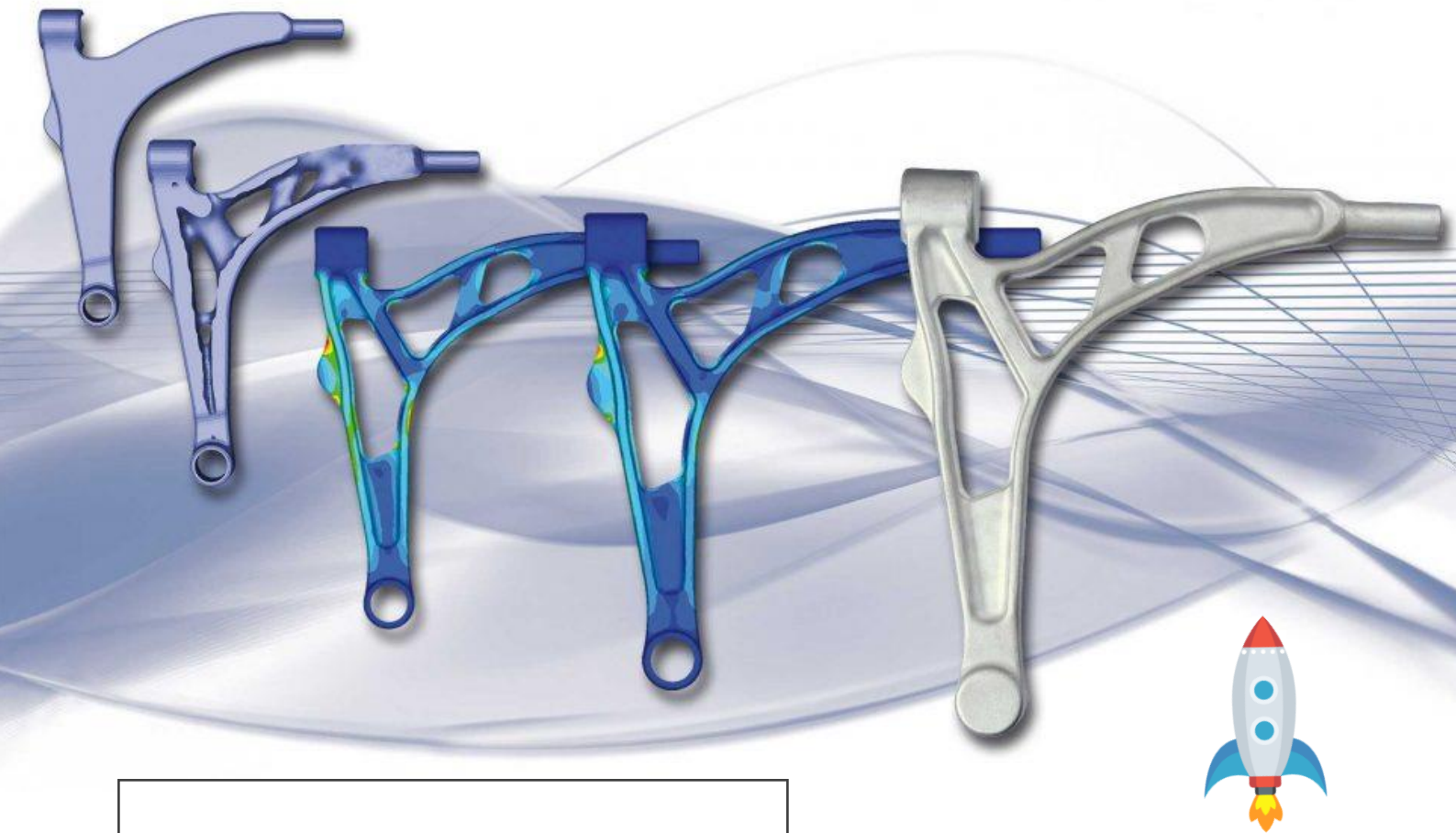




Topology: ULA Rocket Hardware Design

Advanced Computational
Methods (M6X00408)

Shaun Kilbane - 1705015

1. Index

Table of Contents

1. Index.....	2
2. Abstract.....	3
3. Introduction	4
4. Parameters.....	5
5. Analysis	9
6. Conclusion.....	18
7. References	19

2. Abstract

This report examines the use of topology optimization as a tool to aid with the design of a ULA rocket bracket. The report discusses the power of topology optimisation and its use to create a strong and stable structure. Although topology optimisation is used on a relatively basic level here, the report does display the power of the tool and draws parallels between this basic level and wider applications.

3. Introduction

Topology optimisation is a process that takes a basic model, ideally an envelope model from which the product will sit and determine the primary load path of the forces that run within it [1]. The task is to take the envelope model of a ULA bracket and run topology optimisation on it, the model may then be designed fully around the resultant model of the topology optimisation.

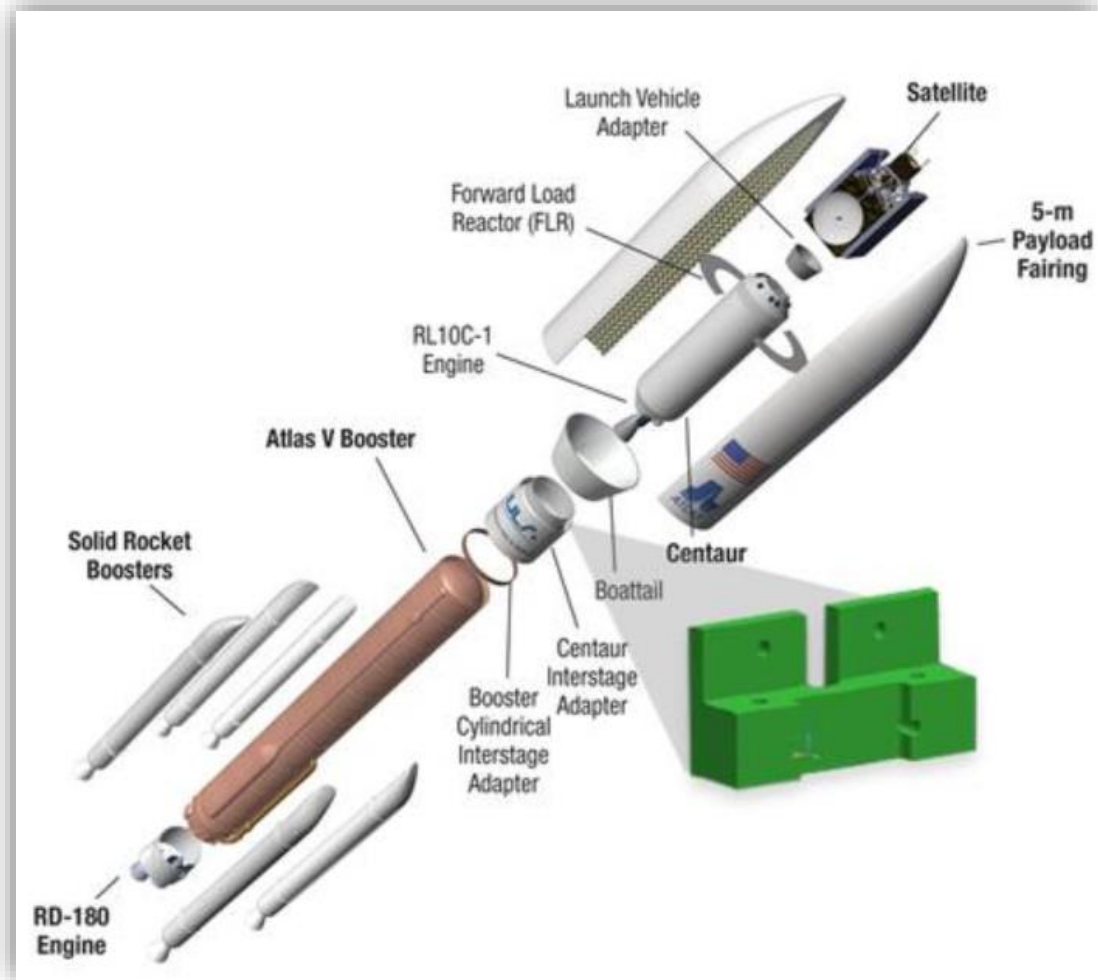


Figure 1 - ULA Rocket exploded view (GrabCAD, 2016)

The bracket attaches to the 'centaur interstage adapter' that supports the 'centaur' which forms part of the fuel system for the rocket, therefore it is important that the design is robust enough to carry the desired load. It will be fastened in place by four 10-32 bolts and two ¼" bolts, this can be determined by the diameter of the clearance holes specified on the drawing [2] [3]. It is important that the material has good flame-retardant properties due to the nature of its application, the specified material Ultem 9085 does meet this criteria [4].

4. Parameters

The task is to carry out a conceptual study of a loadbearing bracket with the use of topology optimisation, this is to be done using an envelope of which to work within [1] [3]. A drawing of the envelope has been provided along with a model, the model has been redesigned due to discrepancies between it and the actual drawing. The remodelling of the envelope was for the purpose of accuracy as to create a fully working product.

There are six given factors that must be adhered to during this exercise:

1. Modelling envelope – As provided in the form of a drawing.

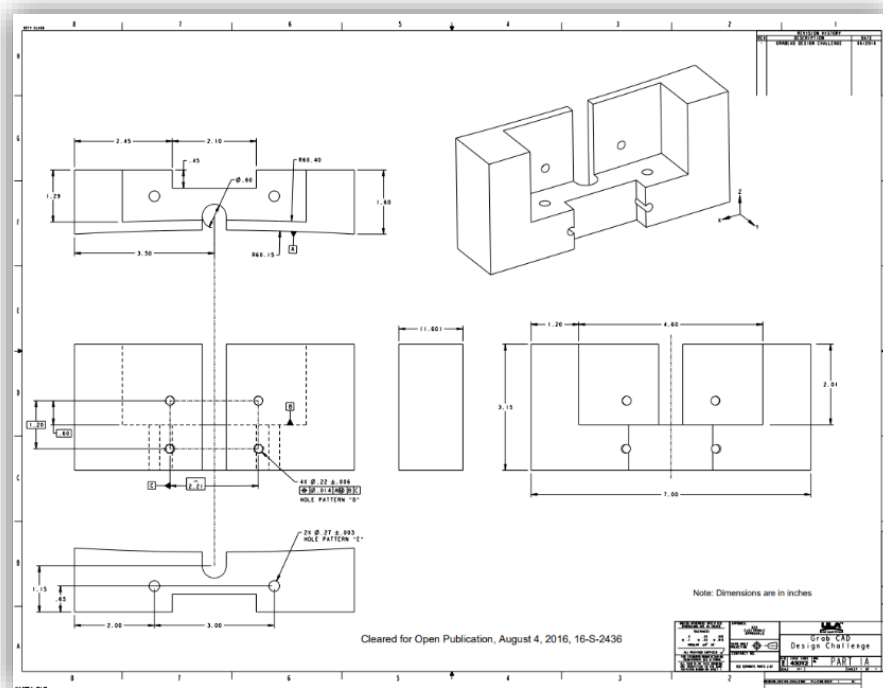


Figure 2 - Envelope drawing (GrabCad, 2016)

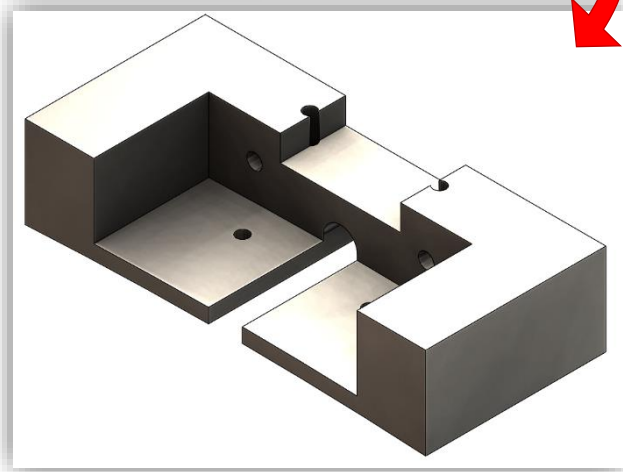


Figure 3 - Model created from drawing (Kilbane, 2019)

2. Material – ULTEM 9085.

- 2.1. Young's modulus 'E': 325KSI (2240.8MPa) $[325 \times 6.895 = 2240.8]$.
- 2.2. Poisson's ratio 'ν': 0.41.
- 2.3. Specific gravity (density 'σ'): 1.34 (1340kg/m³) $[1.34 \times 1000 = 1340]$.
- 2.4. Tensile strength: 4500psi (31MPa) $[4500 \times 145 = 31.02641]$.

 ULTEM 9085 PRODUCTION-GRADE THERMOPLASTIC FOR FORTUS 3D PRINTERS					
ULTEM™ 9085 resin is a flame-retardant high-performance thermoplastic for digital manufacturing and rapid prototyping. It is ideal for the transportation industry due to its high strength-to-weight ratio and its FST (flame, smoke and toxicity) rating. Combined with a Fortus® 3D Printer, ULTEM 9085 resin allows design and manufacturing engineers to produce fully functional parts that are ideal for advanced functional prototypes or end use without the cost or lead time of traditional tooling.					
Certified ULTEM 9085 meets more stringent test criteria and retains material traceability required by the aerospace industry. Certificates of Analysis for both raw material and filament are supplied, documenting test results and identification to match filament manufacturing lot number to raw material lot number. This allows traceability from printed part back to raw material. A Certificate of Conformance certifies that the material is manufactured per specification.					
MECHANICAL PROPERTIES¹	TEST METHOD	ENGLISH		METRIC	
		XZ Orientation	ZX Orientation	XZ Orientation	ZX Orientation
Tensile Strength, Yield (Type 1, 0.125", 0.2"/min)	ASTM D638	6,800 psi	4,800 psi	47 MPa	33 MPa
Tensile Strength, Ultimate (Type 1, 0.125", 0.2"/min)	ASTM D638	9,950 psi	6,100 psi	69 MPa	42 MPa
Tensile Modulus (Type 1, 0.125", 0.2"/min)	ASTM D638	312,000 psi	329,000 psi	2,150 MPa	2,270 MPa
Tensile Elongation at Break (Type 1, 0.125", 0.2"/min)	ASTM D638	5.8%	2.2%	5.8%	2.2%
Tensile Elongation at Yield (Type 1, 0.125", 0.2"/min)	ASTM D638	2.2%	1.7%	2.2%	1.7%
Flexural Strength (Method 1, 0.05"/min)	ASTM D790	16,200 psi	9,900 psi	112 MPa	68 MPa
Flexural Modulus (Method 1, 0.05"/min)	ASTM D790	331,000 psi	297,000 psi	2,300 MPa	2,050 MPa
Flexural Strain at Break (Method 1, 0.05"/min)	ASTM D790	No break	3.7%	No break	3.7%
IZOD Impact, notched (Method A, 23 °C)	ASTM D256	2.2 ft-lb/in	0.9 ft-lb/in	120 J/m	48 J/m
IZOD Impact, un-notched (Method A, 23 °C)	ASTM D256	14.6 ft-lb/in	3.2 ft-lb/in	781 J/m	172 J/m
Compressive Strength, Yield (Method 1, 0.05"/min)	ASTM D695	14,500 psi	12,700 psi	100 MPa	87 MPa
Compressive Strength, Ultimate (Method 1, 0.05"/min)	ASTM D695	26,200 psi	13,100 psi	181 MPa	90 MPa
Compressive Modulus (Method 1, 0.05"/min)	ASTM D695	1,030,000 psi	251,000 psi	7,012 MPa	1,731 MPa

	METRIC	
	XZ Orientation	ZX Orientation
Flexural Modulus (Method 1, 0.05"/min)	2,300 MPa	2,050 MPa

Figure 4 - Ultem 9085 data sheet (Stratasys, 2015)

- 2.5. Shear modulus 'μ': 2050MPa [5] [6].

3. Load cases 'F': 600lbf (**2.669kN**) [$600 \times 4.448 = 2668.93$].

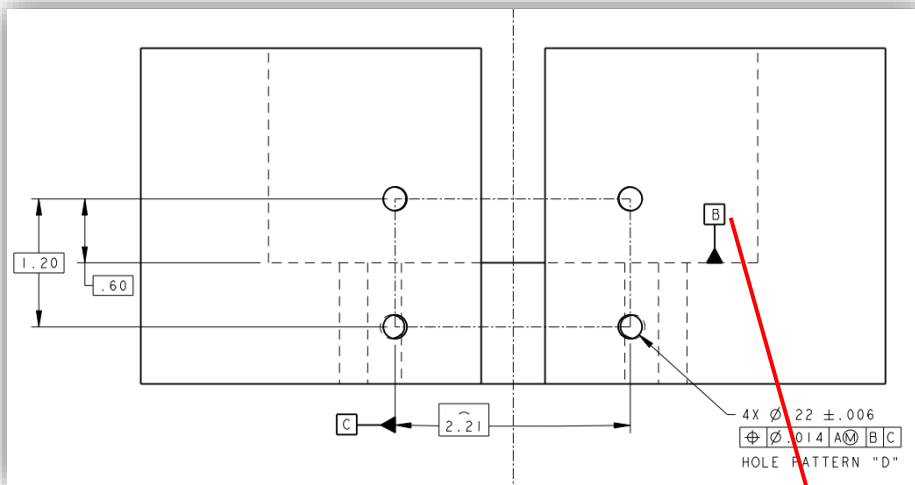


Figure 5 - Front view of envelope drawing (GrabCad, 2016)

The load 'F' is specified as being 2.669kN evenly distributed (UDL) on the surface identified as 'Datum B' in a negative Z direction.

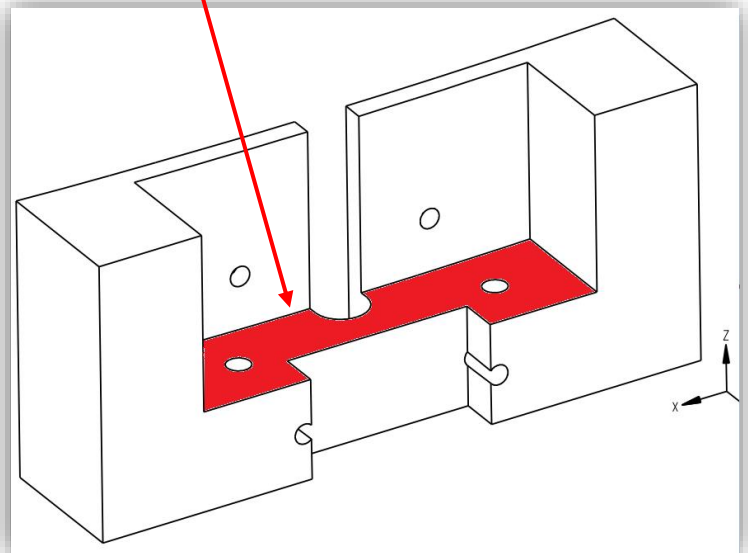


Figure 6 - Isometric view of envelope drawing with force area defined (GrabCad, 2016)

4. Target mass 'm': 0.1lb (**0.045kg**) [$0.1 \times 2.205 = 0.0453592$].

Design Studio will optimize the part as a percentage of its original volume, this may be done by calculating the percentage of the original mass that the target mass will be. One way of doing this is by using SolidWorks, as the material ULTEM 9085 is not available in the SolidWorks material database, measuring the volume of the original envelope model in SolidWorks may be done to calculate the mass.

$$\begin{array}{rcl} (\text{Density}) & \times & (\text{Volume}) & = & (\text{Mass}) \\ 1340 & \times & 0.00032910139 & = & 0.441\text{kg (3dp)} \end{array}$$

$$\therefore 0.045 \div 0.441 = \underline{10.2\% (1dp)}$$

5. Minimum wall thickness ' t ': 0.040" (**1.0mm**) [$0.04 \times 25.4 = 1.016$].

It is important to ensure that all areas of the final design shall not have a thickness of <1mm. A deviation of this may result in the part being damaged easily, therefore is detrimental to the structural integrity of the component.

6. Minimum distance about all holes: $2 \times \text{bolt hole } \varnothing$.

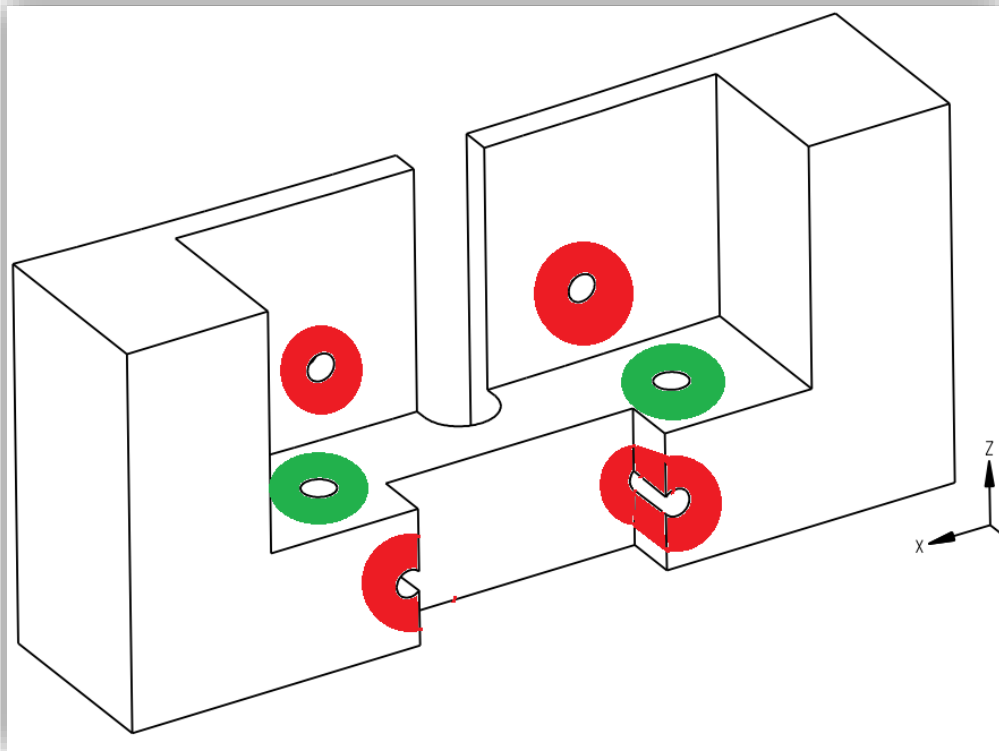


Figure 7 -Isometric view of envelope drawing with hole areas (GrabCad, 2016)

The holes shown above with a red keep zone are $\varnothing.22''$ ($\varnothing 5.6\text{mm}$), the holes shown above with a green keep zone are $\varnothing.27''$ ($\varnothing 6.9\text{mm}$).

- $\therefore$ **Red:** $((5.6 \times 2) \times 2) + 5.6 = \underline{\underline{\varnothing 28.0\text{mm}}}$ keep zone about the centre of the holes.
 $\therefore$ **Green:** $((6.9 \times 2) \times 2) + 6.9 = \underline{\underline{\varnothing 34.5\text{mm}}}$ keep zone about the centre of the holes.

The target is to have a finished product that adheres to this requirement.

5. Analysis

Prior to conducting topology optimisation, a mesh must be applied to the model, this can be done using SolidWorks.

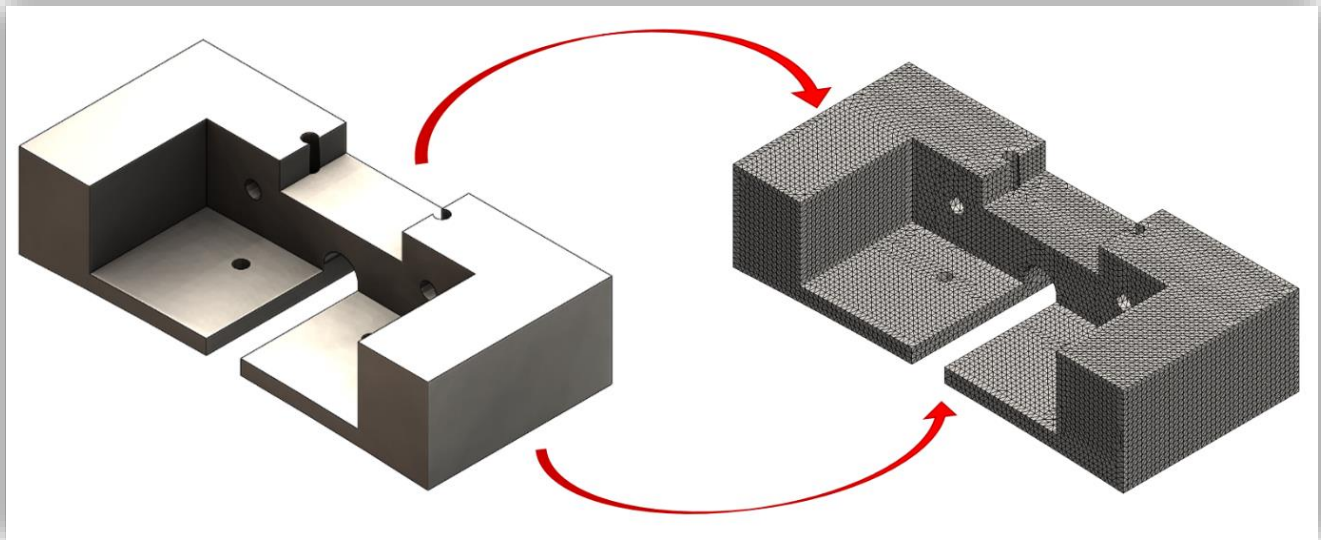


Figure 8 - Envelope model before and after applying mesh (Kilbane, 2019)

Any given mesh will produce results, however a fine mesh would produce the most accurate results, although one consideration when choosing the mesh size is the calculation time, a coarse mesh would be significantly quicker to calculate when compared with a fine mesh [4]. A first order mesh has been selected for this study, this again is due to the relationship between the calculation time and the mesh applied.

It is good practice to apply a mesh of three elements across the thinnest part of the component [4], this is due to the fact that when running an analysis one layer of the mesh elements will be in compression, the opposite layer will be in tension and the middle layer of element will act as a neutral axis [7].

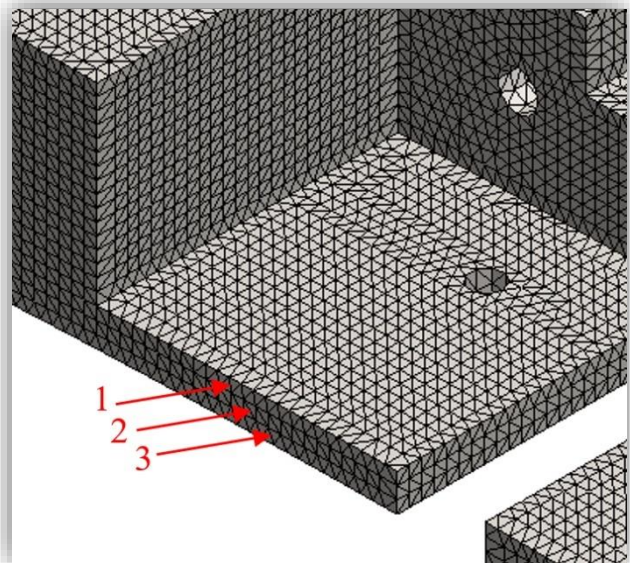


Figure 9 - Mesh selection (Kilbane, 2019)

The model is saved as a '.dat' file which is a NASTRAN deck format, NASTRAN is a widely used format for CAE as it is interchangeable between many programs and software interfaces [8]. The reason for using this file type in this application is that it is necessary to transfer the model and its mesh from SolidWorks to Genesis Design Studio. The imported model is now ready for further work in preparation for analysis.

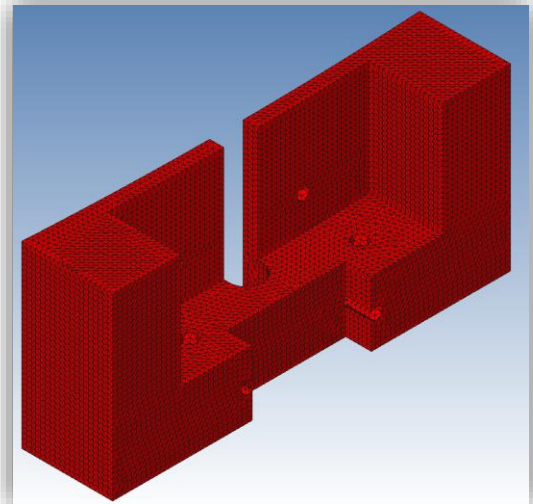


Figure 10 - Model imported into Genesis Design Studio (Kilbane, 2020)

Edit a Material : Isotropic Properties	
E:	2.2408E+9
G:	2.05E+9
Nu:	0.41
Rho:	1340.0
A:	
Tref:	
GE:	
ST:	3.1E+7
SC:	
SS:	

The material properties must first be set in order to run an accurate analysis, this is done by simply imputing the material values that were obtained previously:

E = 2.2408GPa
 G = 2.05GPa
 Nu = 0.41
 Rho = 1340kg/m³
 ST = 31MPa

Figure 11 - Material properties applied in Genesis Design Studio (Kilbane, 2020)

Under the command 'Grid-Component Sets', the part must be secured using the four bolt holes as identified on the drawing. There are several ways of doing this, one way is to constrain the upper edge of the hole as this is where the bolt head rests. A second option is to consider the washer that may be used and then constrain the nodes beneath the area of the washer. A third option is to constrain the entirety of the inner surface of the hole, one may argue this is a more realistic approach as the part may move as such that the inner surface of the hole contacts the bolt shaft, this third method was used. The nodes that are fixed are then identified as having six degrees of freedom by adding small white text boxes next to each of the nodes.

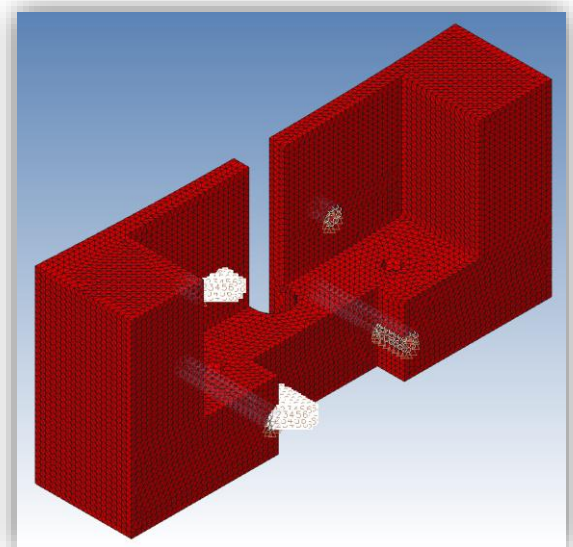


Figure 12 - Hole nodes fixed (Kilbane, 2020)

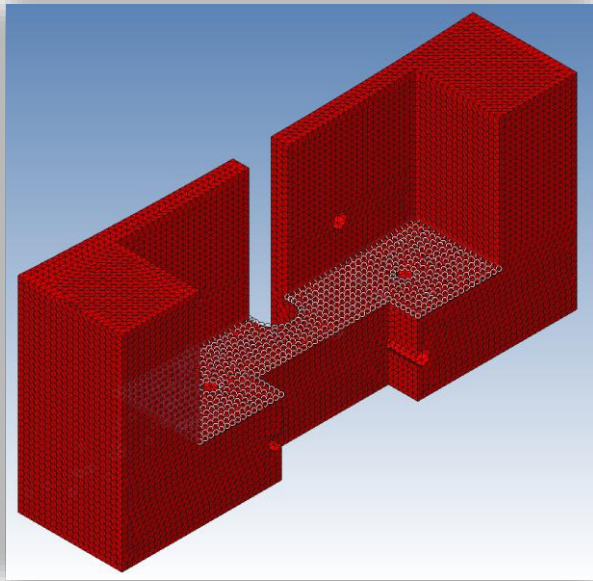


Figure 13 - Nodes selected for applying force (Kilbane, 2020)

The force must be applied across the entire surface as defined previously, it acts perpendicular to the face and directly upon it. The force is a UDL, this may be calculated by taking the total force and dividing it by the number of selected nodes on which it will act. This is necessary as Design Studio will apply the specified force to each of the selected nodes.

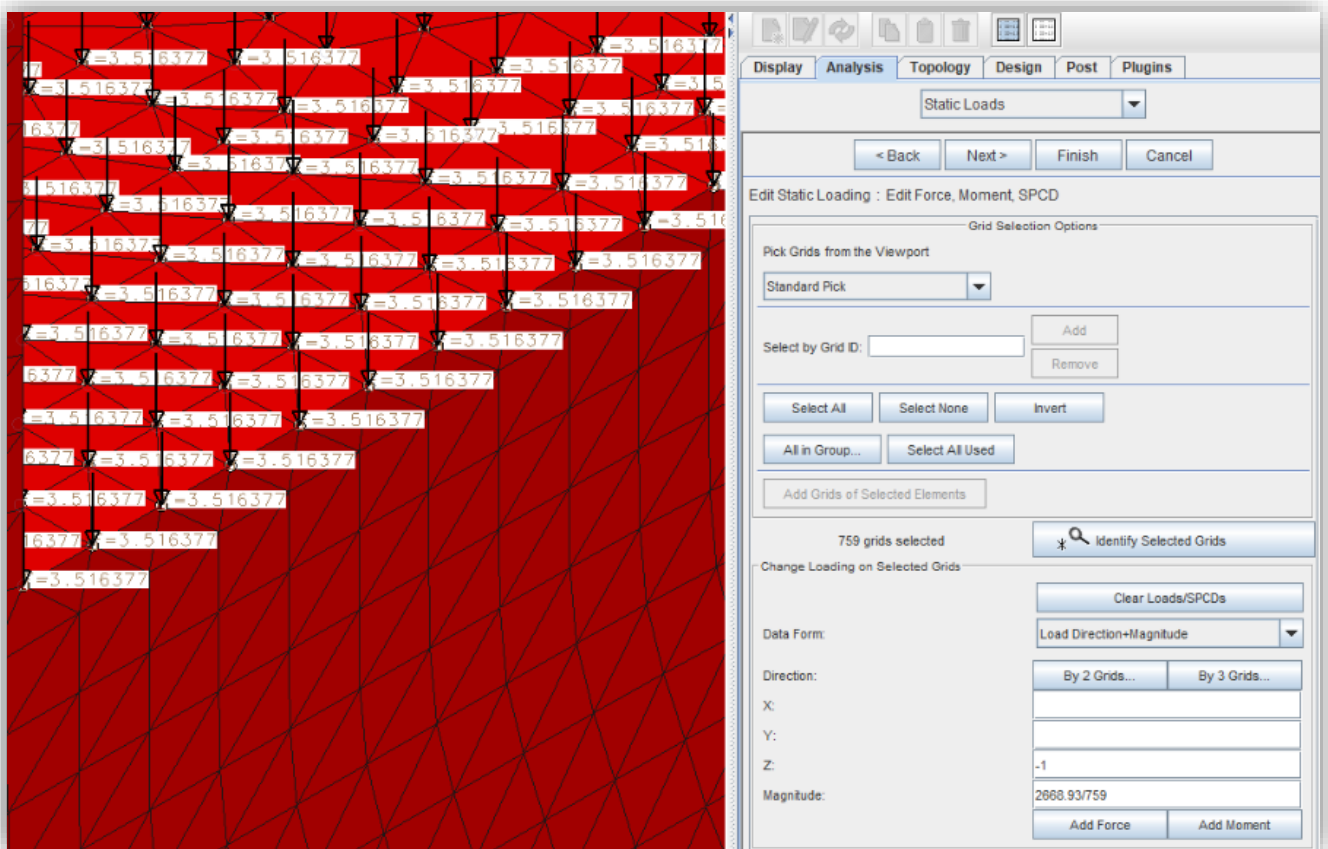


Figure 14 -Nodes with force applied (Kilbane, 2020)

The image above shows the force acting upon each node, being 759 of them in total.

$$\begin{aligned} \text{(Total Force)} \quad & \div \quad (\sum \text{ Nodes}) \quad = \quad \text{(Force per node)} \\ 2668.93 \quad & \div \quad 759 \quad = \quad \underline{\underline{3.516377\text{N}}} \end{aligned}$$

Having defined the parameters of the material properties, fixing holes and the force, it is time to tie this together in what is known as a 'loadcase'. This is done using a simple command within Design Studio, it defines what will form part of the study. At this stage, it is possible to run a 'Single Analysis', this is not the topology although it runs a simulation on the model given the material and the forces.

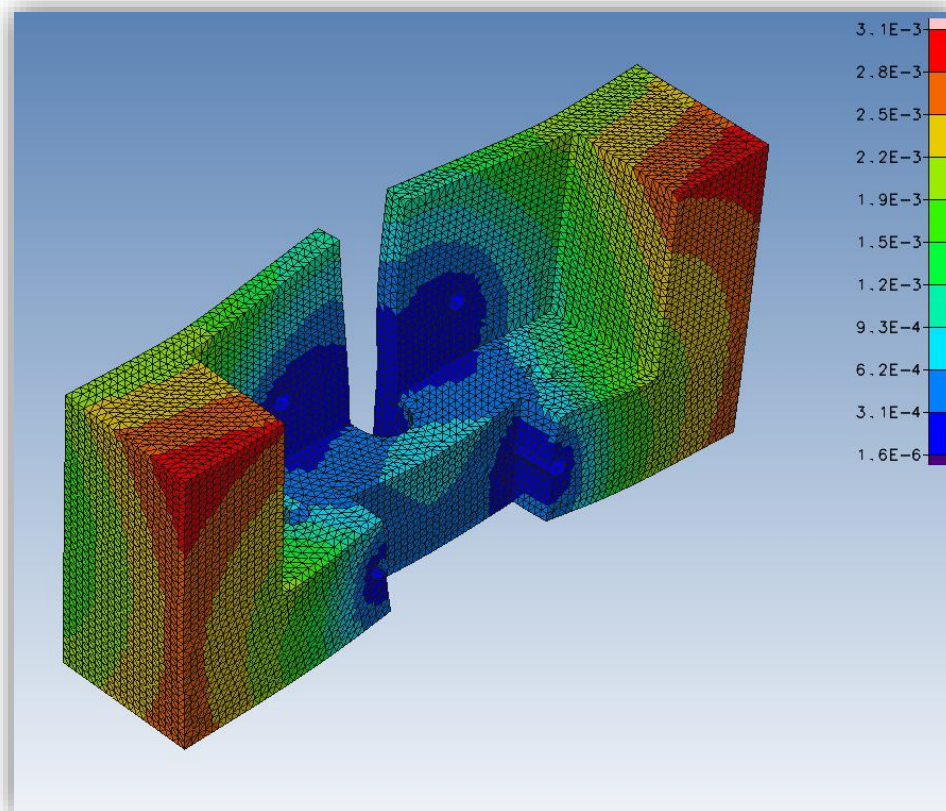


Figure 15 – Displacement results from single analysis (Kilbane, 2020)

The results display the displacement of the component under the given loading condition, with a maximum displacement of 3.1mm at the uppermost far corners. Whereas this displacement may not be desirable, the outer most corners are not in contact with the mating parts therefore this has little consequence in reality. More concerning is the surface where the load itself is acting, because this area will be in contact with the mating parts, and a displacement of 1.5mm is seen at the edges, therefore further investigation may be required in order to understand the impact of this within the larger assembly.

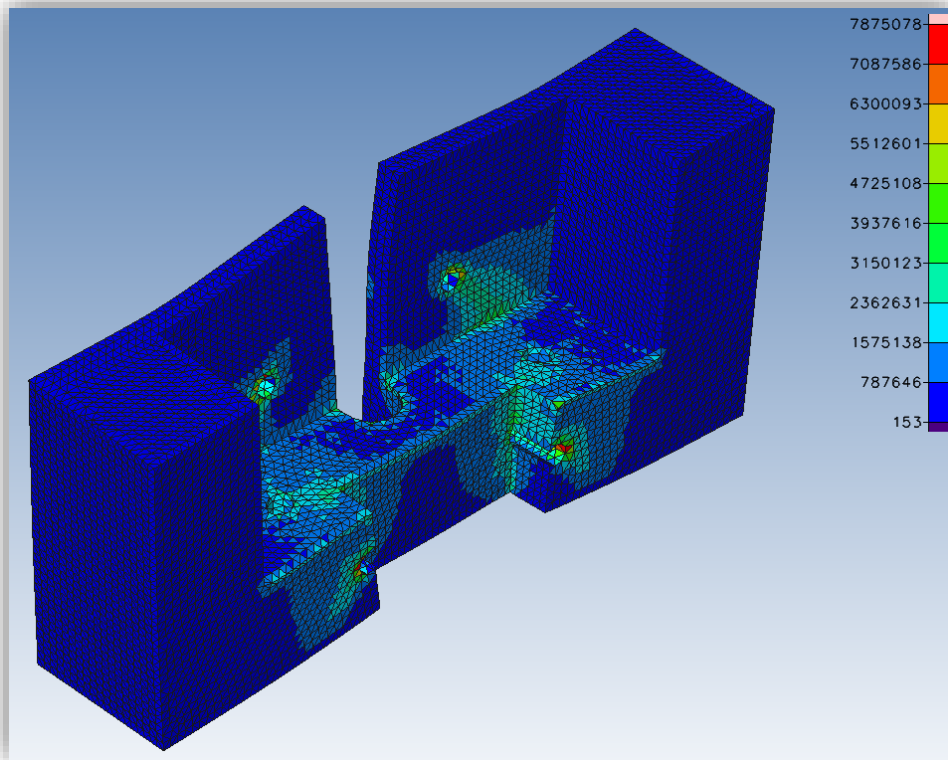


Figure 16 - Stresses from single analysis (Kilbane, 2020)

The resultant stresses of the loading are displayed above, showing high levels of stress about the holes, this is to be expected as there is a concentration about the tangent of the holes as this is the area constrained by the bolts. The stresses in these areas are 7.875MPa, this is <31MPa which is the allowable stress or tensile strength of the material [5], so the stress concentration in this region is not a concern. However, if the stress was relatively higher, due to its concentration it may be possible for such a condition to give rise to a crack and over time through vibration the crack may propagate, therefore fatigue failure may be possible [9] [10].

The analysis carried out thus far is not a step of the topology optimisation process, but it provides a good indication of what is happening with the model [11], it may also aid with the decisions made post optimisation. For example, if an area is highly stressed, in a corner for example, it may be permissible to add a simply fillet to alleviate some of the stresses [10], or in this case, a larger bolt may be desired to distribute the load somewhat.

As the results of the single analysis are imported into Design Studio, they will remain saved for reference at a later stage if required, or they may be deleted manually by the user if they were to run another study. The next step is to prepare for topology optimisation, this is done by setting three elements:

1. 'Topology Regions' – The model that is to be analysed, and the minimum thickness is set to 1mm, as defined in the parameters.
2. 'Topology Objectives' – The loadcase that has been defined previously.
3. 'Topology Constraints' – This is set to have an upper bound constraint of 0.1, this means that the desired result is to have an optimised model that is 10% of the size of the original model.

The next step is to run the optimization, the results are returned in the form of a text data window with two graphs.

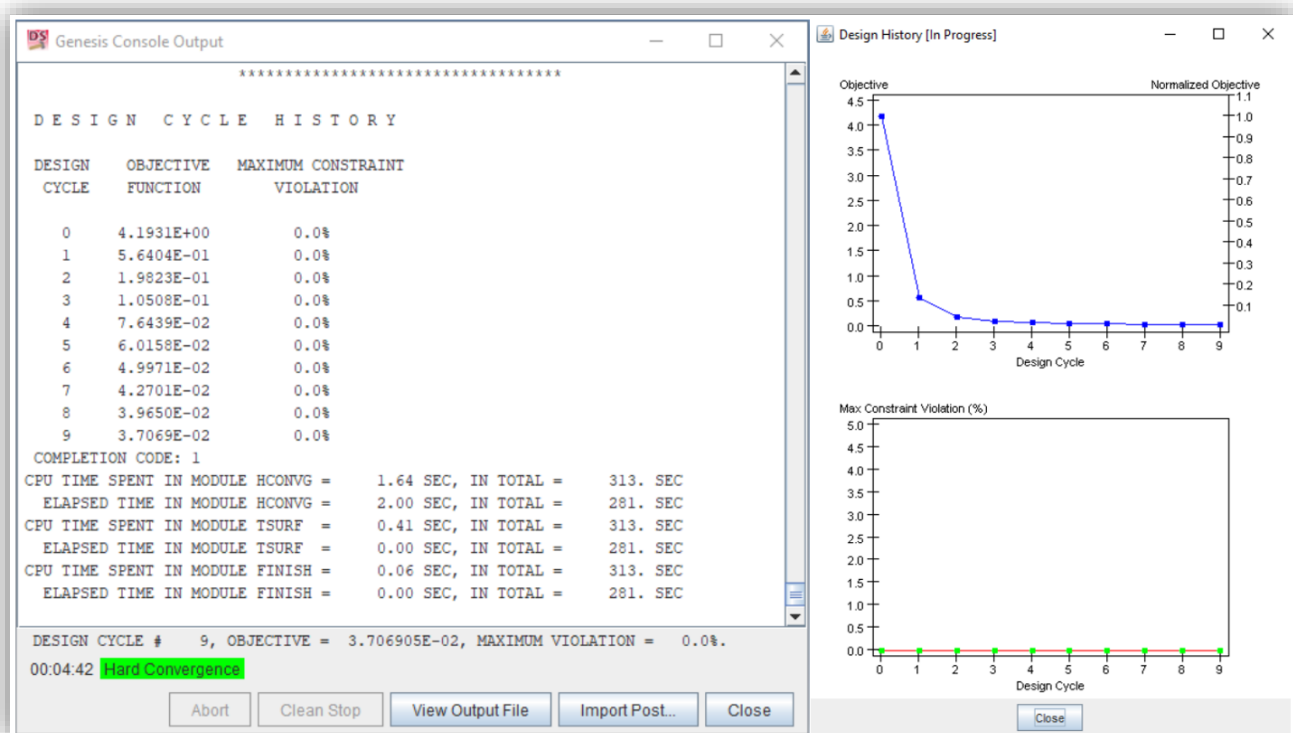


Figure 17 - Topology Optimisation graphs & script (Kilbane, 2020)

The 'Objective' graph displays that the number of iterations or design cycles the topology optimisation has run, each time attempting to optimise the model to achieve the target. The vertical scale on the left displays the mass of the model, the adjacent scale displays the percentage compared with the original model, the horizontal scale simply defines the number of cycles that the optimisation has run.

The first returned model at 10% of the original mass looks good, although model does not adhere to the $2 \times \text{bolt hole } \varnothing$ of material surrounding the holes, therefore this is a violation. The uppermost holes shown on the vertical face do not have the desired hole to edge distance, where the holes on the horizontal face does not have material following throughout the part, meaning a great portion of the holes are missing.

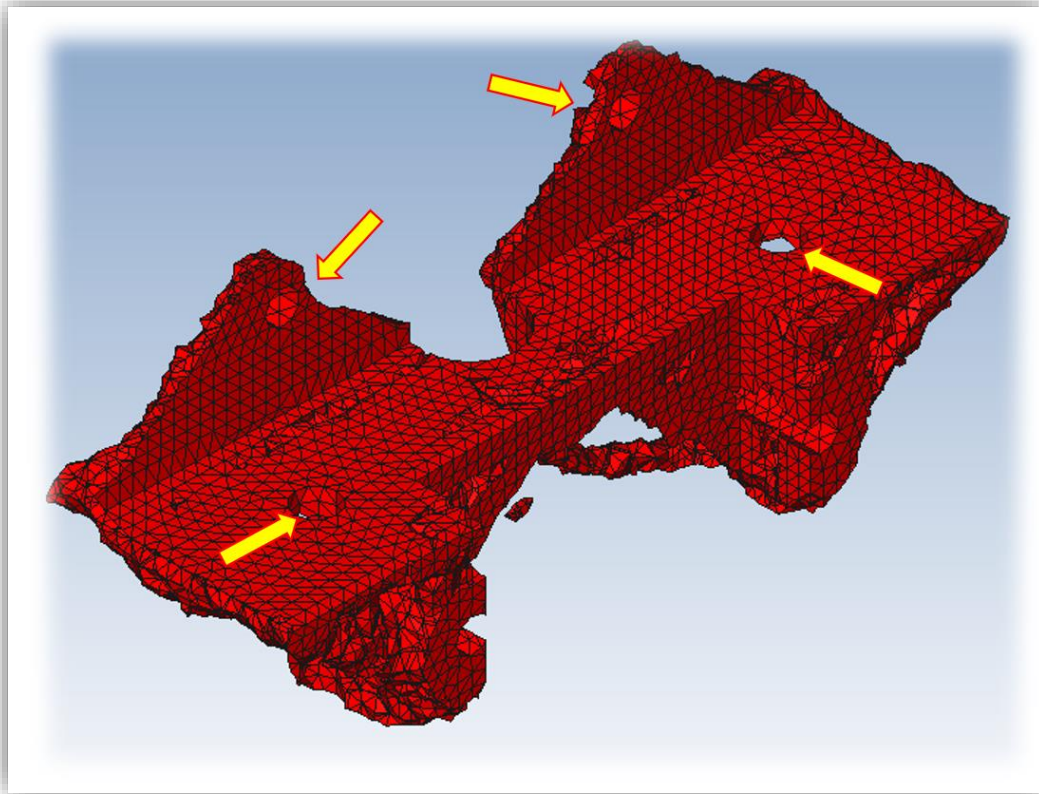


Figure 18 - Topology result without hole area consideration (Kilbane, 2020)

In order to obtain a more accurate result, a new region may be set up within the parameters in order to maintain the areas surrounding the holes. This new area is set to $2 \times \text{bolt hole } \varnothing$ and will be excluded from the optimisation, the optimisation is run again.

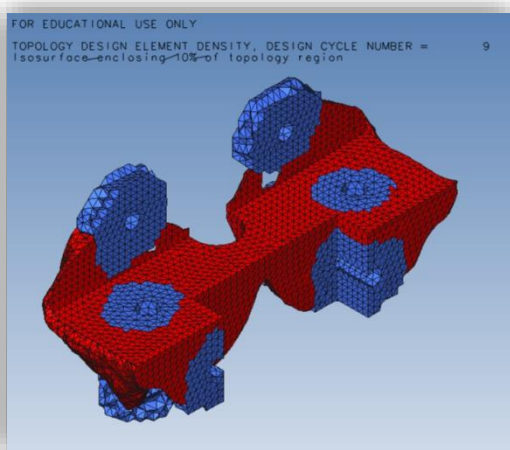


Figure 19 - Topology result with hole areas adhered to (Kilbane, 2020)

This is the returned model that is 10% of the original size, although it is important to note that this 10% figure is of the optimised area alone (in red), not the area that was excluded from the optimisation (in blue). It is necessary to reduce the model to a lower percentage, then measure the mass.

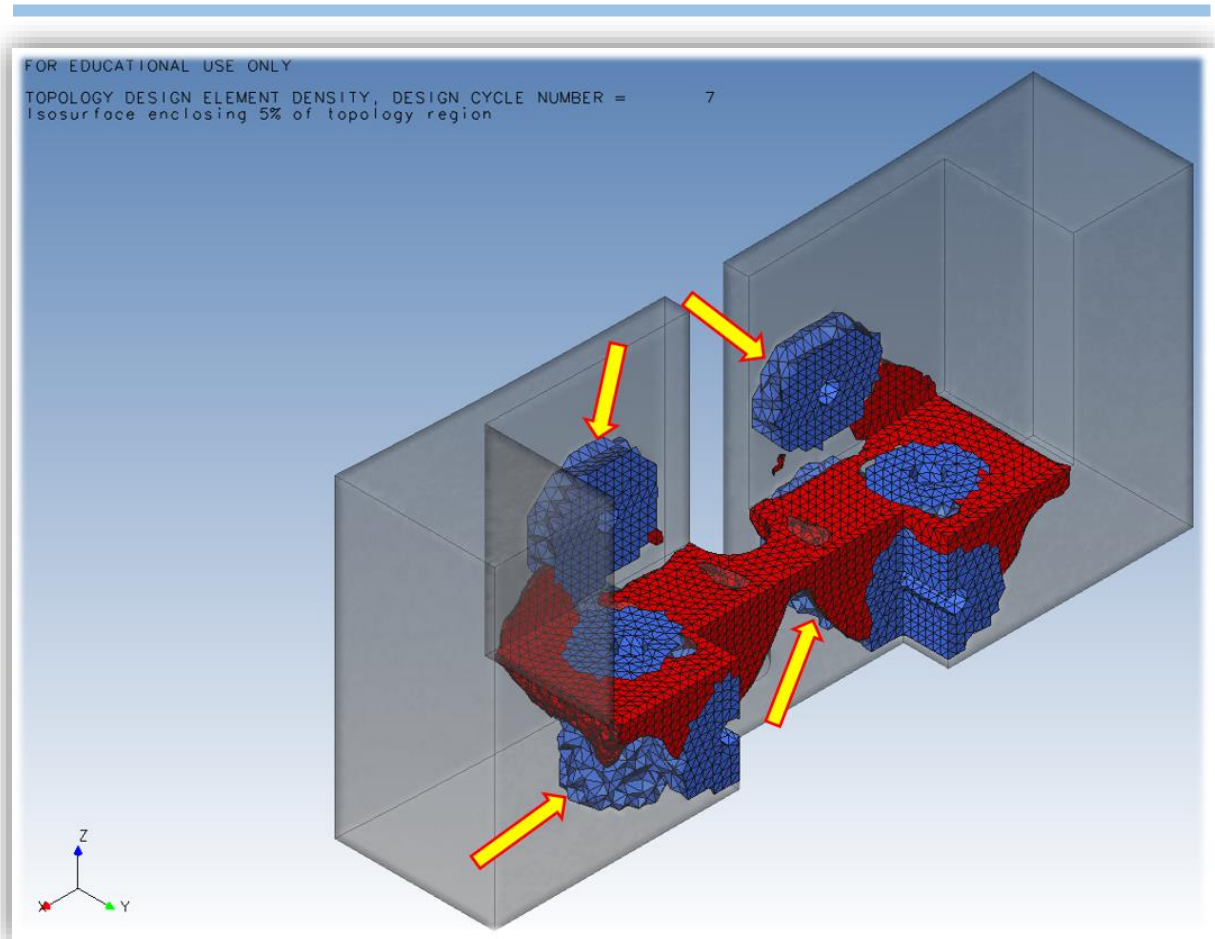


Figure 20 - Topology result with 5% model volume and hole areas (Kilbane, 2020)

The optimised area of this model (in red) is 5% of the original size, it accounts for approximately 50% of the entire model, making this very near the desired result. Note that the area surrounding the holes are good to satisfy the $2 \times \text{bolt hole } \varnothing$ rule, and the area follows the depth of the holes. It is now possible to export the model back to SolidWorks for further work.

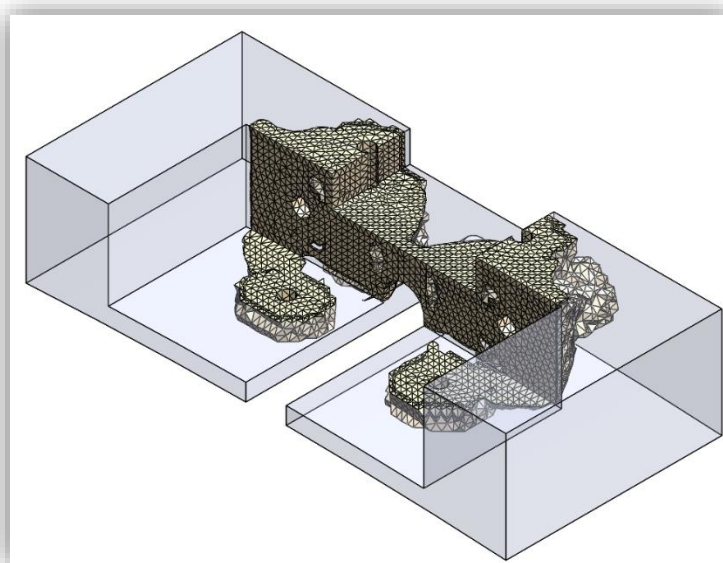


Figure 21 - Topology model overlaid with initial envelope (Kilbane, 2020)

Analysing the part in SolidWorks, the volume is 0.00038163489, it is now necessary to calculate the mass:

$$\begin{array}{rcl} (\text{Density}) & \times & (\text{Volume}) & = & (\text{Mass}) \\ 1340 & \times & 0.00038163489 & = & 0.511\text{kg (3dp)} \end{array}$$

Considering the target mass of 0.441kg, there is an infringement of 70g above the desired mass, this is not a concern as the model has not yet been modified.

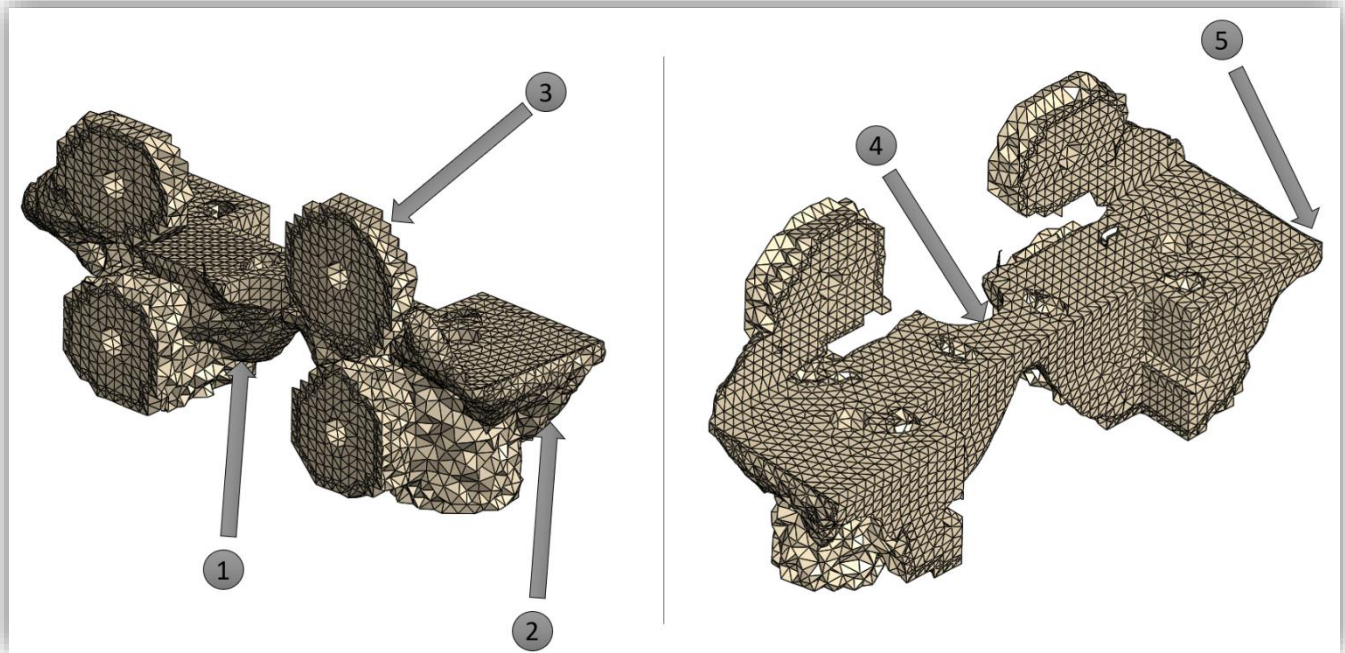


Figure 22 - Topology model with suggested design solutions (Kilbane, 2020)

List of suggestions

1. Reduce the thickness on the middle of the underside and add a few ribs to compensate for the reduced material [12].
2. Reduce the thickness on the edges of the underside and add a few ribs to compensate for the reduced material [12].
3. Smooth and reduce the rough area surrounding the holes, they are slightly greater than $2 \times \text{bolt hole } \varnothing$.
4. This area may be removed completely, making two brackets instead of one. The fact that there are three fastening holes in each part, the part will not rotate therefore it is possible to split the part into two.
5. Add fillets to the outer corners where possible, thereby removing further mass.

It is not within the scope of this assignment to carry out this work, however in reality this is what could be done to achieve a good end product.

6. Conclusion

The topology optimisation returned results that provided the basis for a stable structure, whereas the resultant model itself may be used to identify the primary load paths through the product, additional work must be done to design a workable product. The returned model does not appear to be aesthetically attractive, hence why design work must be carried out to finalise the product. The process of topology optimisation in itself is a very useful tool, by running this process on a simple envelope model at an early stage, it enables the ability to design a better product that should meet the requirements with regards to loading factors.

Upon review of the work done here, perhaps different meshes should have been used, a finer mesh would return a smoother end model to design around, with a finer mesh may simply be additive manufactured using ever growing technologies [13] [14], this would however depend on factors such as the cost and aesthetic requirements of the product. In this instance the subject is a bracket, topology optimisation may be used to develop larger structures such as bridges, motor car frames and even aircraft structures [15], it can save a lot of money when compared with the usual method of design and then running stress analysis on FEA models.

More work could be carried out on this product, the areas around the holes could have been measured more accurately when setting up for topology optimisation, a more experienced Genesis Design Studio user could possibly modify the parameters as such to return better results.

7. References

- [1] X. Huang and Y. M. Xie, *Evolutionary Topology Optimization of Continuum Structures*, Chichester: Wiley, 2010.
- [2] E. Oberg, F. D. Jones, H. L. Horton and H. H. Ryffel, *Machinery's Handbook*, South Norwalk: Industrial Press Inc, 2016.
- [3] GrabCAD, “GrabCAD Community,” GrabCAD, 2016. [Online]. Available: <https://grabcad.com/challenges/3-2-1-liftoff-ula-rocket-hardware-challenge>. [Accessed 3 January 2020].
- [4] stratasys, “stratasys.com,” stratasys, 2020. [Online]. Available: <https://www.stratasys.com/materials/search/ultem9085>. [Accessed 5 January 2020].
- [5] Stratasys, *ULTEM 9085 - Data Sheet*, Eden Prairie Minnesota: Stratasys, 2017.
- [6] “Stratasys,” Stratysys, 2020. [Online]. Available: https://www.stratasys.com/?utm_source=logos&utm_medium=pdf&utm_content=footer-stratasys-logo. [Accessed 3 January 2020].
- [7] J. E. Brancheau, *Practical Aspects of Finite Element Simulation*, Manchester: Altair, 2015.
- [8] A. O. Cifuentes, *Using MSC/NASTRAN Statics and Dynamics*, New York: Springer Science & Business Media, 2012.
- [9] M. P. Norton and D. G. Zarczub, *Fundamentals of noise and vibration analysis for engineers*, Cambridge: Cambridge University Press, 2003.
- [10] E. J. Hearn, *Mechanics of Materials 1*, Oxford: Elsevier, 1997.
- [11] B. Bian, *Topological Optimization of Buckling*, Berlin: de Gruyter, 2018.
- [12] K. L. Tucker, M. Kemp and R. Boyd, “Topology optimization of aircraft wing box ribs,” in *10th AIAA/ISSMO Multidisciplinary Analysis and Optimization Conference*, New York, 2004.
- [13] D. J. Munk, D. J. Auld, G. P. Steven and G. A. Vio, “On the benefits of applying topology optimization to structural design of aircraft components,” *Structural and Multidisciplinary Optimization*, vol. I, no. 60, p. 1245–1266, 2019.
- [14] W. Zhang, J. Zhu and T. Goa, *Topology Optimization in Engineering Structure Design*, London: ISTE Press Ltd, 2016.
- [15] J. Zhang, Z. Li, Q. Fang and C. Chen, “Topological optimisation design of passenger car seat backrest frame based on multiple-loading conditions,” *International Journal of Crashworthiness*, vol. I, no. 26, pp. 1083-1093, 2019.

Table of Figures

Figure 1 - ULA Rocket exploded view (GrabCAD, 2016).....	4
Figure 2 - Envelope drawing (GrabCad, 2016)	5
Figure 3 - Model created from drawing (Kilbane, 2019).....	5
Figure 4 - Ultem 9085 data sheet (Stratasys, 2015).....	6
Figure 5 - Front view of envelope drawing (GrabCad, 2016)	7
Figure 6 - Isometric view of envelope drawing with force area defined (GrabCad, 2016)	7
Figure 7 -Isometric view of envelope drawing with hole areas (GrabCad, 2016).....	8
Figure 8 - Envelope model before and after applying mesh (Kilbane, 2019).....	9
Figure 9 - Mesh selection (Kilbane, 2019)	9
Figure 10 - Model imported into Genesis Design Studio (Kilbane, 2020).....	10
Figure 11 - Material properties applied in Genesis Design Studio (Kilbane, 2020).....	10
Figure 12 - Hole nodes fixe (Kilbane, 2020)	10
Figure 13 - Nodes selected for applying force (Kilbane, 2020)	11
Figure 14 -Nodes with force applied (Kilbane, 2020)	11
Figure 15 – Displacement results from single analysis (Kilbane, 2020)	12
Figure 16 - Stresses from single analysis (Kilbane, 2020)	13
Figure 17 - Topology Optimisation graphs & script (Kilbane, 2020)	14
Figure 18 - Topology result without hole area consideration (Kilbane, 2020).....	15
Figure 19 - Topology result with hole areas adhered to (Kilbane, 2020).....	15
Figure 20 - Topology result with 5% model volume and hole areas (Kilbane, 2020).....	16
Figure 21 - Topology model overlaid with initial envelope (Kilbane, 2020).....	16
Figure 22 - Topology model with suggested design solutions (Kilbane, 2020).....	17