

1708327 Paul Thomas

by Paul Thomas

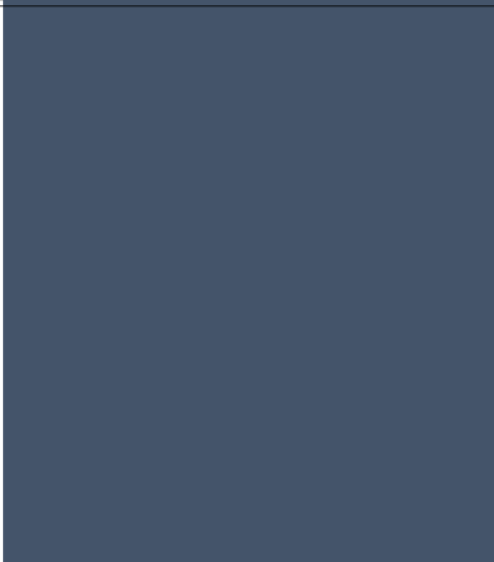
Submission date: 26-Jan-2020 04:48PM (UTC+0000)

Submission ID: 119121397

File name: 6171_Paul_Thomas_1708327_Paul_Thomas_231305_2084939408.pdf (2.06M)

Word count: 4185

Character count: 26225



BEng Engineering

Advanced Computational
Methods

University Of Wales Trinity Saint David
School of Logistics and Manufacturing

M6X00408

Paul Thomas –1708327
Manufacturing Systems Engineering

Executive Summary:

In 2016, the United Launch Alliance (ULA) collaborated with GrabCAD to launch a topology optimisation design competition. This is a study detailing the process using topology optimisation to redesign a Boattail support bracket. Finite Element (FE) analysis is employed to simulate the components performance in a specific loading scenario.

The design process used three design iterations to optimise the component: Material positioning was prioritised in regions of high stress and removed in areas of low stress. An 89.4% mass reduction was accomplished with a subsequent effect on displacement and the stiffness-to-weight ratio. The final component, though considerably lower in mass was more efficient at distributing the applied loads.

The study concludes that the organic designs generated are perfectly suited for modern additive manufacturing techniques. Although further thought is required, as the fixed geometry and loading conditions are pinpointed as potential limitations that need further insight due to their effect on result accuracy.

1 Table of Contents

Executive Summary:	1
2 Introduction	3
2.1 Challenge Criteria	4
3 FE Model Considerations.....	5
3.1 Part function.....	5
3.2 Loading Conditions	6
3.3 Fixture Conditions.....	7
3.4 Material Considerations	8
4 Topology Optimisation	11
4.1 Mesh Strategy.....	11
4.2 Baseline Static Study.....	11
4.3 Topology Optimisation	13
4.4 SolidWorks Topology Constraints.....	13
5 Topology Optimization Design Iterations	14
5.1 Final Design.....	16
6 Analysis and Discussion	18
6.1.1 Von Mises Stress Analysis	18
6.1.2 Stiffness to Weight Ratio	20
7 Study Limitations	20
7.1.1 Fixed Geometry	20
7.1.2 Multiple Load Cases.....	21
8 Conclusions.....	21
9 Recommendations.....	21
10 References.....	22

2 Introduction

The United Launch Alliance (ULA) was announced on December 1, 2006. It is an amalgamation of Boeing and Lockheed Martin Corp (LMC), with the purpose of uniting their skills and experience to produce dependable and cost-effective spacecraft. Both companies have in excess of 50 years' experience in the aerospace industry (United Launch Alliance, 2010).

In 2016, GrabCAD collaborated with ULA to launch a competition to design a launch support bracket for ULA's Atlas V rocket. The competition required the component to meet strict criteria for size, weight and performance (Evotech, 2019).

This study will document the design of a suitable rocket launch bracket, with the use of topology optimisation (TO) software within SolidWorks. The objective of the study is to design a component which meets all the competitions criteria. TO software can examine a design space and identify the critical load paths and distribute material accordingly, this makes it an incredibly useful design tool. (Bodkhe & Jain, 2014) discuss the importance of TO software within the design process, this is demonstrated in Figure 1 below. TO utilises gradient computation methods to assist the design process, this involves repeated design iterations until user criteria is met (Sigmund & Maute, 2013).

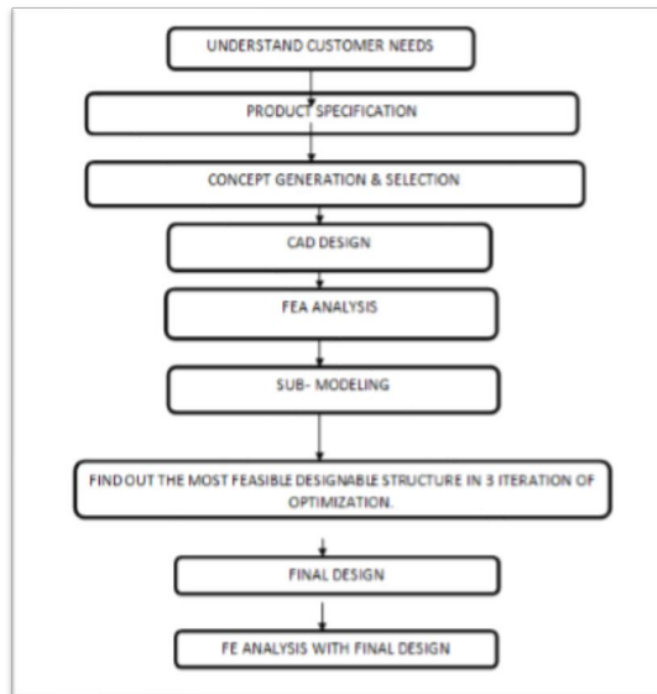
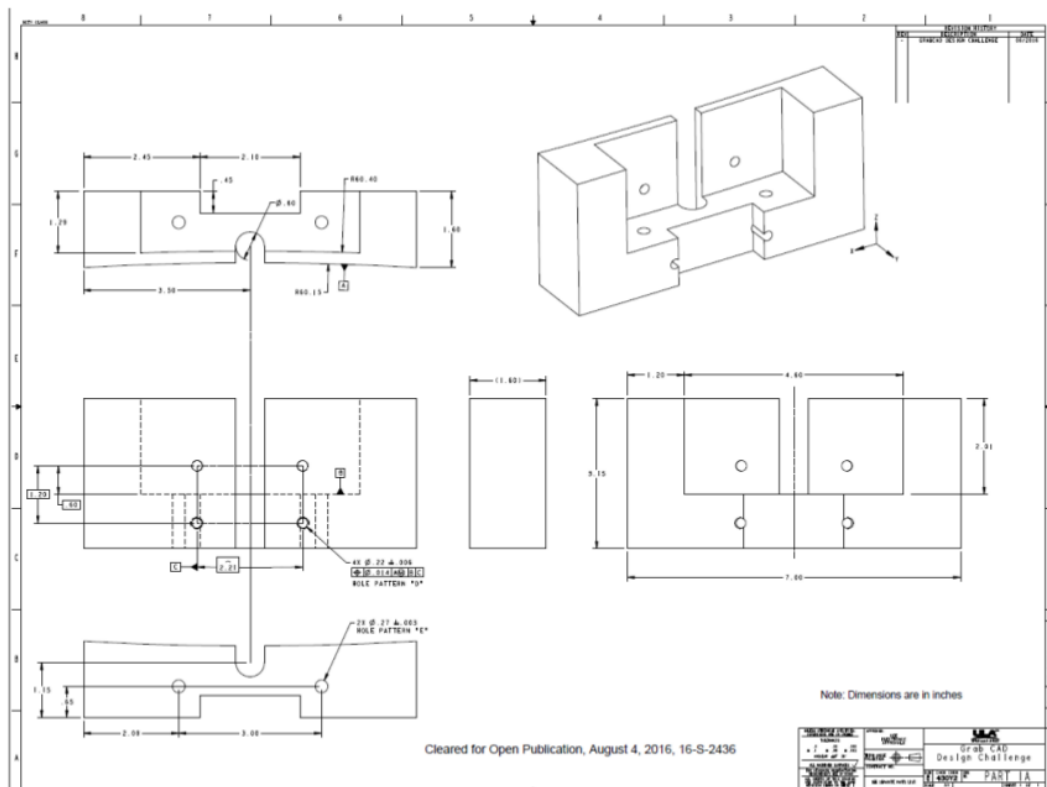


Figure 1 - FE design process including topology optimisation (Bodkhe & Jain, 2014).

2.1 Challenge Criteria

Part Requirements	Maximum part weight: 0.1 lb Minimum wall thickness: 0.040 inches Ultimate load (due to work platform): 600 lbf applied uniformly in the negative Z direction (illustrated on the envelope drawing) The minimum edge distance surrounding all bolt holes shall be 2 times the diameter of the bolt hole
Required material: Ultem 9085	Material Properties Ultem 9085 Material Strength Allowable: 4500 PSI Specific Gravity: 1.34 Young's Modulus: 325 KSI Poisson's Ratio: 0.41



3 FE Model Considerations

Development of an accurate FE model requires a complete understanding of a part, how it interacts with mating components and their environment. Once this is understood a suitable simulation of any loading conditions can be generated.

3.1 Part function

The boattail support for the challenge is located at the upper stage of the rocket. Its main function is as a support location for work platforms used before launch (Figure 2). As it remains with the spacecraft during its flight, its alignment with fundamental spacecraft design requirements is critical. Primarily, it must be lightweight due to its first order effect on payload and performance. Furthermore, it must withstand the extreme loads involved in the initial stages of the rocket launch of the ULA Atlas V (NASA, 2011), (Watson, et al., 2019).

The boattail forms part of the spacecrafts upper stage. It is a transitional section designed to reduce the drag created by the payload fairing. An efficient boattail design forces air into the area behind the fairing reducing drag to below zero and helping produce thrust. The fairing is used to protect the rocket on its launch and ejected once the atmospheric phase is complete (Ramamurti, et al., 2001).

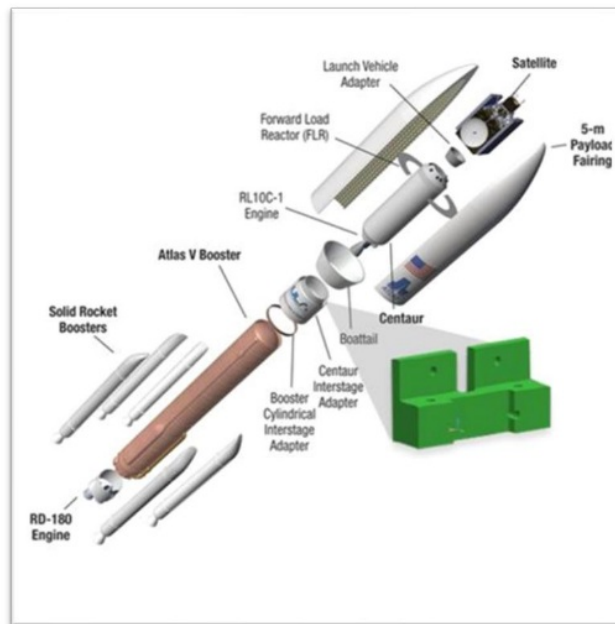


Figure 2 - Boattail support bracket location (Evotech, 2019)

1.

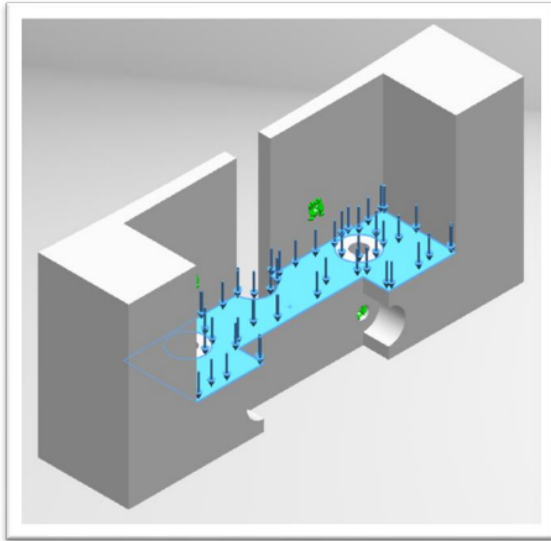


Figure 3 - FE Model vertical load

3.2 Loading Conditions

The loading conditions identified while generating an FE model, will define the accuracy of the simulation (Sigmund & Maute, 2013). This is collaborated by (Bodkhe & Jain, 2014), who state that the accuracy of an FE simulations result, is diametrically dependant on the loading conditions created. The challenge criteria specified a definitive load of 600 lbf applied uniformly in the negative Z direction(Figure 3).

In addition, loads generated by fasteners must also be considered. Fasteners used within a spacecraft launch environment are subject to extreme vibration levels (Li, et al., 2018). Therefore, the correct determination of fastener torque is essential, (NASA, 1999) state that increased fastener torque, increases resistance to vibration. Torque is defined as the function of friction coefficients between mating surfaces. 50% overcomes the nut bearing friction , 40% dissipates the thread friction and the final 10% generates preload (NASA, 1999).

Consequently, (NASA, 1999) have developed standard torque levels for components used within spacecraft. These state that for the bolt size required (10-24 pitch), it is essential that any fasteners be torqued between 23-34 in/lb. In addition, the effect of washers need consideration. A washer will distribute the preload over an increased surface area. Table 1 below displays the calculations used to generate loading conditions for the FE simulation.

1.

T = Torque (lb/in) F = Bolt tension D = Nominal Bolt Diameter C = Coefficient of Friction constant Aw = Area of Washer	$T = cDF$ $Fb = \frac{T}{cD}$ $F = \frac{23}{0.2 \times 0.19}$ $F = 605 \text{ lbs}$	$Aw = \frac{\pi}{4} (0.62^2 - 0.19^2)$ $A = 0.27 \text{ in}^2$ $\sigma_w = \frac{F}{A}$ $\sigma_w = \frac{605}{0.27}$ $\sigma_w = 2240$
---	--	---

Table 1 - Fastener Calculations (NASA, 1999).

The load generated by the fastener will be distributed using the split lines created around the bolt holes, therefore simulating the area of a washer (Figure 4, Figure 5).

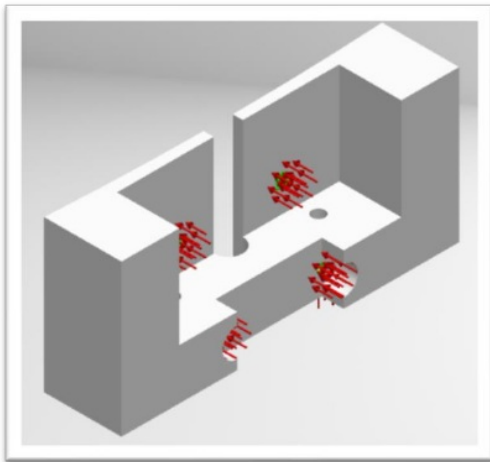


Figure 4 - Location of fastener loads

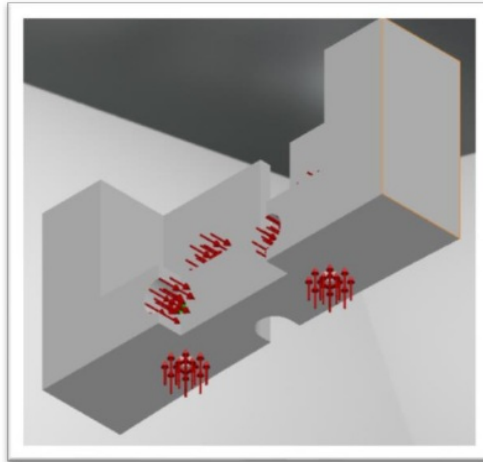


Figure 5 - Location of fastener loads

3.3 Fixture Conditions

The generation of suitable fixture conditions is critical to simulation accuracy, therefore careful consideration of how the part will attach to any mating parts is essential. The part's function indicates it is fastened to the large diameter of the rocket boattail. It was presumed that the component is fastened using washers and has full contact on the brackets rear surface with the boattail body. Therefore, to determine fixed geometry within the model I have utilised the split lines modelled when generating the fastener load surface (Figure 6).

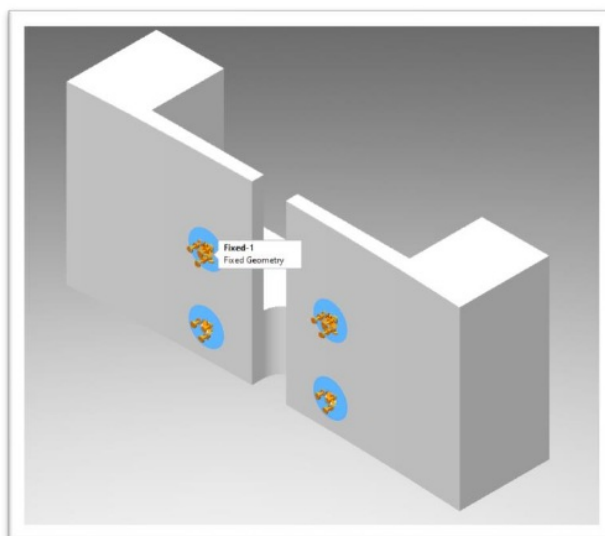


Figure 6 - Fixed geometry location

3.4 Material Considerations

Ultem® is an amorphous thermoplastic polyetherimide (PEI) resin, it is easily fabricated via FDM or injection moulding (Aerospace.org, 2019). (Stratasys, 2019) describe how its high strength/weight ratio and high stiffness properties are ideal for aerospace components.

Material properties of Ultem®	Applications for Ultem®
Heat, solvent and flame resistant	Jet engine components
High dielectric strength and stability	Microwaves
Excellent mechanical properties	Medical components
Rigid and strong	Manifolds
Exceptional resistance to environmental forces	Electronic/electrical insulators

Table 2 below lists Ultem's® material properties and typical applications. Materials selected for spacecraft components must be able to withstand exposure to the severe launch environment. Temperatures range from -185 to 425 degrees Celsius with intense acoustic vibrations ranges of 5-3000hz (NASA, 2011), (Cherian, et al., 2015). Typical spacecraft materials used include, aluminium, titanium, molybdenum, nickel and beryllium. Their characteristics of high strength and high-heat resistance are essential in spacecraft component design (Henson, 2019). Ultem® shares many of these characteristics but most importantly is very low density.



Material properties of Ultem®	Applications for Ultem®
Heat, solvent and flame resistant	Jet engine components
High dielectric strength and stability	Microwaves
Excellent mechanical properties	Medical components
Rigid and strong	Manifolds
Exceptional resistance to environmental forces	Electronic/electrical insulators

Table 2 - Ultem material properties (Stratasys, 2019)

The stress-strain curve shown in Figure 7, demonstrates the relationship between the tensile stress/strain measured in FDM and injection moulded Ultem. The graph clearly demonstrates the enhanced abilities of injection moulded Ultem in relation to strain. It also demonstrates the importance of optimising FDM print direction for increased tensile strength (Fischer & Josupeit, 2019). As a result, injection moulding the final component would be advantageous, owing to its increased resistance to strain.

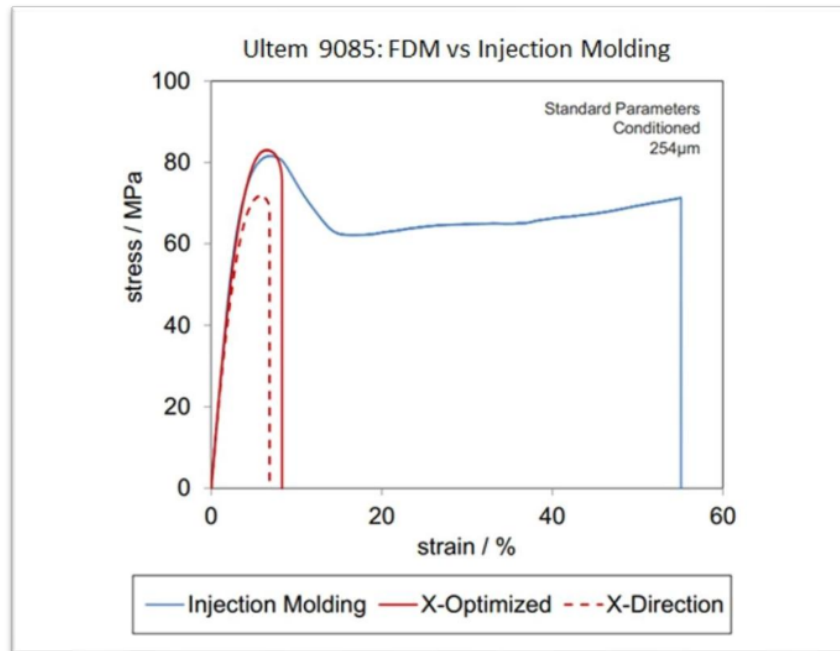


Figure 7 - Ultem FDM vs Injection Moulding Stress/Strain (Fischer & Josupeit, 2019)

The plots below clearly demonstrate Ultem's advantageous capability to combine low density with high strength and stiffness in relation to other materials used within aerospace.

Figure 8, Figure 9 plot Young's modulus and Tensile Modulus vs density. According to (Hibbeler, 2013), this determines a materials capability in terms of strength-to-weight and stiffness-to-weight ratio.

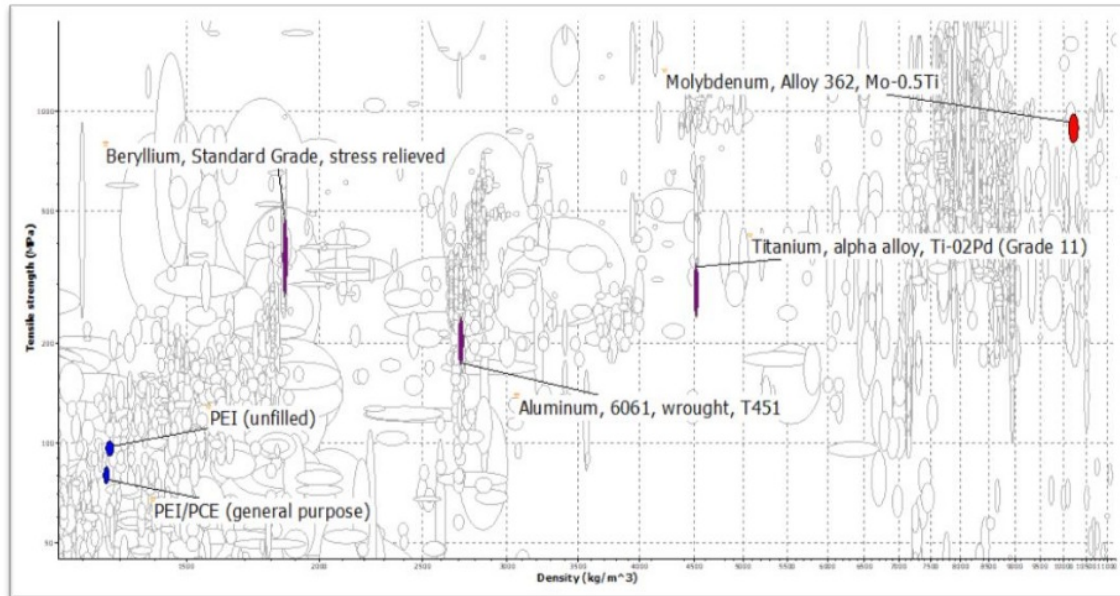


Figure 8 – Tensile Strength vs Density Plot (CES edupack, 2018)

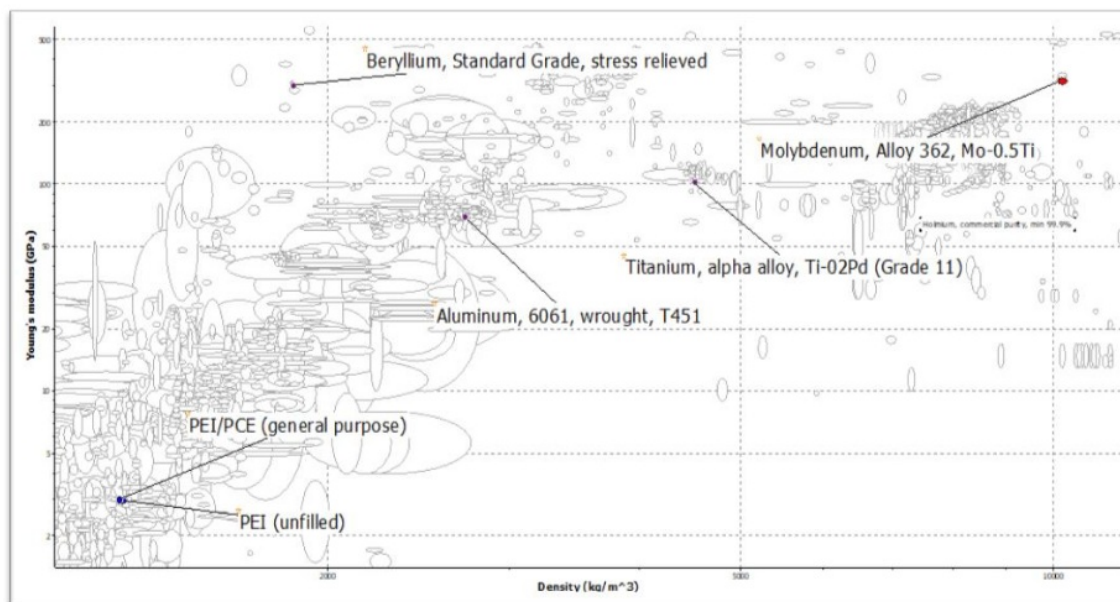


Figure 9 - Youngs Modulus vs Density Plot (CES edupack, 2018)

4 Topology Optimisation

SolidWorks has topology optimisation included in the software; this has been used to optimise the rocket bracket. Initially a static simulation closely replicating its loading scenario was undertaken to identify the stress within the component. Subsequently, a topology optimisation study was performed to remove unnecessary material but still achieve a satisfactory results in terms of part strength as specified by the challenge criteria.

4.1 Mesh Strategy

The mesh strategy adopted for the study, included an initial study utilising a coarse mesh. Mesh refinement was implemented in the identified areas of high stress concentration. This was in accordance with (Comsol, 2019) who maintain the requirement for element concentration in any area with high stress. A final mesh size of 3mm with mesh control of 1mm in areas of high stress was chosen. Since additional mesh refinement produced similar maximum Von Mises results with large increases in solve time. The quality of the mesh can be quantified, with 99.3% of elements with an aspect ratio below 3 (Figure 10). In addition, the TO within SolidWorks uses draft quality mesh as its default, with no option to choose between nodal or element analysis.

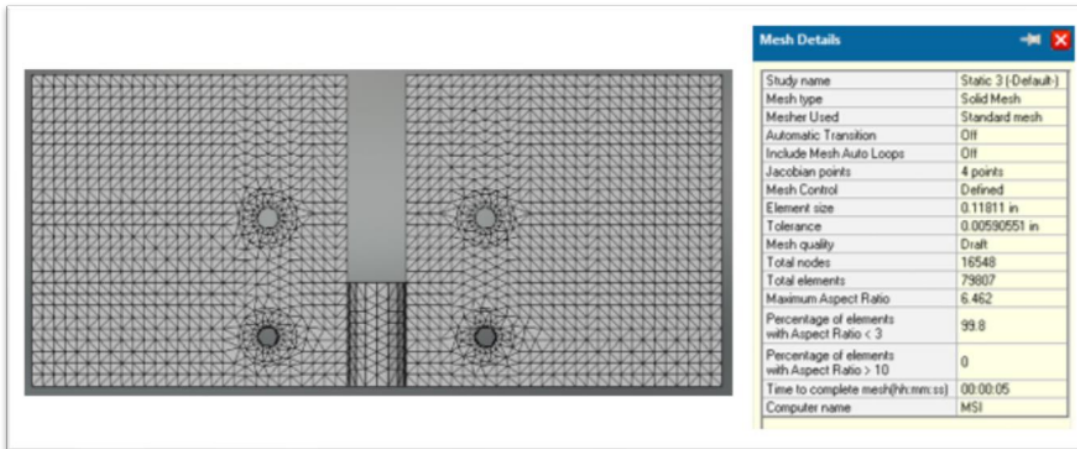


Figure 10 - Model mesh with mesh control around fastener locations

4.2 Baseline Static Study

An initial baseline simulation was undertaken to establish the nominal stresses for the design package under maximum loading conditions. This baseline will be used to establish the performance of the optimised part. (Lowrie & Ngaile, 2016) discuss the merits of establishing a baseline to evaluate performance of lower weight designs evolved during TO. The results of the baseline study can be seen in Figure 11, Figure 12 and Table 3 below.

Simulation Details	Baseline	Max Value
DOF's = 48834 Nodes = 16548 Elements = 79807 Part Mass = 0.9lb Mesh = 3mm with 1mm mesh control Solve Time = 00.31	Stress Displacement	9.8163 e06 7.036 e-02

Table 3 - Baseline Results

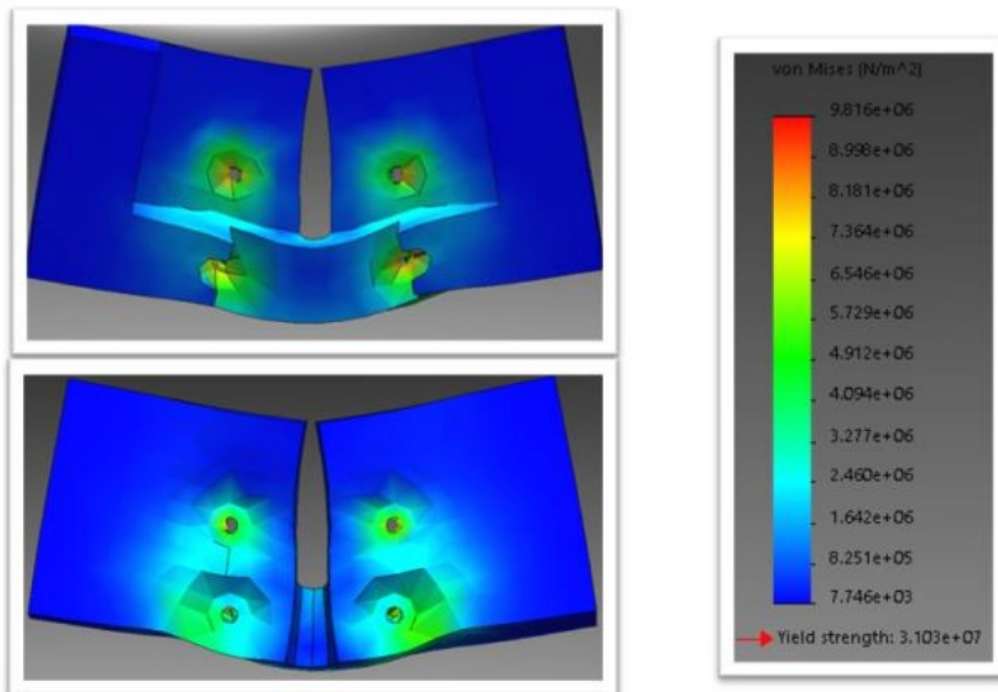


Figure 11 - Von Mises Stress Baseline

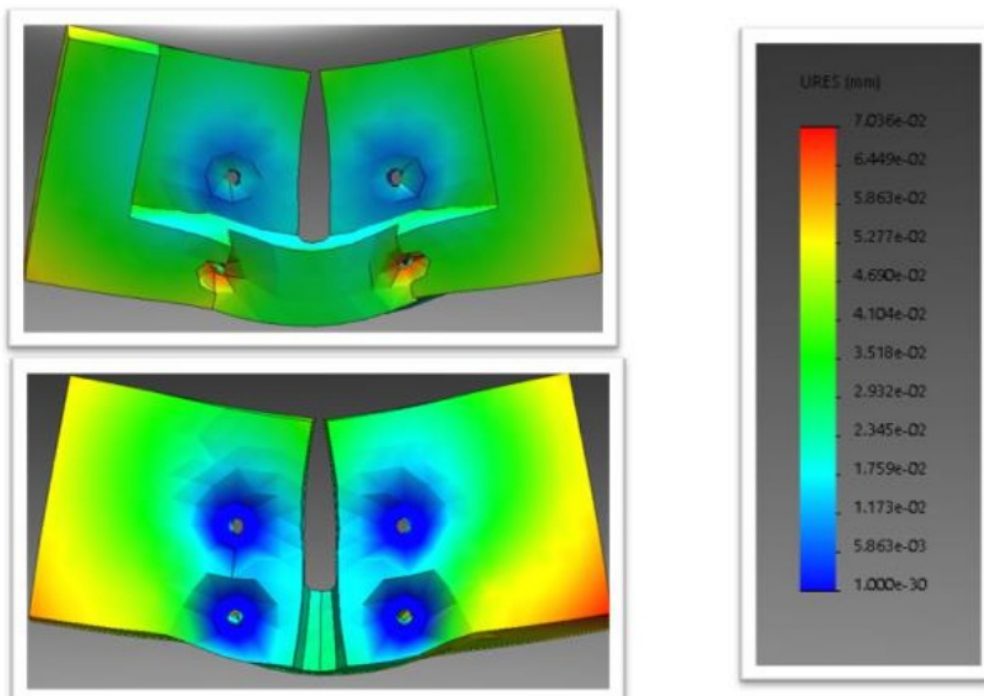


Figure 12 - Displacement Baseline

4.3 Topology Optimisation

Once a baseline was established, SolidWorks TO software was utilised to achieve a specified target mass. The aim of the study was to accomplish the required mass while maintaining the components strength/stiffness. The TO software generates load paths and identifies critical areas which must be retained while also identifying areas of material to be removed (Bodkhe & Jain, 2014) (Solid Solutions, 2016).

4.4 SolidWorks Topology Constraints

SolidWorks TO software enables the user to define 'Manufacturing Controls'. These were used to identify areas that must be retained, such as the area around the holes (Figure 13). In addition, 'Goals and Constraints' can be determined, such stiffness-to-weight ratio and mass reduction (Figure 14). Figure 15 is the colour guide generated by SolidWorks for material removal.

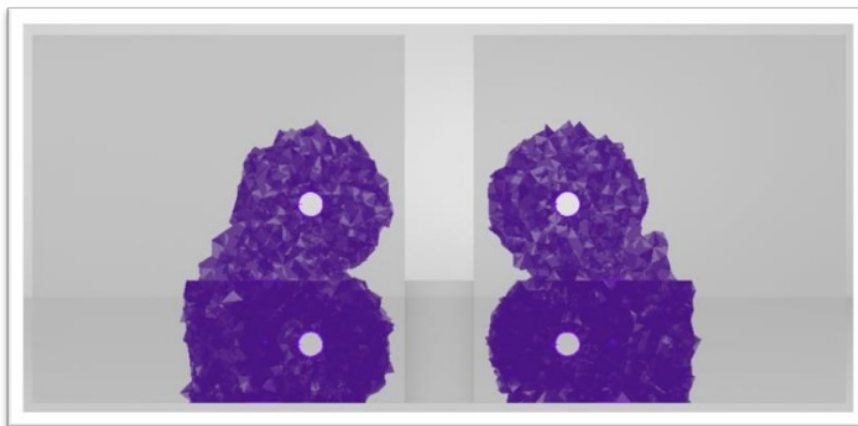


Figure 13 - Manufacturing Controls - Preserved regions

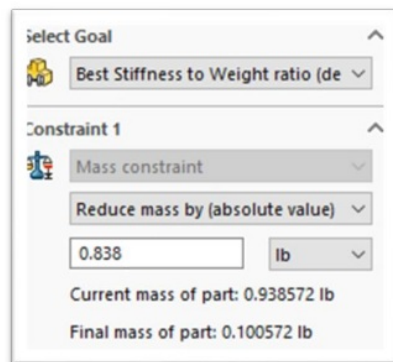


Figure 14 - Goals and Constraints - Reduce Mass

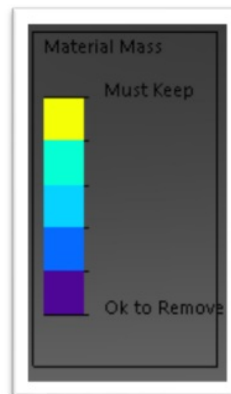


Figure 15 - Material removal guide

5 Topology Optimization Design Iterations

The final component has a required mass of 0.1lb, this is a 90% mass reduction. When the TO constraint was set to achieve this target mass, the study failed. This failure was the software's inability to converge on a solution which met all the user's constraints. This is described by (Dassult Sytems - Solidworks, 2019), "for each iteration the solver checks if either the objective or the constraint has converged to stop the solution. The numerical tolerances between subsequent iterations for reaching convergence are relaxed, and both the objective and constraint need to be satisfied for the solver to stop".

The TO software couldn't accept anything above a 70% reduction and still sustain the preserved regions. The TO result can be seen below in Figure 16. The left displays a model of suggested material removal, the right is a smoothed SolidWorks model. The TO's inability to achieve the target mass is the consequence of the inaccuracy of preserved regions. The software has retained the wall thickness around the holes through the entire part thickness.

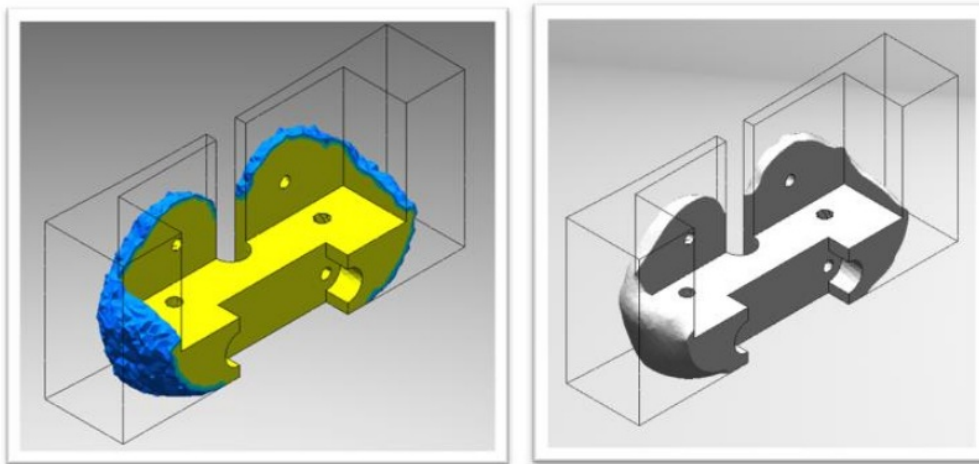


Figure 16 – Iteration 1 -70% reduction

To support the TO abilities to meet its target mass and retain the preserved regions, modifications were made to the thickness of the original design space in selected areas. The modified model has been reduced to 0.499lbs (Figure 17).

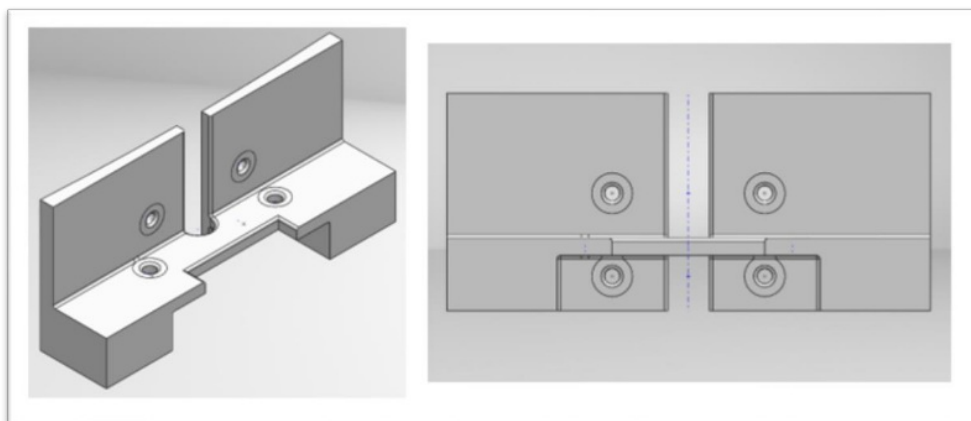


Figure 17 - Iteration 2 - Modified design

A second TO study was implemented and once again the study failed, the lowest mass achievable was 0.199lb. The modifications have clearly changed the effect of the loading conditions on the optimised shape (Figure 18).

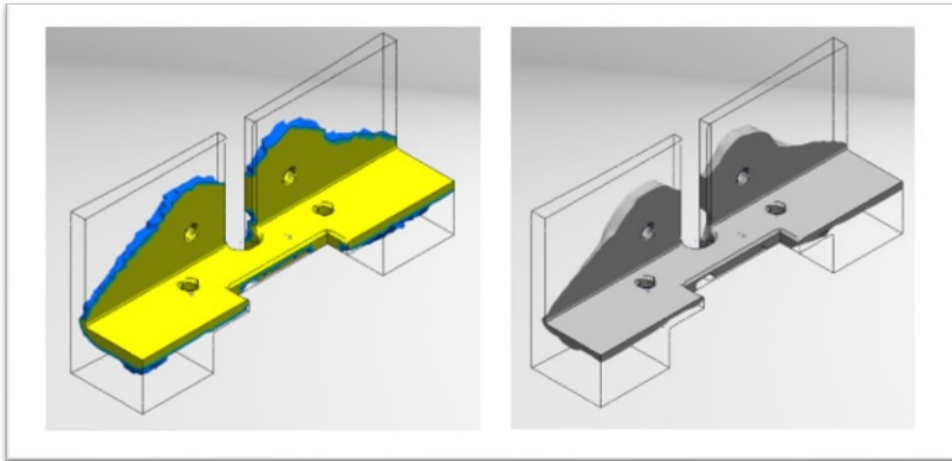


Figure 18 - Iteration 2 TO results

As the optimisation process is still unable to achieve the target mass, further modifications were made to the design. The TO results were used as a guide of what material to remove or retain, the 3rd iteration design can be seen below Figure 19, with the TO results shown in Figure 19.

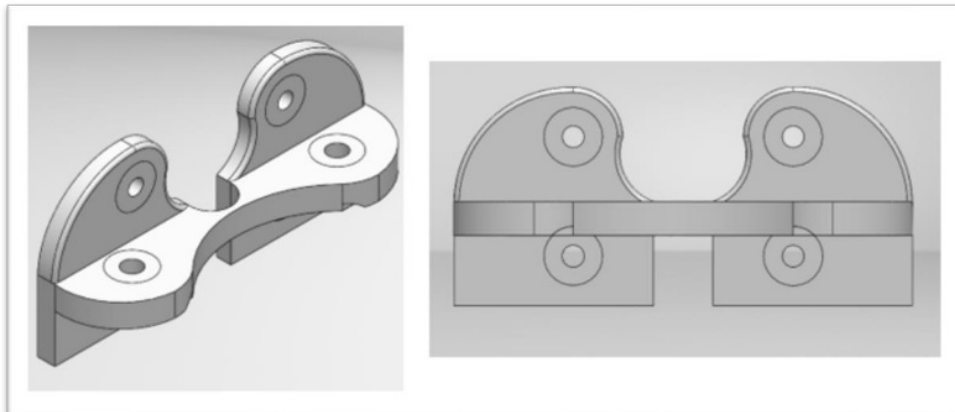


Figure 19 - Iteration 3

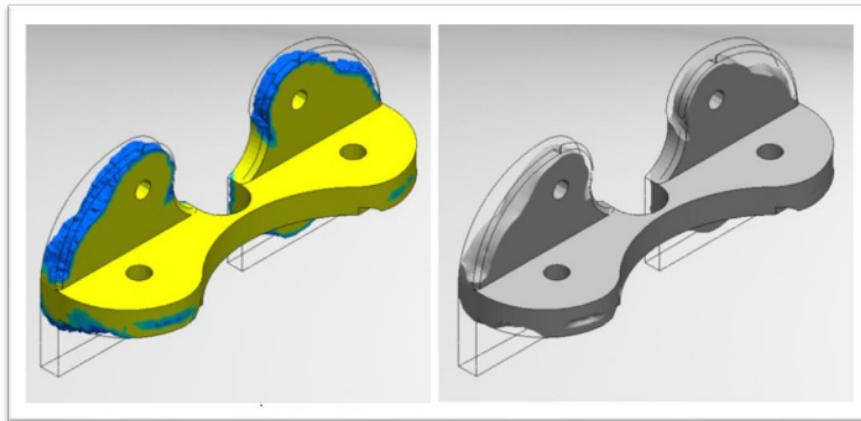


Figure 20 - Iteration 3 TO results

5.1 Final Design

After 3 iterations of topology optimisation and the subsequent design changes, a final design was produced. The design meets all the competition criteria and closely resembles the winning design. This clearly demonstrates the effect of TO on the design process. Although different software has been used, the common constraints and objectives have produced an almost identical result (Figure 21).



Figure 21 - Final design isometric view and competition winner



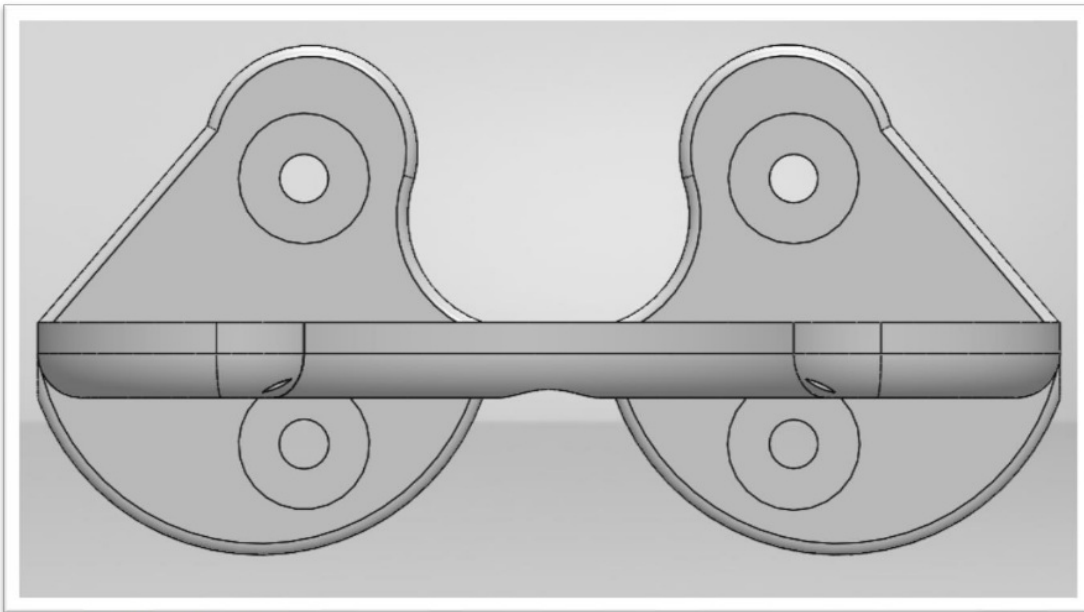


Figure 23 - Final design front view

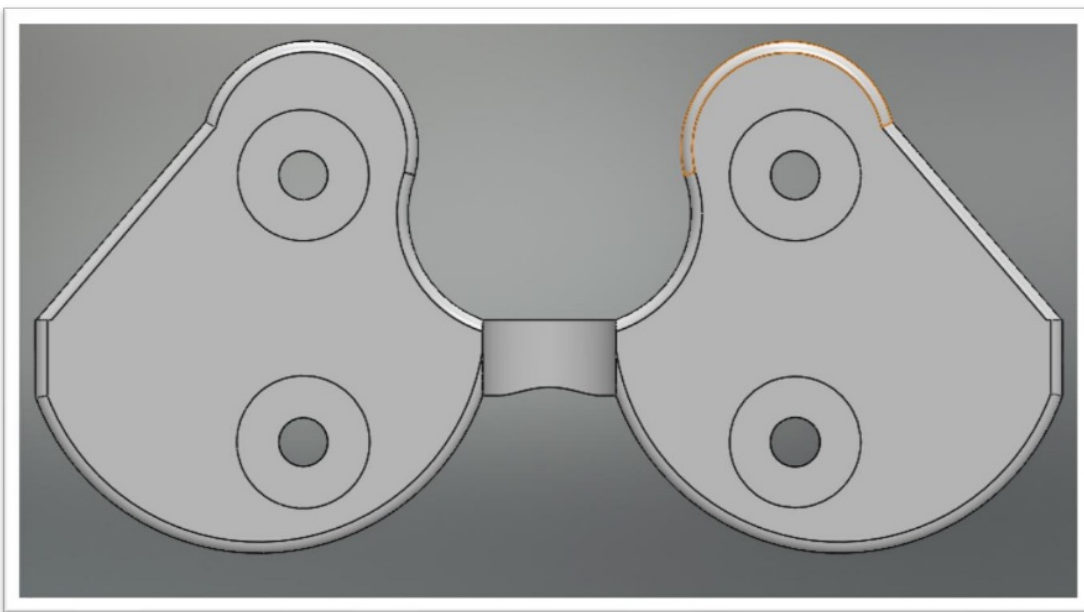


Figure 22 - Final design rear view

6 Analysis and Discussion

6.1.1 Von Mises Stress Analysis

The Von Mises stress results of the original and final designs are shown in Figure 24, Figure 26 . There are an elevated number of elements indicating higher stress values in the final design. This is expected, as the final design must endure the same loading conditions but with a 90% reduction in mass. The maximum stress in the final design (26MPa), is a 58% increase than in the original design (15.3MPa). This is 84% of the yield strength of the material (31 MPa). Therefore, any increase in load conditions could potentially exceed the elastic limit, initiating plastic deformation (Hibbeler, 2013) (Lowrie & Ngaile, 2016).

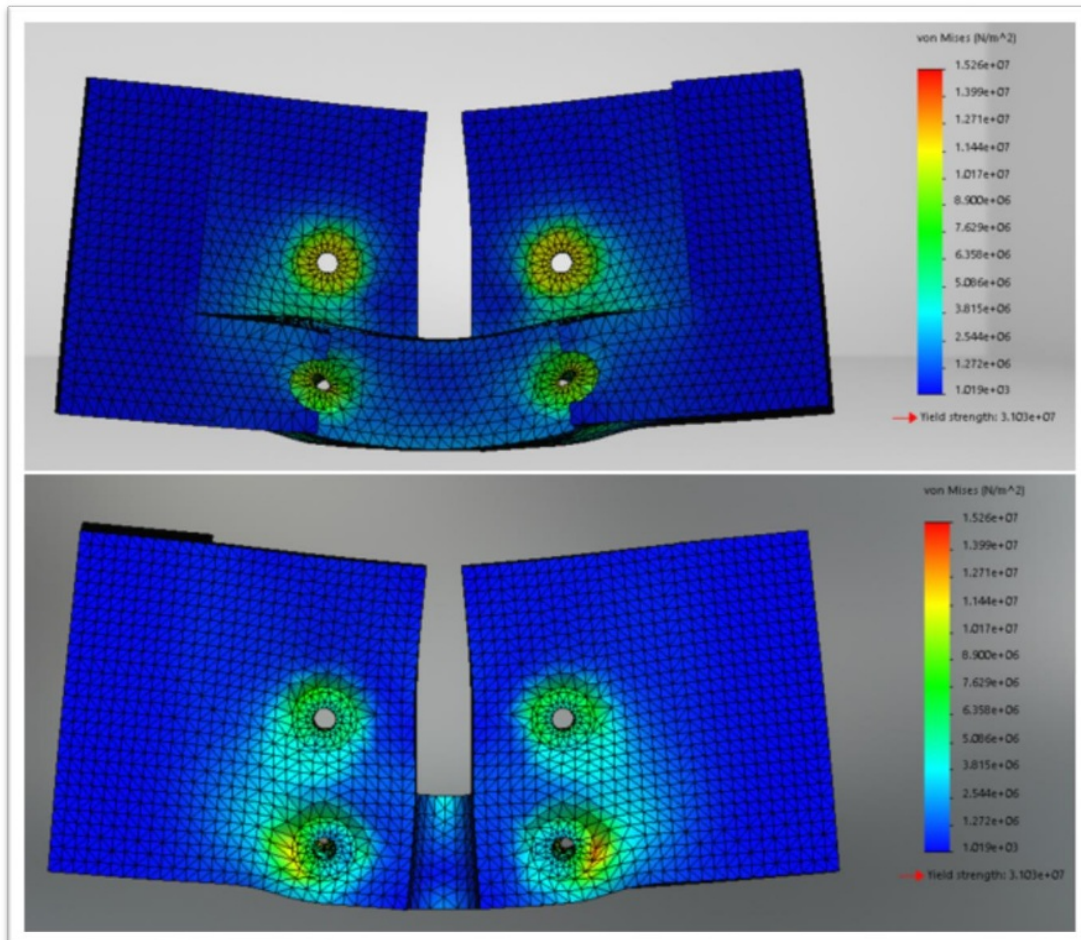


Figure 24 - Von Mises result - Original

8

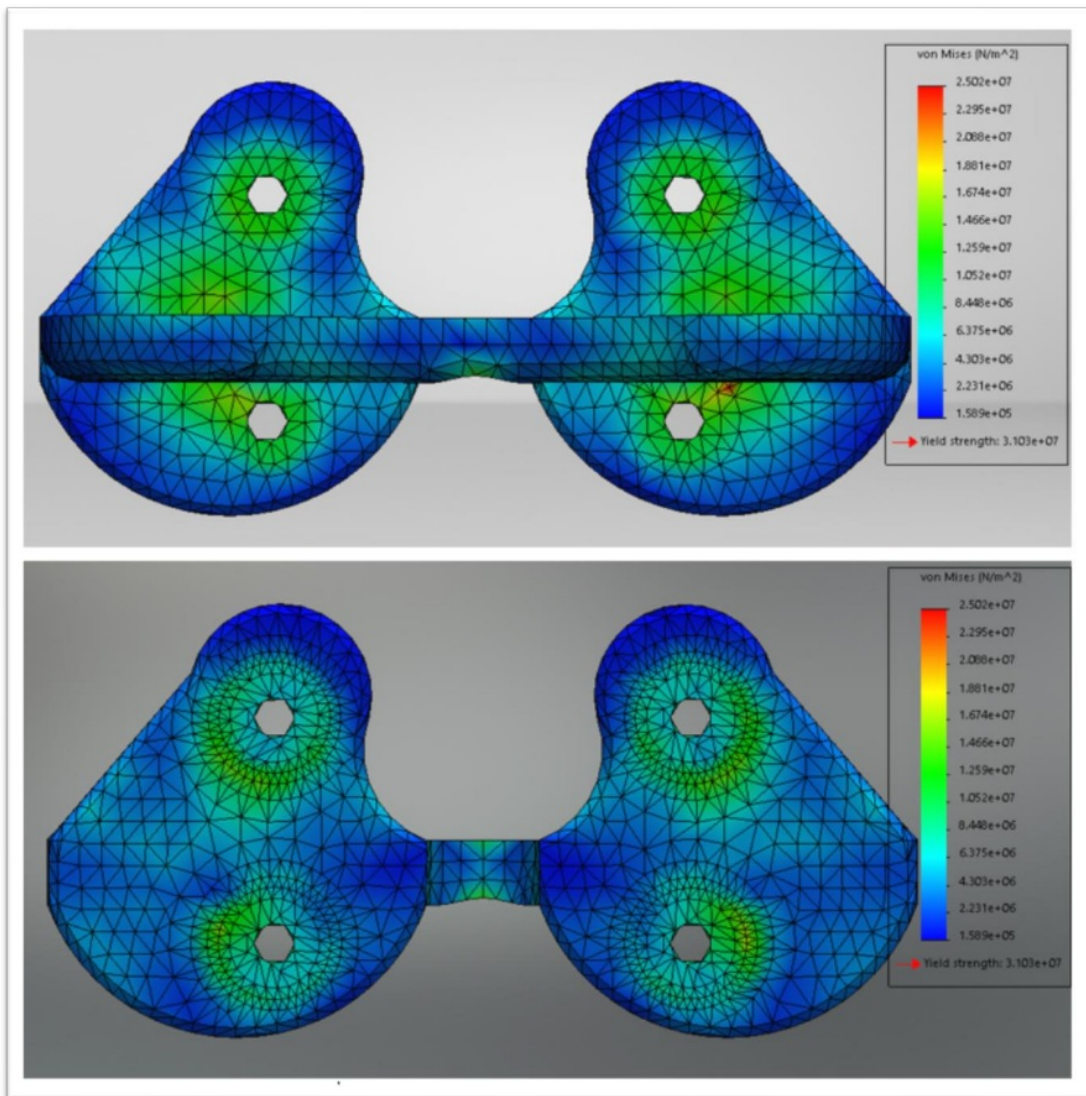


Figure 26 - Von Mises result - Final

The area of highest Von Mises stress, is located at the perimeter of one of the split lines created around the holes (Figure 25). This does not appear to be consistent with results recorded in component. Further analysis reveals suspect element aspect and shape at that location. The split lines around the other holes have more consistent elements and stress results.

9

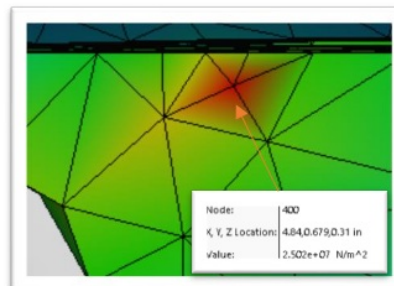


Figure 25 - Suspect mesh element

6.1.2 Stiffness to Weight Ratio

The stiffness-to-weight ratio of the bracket has been calculated by using information generated in the static studies of both baseline and final designs. The mass of the bracket has been reduced by 89.4% from 0.94lb to 0.1lb, the resultant effect of this is an 75% increase in the max displacement and a 67% reduction in its stiffness-to-weight ratio. Although an increase in max displacement and a reduction in stiffness-to-weight are undesirable, the first order effect of the components mass is of the utmost importance.

Original Design Package – Reaction Forces		
Sum X = 0.004 Sum Y = -1939 Sum Z = 9501 Resultant = 9697N	Mass = 0.94lb = 0.426kg Max Displacement = 1.176e-01	Stiffness = $\frac{9697}{1.176e-01} = 550966 \text{ N/mm}$ Stiffness/weight ratio = $550966 / (0.426 \times 9.81)$ = 131810 N kg/mm
Final Design – Reaction Forces		
Sum X = 0.008 Sum Y = -1702 Sum Z = 8858 Resultant = 9020N	Mass = 0.10lb = 0.045kg Max Displacement = 4.748e-01	Stiffness = $\frac{9020}{4.748e-01} = 18998 \text{ N/mm}$ Stiffness/weight ratio = $18998 / (0.045 \times 9.81)$ = 43079 N kg/mm

7 Study Limitations

10

7.1.1 Fixed Geometry

The FE study has predominantly generated the highest stress elements at the location of the fixed geometry (Figure 27). All fixed geometry within an FE model is infinitely stiff, due to being fixed in 6 DoFs. A more realistic simulation would have benefited from creating a larger diameter for the fixed geometry, which more closely replicated the effect of the fasteners load on the back face. This has limited the accuracy of the FE study as the effect of the fasteners load would be distributed over a larger area than just the split line created.

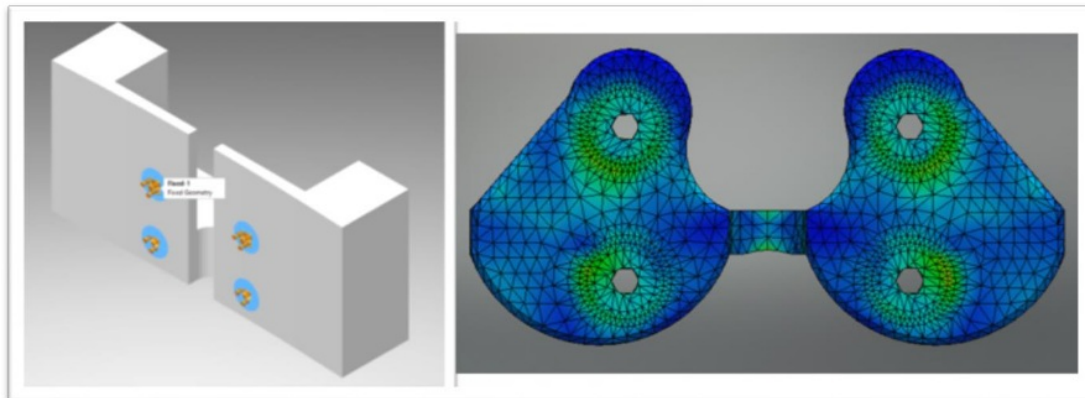


Figure 27 - Fixed geometry effect on von mises plot

7.1.2 Multiple Load Cases

This study was conducted using multiple load cases, it can be assumed the load provided by the competition criteria is accurate. But further consideration is needed of the load cases generated by the simulation's fasteners. The fasteners loads were calculated by converting known preload torques and into a simulated pressure, distributed over washer contact area. This is corroborated by (Dassult Sytems - Solidworks, 2016) who state "then nonlinear solver must be used in conjunction with bolt connectors, as a structure's dynamics response is shifted, non-linearly, by pre-loads and other background stresses".

8 Conclusions

To conclude, the topology optimisation process has clearly had a palpable effect on the design process. The final design was generated via just three design iterations, with a design that met all competition criteria. The design has a final mass of 0.10lb, 2.5Mpa maximum recorded Von Mises stress, a double hole diameter minimum wall thickness and finally, is manufacturable using standard FDM or injection moulding techniques. The design achieved an 89.4% reduction in mass, but subsequently there was a 75% increase in the max displacement and a 67% reduction stiffness-to-weight ratio. This is unavoidable, as Hooke's law dictates that any reduction in mass will decrease structural stiffness. Although these figures look detrimental, the component is utilising its mass more efficiently in resisting its loading conditions.

The power of TO as a conceptual design tool is evident from these results. Although an experienced designer could have produced a suitable design, the software has dramatically reduced the time and experience required for the design process. TO clearly reduces the trial and error in design and the influence of a designer's style. This is apparent when comparing my design to the winning design. The influence of the gradient computation method used within TO software has produced almost identical results.

However, the validity of my design is dependent on the accuracy of my TO study. The profound effect of any inaccuracy when defining fixed geometry, loading conditions and design constraints will be synonymous with my results. As discussed, my study has identified the need for additional insight for the correct application of any fastener loads.

The goal of this study was the optimisation of weight and stiffness. The organic shapes generated impart themselves perfectly to modern additive manufacturing techniques. The common constraints of conventional manufacturing techniques will no longer restrict the design process. The rapid increase of investment in these manufacturing techniques will clearly be advantageous to the uptake of TO design software.

9 Recommendations

The component study records a max Von Mises stress at 84% of the material yield strength. Therefore, further research to investigate the influence of print direction in FDM Ultem is essential, as this could be critical to the tangible performance of the component.

10 References

Adkine, A. S., Overikar, P. G. & Surwase, P. S., 2018. Static and Modal Analysis of Engine Supporting Bracket - A Literature Review. *IJMTET*, 3(11), pp. 76-83.

Aerospace.org, 2019. *3d-printing-creating-future-space*. [Online]
Available at: <https://aerospace.org/article/3d-printing-creating-future-space>
[Accessed 08 01 2020].

Atlas, 1999. *ATLAS LAUNCH SYSTEM MISSION PLANNER'S GUIDE*, San Diego: Lockheed Martin.

Bodkhe, P. & Jain, A., 2014. Design and Development of Stamping Bracket of Snowmobile Using Computer Aided Engineering. *International Journal of Engineering Inventions*, 3(9), pp. 12-21.

Calolsa, R., Ramachandran, A. & Ramachandran, N., 2013. DESIGN AND ANALYSIS OF PRESSURE CHAMBERS FOR STRUCTURAL QUALIFICATION TESTING OF THE PAYLOAD FAIRING OF A LAUNCH VEHICLE. *IJMERR*, 2(2), pp. 241-246.

CAMX – The Composites and Advanced Materials, 2015. *ADDITIVE MANUFACTURING AND CHARACTERIZATION OF ULTEM POLYMERS AND COMPOSITES*. Dallas, Nasa.

CES edupack, 2018. *CES edupack*. London: Granta.

Cherian, A., George, P. & Prabha, D. A., 2015. Response Analysis Of Payload Fairing Due To Acoustic Excitation. *INTERNATIONAL JOURNAL OF SCIENTIFIC & TECHNOLOGY RESEARCH*, 4(11).

Comsol, 2019. *Mesh Refinement*. [Online]
Available at: <https://www.comsol.com/multiphysics/mesh-refinement>

Conway, J., 2016. *Finite Element Analysis simulation of Rocket Bracket*, London: Conway.

Dassult Sytems - Solidworks, 2016. *Fasteners In FEA Dynamics Simulation*. [Online]
Available at: http://help.solidworks.com/2018/english/SolidWorks/cworks/hidd_topology_options.htm
[Accessed 12 01 2020].

Dassult Sytems - Solidworks, 2019. *Topology Options*. [Online]
Available at: http://help.solidworks.com/2018/english/SolidWorks/cworks/hidd_topology_options.htm
[Accessed 20 01 2020].

Evotech, 2019. *ULA Rocket Bracket Case Study*, London: www.evotechcae.com.

Fischer, M. & Josupeit, S., 2019. *Tensile strength: Additive manufacturing vs Injection molding*. [Online]
Available at: <https://www.sculpteo.com/en/>
[Accessed 12 01 2020].

Gebisia, A. W. & Lemu, H. G., 2017. A case study on topology optimized design for additive manufacturing. *Materials Science and Engineering*, 276(1), p. 12026.

Henson, G., 2019. *Materials for Launch Vehicle Structures*, Ohio: NASA.

Hibbeler, R. C., 2013. *Mechanics of Materials*. Ninth ed. Singapore: Pearson.

Krivanek, T. & Yount, B., 2019. *COMPOSITE PAYLOAD FAIRING STRUCTURAL ARCHITECTURE ASSESSMENT AND SELECTION*, Ohio: NASA.

- Li, C., Wang, X., Liu, M. & Li, C., 2018. Topology Optimization Design of Launcher Bracket Based on Multi-body Dynamics. *Journal of Physics: Conference Series*, 1(1), pp. 1-7.
- Lowrie, J. & Ngaile, G., 2016. *Weight Reduction Finite Element Study of Selected Power Train Components for Heavy Duty Vehicles*, North Carolina: AMTL.
- Maski, S. & Basavaraj, Y., 2015. FINITE ELEMENT ANALYSIS OF ENGINE MOUNTING BRACKET BY CONSIDERING PRETENSION EFFECT AND SERVICE LOAD. *IJRET: International Journal of Research in Engineering and Technology*, pp. 2321-7308.
- NASA, 1999. *Fastener Design Manual*, Houston: NASA.
- NASA, 2011. *Launch Environment*, Houston: National Aeronautics and Space Administration .
- Ramamurti, V., Rajarajan, S. & Venkateswara, G., 2001. Dynamic Studies of a Typical Payload Fairing for different Boat Tail Configurations. *Indian Journal of Engineering & Material Science*, 8(1), pp. 247-254.
- Sigmund, O. & Maute, K., 2013. Topology optimization approaches: A comparative review. *Structural and Multidisciplinary Optimization*, 48(6), pp. 1031-1055.
- Solid Solutions, 2016. *Truform - Topology Optimisation*. [Online]
Available at: <https://www.solidsolutions.co.uk>
[Accessed 3 January 2018].
- Stratasys, 2019. *Ultem - Polyetherimide Characteristics*. [Online]
Available at: <https://www.stratasys.com/materials/search/ultem9085>
[Accessed 10 01 2020].
- United Launch Alliance, 2010. *ATLAS V LAUNCH SERVICES USER'S GUIDE*, Littleton: ULA - United Launch Alliance.
- Watson, M. et al., 2019. *Launch Vehicle Control Center Architectures*, Houston: NASA.

FINAL GRADE

GENERAL COMMENTS

Instructor

77 /100

PAGE 1

PAGE 2

PAGE 3

PAGE 4



Comment 1

resolution

PAGE 5

PAGE 6

QM

1. Secondary(+)

Good secondary research

PAGE 7

QM

1. Secondary(+)

Good secondary research

PAGE 8



Comment 2

Use third person narrative.



Comment 3

These are all names of materials, so should start with capitals.



1. Secondary(+)

Good secondary research



Comment 4

When showing plots such as this you should show the underlying mesh so that the influence of the mesh can be seen on your results.



Comment 5

Yes, it is possible to set unachievable constraints, given the variables.



Comment 6

Yes. The Topology will only ever give design direction.



Comment 7

Good comparison of your design to the winning design.



Comment 8

You were not expected to carryout analysis on your proposed design. But your results show a part that is fairly evenly stressed which suggests an efficient structure. Well done.



Comment 9

Excellent consideration of the element shape on your results.

PAGE 21



Comment 10

Good

PAGE 22

PAGE 23

PAGE 24

GRADING FORM: ACM ROCKET BRACKET

PAUL
THOMAS

76.80

RESEARCH-INFORMED LITERATURE



15

KNOWLEDGE AND UNDERSTANDING



16

ANALYSIS



24

PRACT. APP. & DEPLOYMENT



15

SKILLS FOR PROF. PRACT.



6.8