Project Proposal and Description

The initial project proposal was submitted to optimize the landing gear of a medium-sized RC Plane. The span of the plane is assumed to be 3m, with the maximum length of landing gear in the wing direction at 450mm. The location of the attachment of the landing gear to the main airframe is through 4 struts. The space for the struts is provided at the top (4 mounts). The hole for the wheel attachment is provided on the side bottom.

Furthermore, since the plane is entirely symmetrical from the wing bisectional plane, the landing gear is only simulated for one side. The three load cases were briefly mentioned in the original proposal to be three different landing scenarios. The complete description of the part, the main objective of the optimization, and the loading conditions are as follows.

Description of the part to be optimized

The Landing gear is an important structural member of the Primary Aircraft. The landing gear must be designed to handle the impact loading conditions of the aircraft. The Landing gear's dimensions have been identified based on the location of attachment to the airframe and the desired ground-wing clearance(290mm). These constraints fix some of the width and the height of the design domain. But leave the breadth of the design domain to be an open-ended dimension that can help accommodate the design requirements alongside the constraints.

Basic Information on the plane:

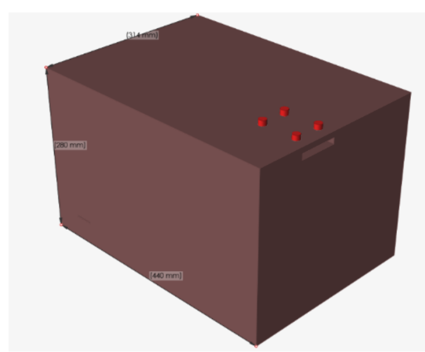
1. The entire weight of the aircraft is 14 Kg
2. Location of CG with respect to a wheel:
    a. 450 mm (from the plane of symmetry)
    b. 290 mm (height of CG from the wheel center)
    c. 160 mm (backward from the wheel)

Vectorized format of CG (considering wheel as the origin): **<0.8054, 0.2864, 0.5190>**

Specification of the objective function

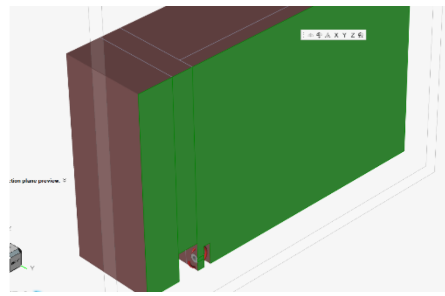
The landing gear is an essential structural member and should handle harsh impact landing load conditions. This requires the landing gear to be as strong as possible. Thus, the primary objective function I am trying to solve the optimization problem for is the stiffness function. The main goal is set to Maximize stiffness in the Optimization runs. Furthermore, the landing gear is a part of an aircraft, and hence the weight of the landing gear is also an important aspect. Also, the material is chosen to be aluminum, and the volume fraction is tried to be kept as low as possible with a significant minimum factor of safety.

A drawing of the design region



The initial design space was chosen to be a cuboid with the height and length taken from the problem statement and the smallest allowable setup. The width was set to the minimum possible bounding box and was increased upon the subsequent runs. Even the height and the length were later modified to the possible extent so that it doesn't break the constraints but at the same time doesn't stop the material from flowing. The final design space used for the optimization can be seen here. Furthermore, as can be seen, an opening is provided on the bottom of the strut holes for easy accessibility. An Engineering Drawing of the design region has been attached for your reference.

How it connects to non-designable components

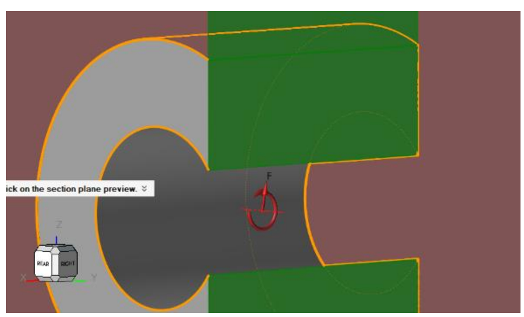


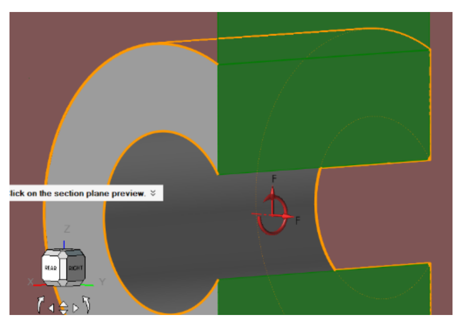
The main connecting point of the landing gear to the airframe is the holes on the top surface. The landing gear will be bolted to the airframe base with four bolts (four partitions here). The slot for accessing nuts at the bottom of the structure is also provided in the design space. Furthermore, the wheel is attached to the right-hand base of the design space, as can be seen in the figure here. Extra slot is also provided behind the wheel attachment location for accessibility. The Design space was subsequently extruded further to accommodate the material flow. This will be discussed in detail in the problem setup.

A set of load cases (You must provide at least three distinct load cases)

Three independent load cases were identified at the start and provided in the initial proposal. Since we want to design for the worst-case scenario, the possible loading cases are all one-wheel landings, with the entire weight of the plane (14 Kg) on the wheel. Furthermore, the aircraft will be tilted during an imperfect landing, and thus, three preliminary design considerations are

1. The plane should be able to land with a left tilt – maximum left tilt angle calculated based on the wingspan and the ground clearance.
2. The landing gear should be able to handle a nominal amount of front tilt – the front tilt angle is decided heuristically and fixed to a full front tilt thumb rule available through online resources.
3. The landing gear should also withstand the reaction forces if the brakes have already been applied on the wheels. Along with the regular forces moments, a substantial frictional force is also experienced.





Geometric and other constraints must be satisfied.

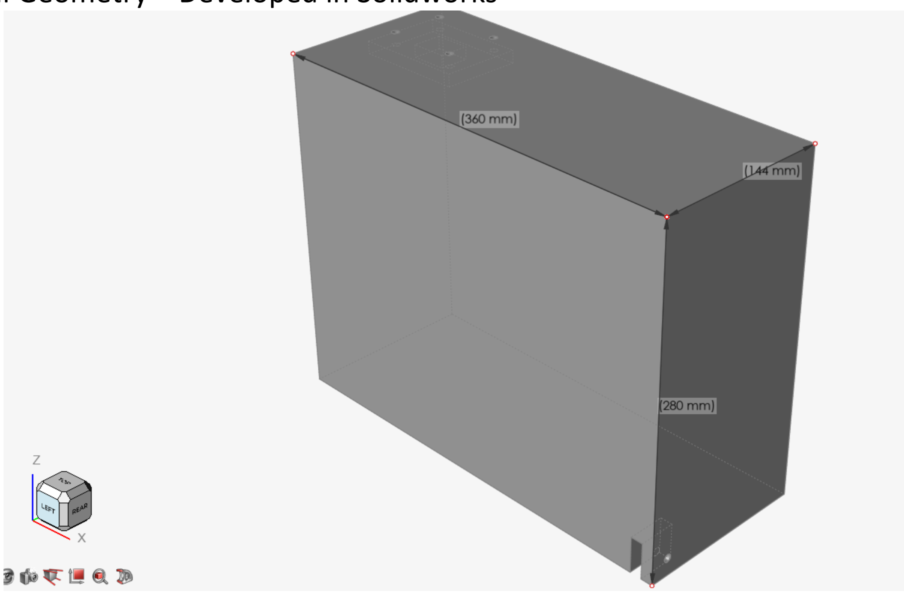
I want my plane to be as stable as possible during the landing. Thus, I have taken a different approach and have fixed the airframe. I have considered the airframe as the static part and the four bolts to be the fixed support. The wheel is where the reaction forces from the ground will be applied. Thus, I have added the static support constraints at the four bolted holes, apart from the geometric constraints already mentioned.

The Support Constraints at the top are the same for all the loading cases. The main difference is at the bottom contact point with the wheel. The force and the Moments applied can be seen in the diagrams below. – (left tilt 8^0, left tilt 12^0, High Frictional Force It is necessary to make sure that the landing gear does not contain extremely thin members with a grid-like structure, and thus, a minimum thickness of 24 mm was given in the runs. This ensures that the minimum size of any member is at least 2 cm, thus ensuring it is a feasible engineering design. Furthermore, the objective was to increase the stiffness, and thus, high strength is ensured as a result. The minimum safety factor is maintained around 4, which will be further discussed in the post-processing section.

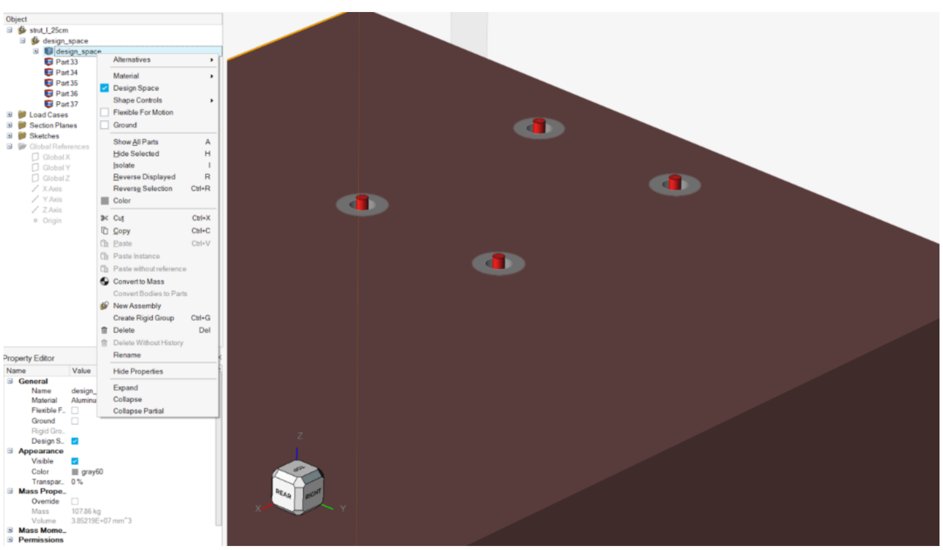
Problem Setup

Describe how you set up this problem for a solution using Inspire. Take screenshots of the process as needed.

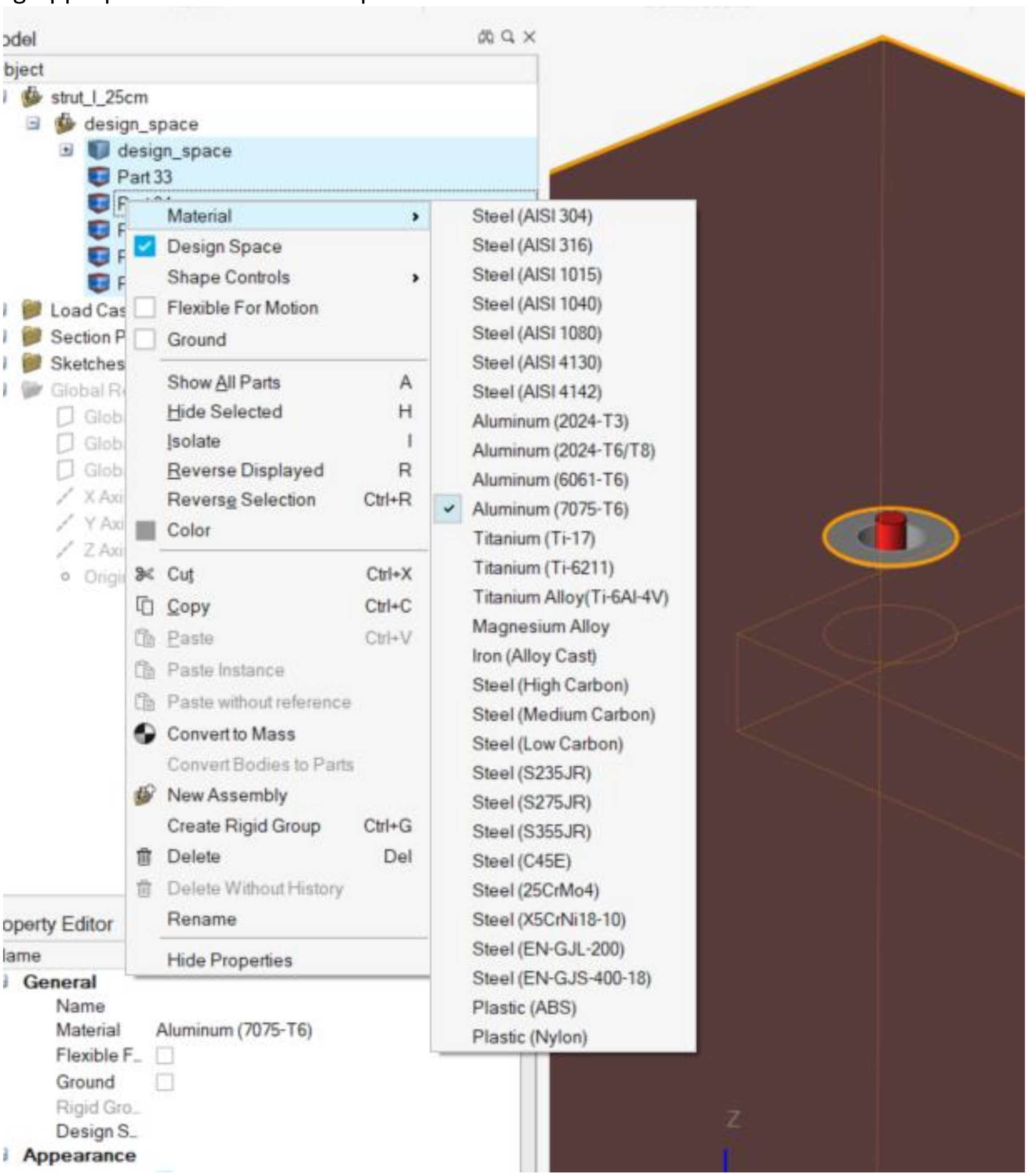
1. The initial Geometry – Developed in Solidworks



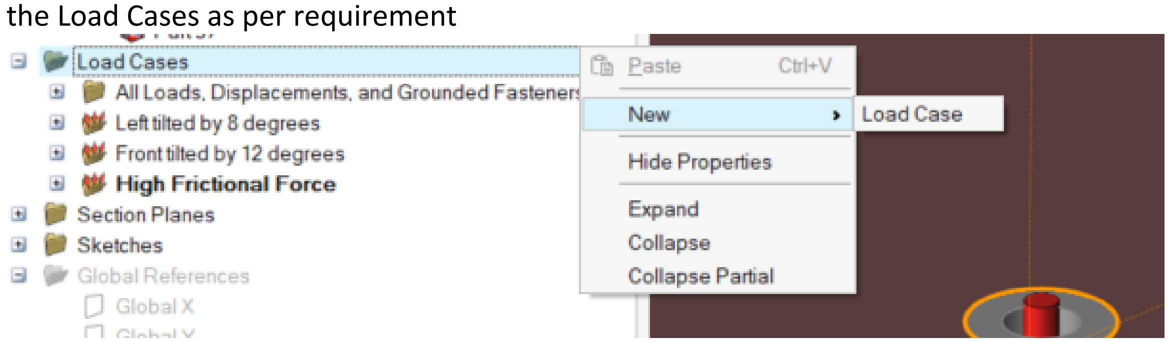
2. Creating Partitions from the Original Geometry & generating the design space



3. Assigning Appropriate Material to the parts



4. Adding the Load Cases as per requirement



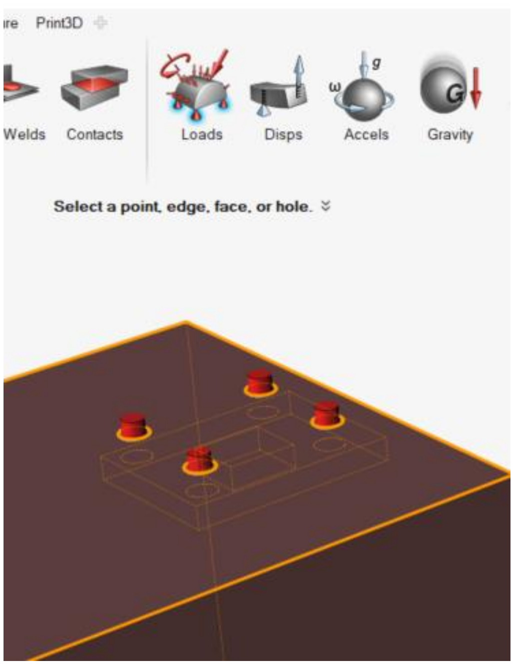
5. Load Calculations

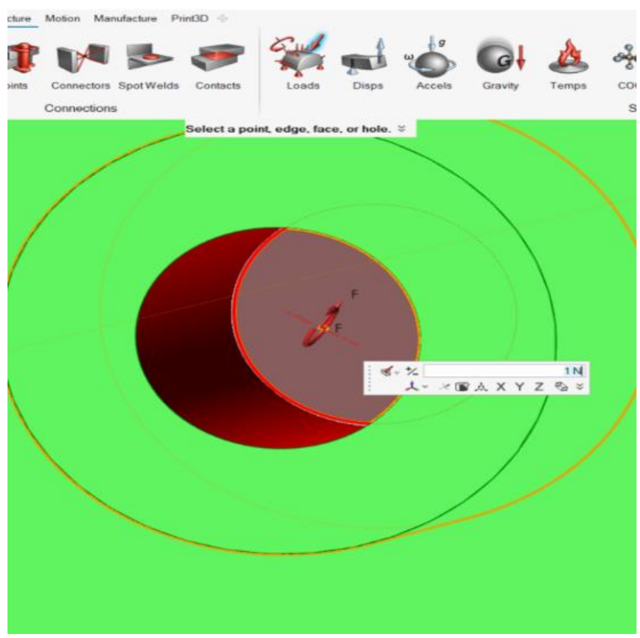
Tilt Angle calculations were done using dot product formula on normalized vector location of CG, considering the wheel of the origin.

Load Cases:

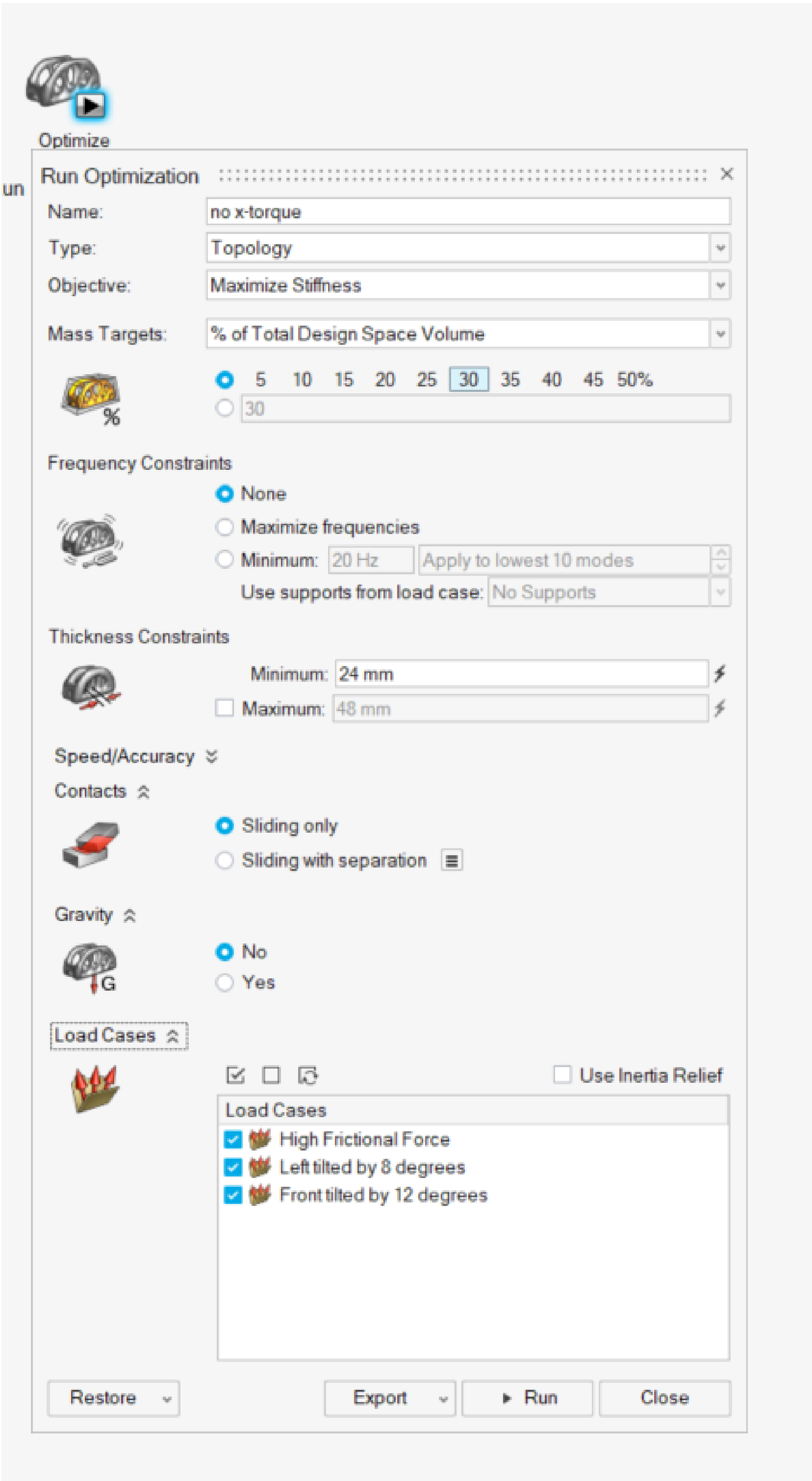
1. Leftwards Tilted Plane
    a. New location of CG: <400,160,355.81> - App. 8 degrees of tilt
    b. Normalized Vector: <0.7159, 0.2864, 0.6368>
    c. Net Moment on the wheel: <22.4, 49> Nm
2. Forward Tilted Plane
    a. New location of CG: <450,50,327.41> - App. 12 degrees of front tilt
    b. Normalized Vector: <0.805371, 0.08946, 0.585970>
    c. Net Moment on the wheel: <7,63>
3. High speed Landing with brakes
    a. location of CG: <450,160,290>
    b. Frictional Coefficient: 0.75
    c. Net Moment: <22.4,63> Nm
    d. Frictional Force on the wheels: -y Dir: 105

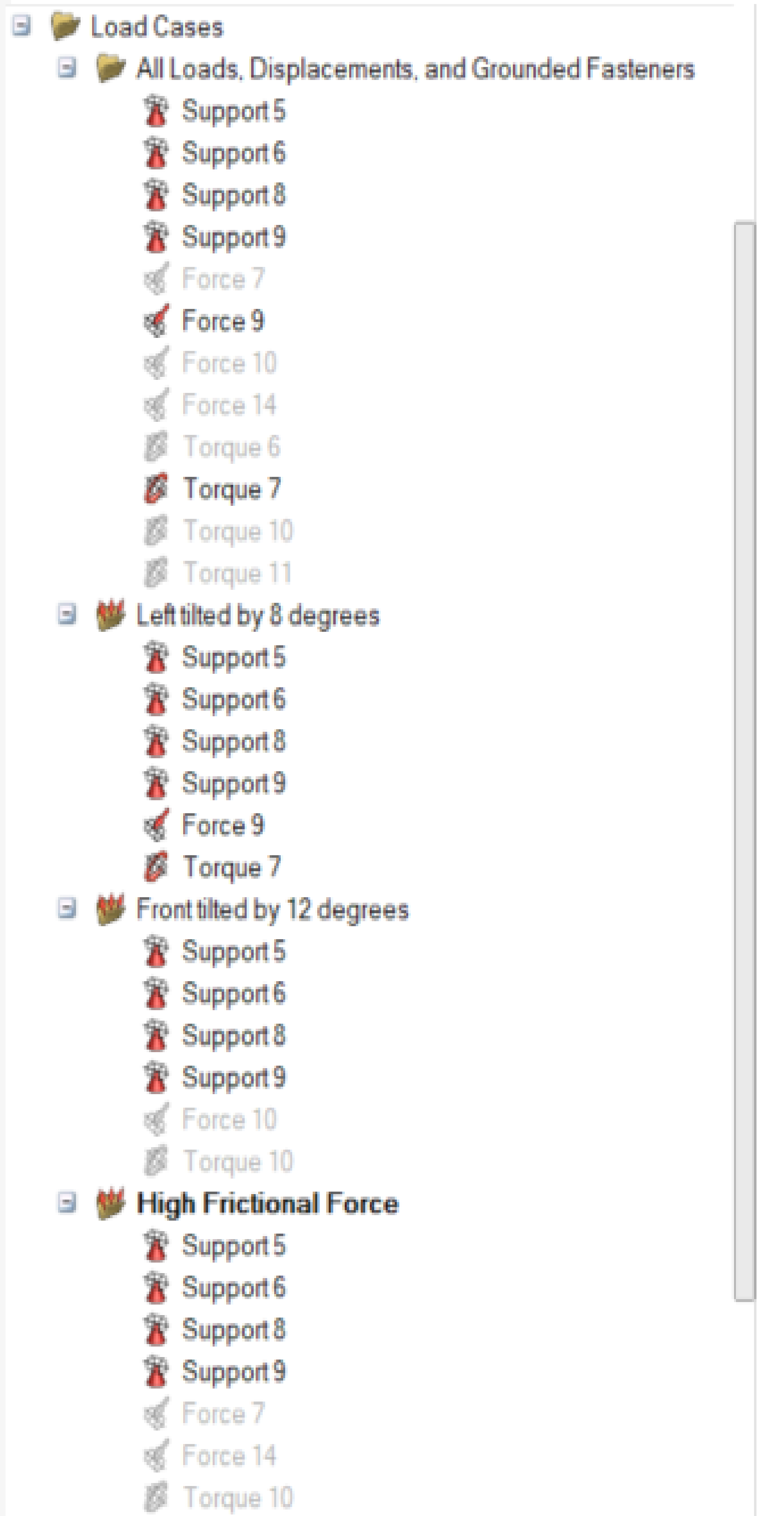
6. Filling in Appropriate support constraints and loads



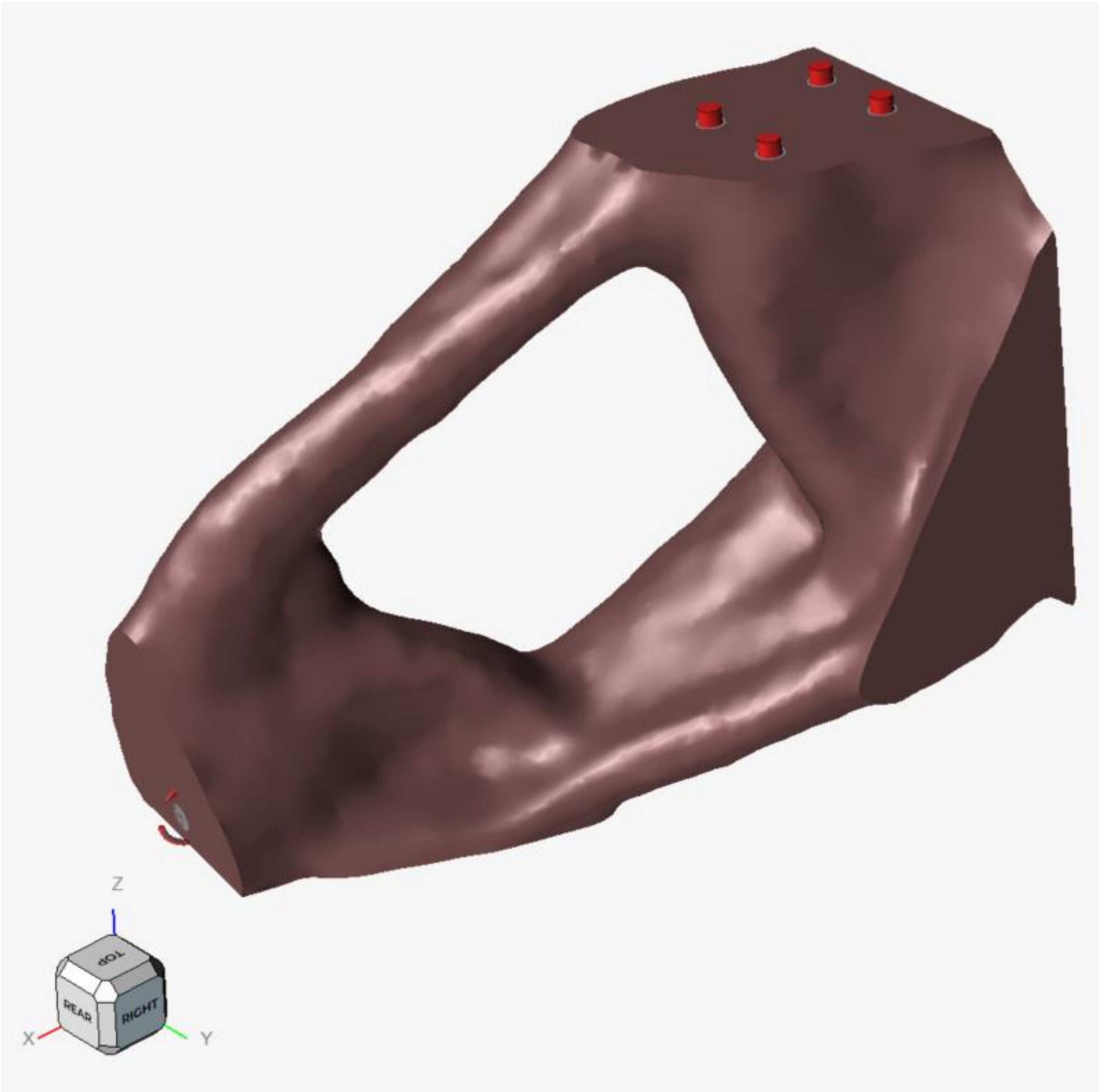


7. Running the Optimization cycle



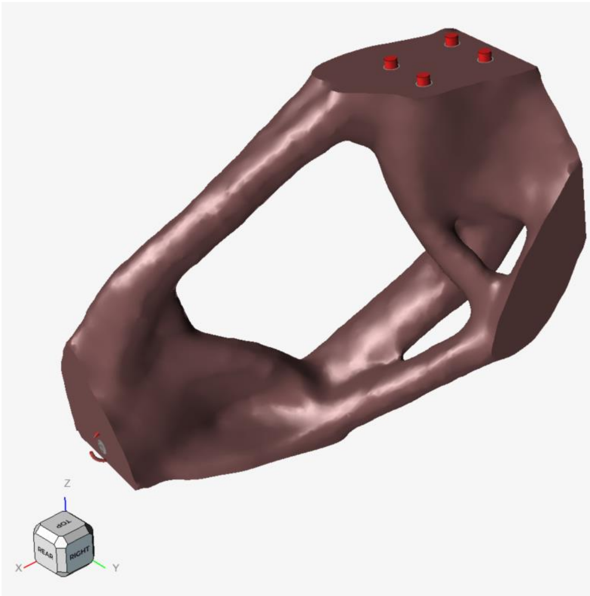


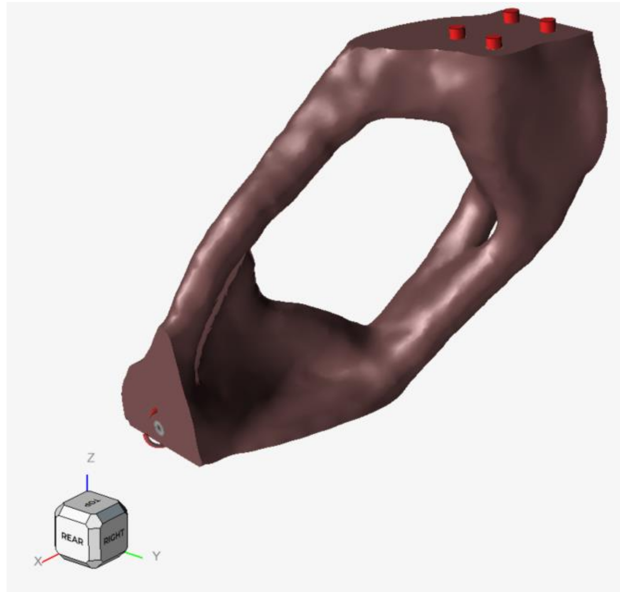
The result from the first iteration:

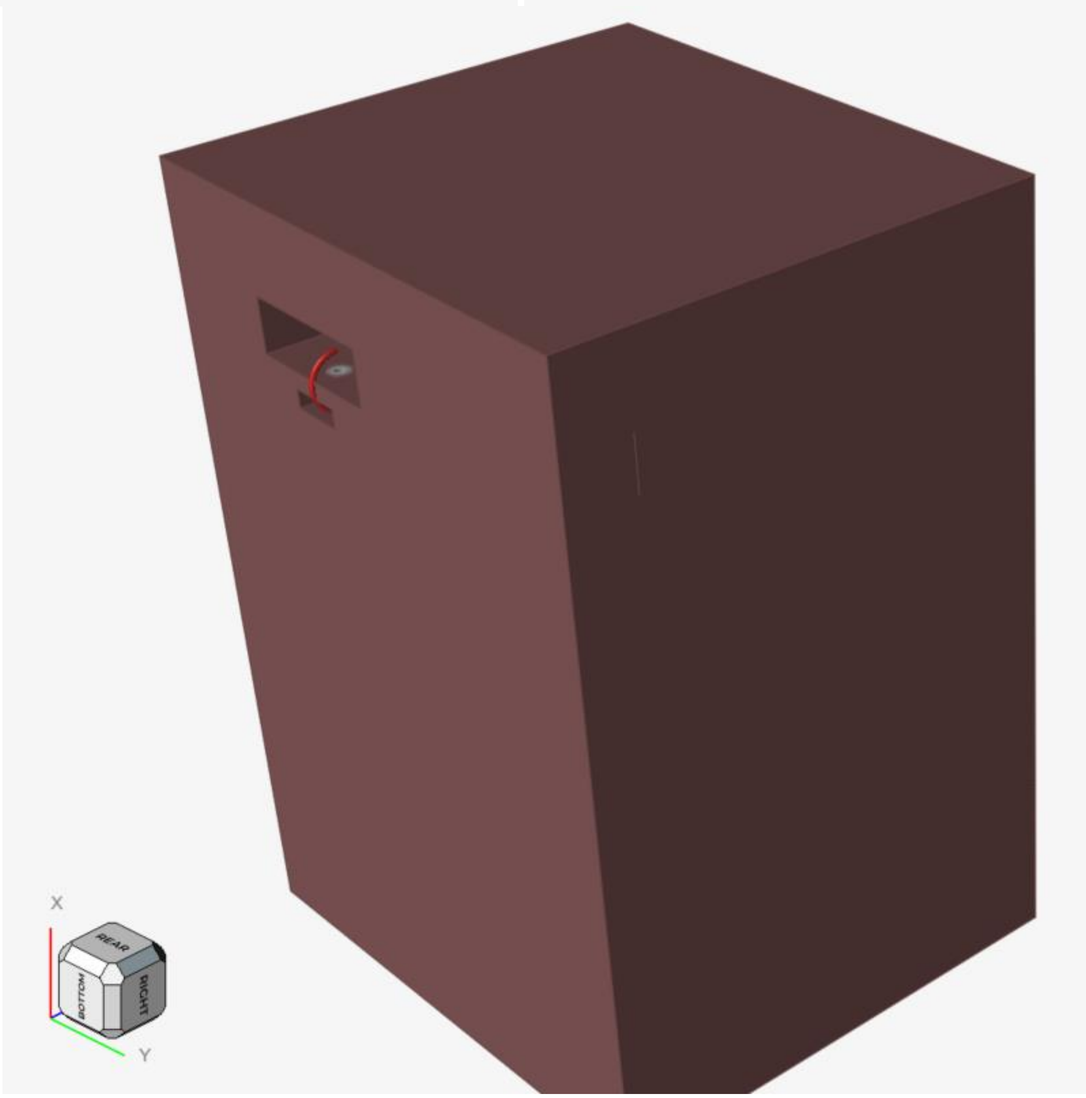


As can be seen, the material is stuck on the outer edges of the boundary and is trying to flow out on all four sides. A key observation was that the material amount is quite high and can be further reduced to save o the weight. This was run with a 15% volume fraction. The next iterations are run with 12 % & 10% subsequently.

The material is still constrained at the end near the wheel. Due to constraints, the wheel's location is fixed, but the material can still form an envelope and support it from both directions. Thus, the design space was extended to accommodate this. The figures on the next page include the result for the first two optimization runs and the updated design space.

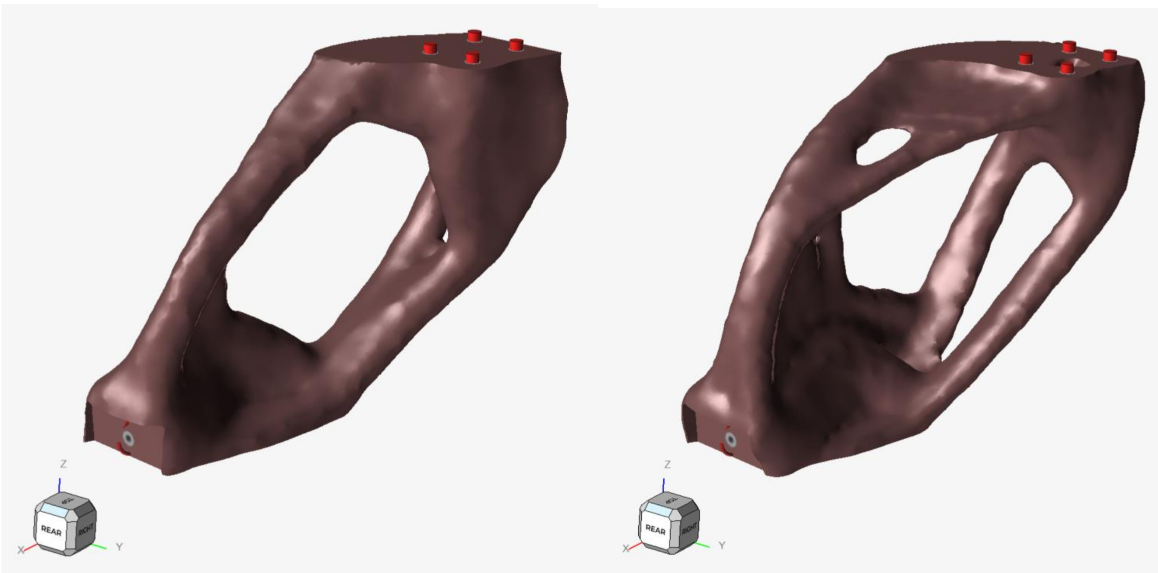



Z
REAR
RIGHT
X
Y


X
REAR
BOTTOM
RIGHT
Y

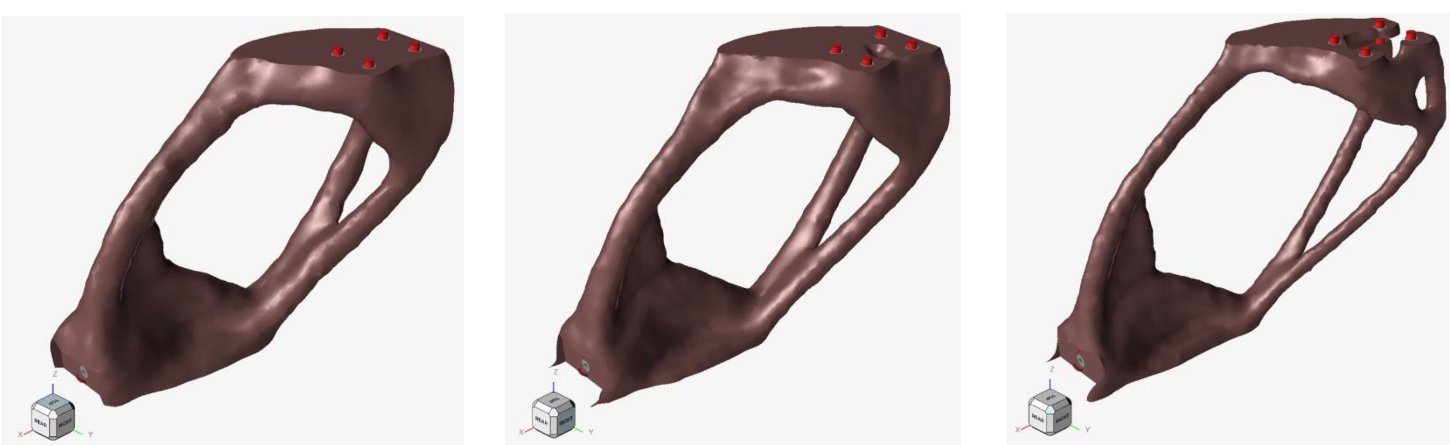
The updated design after the improved design space looks as below. Following this, I tried two different things to improve the design:

1. Reducing the volume fraction further to check how the results evolve – 8%
2. Reducing the minimum thickness to allow smaller features into the outcome – 8% with 12 mm



The left side is the one with a higher minimum thickness. The result on the right might be better structurally. Still, the critical issue is that it covers a lot of surface area, which is both bad aerodynamically and in terms of manufacturing. Thus, reducing the minimum thickness was not considered optimal, and hence 24 mm minimum thickness was considered appropriate for further runs.

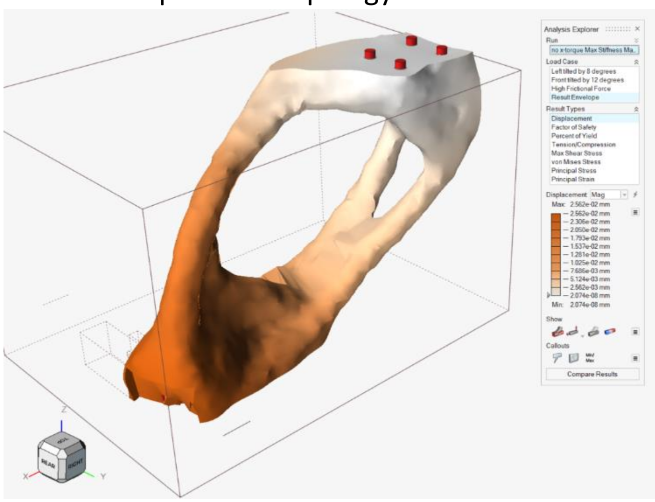
Though the optimization runs already look good enough, with no extra material forced at the boundary, and a decent manufacturable structure, I wished to look through more options and further check the result of decreasing volume fraction. This is a part of Aeroplane, and even grams of decrease in weight makes a huge difference, and thus, I decided to further run the optimization on 7%, 6%, and 4%.

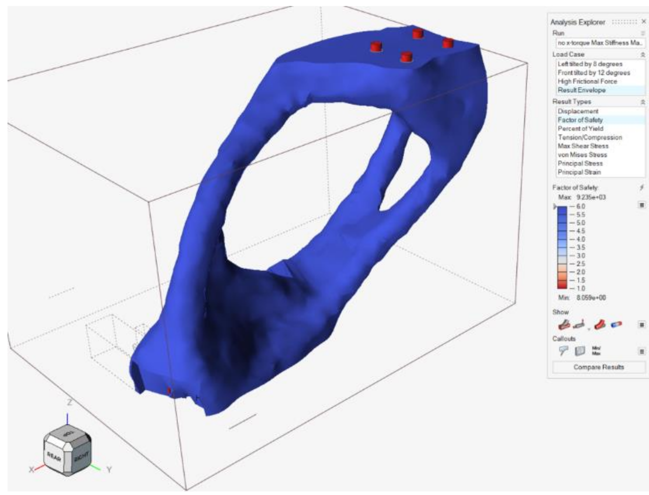


As can be seen, the optimization run in 7% improves the solution, but in 6% and 4%, the material is not enough to fill up around the support structures. The solution deteriorates after 7%, and thus, the optimization run with a volume fraction of 7% and a minimum thickness of 24mm is considered the final optimized structure. This structure is now passed on for FEA Analysis to the in-built FEA solver. This analysis and final optimization result have been discussed in the next section.

Finite Element Analysis on the Optimized Topology

Given below are the displacements and factor of safety acquired from the FEA applied directly on the optimized topology achieved.





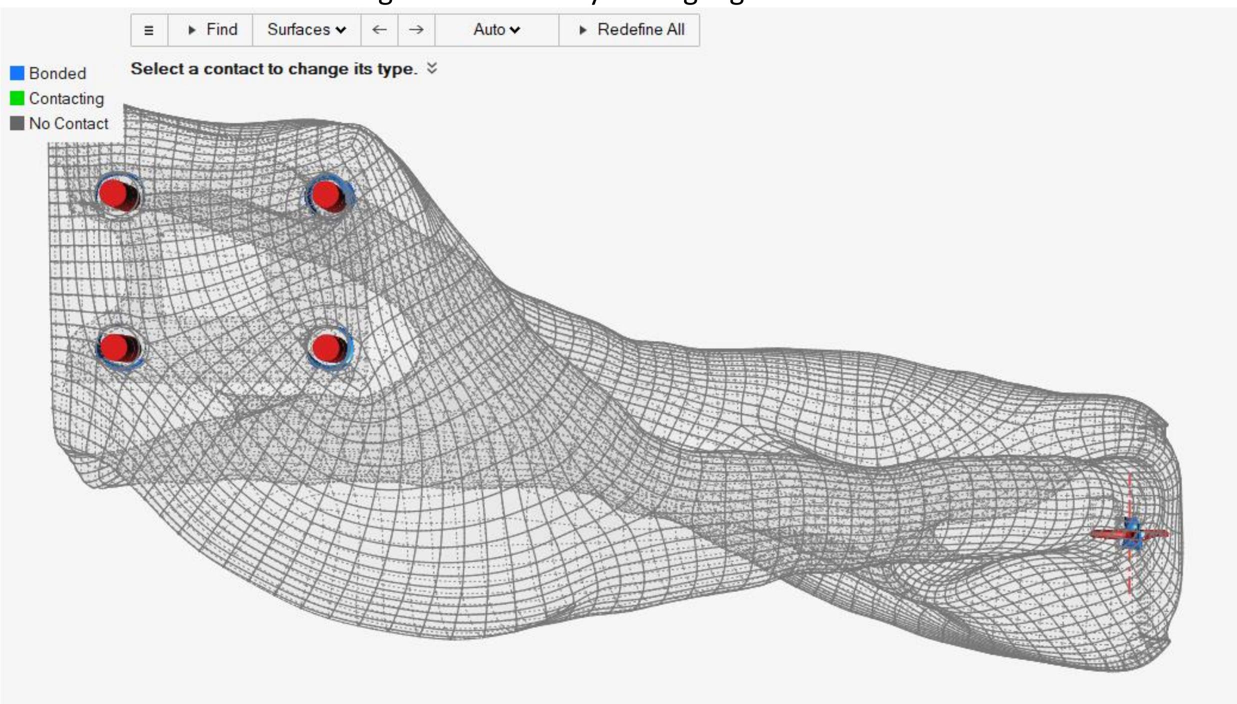
As can be seen in the figure above, the displacements are in the range of microns, and thus the landing gear is very stiff. The factor of safety is also good at all places. The minimum factor of safety is 8, which looks very high at first glance. The forces applied here are primarily static, and hence, they don't exactly replicate the actual conditions. The force on the landing gear will be more of a dynamic load, with an impact, rather than a force. This can lead to forces 2 or 3 times higher in general.

Furthermore, these are the raw results for the optimization. This will be post-processed, converted to CAD using polynurbs, and then sent for manufacturing. The conversion to polynurbs will also play a role in deciding the final FOS, and the current FOS of 8 will be reduced. It will be shown down in the post-processing section that the part will still be good enough to satisfy the original constraints. Considering that the minimum FOS reduces by around 30% in the post-processed CAD, the final FOS should be around 4 which is not very high, given the impact loading conditions. Thus, I finalized this part and moved on to making the CAD.

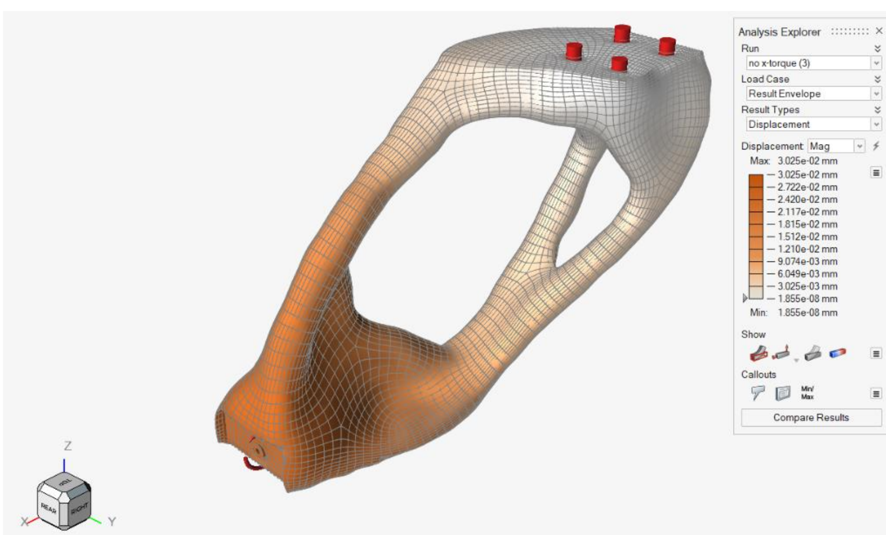
Post-processing

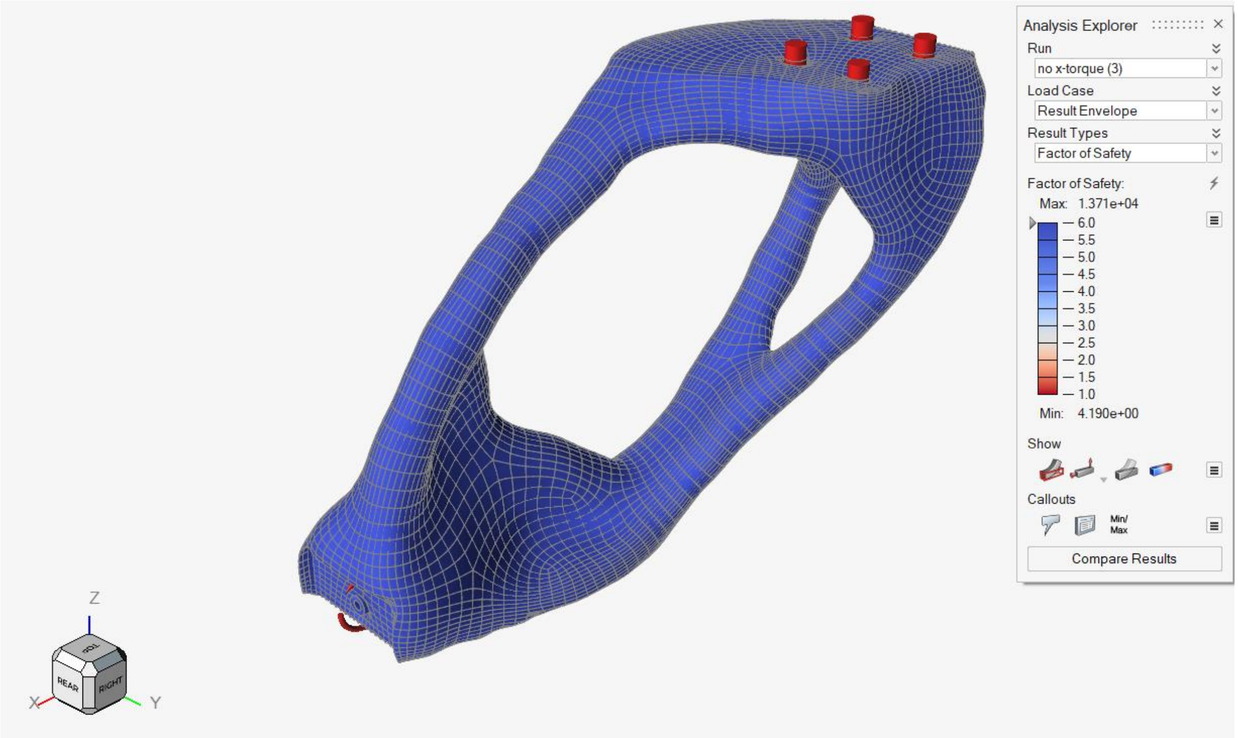
The first part of the post-processing was to create a combination of polynurbs that can appropriately represent the current optimization result. I utilized the polyfit functionality of Inspire to complete this step. I ran several trials by fidgeting with the variables, Number of polynurbs Faces, and curvature. Increasing the Number of Polynurbs faces captures the minor details from the real solution. Still, it becomes very computationally expensive and, at the same time, has a very wavy shape, which is undesirable in terms of manufacturing. The curvature was left to the default value as that provided appropriate solutions, and not much change was observed after variating it.

Once the CAD was generated using the polynurbs, another critical post-processing activity was to identify the contact points, if any, present them accurately and if not, make them. The forces will not be transferred appropriately without appropriate contact between the partition and the polynurb generated CAD. This is essential for performing an accurate FEA on the CAD. This operation was performed by selecting the polynurb surfaces on the corner and manually adjusting the corner points to generate contact between the partition circles and the CAD. The contacts can be seen in the figure below. They are highlighted in blue.



Once this process was done, an analysis was done for the CAD, and the results are as follows





As can be seen in the displacement figure, it still has a maximum displacement of 0.024 mm, i.e., 24 microns. Thus, the initial requirement of a highly stiff part is satisfied. Furthermore, two important observations can be inferred from the figure on FOS. The FOS, as predicted earlier, has dropped from 8 to 4. The second is the strength of the landing gear. Our aim was to develop a strong landing gear that could withstand many landing forces. The results above show the landing gear attains the standards set earlier in the report.

Altair is the post-processing industry leader, and PolyNurbs is possibly the best shot at creating an almost exact CAD replica from the optimized part. Improving the results would invariably require a better implementation of the PolyNurbs package. Though not efficient, as discussed in the class, an alternate way is to import the geometry as a reference and make a cad manually.

Conclusion

The main goal of my project is to determine a structure for the landing gear that is exceptionally stiff, very strong, and weighs less than conventional landing gear. The process of topology optimization was iterative. The overall process was repeated a couple of times to reach the optimized structure. Furthermore, some subprocess, like choosing an appropriate design space, was also an iterative process. The first step was to select an appropriate design space, followed by applying appropriate constraints and loads at the correct locations. Many runs were made, with each progressive run taking me towards my final solution and giving me an idea of where to move forward and how to tune different parameters like minimum member thickness. Last but not least, I used the Polynurbs functionality of Altair Inspire to make the actual CAD and ran an FEA on it to confirm that the Landing Gear satisfies the constraints.

My major technical learning from the project was setting up the topology optimization problem, iteratively progressing towards the solution, and finally validating the result. These steps required logical reasoning, critical analysis, and base-level engineering background. This project also helped me understand how to logically break down the entire design process and evolve design through iterative improvements.

The Drawing for the design space can be seen below.