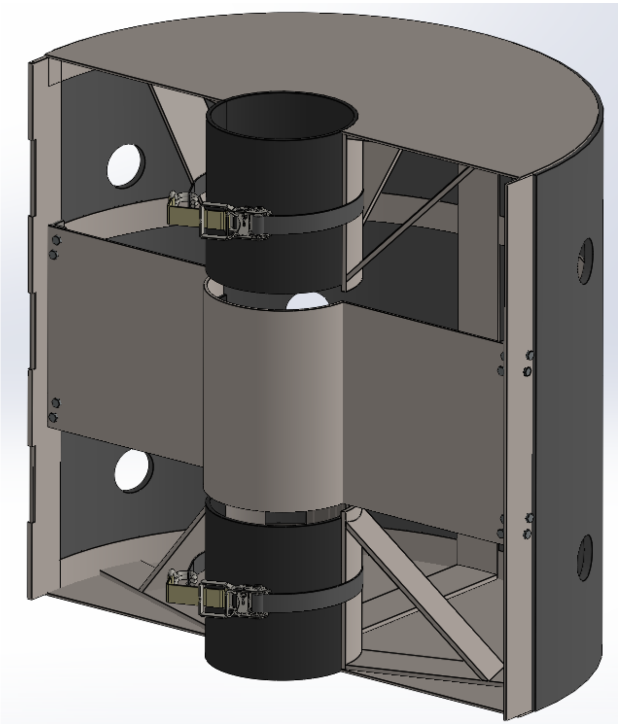
SMASHY - Functional Tests

FEM – Solidworks Simulations

In the following, simulations regarding the functionality of the SMASHY module are going to be conducted. It consists of FEM simulations, that were generated in Solidworks to determine the displacement, the stress, and the compliance with the factor of safety, equalling 2.

In the simulations, only the front part of the module is going to observed, as this is the part where external forces are going to be applied. Apart from that, the rest of the structure just gives an extra support from the other side and therefore only improves the overall sturdyness, which means that a successful simulation like this gives evidence, that the module is also functional as a whole. Apart from that, all unnecessary parts that are not structural and do not contribute to the stability were removed.

Used fix points are the rubber-rings that have contact with the traffic light pole, as well as the vertical components of the triangles which are strapped on, as displayed in the figure below. For the simulation, the ratchet straps and outer rubber-rings were removed as shown.



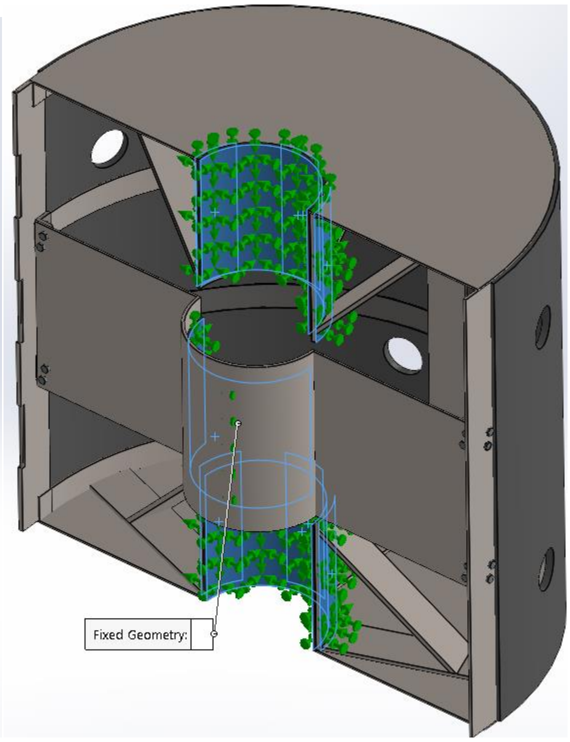
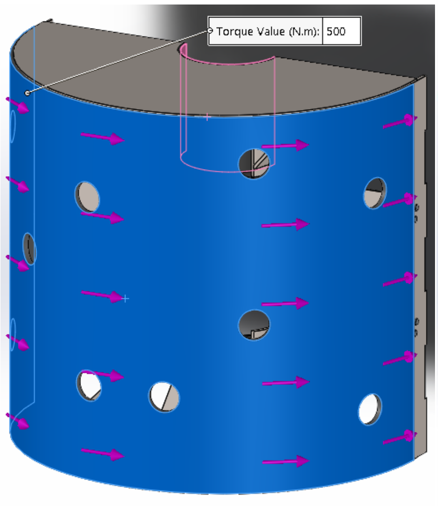


Figure 1 – Ratchet and strap

Figure 2 - Fixtures

The forces that were applied in the simulations are 1000 N from the top, a torque of 500 Nm applied around the vertical axis in the centre and a force of 350N in each of the three button positions that were selected. This equals a person standing on top of the module with his bodyweight evenly distributed, another or multiple persons hitting 3 buttons at the same time with a lot of power and someone pulling it back on the side, also with a force equal to his complete weight. These 3 buttons were selected as they display the region with the biggest risk of high displacement and stress, regarding their position relative to the framework.

The simulations were also performed with each load only by itself, aswell as in combinations with multiple loads at the same time, as it’s possible that they influence themselves and result in a decrease of displacement or stress. In the following, the highest simulated deformations and stresses that resulted are presented, which occurs with all loads applied simultaneously.



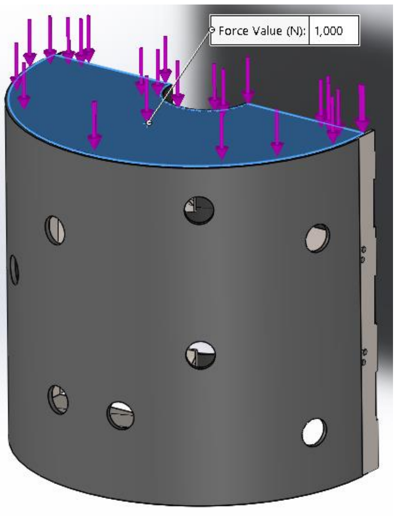


Figure 3 - Applied downward force

Figure 4 - Applied Torque

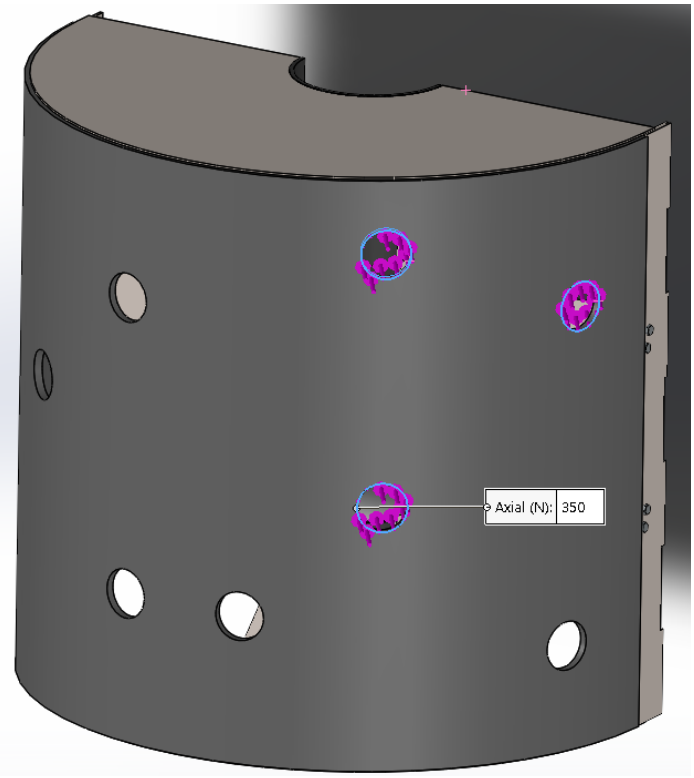
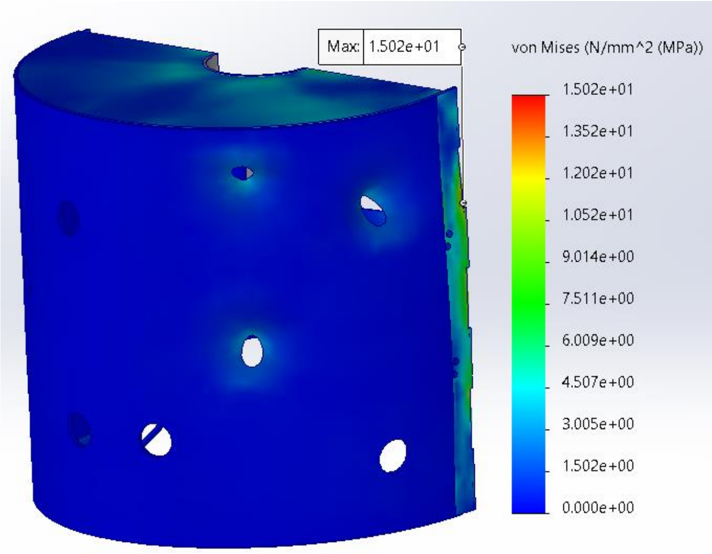


Figure 5 - Force applied to buttons

Stress Analysis (von Mises)



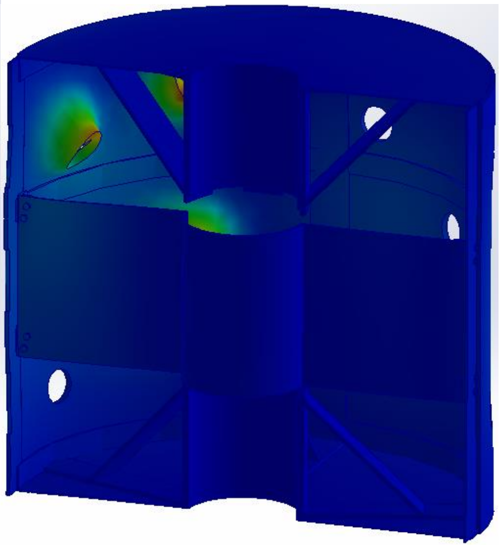


Figure 6 – Von Mises stress (frontside)

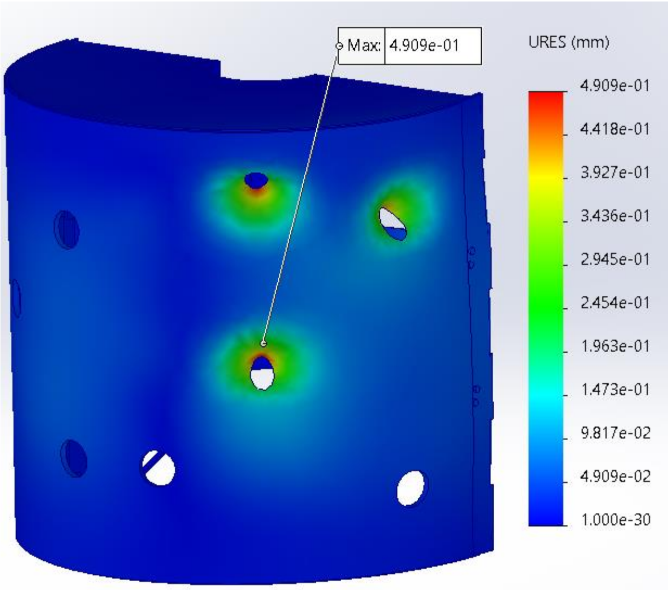
Figure 7 – von Mises stress (backside)

As to be seen in this simulation, the maximum stress does not surpass 15.02 N/mm^2 , which applies to the annotated point. The main internal structure is made of stainless steel with a yield strength of 172.34 N/mm^2, which shows that its going to withstand the applied forces with a high factor of safety.

Property	Value	Units
Elastic Modulus	200000	N/mm^2
Poisson's Ratio	0.28	N/A
Shear Modulus	77000	N/mm^2
Mass Density	7800	kg/m^3
Tensile Strength	513.613	N/mm^2
Compressive Strength		N/mm^2
Yield Strength	172.339	N/mm^2
Thermal Expansion Coefficient	1.1e-05	/K
Thermal Conductivity	18	W/(m·K)

Table 1 – Properties of stainless steel (ferritic)

Displacement



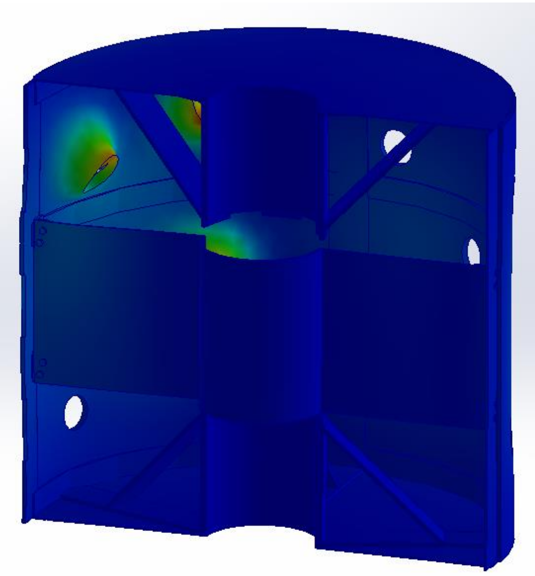
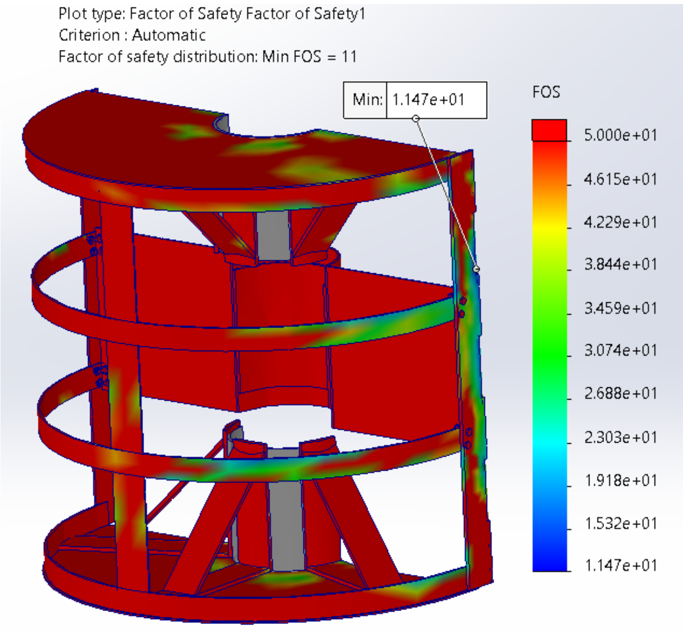


Figure 8 – Displacement (backside)

Figure 9 – Displacement (frontside)

Here the maximum displacement is visible, which occurs on the side of the HDPE plate, which equals 0.4909mm. Regarding the fact that the forces in this simulation were applied to the edges where the buttons sit, a lower displacement can be expected in the actual model. This is because the force will be applied on the button and then to a rubber ring, which distributes the force of hitting the button on to a bigger surface area. Therefore this displacement is still acceptable.

Factor of safety



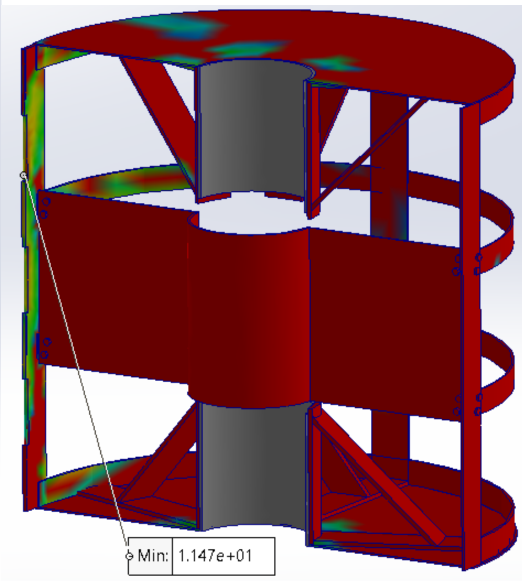


Figure 10 - Factor of Safety (backside)

Figure 11 - Factor of Safety (frontside)

This figure displays the factor of safety (FOS), with the FOS of above 50 being shown in red. The minimal factor of safety values 11, which exceeds the minimum value of 2 we set.

In this case it would be possible to use different materials with a lower yield strength or design the parts to be thinner, so less material is used. But because this is only a static simulation, dynamic and continuous loads and stresses over a long period of time are not simulated but occur in the real use case. That’s why the structure like this is acceptable and reasonable and due to fact that the other simulations were successful too, no changes to the structure must be made.