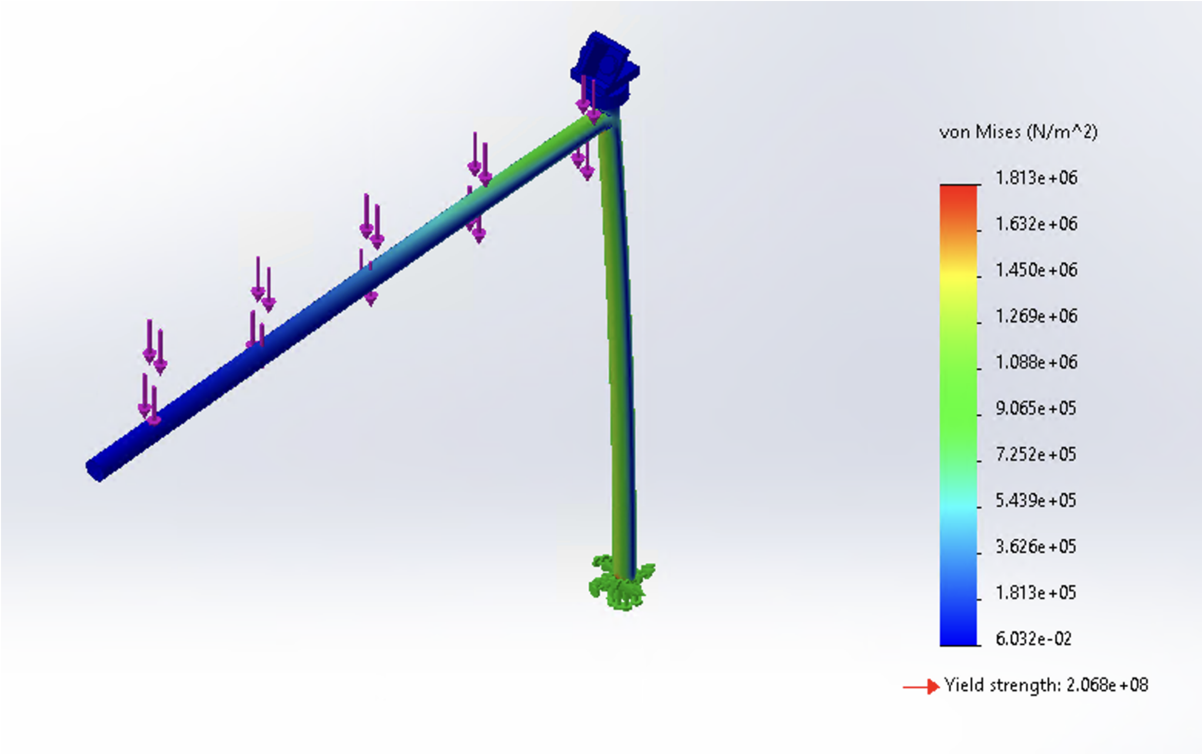
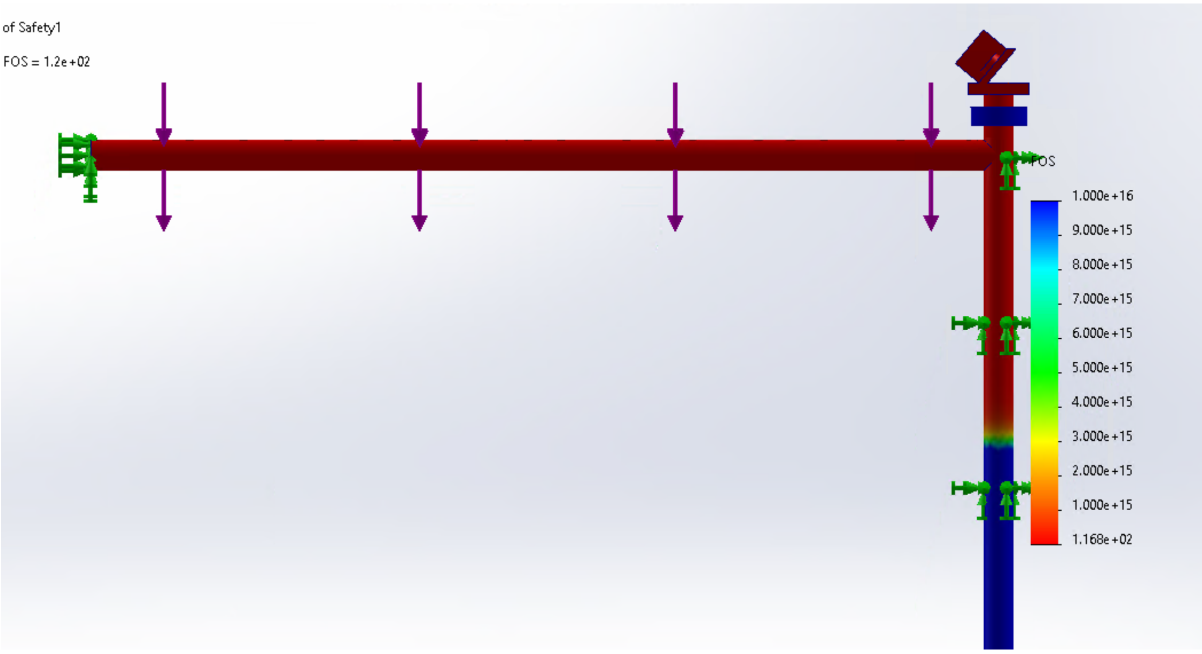


Figure 6: Von mises stress on mirror arm



Part 3: Base

The last part on the design for the simulations was the bass where most of the components will be concealed in an enclosure including the breadboard, arduino, motor, battery, wires and also the mirror arm. Having all this weight on top of the base in the middle, simulations had to be taken into account as we did not want the design to fail when having all this weight on top of the base. This is the part of the mechanism which will hold the most weight so it is essential that this component meets mechanical standards. For this analysis, a downwards force of 2.98N was applied to the op of the base plate. This value represents the estimated weight of the entire mechanism. In terms of fixtures, the metal collars were set to fixed as these parts of the base plate will be attached to the floatation device which is among one of the most sturdy parts of the design. By keeping these points fixed all of the force would stay on the base plate and would not be distributed to other bodies. This helps the design prepare for the worst possible case scenario when making a simulation. In terms of materials, ABS was used as this part was expected to be 3D printed. ABS is among the most common 3D printer filaments so choosing it allowed for the most realistic results. In terms of results it was clear that the base plate far exceeded expectations and also has a displacement of less than 1mm. With a high von mises stress and low tensile stress, this shows that the base design is very suitable for this application. In terms of calculations, it was originally assumed that AISI 304 steel would be used for this component as well, but for convenience ABS was used instead. This would imply that the values which were achieved from calculations were much higher then what was simulated due to the difference in material properties.

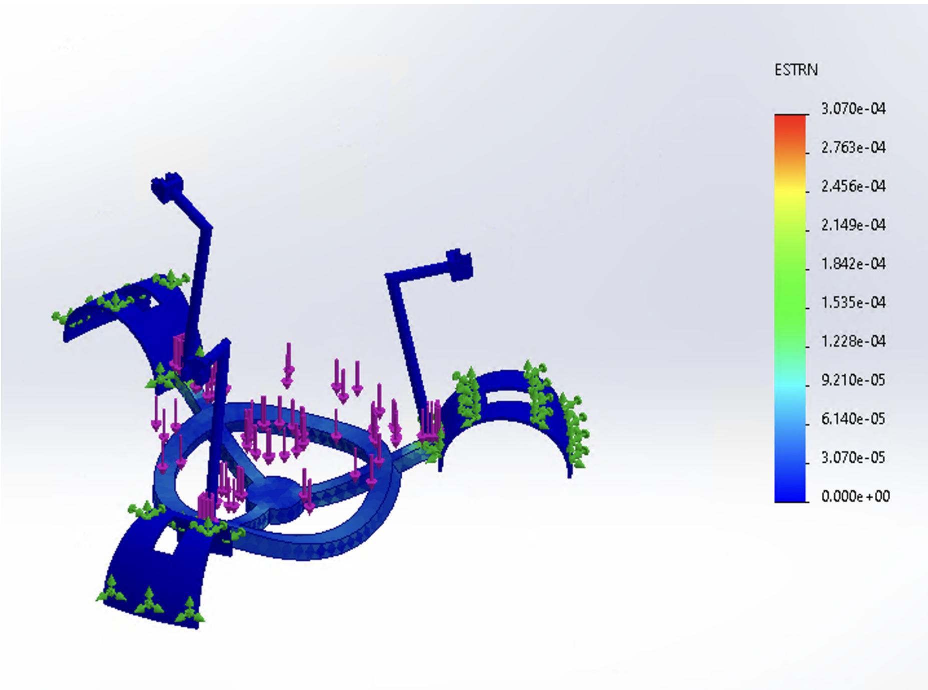


Figure 7: Tensile stress on base

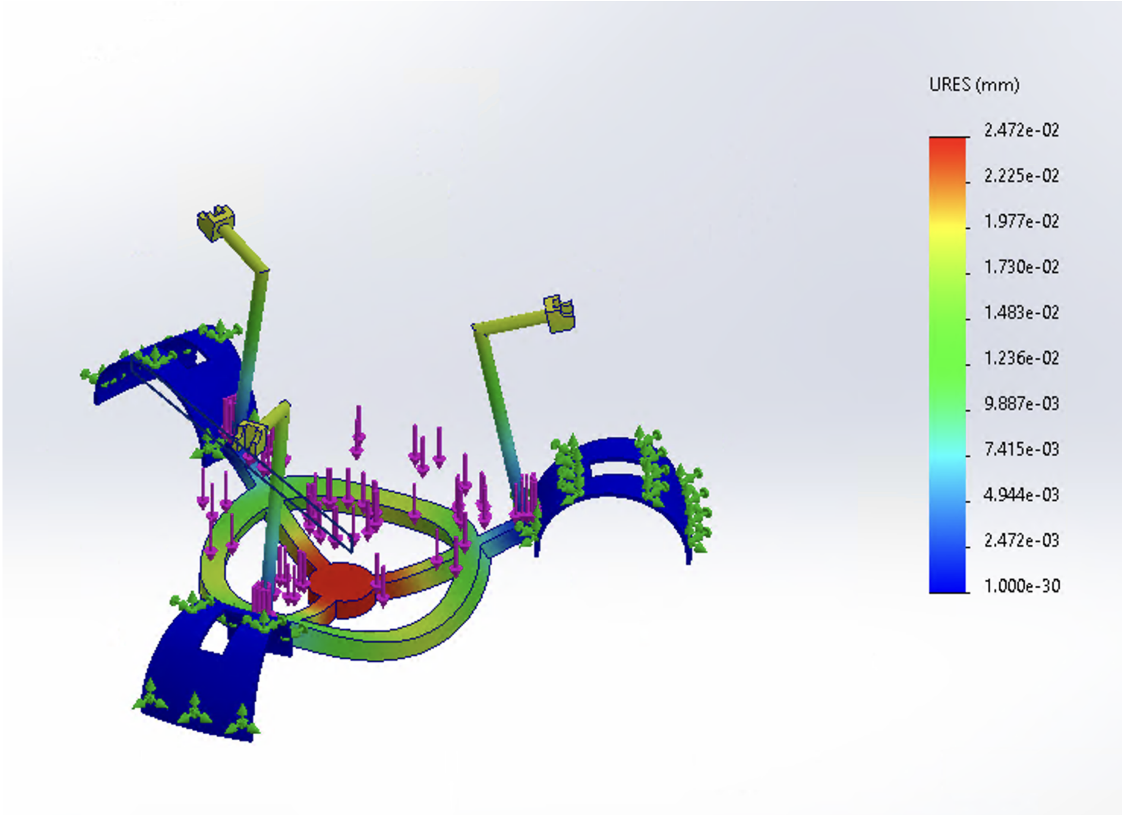


Figure 8: Yield stress on base

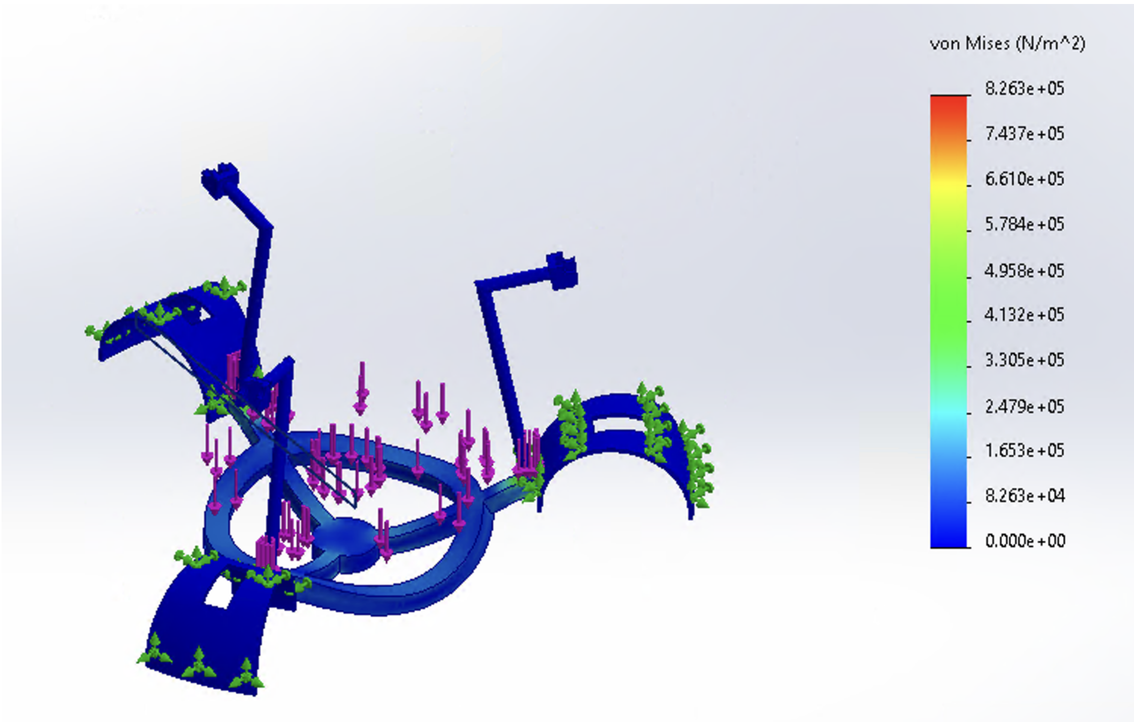
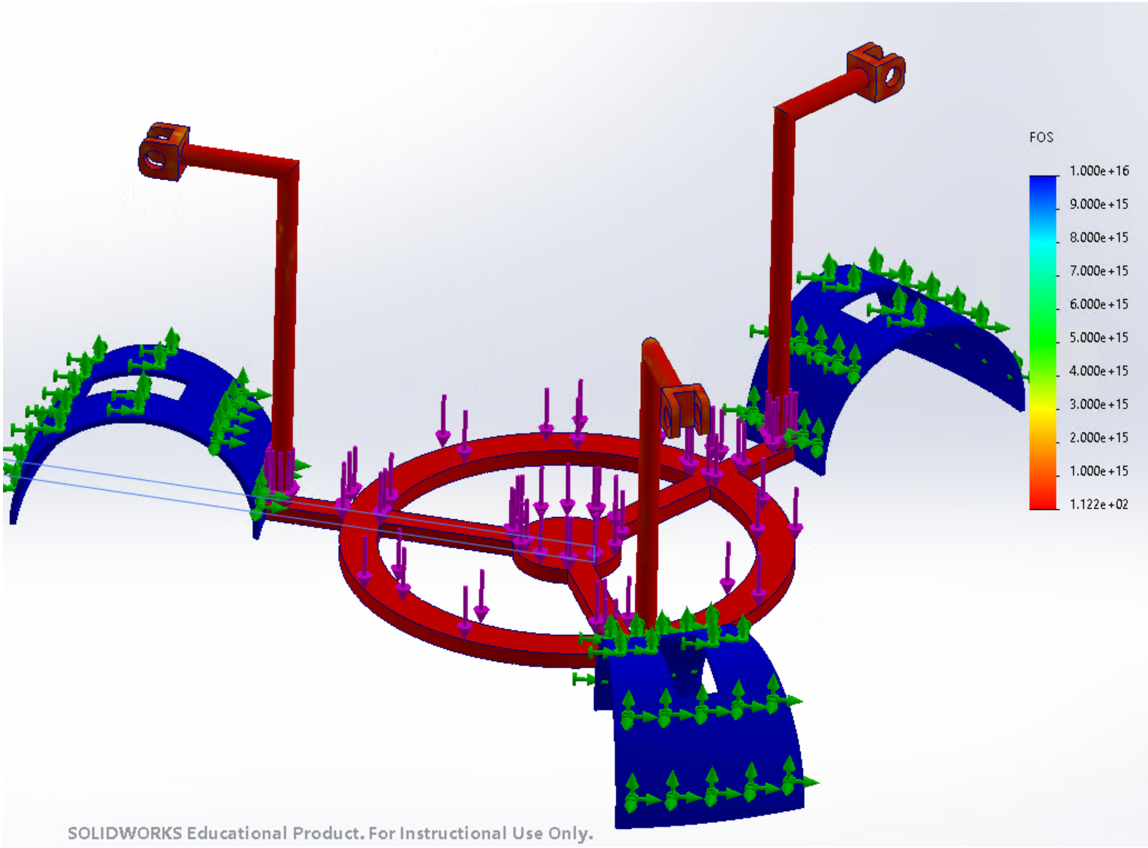


Figure 9: Von mises stress on base


FOS
1.000e+16
9.000e+15
8.000e+15
7.000e+15
6.000e+15
5.000e+15
4.000e+15
3.000e+15
2.000e+15
1.000e+15
1.122e+02
SOLIDWORKS Educational Product. For Instructional Use Only.