



ULA Rocket Hardware Design

Advanced Computational Methods Assignment

Contents

1.0	Introduction.....	2
2.0	Research.....	2
2.1	Material Properties	2
2.2	Applied Load	3
3.0	Topology Optimisation Study	4
3.1	Mesh Setup.....	4
3.2	Material Setup	4
3.3	Grid Component Setup.....	5
3.4	Static Load Setup.....	5
3.5	Loadcase Setup	6
3.6	Topology Objectives	6
3.7	Group Properties.....	7
3.8	Topology Regions	8
3.9	Topology Constraints	8
3.10	Results	9
4.0	Critical Appraisal.....	12
5.0	Conclusion	14
	Bibliography	14

1.0 Introduction

This objective of this project was to produce an optimised version of a work platform bracket to be used on the Atlas V rocket during pre-launch assembly as outlined in the GrabCAD 'ULA Rocket Hardware Challenge' (GrabCAD, 2016). This would involve modelling the solid CAD file from given technical drawings, researching the given material and loadings and performing a topology optimisation using the Design Studio software. These steps are documented within this report, with the final output of the project being an optimised model that could be used to generate the CAD model required to create a manufactured part. In this regard the project was successful, as the generated optimised part was very similar to that of the winning entry for the competition that was chosen by NASA to be used on the rocket (Lo, 2016). The key constraints of the project were as follows:

- Maximum final mass – 0.0453Kg
- Part loading during use – 2668.93N (600lbf) in the negative Z direction
- Minimum wall thickness – 1.016mm
- Minimum wall thickness around mounting holes – 2 x diameter of holes

These constraints would be used to set the study constraints and objectives for the topology optimisation run within Design Studio. An additional assumption that was made was that fasteners to attach the part to the rocket had already been chosen, and so the overall length of the mounting holes would not be allowed to change – this would affect the setup and results of the topology optimisation study, as it would mean no material could be removed from these areas to meet the final mass constraint.

2.0 Research

2.1 Material Properties

The first step of the research was to learn more of the material properties of the Ultem 9085, as those given in the design brief were not sufficient to create a working stress analysis model within Design Studio, and those that had been given were not in SI units so would have to be converted. The material was found to be a resin used in Fused Deposition Modelling (FDM) additive manufacturing, in which individual strands of fibre are laid down in layers to generate the final part. By using the material datasheet (Stratsys, 2020) and the design brief, the following material properties could be found:

- Ultimate Tensile Stress (UTS) – 69.0MPa
- Maximum Strength Allowable – 31.0MPa
- Specific Gravity (relative density) – 1.34g/cm³
- Young's Modulus (modulus of elasticity) – 2240.8MPa
- Shear Modulus – 2050MPa
- Poisson's Ratio – 0.41

These values would then be able to be imported as the material properties of the CAD model, allowing stress analysis to be performed.

2.2 Applied Load

As the part in the study has a single purpose – to attach work platforms that can be used by engineers during ground processing of the rocket stage (GrabCAD, 2016) – the applied load would be relatively simple. The bracket must be able to be bolted to the rocket stage using the designated mounting holes, so these would act as the fixed geometry in the stress analysis, while the load would be the mass of the loaded platform. An assumption was made that the load force would therefore equal the mass of the engineer (assumed at 85Kg as the mass of an average North American male (Medical News Today, 2018)), the mass of required tooling and safety equipment (assumed at 20Kg consisting of a laptop, Personal Protective Equipment and a tool kit) and applying a factor of safety (2.5). This came to a total force of 2575N, which was approximately equal to the 2668.93N stipulated by the design brief. This similarity implies that the assumptions were correct and that the load was reasonable.

Under some circumstances the force of gravity would be applied to the simulation and resulting optimisation, but this is generally a concern for much larger, heavier parts. Given that the studied part had an initial mass of only 0.436Kg and a target mass of 0.0453Kg, gravity could be discounted in the study to improve computation time.

The face for the load to be applied to (shown in Figure 4) was designated in the design brief and technical drawings. The total load force would be applied as a uniformly distributed load across this face to simulate the work platform attachment.

3.0 Topology Optimisation Study

3.1 Mesh Setup

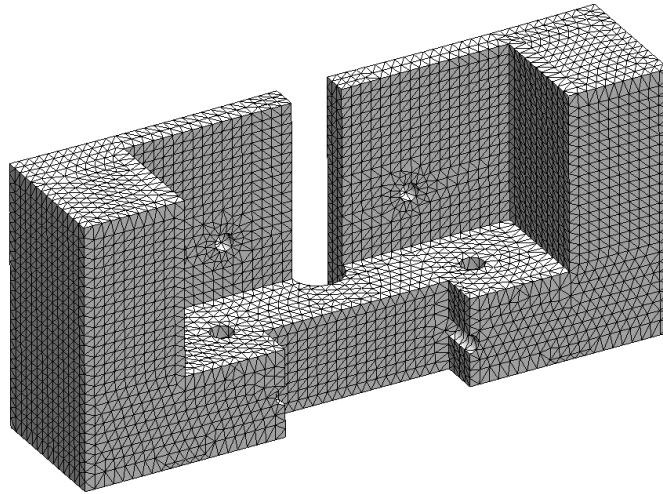


Figure 1 - CAD Model Mesh

The first step of the optimisation was to model the design envelope in the Solidworks CAD package and apply a mesh to the body, as this would allow the part to be imported to the Design Studio software. A 3mm, first-order element size was chosen as this would provide a reasonable balance between accuracy and computational run time – this was based on the general rule that element size should be approximately one third of the smallest wall thickness on a given part. On this part one third of the minimum thickness was calculated to be $\sim 2.3\text{mm}$ but given that multiple attempts would likely be taken to achieve satisfactory results with the optimisation software this value was rounded up to 3mm. This would decrease the computational run time whilst maintaining a reasonable level of accuracy. This mesh could then be exported from Solidworks as a NASTRAN deck and imported in to Design Studio.

3.2 Material Setup

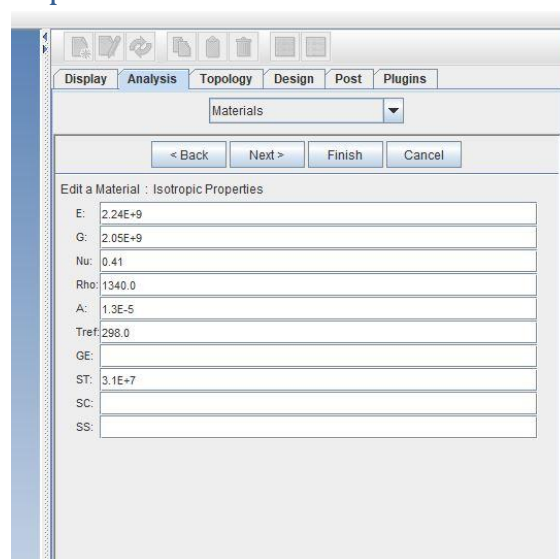


Figure 2 - Ultem 9085 Material Properties

Using a combination of the given values and those found from the Ultem 9085 datasheet (Stratsys, 2020), the material properties were entered and applied to the imported mesh model. It was vital that these values be entered correctly as they would greatly affect how the model would behave under load and thus how much material could be removed to optimise the part – the values used were those found in Section 2.1.

3.3 Grid Component Setup

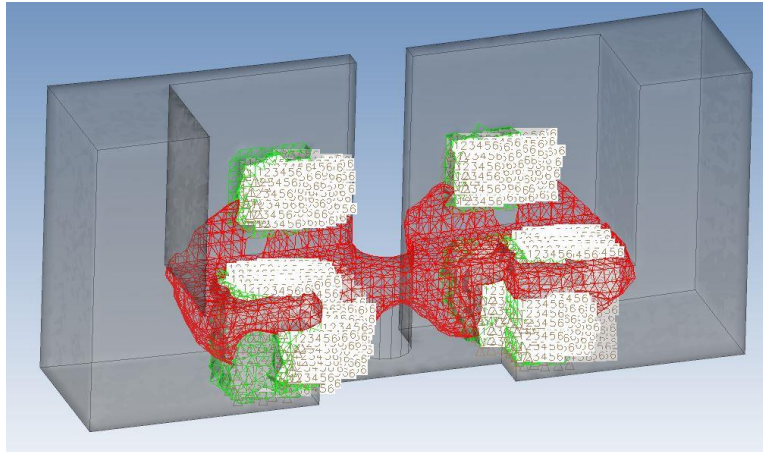


Figure 3 - Grid Component Setup

Next came the defining of the grid point components. This step defines the fixed geometry that the load will be supported by, and as stated in the brief an area approximately twice the size of the hole diameters was chosen around each mounting hole, around the entire depth of each hole. It was assumed at this point that fasteners had already been chosen so the entire depth of each hole was retained and would not be considered for material optimisation at this time.

3.4 Static Load Setup

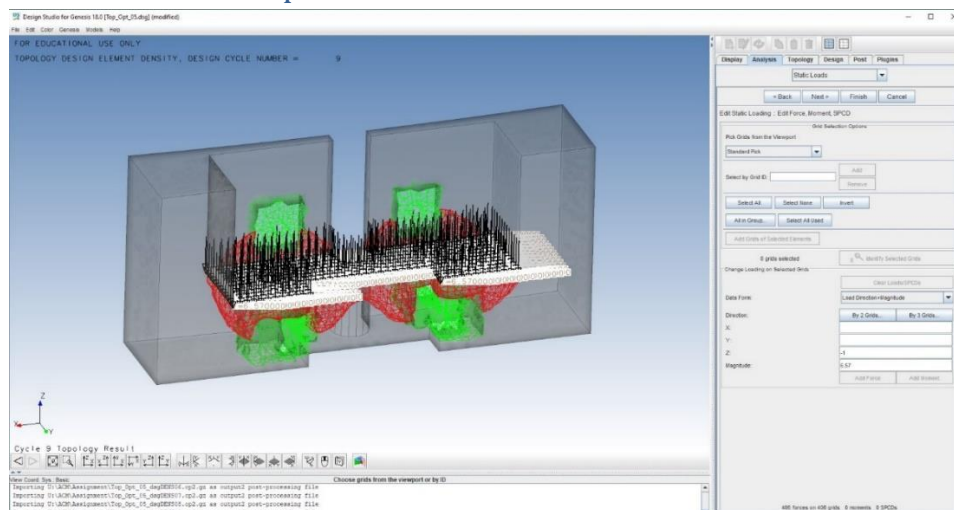


Figure 4 - Static Load

The static load could then be applied to the necessary surface. The load value had been given at 2668.93N (600lbf), which had to be applied as a Uniformly Distributed Load (UDL) across the platform mounting face in the negative Z direction. As this load was a UDL, the entered value had to

be given as the total UDL value divided by the number of grids selected on the face – in this case 406 selected grids, each bearing a load of 6.57N.

3.5 Loadcase Setup

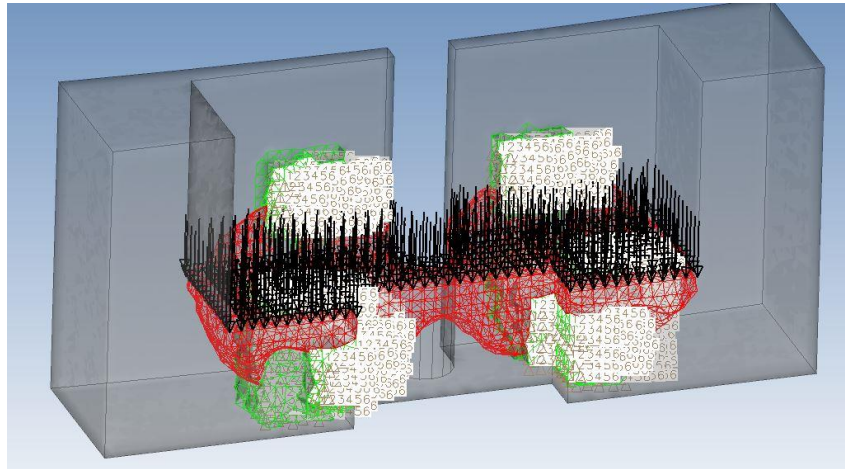


Figure 5 - Loadcase

The static load and grid component setup were then combined to create the loadcase. This is what the stress analysis and topology optimisation tools would refer to when generating results, as it stipulates the loads and fixtures of the part under a given condition. This is a simple but useful tool, as it means that multiple scenarios can be generated for a part, and both the stress analysis and topology optimisation can be run considering any combination of these scenarios desired. In the case of this part, only one significant load was to be considered, so it was simply a case of combining the Z direction load and the fixed grids.

3.6 Topology Objectives

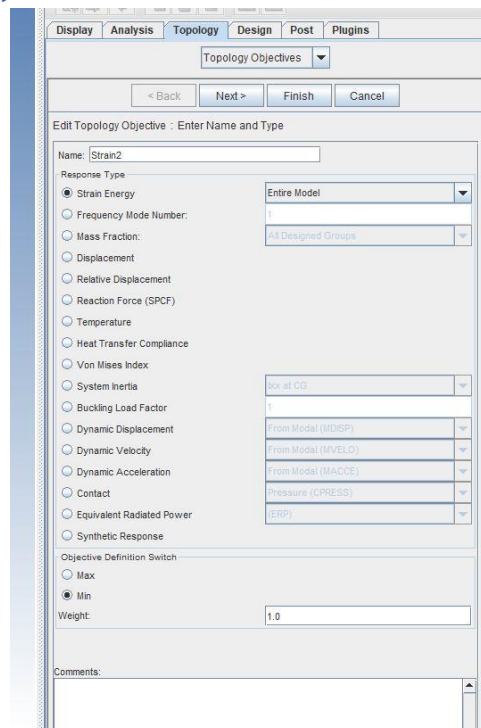


Figure 6 - Topology Objective

The topology settings could now be selected, beginning with the Topology Objectives; the only objective required for this part was to minimise the strain energy within the part whilst under load. This objective would then be targeted during the optimisation, considering the constraints and regions that would be used as the defined study conditions.

3.7 Group Properties

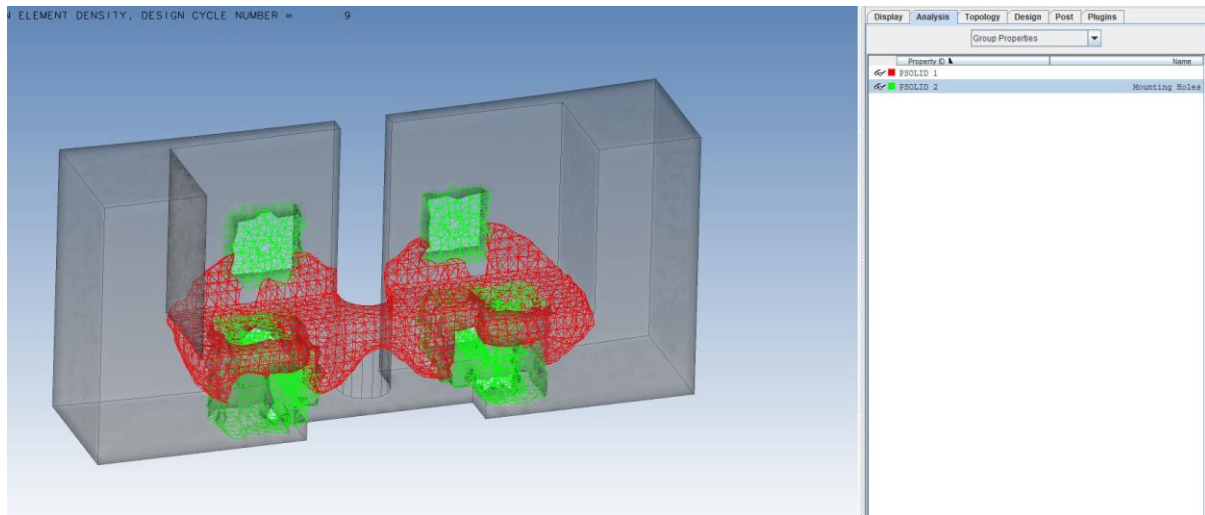


Figure 7 - Group Properties

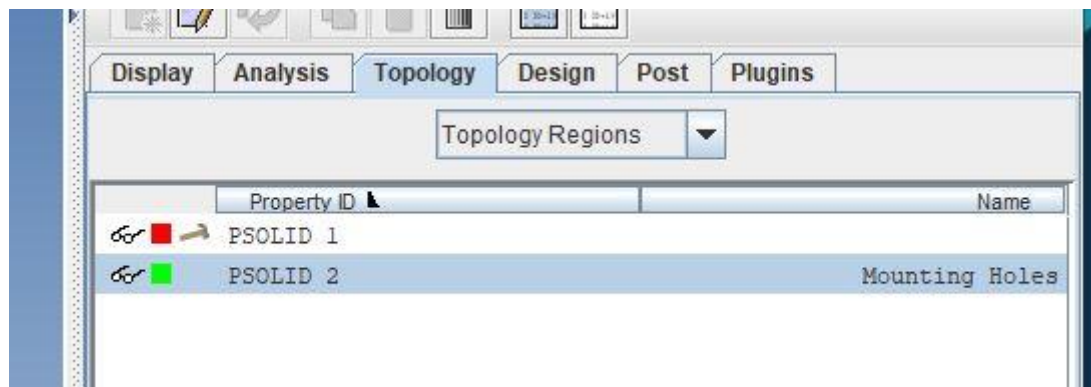


Figure 8 - Topology Regions

The group properties tool was used to define the regions that the topology optimisation would be allowed to consider when removing material from the initial part. This was key as the brief had stipulated a minimum area around the mounting holes, so by moving the selected elements (those shown in green in Figure 7) to a separate group this would prevent this material from being lost.

3.8 Topology Regions

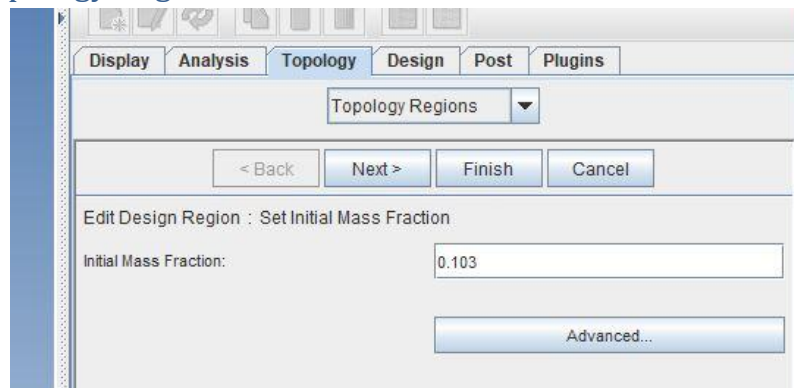


Figure 9 - Initial Mass Fraction

Within the topology regions the initial mass fraction could be defined. This is the total fraction of starting mass that would be the maximum allowed in the final optimised part. The initial mass was calculated to be 0.436Kg from the given material density and part volume, and the target mass was given as 0.0453Kg by the brief; this meant the initial mass fraction was to be set at 0.103. As stated in section 3.7, this region was defined only by the group PSOLID 1, so as to prevent the removal of required material around the mounting holes.

3.9 Topology Constraints

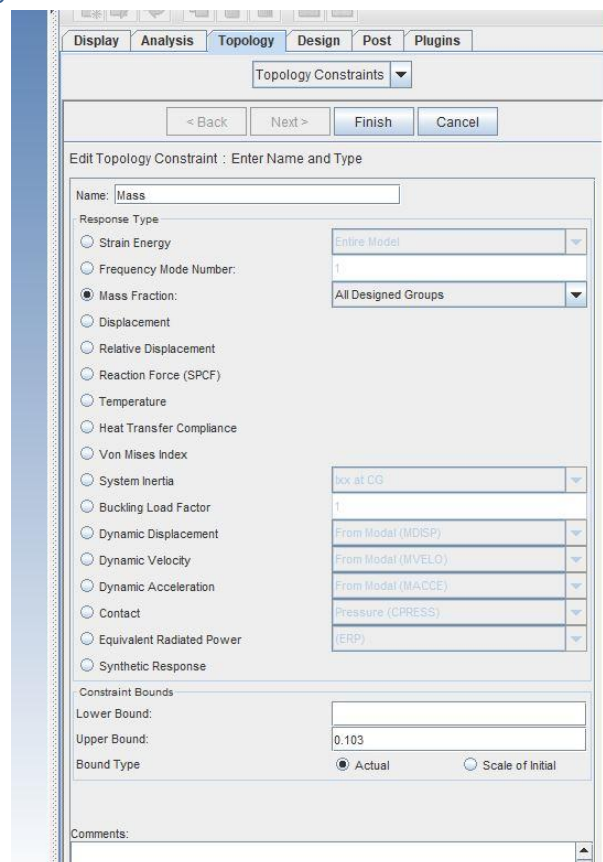


Figure 10 - Topology Constraints

The initial mass fraction would also be entered as a maximum mass within the topology constraints. Again, this is used by the topology optimisation tool to define the maximum amount of material it was permitted to use for the final part.

Once all these objectives and constraints had been entered, the stress analysis simulation could be run and the results imported to the model, allowing the topology optimisation study to be run.

3.10 Results

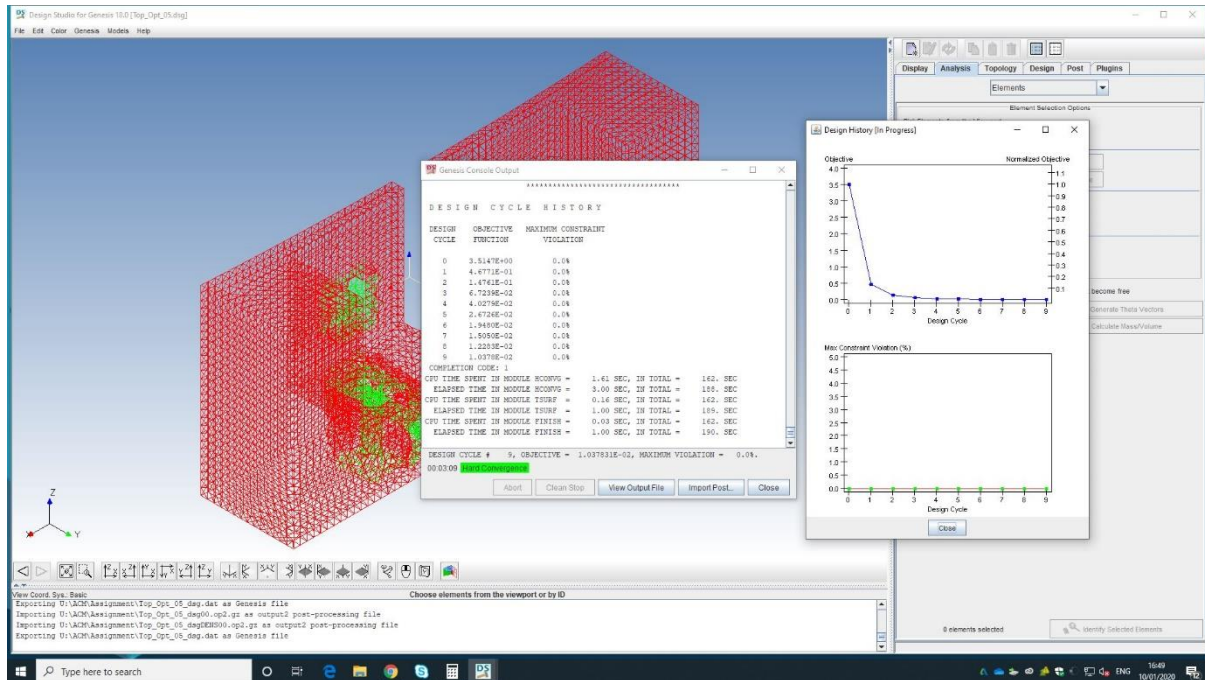


Figure 11 - Design Cycles

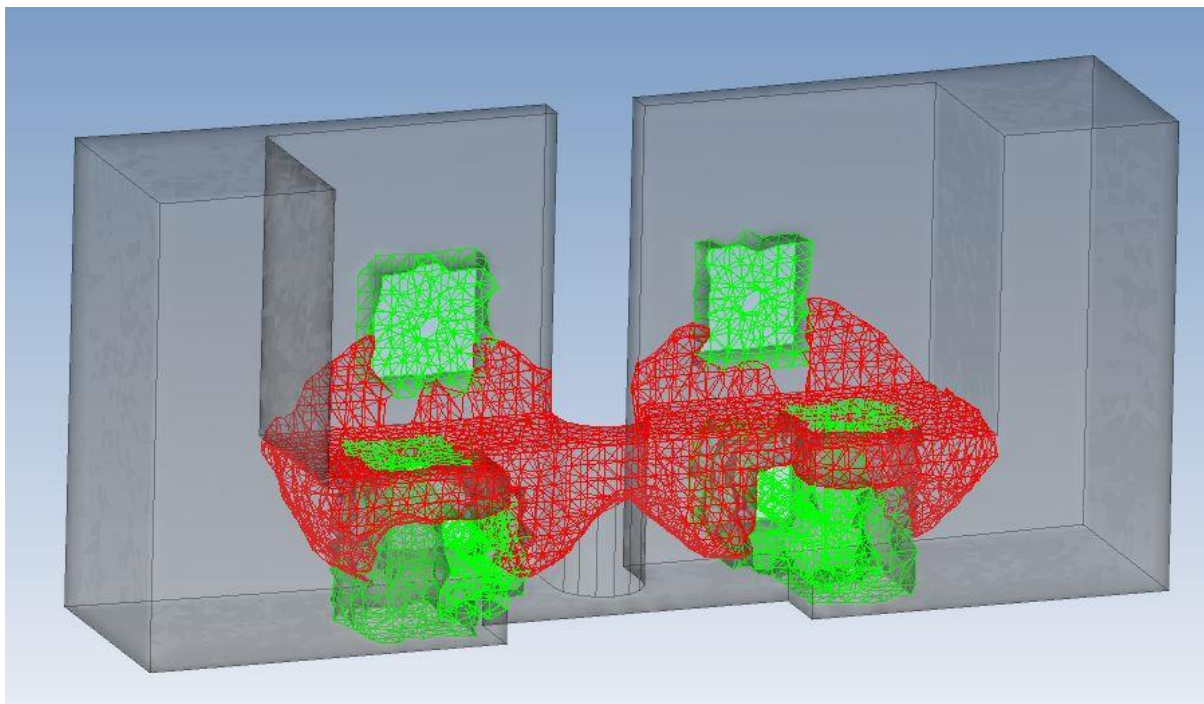


Figure 12 - Optimisation Results 1

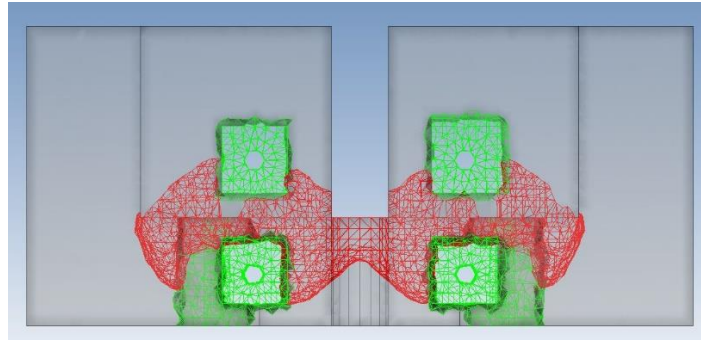


Figure 13 - Optimisation Results 2

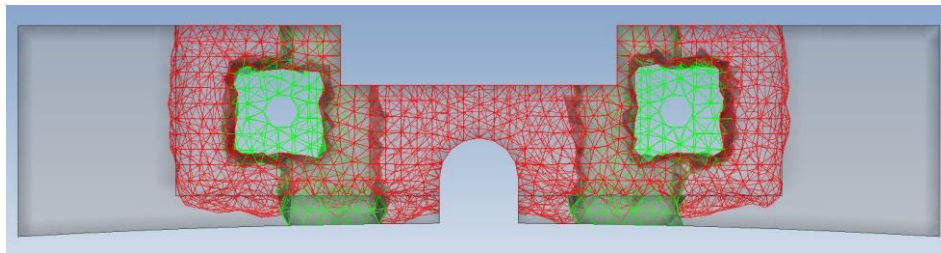


Figure 14 - Optimisation Results 3

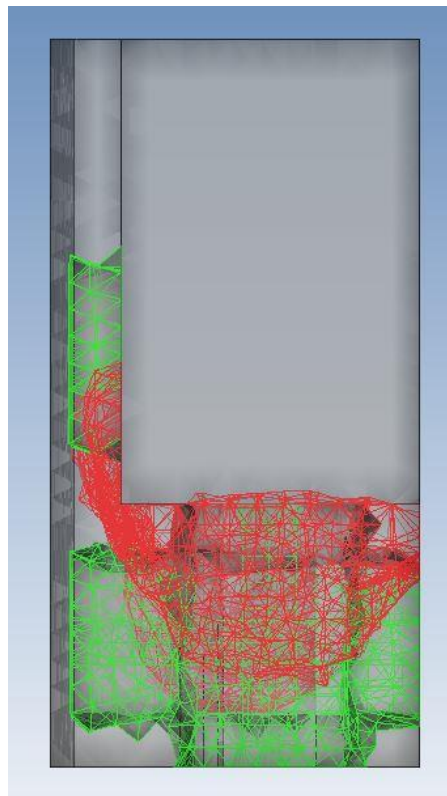


Figure 15 - Optimisation Results 4

Figures 11-15 show the final remaining material after the topology optimisation study after 9 design cycles resulted in a hard convergence. Note the green areas defined within region PSOLID 2 have remained unchanged, while the amount of material from the original design envelope (shown in

grey) has reduced drastically to meet the topology requirements by removing the material not being used to support the static load within the stress analysis.

This optimisation works by analysing the load paths through the stressed part, removing those parts that least stressed and reassessing the model to check that the resultant part remains stable (Abbey, 2018). This process is repeated continuously in design cycles (see figure 11, design cycles shown on x axis), removing more material each time until the optimisation tool reaches a convergence point where each new design iteration is not providing a significant enough improvement to warrant proceeding with the study (see figure 11, line gradient decreases with each cycle).

However, simply removing the elements deemed unnecessary is complicated and computationally expensive. In order to achieve a similar result, elements thought to be unnecessary have their stiffness and density set to near-zero by the optimisation tool and the stress analysis is run again. The optimiser “determines how to distribute material by removing the “soft” elements that do not contribute to the stiffness of the component for the particular load scenario, boundary conditions, and manufacturing controls. For each element, the optimization algorithm couples the material’s Young’s modulus with a relative mass density factor ranging from 0.0001 (for a void element without any load-carrying capacity) to 1.0 (for a solid element with load-carrying capacity). Elements with low relative mass densities (< 0.3) are considered “soft” elements. These elements do not contribute to the overall stiffness of the component, and they can be safely removed. Elements with high relative mass densities (> 0.7) are considered “solid”. These elements contribute the most to the overall stiffness (as a measure of the load-carrying capacity) of the component, and they should remain intact in the final design. The “solid” elements distribute the applied loads more effectively than the “soft” elements” (D'Assault Systemes, 2018). This has a similar effect on the stress analysis results as actually removing the elements at an improved time cost, so the optimiser only removes elements after the final design cycle has been reached (Abbey, 2018).

4.0 Critical Appraisal

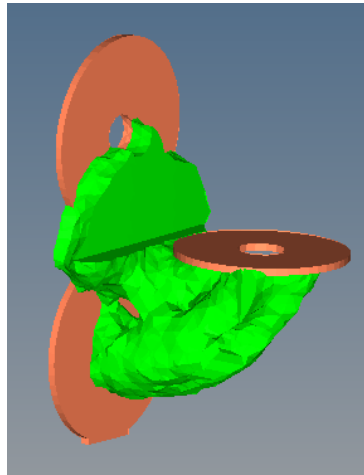


Figure 16 - Topology Optimisation from Winning Contestant



Figure 17 - Final CAD Model from Winning Contestant

Overall, the optimisation study was considered a success. The target mass was achieved whilst maintaining the required geometry, and the load was capable of being supported. As can be seen in Figures 16 & 17 of the winning contestant's entry (Lo, 2016), the same features of the majority of material being centred around the mounting holes with a central connecting beam are present on both studies. One noticeable difference is that the winning contestant more accurately defined the radius around the mounting holes that would not be considered by the optimiser, while the study in this report used a rectangular estimation of these regions – by defining these areas more accurately, the study could have potentially removed more unnecessary material.

Another aspect that could potentially improve the results would be allowing the optimiser to remove unnecessary material from the length of the mounting bolt holes to further reduce the mass, but this would affect the required fasteners associated with the part so would have to be approved by other project personnel. Additionally, this study used a draft-quality first-order mesh setup with an element size chosen for speed of computation as a major consideration; for greater accuracy and mass efficiency, this study could be run again with a finer mesh size (1-2mm) and second-order elements. While this would take considerably longer to run as a first study, by using the draft-quality topology study as a baseline for a new CAD model and re-meshing this with a finer

element size, this time could be computation time could be reduced as the optimiser would not have to consider the material already considered unnecessary and a more refined final study could be created. This would allow for more accurate study of the load paths and as a result a more mass-efficient topology optimisation.

Topology optimisation has a reputation for producing parts that would be incredibly difficult or expensive to manufacture via traditional techniques (Abbey, 2018). This can be countered by applying manufacturing constraints to the study, such as symmetry along planes or uniform geometry in a certain direction as these will be considered by the optimiser when removing material from a build envelope. For this study however, the material was given as a resin used within modern additive manufacturing so no such constraints were required and the output model could be used directly if desired. The Design Studio software could export the final design as a .STL file which can be sent directly to a 3D printer for manufacture, and a final part with the exact geometry of the topology optimisation study could be created.

The optimisation software has no capability for considering aesthetics, however. The raw optimisation file is still based on the element mesh that was initially imported, so the model geometry is still inherently tetrahedral. This geometry is why the optimisation model – the optimised shape - should be used to create a second final “optimal shape”, to improve the final parts’ aesthetics and handling characteristics (D'Assault Systemes, 2018). This final CAD model can also ensure that the exact specifications of the design brief are met, such as the minimum wall thickness surrounding the mounting hole. While this thickness was considered for the optimisation study, because the area to designate was based on the tetrahedral mesh the area had to be selected to the nearest element, rather than a specified numeric value.

Another reason for creating this second-generation CAD model is to perform the original stress analysis on the new part – while the optimisation software is accurate, it is still subject to user error and so it would be good practice to perform a fresh secondary stress analysis simulation before going to production with the new geometry. This would hopefully catch any input errors or any undesired geometry changes made by the optimiser. By creating this second CAD model the second stress analysis would also likely be performed in a different software package to the topology optimisation, so the two could be compared directly to verify the results.

5.0 Conclusion

Overall the results of the topology optimisation study were satisfactory. The material properties, loadings, topology objectives and constraints combined to create a viable optimised part that used significantly less material than the original. As the performance efficiency of many systems – such as rockets - is directly correlated to mass (GrabCAD, 2016), this project demonstrates the usefulness of optimisation tools within engineering to create parts that can deal with high stresses whilst utilising minimal material. In the case of the platform mounting considered here, this part is used at multiple locations throughout the rocket assembly, so reducing the part mass has a considerable effect on the overall launched mass. Especially when combined with modern additive manufacturing methods, optimisers can be used to create much more mass-efficient parts that can reduce both material mass and monetary costs. This is a great benefit given the modern environmental concerns as it can help companies to produce systems that are less costly both in terms of harmful emissions and material wastage.

Bibliography

- Abbey, T. (2018, February 4). *What Is Topology Optimization and Why Is It Useful?* Retrieved from PTC.com: <https://www.ptc.com/en/cad-software-blog/what-is-topology-optimization>
- D'Assault Systemes. (2018). *Design Through Analysis: Improving Product Design and Automating Manufacturability with Topology Optimization*. Waltham: D'Assault Systemes.
- GrabCAD. (2016, August 11). *3-2-1 Liftoff! ULA Rocket Hardware Challenge*. Retrieved from GrabCAD.com: <https://grabcad.com/challenges/3-2-1-liftoff-ula-rocket-hardware-challenge>
- Hale, S. (2017, May 22). *What is Topology Optimization and Why Use It?* Retrieved from CAEAI.com: <https://caeai.com/blog/what-topology-optimization-and-why-use-it>
- Lo, K. (2016). *ULA Challenge Topology Optimized Design*. Retrieved from GrabCAD.com: <https://grabcad.com/library/ula-challenge-topology-optimized-design-1>
- Medical News Today. (2018, February 14). *Is there an average weight for men?* Retrieved from medicalnewstoday.com: <https://www.medicalnewstoday.com/articles/320917.php>
- Stratsys. (2020). *Ultem 9085 Resin*. Retrieved from Stratsys.com: <https://www.stratsys.com/materials/search/ultem9085>

Figure 1 - CAD Model Mesh	4
Figure 2 - Ultem 9085 Material Properties	4
Figure 3 - Grid Component Setup	5
Figure 4 - Static Load	5
Figure 5 - Loadcase	6
Figure 6 - Topology Objective	6
Figure 7 - Group Properties	7

Figure 8 - Topology Regions.....	7
Figure 9 - Initial Mass Fraction	8
Figure 10 - Topology Constraints	8
Figure 11 - Design Cycles	9
Figure 12 - Optimisation Results 1	9
Figure 13 - Optimisation Results 2	10
Figure 14 - Optimisation Results 3	10
Figure 15 - Optimisation Results 4	10
Figure 16 - Topology Optimisation from Winning Contestant.....	12
Figure 17 - Final CAD Model from Winning Contestant	12