Finite Element Analysis of Arctic Plant Growth System Base Plate

Sudip Karmacharya Brenna McKim

December 7^{th} 2020

1 Abstract

Statement of the Problem: The Base plate of the Arctic Aeroponics System is an important part of the system that takes the weight of entire system. Deformation of the Base plate can cause disruption of the function of the system, however, it also has to be as light as possible to keep the weight of the system down.
Purpose: Mathematical simulation of the Base plate to optimise the thickness of the Base plate to reduce weight, while ensuring the stress does not exceed the material's yield strength.
Material and Methods: Stress values were simulated in the Base plate with Finite Element Analysis software ABAQUS CAE. Four varying thicknesses were analyzed: 5cm, 2.5cm, 2cm and 0.5cm. A distributed load of 4203.22 N was applied to the Tank Base and a distributed load of 196 N was applied to the Top Surface. The Feet Supports Base had a fixed boundary condition. Values of von Mises equivalent stress were computed for each variation.
Results: Maximum stress was found to be along the edge where the Feet Base comes into contact with the plate. The stress levels increased by 2291.45% by decreasing the thickness by 90% from 5cm to 0.5cm. For the 0.5cm plate, its maximum stress was 79.38% under the yield of Al-6061.
Conclusion: The study has successfully reduced the weight of the Base plate while ensuring acceptable stress level. The design of the Base plate can be further improved to lower the weight, as the stress levels is low. However, precautions need to be taken as the sheet metal also acts as an insulating layer.

There exists a need for a system to maintain the least acceptable temperature, humidity, and nutrients to grow plants in the temperatures range of -50 deg C to 10 deg C, to provide a family of 3 people with produce and fight food insecurity in remote arctic communities. Therefore, such a system is being developed by University of Waterloo 4^{th} year Mechanical Engineering students for their Capstone Project. The system is depicted in Figure 1. A critical part in this system is the Base plate that supports the design.

The Base plate is loaded with weight from: water in the 50 gallon tank, PVC pipes, the top plates, and necessary sensors. The weight of everything except the sensors rests on the area of the water tank projected onto the Base plate. The weight of the sensors rests on the outer body. The system has five support feet connected to the Base plate. This provides an air gap between the system bottom and the ground for added insulation. Lastly, the sensors are screwed to the Base plate, however, as the type of sensors is not finalised, they are omitted from the CAD models. The sensors need to be screwed to the base, constraining the plate thickness to 0.5cm.

This simulation study compares the influence the Base plate thickness has on its stress distribution to identify maximum possible weight reduction. For this purpose, the Base plate was modeled used 3D graphics and the plate stress distribution was computed by Finite Element Analysis (FEA).

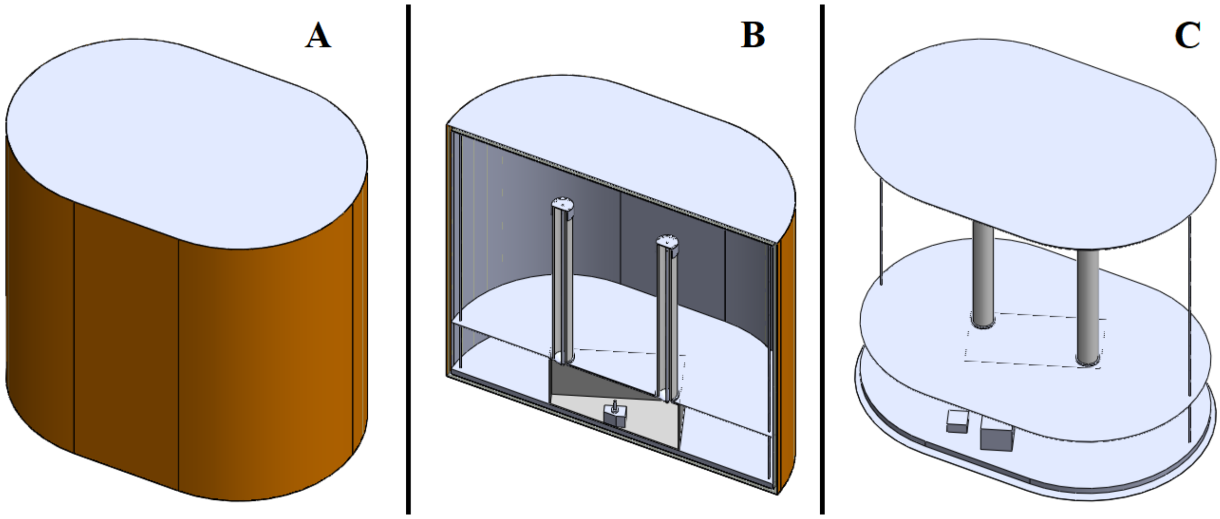

Figure 1: Arctic Plant Growth System CAD Model depicting the Base plate, placed right above the bottom most plate (exterior plate)

2 Material and Methods

The selected material is 6061 aluminium, with a preliminary thickness of 5 cm. For defining the material in ABAQUS, the Al-6061 was assumed to be homogeneous and isotropic. CES EduPack was used to determine the values for yield strength (σ_y) and Young's Modulus (E) within the described temperature range [1]. The smallest σ_y value over the temperature range, 140 MPa, was used as it was the most conservative values. The same reasoning was used to select E = 68.3 GPa; underestimating E results in a less ductile behaviour. For this loading scenario this causes the resultant stress to increase, giving a conservative estimate as well.

Four varying plate thicknesses were analyzed: 5cm, 2.5cm, 2cm and 0.5cm. Each study had five key regions for developing the mesh, adding boundary conditions (BC) and applied loading, shown in Figure 2: where the water tank was placed (Tank Base), location of the support stands holding the plate (Feet), the area around the edge of the Feet Supports Base (Feet Rings), the top face of the plate excluding the Tank Base region (Top), and the bottom of the plate (Bottom).

In ABAQUS, the initial step defines the plate BC. The plate is placed on the five Feet that are secured to the ground. Thus, for the plate, the rectangular area that the Feet are in contact with the plate were defined using Fixed BC.

There were two different loadings that needed to be considered: the weight directed from the water tank, and the total sensor weight. A projection of the tank was partitioned from the top surface of the CAD model, and a distributed load of 4203.22 N was applied only to that surface. The total sensor weight was approximated to 196 N, and was applied as a distributed load to the Top as specific sensor locations are unknown at this time.

A symmetry BC was not used for this problem due to the asymmetrical position of the Tank; resulting in asymmetrical loading. If the Tank was centred on the plate, a quarter model of the plate could be created; then two symmetry BCs could have been used to reduce the computational expense of the model.

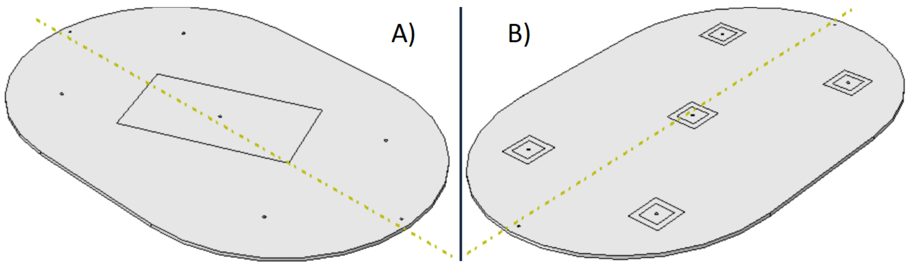

Figure 2: A) Top view of the Base plate depicting the Tank Base and Top regions. B) Bottom view of the plate depicting the Feet Supports Base, Feet Rings and Bottom Surface regions.

Using engineering judgement, the critical locations for the plate were identified as the Tank and Feet Rings (closely followed by Feet Base). These regions are designed with a finer mesh while the rest would have a coarser mesh as they do not require a high level of accuracy a fine mesh provides. Additionally, it offsets the computational expenses from refining the mesh for the Tank, Feet, and Feet Rings. Next, a mesh convergence analysis was completed using P-refinement and H-refinement. For P-refinement of the Tank Base mesh, the different max stress values are shown in Table 1. There is a significant difference between the element orders, thus quadratic elements will be used for meshing the part.

Element Order	**Maximum Stress (Pa)**
Quadratic	650600
Linear	59890
Difference =	590710

Table 1: P-refinement for Tank Base

The results of the H-refinement mesh convergence analysis for the Tank are shown in Figure 3. It can be seen that, after a point, the values of stress stabilise. This point was element size of 75mm. However, it can be observed that after the mesh size of 50 mm the values are even more stable; therefore, the most suitable mesh size was determined to be 50mm. The mesh size could not be reduced lower than 5mm because it would exceed the academic element limit of ABAQUS. From these results, the optimal mesh size for each section is shown in Table 2.

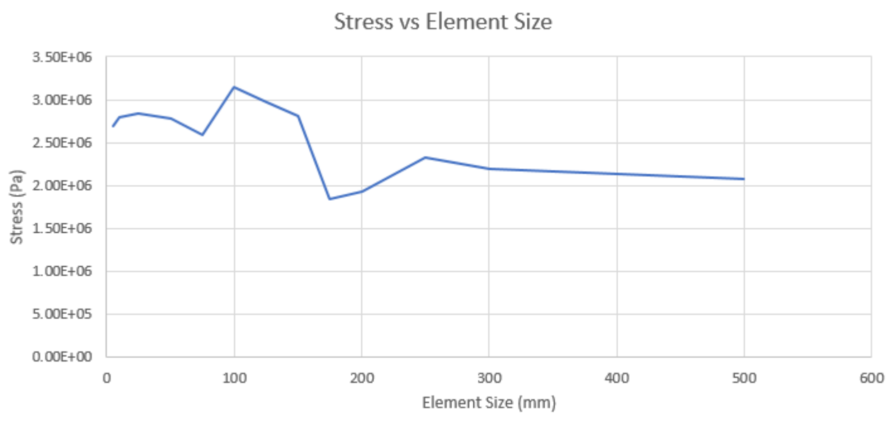

Figure 3: Stress vs Element size in the Tank Base

Part Section	Mesh Size (mm)
Feet Base	50
Tank Base	50
Feet Rings	75
Top Surface	150
Bottom Surface	150

Table 2: Final Mesh Sizes for Part Sections

3 Results

The first simulation that was run was with the 5cm Al-6061 plate with the mesh sizes for each section outlined in the previous section. As expected, areas of higher stress were located near where the Feet Base come into contact with the plate and directly underneath the Tank Base, seen in Figure 4. The highest stress was located in the corners of the Feet Base on the Bottom Surface at a value of 1.321 MPa. The regions exposed to the 20 kg distributive load experience very low stresses. The maximum stress is 0.94% of Al-6061's yield strength, thus it was determined that the thickness of the plate can be reduced as there is sufficient room for subsequently higher von Mises stresses in the part.

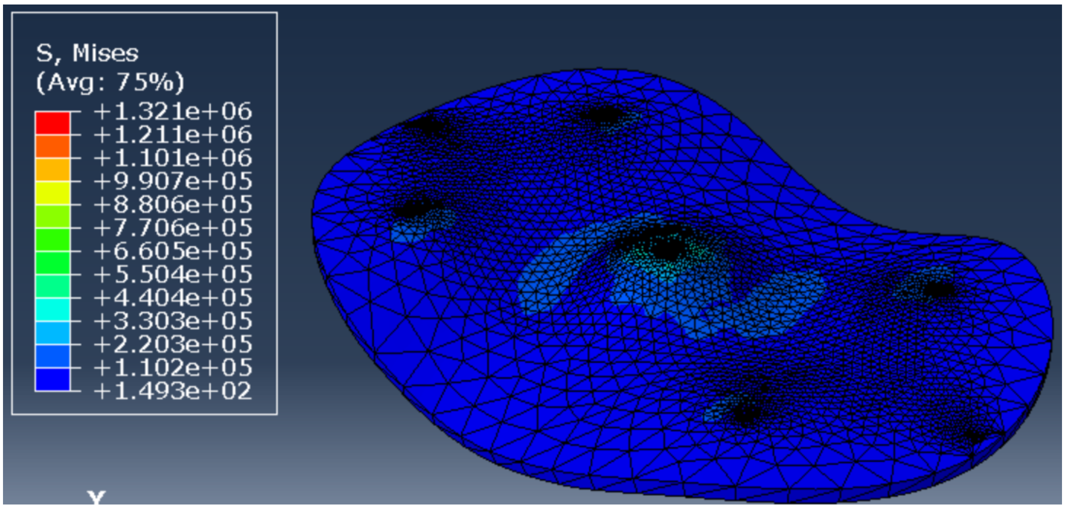

Figure 4: 5cm Thick Plate von Mises Stress Results Isometric View

For the second simulation, the thickness of the plate was reduced by half to 2.5cm, with the von Mises stress plot shown in Figure 5. With half the material, the maximum stress increased by 347.5% to a value of 4.59 MPa and still occur in the same critical regions as before. The regions exposed to the 20 kg distributive load increased approximately by a factor of 2; however, the stress in the Top Surface is still significantly lower compared to the Feet Base, Feet Rings, and Tank Base regions. The maximum stress for the 2.5 cm plate is 3.28% of Al-6061's yielding point. It was determined that there was still room to reduce the thickness of the plate while staying underneath the material's σ_y.

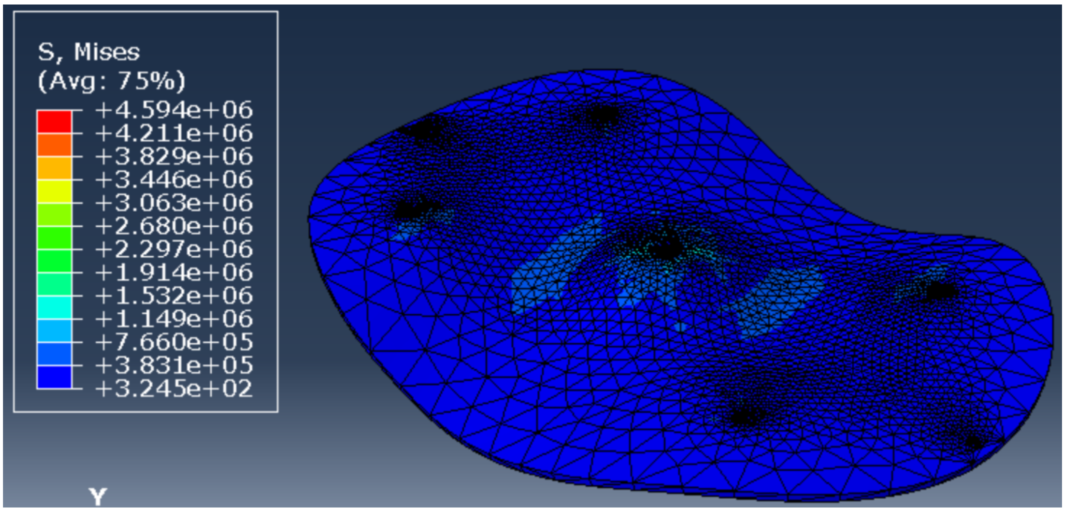

Figure 5: 2.5cm Thick Plate von Mises Stress Results Isometric View

In the third simulation, the plate thickness was reduced slightly from the previous run. The maximum stress is 6.376 MPa, 138.9% increase from the 2.5cm plate, seen in Figure 6. The maximum stress in the 2cm plate is 4.55% of the yield stress.

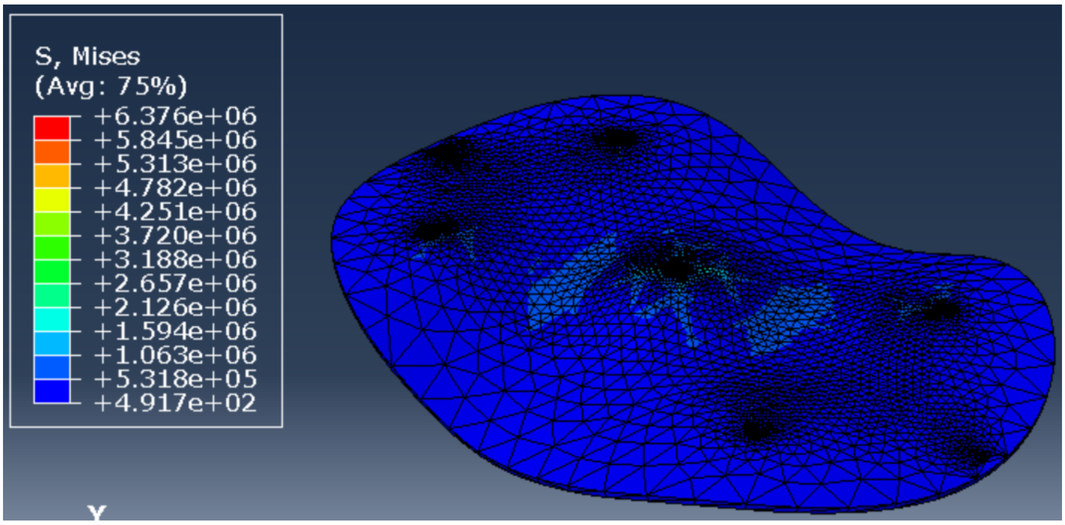

Figure 6: 2cm Thick Plate von Mises Stress Results Isometric View

In the final simulation, the plate thickness was reduced to 0.5cm. A large factor of safety is still desired for this design; thus, it is desirable to have the maximum stress in the part to be considerably below the yield stress of Al-6061. That being said, there was justification to save more material, equating to decreasing expenses, and reduce the plate to a standardized plate thickness of 0.5cm. The maximum stress, by further reducing the plate thickness by 1.5cm, increased by 474.75%. As seen in Figure 7, the maximum stress in the 0.5cm plate is 30.27 MPa, 20.62% of the yield stress. This was used, coupled with the FEA results, as a deciding factor that 0.5cm is a safe plate thickness to finalize the plate design.

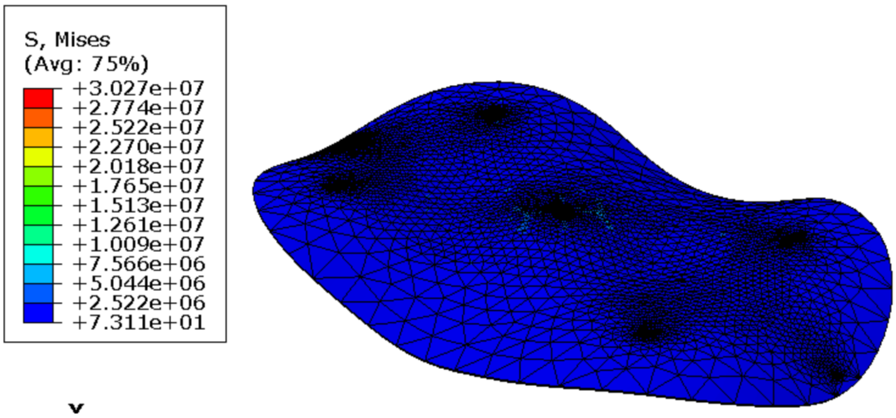

Figure 7: 0.5cm Thick Plate von Mises Stress Results Isometric View

4 Discussion

As stated previously, the largest stress seen in all the studies, regardless of the thickness, was along the edge of where the Feet Base came into contact with the plate. This is because the downward distributed load causes the aluminum plate to want to bend as shown in Figure 6. Due to the presence of the supports, the plate is constrained to bend around the feet. Thus, the corner of the feet press into the aluminum plate, increasing the stress.

The results from the FEA model are cohesive to what is known within solid mechanics. The cross-sectional height of a structure is proportional to the moment of inertia by a power of 3. When the plate thickness decreases, it significantly decreases the plate's moment of inertia. Thus, the plate is can bend more, increasing bending stresses as seen through the increasing stress from the FEA simulations.

From the simulation, it can be seen that the stress levels have increased by 2291.45% by decreasing the thickness from 5cm to 0.5cm. The thickness of the Base cannot be decreased more due to the length of the screws required to attach the sensors to the plate. The density of Al 6061 is 2.71×10^3 [1]. Using this values, we find that weight of the Base plate has decreased from a value of 577.23 kg at 5cm thickness to a value of 57.23 kg at 0.5 thickness. This is a 90% decrease in weight.

As mentioned previously, the largest stress was seen at the Feet locations. However, these were modelled as simple squares on the base of the plate. The BC used on this section was a Fixed BC. The stresses in this area may change with improvements to the FEA model. This improvements could be the inclusion of modelling bolt preload and more accurately representing the connecting face of the Feet supports being sourced for this project.

These results demonstrate that further design improvements can be made to the plate. The stress levels are only 20.62% of yield, thus weight saving techniques for the plate can be done safely. These techniques include hole punching or CNC pocket milling. However, it is important to ensure that the system remains insulated, and further changes to the bottom plate may negatively impact this design factor.

Another future design change would the Feet Base supports holding the plate. Further research can be completed to find feet supports with larger fillets on the corners to decrease the stress that occurs from the plate's bending stresses induced by the water weight. Another consideration is removing the center feet support as the stress levels are very low.

5 Conclusion

The study has successfully reduced the weight of the Base plate while ensuring acceptable stress level by decreasing the thickness from 5cm to

0.5cm. The design of the Base plate can be further improved to lower the weight however, precautions need to be taken as the sheet metal also acts as an insulating layer.

6 References

[1] CES Edupack. (2020). Cambridge, UK. Ansys Inc.