

Andrew Curtis 1706668 ULA Rocket Hardware Challenge

by Andrew Curtis

Submission date: 26-Jan-2020 08:37PM (UTC+0000)

Submission ID: 119131327

File name:

2844_Andrew_Curtis_Andrew_Curtis_1706668_ULA_Rocket_Hardware_Challenge_231305_476110532.pdf
(1.86M)

Word count: 5711

Character count: 36545



JANUARY 17, 2020

ADVANCED COMPUTATIONAL METHODS (M6X00408)
ULA ROCKET HARDWARE DESIGN

ANDREW CURTIS (1706668)
UNIVERSITY OF WALES TRINITY SAINT DAVID
BENG (HONS) MECHANICAL & MANUFACTURING ENGINEERING
KELVIN LAKE

EXECUTIVE SUMMARY

The objective for this project was to design a support bracket for United Launch Alliance's (ULA) Atlas V rocket, as per the challenge set by GrabCAD in 2016 utilising Topology Optimisation to guide the design process. The part was first researched through relevant sources to identify any additional load cases that could be factored into the study. Unfortunately, very little information was available on the function of the part. However, from the description and guidance outlined within the challenge, and the relationship between GrabCAD and additive manufacturing, a credible technology, suitable to manufacture the part was identified as Fused Deposition Modelling, and the process was further described. The material ULTEM 9085 was specified in the challenge's guidance as the material from which the part will be manufactured from. The material was further researched and was identified as a thermoplastic that was available in an Aerospace Certified Filament, suitable for additive manufacturing. The material has a high FST rating (flame, smoke and toxicity) rating, which makes it ideal for use within the aerospace, automotive and military applications.

A conceptual study was undertaken to meet the requirements of the challenge and the finished design was successful in reaching the target weight, and compliance to all preserved regions set within the challenge were met. The finished part has been optimised to give a 'best stiffness to weight ratio', which would have provided the design with a strong structure to support the quantified load applied in the negative Z direction. The conceptual study was critically appraised and lack of software knowledge and limitations was identified that had caused unnecessary over defining of boundary conditions, which did not provide any gains to the outcome, but had increase solution time greatly.

The recommendations from this project would be to manufacture a prototype from the design and perform destructive testing such as fatigue and load testing to confirm the parts suitability for the application.



TABLE OF CONTENTS

EXECUTIVE SUMMARY	1
LIST OF FIGURES.....	3
LIST OF TABLES	3
1.0 INTRODUCTION.....	4
2.0 PROJECT BRIEF	5
2.1 PART DESCRIPTION	5
2.2 PART REQUIREMENTS / TECHNICAL SPECIFICATION	5
2.3 PART LOAD CASE	6
2.4 MATERIAL.....	8
2.5 MANUFACTURING REQUIREMENTS	8
3.0 CONCEPTUAL STUDY	10
3.1 SOLIDWORKS SIMULATION TOPOLOGY OPTIMISATION	10
3.1.1 Creating a Custom Material	10
3.1.2 Boundary Conditions.....	11
3.1.3 Mesh Refinement and Analysis.....	11
3.2 TOPOLOGY RESULTS.....	12
4.0 CRITICAL APPRAISAL	19
5.0 CONCLUSION AND RECOMMANDATIONS	19
REFERENCE LIST	21
APPENDIX A	23
APPENDIX B	26

LIST OF FIGURES

FIGURE 1: BOATTAIL SUPPORT BRACKET LOCATION	5
FIGURE 2: PART SPECIFICATION AND BUILD ENVELOPE	6
FIGURE 3: NASA GROUND MAINTENANCE PLATFORM	7
FIGURE 4: GRABCAD CHALLENGE LOAD CASE INTERPRETATION.....	7
FIGURE 5: FDM PROCESS.....	9
FIGURE 6: SOLIDWORKS MATERIAL PROPERTY MANAGER LIBRARY.....	10
FIGURE 7: TOPOLOGY OPTIMISATION ITERATIONS.....	13
FIGURE 8: PACKAGE SPACE GEOMETRY MODIFICATION FOR HOLE DEPTHS.....	14
FIGURE 9: HOLE PATTERN E. MINIMUM WALL THICKNESS BREACH	14
FIGURE 10. MESH DETAIL QUALITY CHECK.....	15
FIGURE 11: MESH QUALITY PLOT – ASPECT RATION CHECK	16
FIGURE 12: TOPOLOGY VARIABLE STRESS PLOT	16
FIGURE 13: PEAK STRESS REGIONS AROUND FIXTURES	17
FIGURE 14: MODELLED FINAL PART	17
FIGURE 15: MODEL SIMPLIFICATION	18
FIGURE 16: LINEAR STATIC STRESS ANALYSIS	18

LIST OF TABLES

TABLE 1: PART REQUIREMENTS AND GRABCAD'S JUDGING CRITERIA.....	5
TABLE 2: ULTEM 9085 MATERIAL PROPERTIES	8
TABLE 3: BOUNDARY CONDITIONS.....	11

1.0 INTRODUCTION

The objective for this project was to design a support bracket for United Launch Alliance's (ULA) Atlas V rocket, as per the challenge set by GrabCAD in 2016 utilising Topology Optimisation to guide the design process. The software selected to provide the topology optimisation was SolidWorks Simulation Professional 2018 and the machine the simulations were conducted on had an Intel®

 2 Core™ i5-3210M CPU at 2.50GHz, 6.00 GB of RAM and a 64-bit operating system.

‘Topology optimisation is a mathematical approach that optimises material layout within a given design space, for a given set of loads and boundary conditions, such that the resulting layout meets a prescribed set of performance target or predetermined design goals’ (Sudin et al, 2014).

Dassault Systemes SolidWorks Corporation introduced topology optimisation into its simulation framework for SolidWorks Simulation Professional in 2018. The software provides solutions to users with minimal inputs. Loads, design space, constraints, boundary conditions and manufacturing methods are configured to allow the software to run an iterative algorithm that supplies a near optimised part. SolidWorks provides a subtractive method of optimisation, where existing part geometry is refined using Tosca Optimisation software. The software works by creating a mesh onto a model or design space and then performing a simulation to examine the stiffness of each individual element. Any elements that offer little or no structural or manufacturing benefits are removed. The process is then iterated until all constraints and global compliance is achieved. For each element the optimisation algorithm uses the material's Young's modulus with a relative mass density factor ranging from 0.0001 (for an element without any load-carrying capacity) to 1.0 for a solid element with load-carrying capacity. Elements with a relative low density are considered soft elements that do not contribute to the stiffness of the component and are removed. High mass density elements are considered solid elements and are considered to provide most of the overall stiffness of the component. Solid elements distribute applied loads more effectively than soft elements (Engineersrule.com, 2020; SolidWorks.com, 2018a).

Topology is used to provide an initial design concept for further downstream applications, which lead to more efficient designs using computer-aided techniques. The approach creates an initial design concept which has an optimal structural layout. The benefits of the approach consist of reduced material costs, improved rigidity and mass, weight reduction. Topology provides an alternative to the expensive classical trial and error approach to design and enhances the design process early on (Sudin et al, 2014).

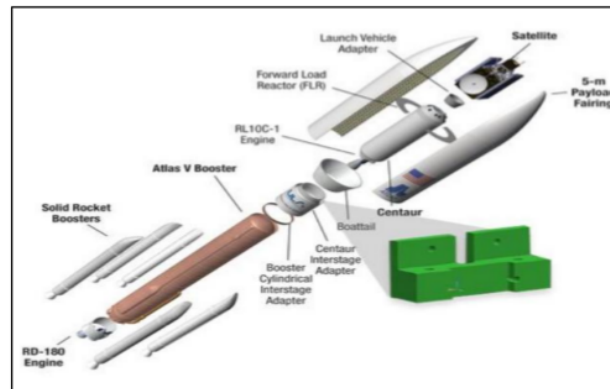


2.0 PROJECT BRIEF

2.1 PART DESCRIPTION

The Boattail bracket's function is to provide support for work platforms during ground maintenance and is fitted to multiple locations internally and around the vehicle's upper stage, as illustrated in Figure 1. The bracket will remain as part of the rocket during flight and will add to the vehicles weight, which will have a first order effect on the performance of the system (GrabCAD.com, 2016).

Figure 1: Boattail support bracket location



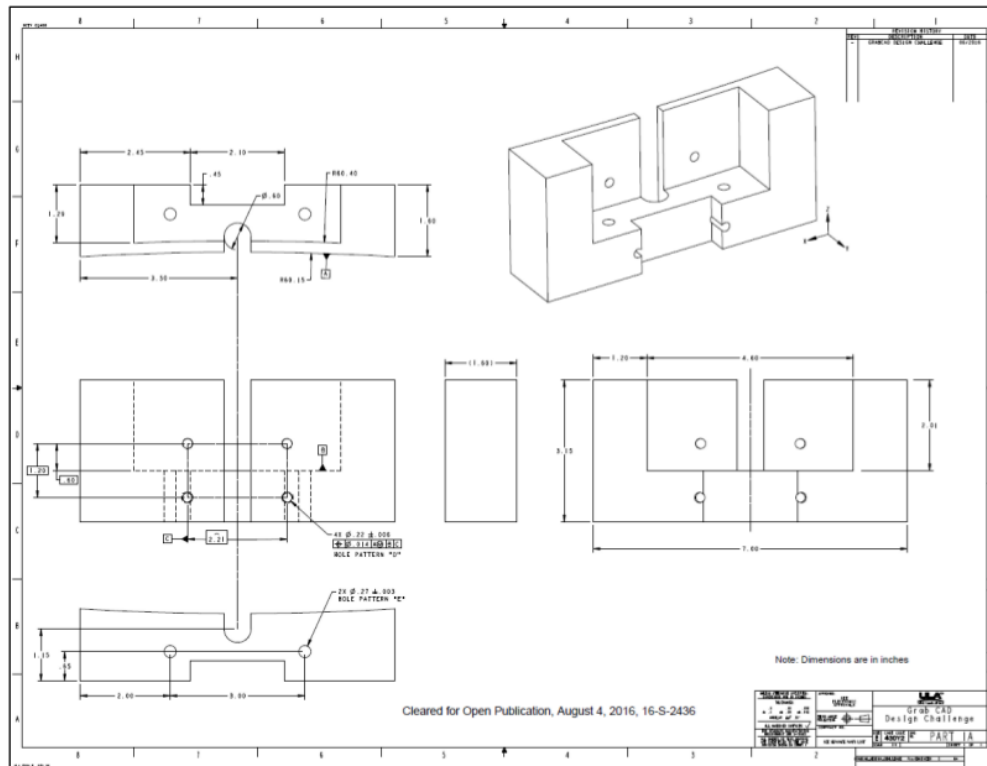
(GrabCAD.com, 2016).

2.2 PART REQUIREMENTS / TECHNICAL SPECIFICATION

Table 1: Part Requirements and GrabCAD's Judging Criteria

PART REQUIREMENTS & TECHNICAL SPECIFICATION	
1	The finished part must be contained within the part envelope illustrated in Figure 3.
2	The part will be secured to the vehicle using four bolts at Datum A and using Hole Pattern D in Figure 2.
3	The load will be applied uniformly on a surface located at Datum B and will be fastened using two bolts shown as holes (Hole Pattern E).
4	Ultimate load (due to work platform): 2668.93 N (600 lbf) applied uniformly in the negative Z direction
5	The minimum edge distance surrounding all bolt holes must be two times the diameter of the target bolt hole.
6	Required material: UITEM 9085
7	Maximum finished part weight 0.0453 Kg
8	Minimum wall thickness: 1.016 mm
JUDGING CRITERIAL	
1	Pass/Fail – meets the part design requirements
2	Minimize final part mass
3	Minimize total part material volume including print support material as measured in cubic inches
4	Unique features that leverage additive manufacturing technology
5	Use of good design practices (e.g. proper load path, sufficient radii, etc)
6	Ease of installation – takes into account tool access clearance, bolt/nut clearance, etc

Figure 2: Part Specification and build envelope



(GrabCAD.com, 2016).

2.3 PART LOAD CASE

Extensive research into any additional load cases that could be factored into the GrabCAD's challenge proved unsuccessful. However, NASA did release information pertaining to a new work platform that was to be used to access, test and process rockets prior to launch in 2015. The platform photograph



Figure 2), provided an insight into how areas around the rocket are accessed and maintained. The platforms are designed to move in and out, and translate up and down as needed to accommodate different configurations of government and other commercial rocket vehicles (Nasa.gov, 2015). The Boattail bracket details provided from the challenge presented many questions on how the bracket would support a work platform based on its geometry and size. After discovering the aforementioned platform detail, a reasonable assumption would be that the bracket only provides an anchor or attachment point to the rocket for safety, or to provide stabilisation to the work platform in adverse weather, etc. The majority of the load is supported by the platform's infrastructure. Therefore, no additional load cases were factored into the study, as these would have been further assumptions.

Figure 3: Nasa Ground Maintenance Platform

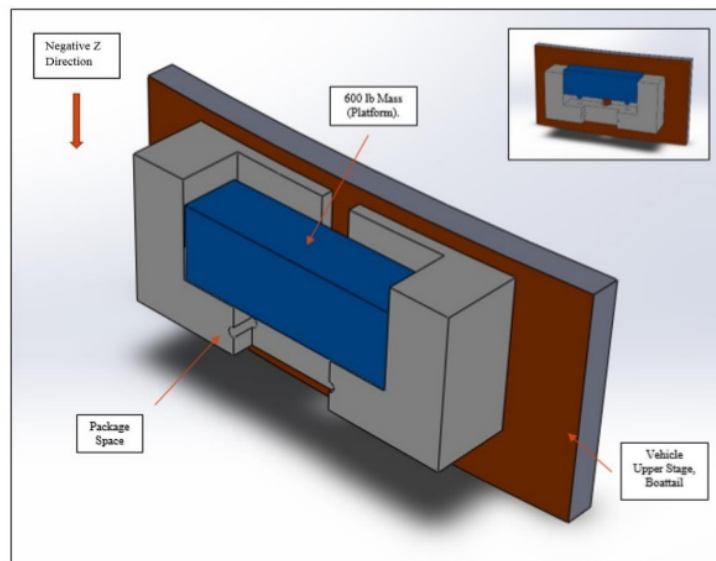


1.

(NASA.gov, 2015).

An interpretation of the package space and load case requirements based on the GrabCAD criteria is illustrated in Figure 2.

Figure 4: GrabCAD Challenge Load case interpretation



2.4 MATERIAL

The material specified for use within the challenge was ULTEM 9085, and the full material description and properties are detailed in Appendix 1. ULTEM 9085 is a flame-retardant high-performance thermoplastic used in digital manufacturing and rapid prototyping. The material has a high strength-to-weight ratio and because of its FST (flame, smoke, and toxicity) rating, the material is ideal for use within aerospace, automotive and military applications. The material is also available in Aerospace Certified Filament, which is in full compliance with this industries quality and traceability requirements. The thermoplastic expands the use of additive manufacturing into applications that require chemical and thermal resistance. ULTEM 9085 also works with breakaway support material which is required for some complex designs (Palermo, 2013; 3Dprinting.co.uk, 2020).

Table 2: ULTEM 9085 Material Properties

MATERIAL PROPERTIES	VALUES
Ultimate Tensile Stress (UTS)	69.0 MPa
Material Strength Allowable	31.0 MPa
Specific Gravity (Relative Density)	1.34 g/cm ³
Young's Modulus	2240.8 MPa
Shear Modulus	2050 MPa
Poisson's Ratio	0.41

(GrabCAD.com, 2016; 3Dprinting.co.uk, 2020).

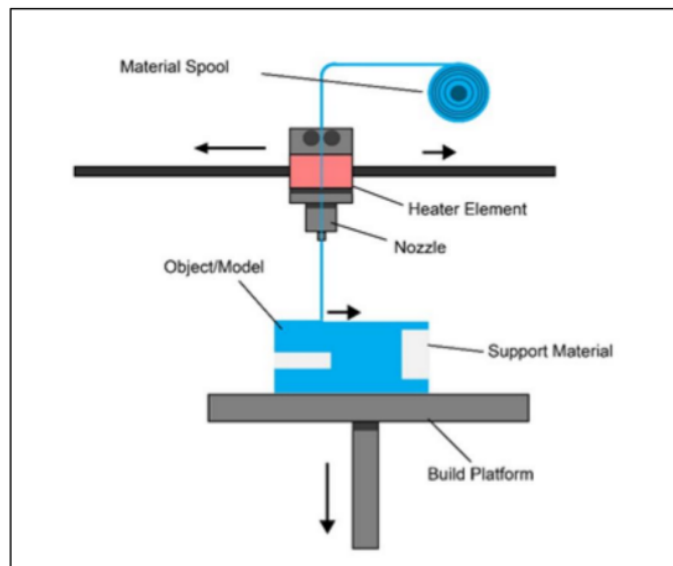
2.5 MANUFACTURING REQUIREMENTS

The original United Launch Alliance GrabCAD design challenge asked participants to design a support attachment bracket for ULA's Atlas V rocket. The challenge was promoted as an additive manufacturing competition hosted by GrabCAD. GrabCAD is a division of Stratasys Inc., which is an organisation specialising in the additive manufacturing field. The material specified for use was ULTEM 9085, which as previously stated is a thermoplastic and is available in an aerospace certified filament. Therefore, all evidence and supported sources suggest that the Boattail support bracket was to be designed for manufacture using an additive manufacturing method (Linkedin.com, 2020; Additivemanufacturingtoday.com, 2016; Marketwatch.com, 2016). The organisation Stratasys Inc. own a trademark title for an additive manufacturing technology called Fused Deposition Modelling (FDM). Which is a Material Extrusion type of additive manufacturing. To avoid infringement issues, this method of additive manufacturing also identifies as Fused Filament Fabrication (FFF). The process builds parts from thermoplastic filament material, and would be a suitable candidate to manufacture the bracket based on the package space size and material listed. The process is also

beneficial to Stratasys Inc, who may be able secure future orders for manufacturing and supplying the brackets to ULA (Lboro.ac.uk, 2020).

The FDM process involves heating and extruding thermoplastic filament through a nozzle, which is deposited onto a build platform in a layering process. This process usually utilises a Cartesian coordinate system; the nozzle moves in the X and Y directions, whereas the build platform moves in the Z direction (illustrated in Figure 3). Objects created with an FDM printer are first developed through computer-aided design (CAD) software, the CAD files must then be converted into a compatible format with the printer. Process or printing times are dependent on part size and mass, along with the complexity and precision required. Compared to other additive manufacturing methods such as stereolithography or selective laser sintering, FDM is a fairly slow process. This method is often used to produce prototypes and robust production parts, similar to stereolithography (SLA), FDM parts often require support structure, which can easily be removed post process (Lboro.ac.uk, 2020).

Figure 5: FDM process



(Lboro.ac.uk, 2020).

The advantages of FDM include the readily available acrylonitrile butadiene styrene (ABS) plastic, which provides objects or models with good structural properties. In low production quantities, this method can be more cost effective than injection molding, mainly from the reduced tooling costs. However, this process needs to be carefully controlled to produce high quality finishes. The process prevents perfectly squared corners because of the radius of the nozzle, accuracy and speed are low in

comparison to other processes, and the nozzle thickness has a direct effect on this accuracy. When models or components of high tolerances are needed, gravity and surface tension must be accounted for, typical layer thickness is from 0.178 mm – 0.356 mm (Lboro.ac.uk, 2020).

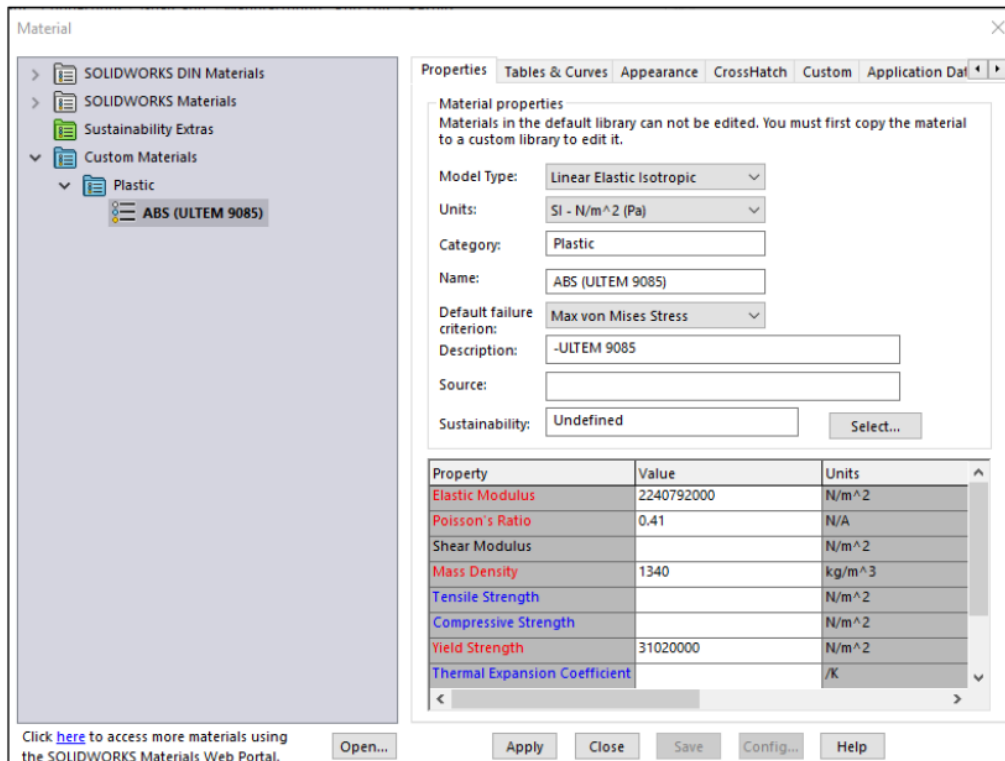
3.0 CONCEPTUAL STUDY

3.1 SOLIDWORKS SIMULATION TOPOLOGY OPTIMISATION

3.1.1 CREATING A CUSTOM MATERIAL

The SolidWorks default library does not contain material ULTEM 9085, therefore, this material was added as a custom material within the library. The material properties given within the challenge were converted to SI units and added to the relevant areas within the property manager. Only the required values highlighted in red, shown in Figure 5 were entered into the property manager, these fields must be entered to enable the material to be saved.

Figure 6: SolidWorks Material Property Manager Library



3.1.2 BOUNDARY CONDITIONS

Table 3: Boundary Conditions

LOADS AND FIXTURES	
Fixtures and loads are utilised to define the environment of the model within a simulation	
Virtual wall	This constrain was defined under the assumption that it would simulate the Atlas V payload upper stage fairing, which the bracket will be mounted on. The Virtual Wall feature restrains the model in a specific way to allow the bracket to separate from the wall, however, the model will not be unable to pass through it (Tms-scotland.co.uk, 2018).
Bolt Connector	This feature was selected to fix the model to the virtual wall. A Foundation Bolt becomes the only option within the SolidWorks Topology Bolt Connector feature, once a Virtual Wall has been created. An assumption was made that the Foundation Bolt feature would provide similar characteristics as a standard bolt arrangement, as the SolidWorks Solver will still account for stiffness and compression from pre-load through mathematical calculations. The Bolt Connector feature was favoured over a Fixed Geometry feature on the holes to provide a more accurate simulation of the loading conditions (Javelin-tech.com, 2018).
Distributed Mass:	This feature was selected to simulate the uniformly distributed load acting in the negative Z direction. The challenge states that the load applied is from an additional structure in the form of a work platform. The distributed mass feature will add a body or structure to the force applied. The distributed mass is assumed to lie directly on the selected faces, so rotational effects are not considered.
GOALS AND CONSTRAINTS	
A SolidWorks topology study presents three options when defining goals and constraints. 'Best Stiffness to Weight Ratio' is the default option and was selected for this study. The option determines the stiffest structure possible for a defined percentage of the required mass removal. The mass of the package space was 0.436 Kg and needed to be reduced to 0.0453 Kg.	
MANUFACTURING CONTROLS	
SolidWorks Manufacturing Controls enable the user to constrain features within the geometry during the optimisation process to comply with part specification, or, that if left unchecked may prevent manufacturing processes from producing the part.	
Preserved Regions	Preserved regions on all hole diameters were added to allow a minimum wall thick of twice the diameter of the target hole.
Symetry Plane	A symmetry plane was added to provide half symmetry to the optimised shape.

3.1.3 MESH REFINEMENT AND ANALYSIS

The model was meshed with linear first order elements by checking the Draft Quality Mesh option within the simulation tree. Draft Quality (first order elements) are the default SolidWorks option for a topology optimisation study. The automatic process generates a mesh with linear tetrahedral solid

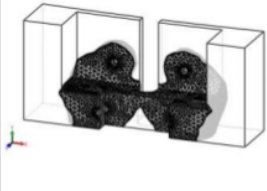
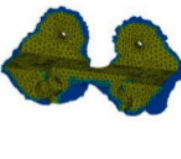
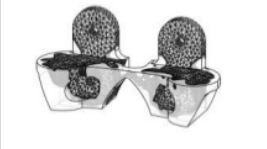
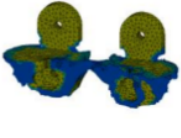
elements, which can be defined by the element having four corner nodes connected by six straight edges, which provides twelve degrees of freedom per element. Second order elements have a greater number of degrees of freedom, which increases the solution or calculation time, however, second order elements give more accurate results (SolidWorks.com, 2020b). A Curvature based mesh was selected for the FE model to create more elements in higher-curvature areas automatically without the need for mesh control. The maximum element size was set to three millimetres and the minimum size to one millimetre. The maximum element size was calculated on the principal that in bending, a part will have a region under compression, a region under tension and a region in the neutral axis. The minimum thickness within the package space was divided by three to provide more accurate information at these regions. Also, the finer the mesh, the more freedom the algorithm has to find an optimal structure. However, for the purpose of this study, the optimised model geometry was exported as a graphic body and used as guidance to provide further modifications to the model to achieve the required mass. The additional length of computational time required to solve a finer mesh and the gains received from the additional detail a finer mesh provided were not warranted for this study (Amiri, 2018).

3.2 TOPOLOGY RESULTS

Once all fixtures, loads, goals and constraints had been defined, the study was run, however, it was discovered that the software could not reduce the mass of the package space in a single iteration when preserved regions were defined within the Manufacturing Controls feature. The software was unable to solve and would fail when a 90% target mass removal goal was defined, which was the required level of mass removal to reach the target mass specified. Without defining preserved regions, the software would optimise the package space and give an end result. Unfortunately, the optimised part without preserved regions did not comply with the challenge's requirements and the minimum hole wall thickness specified in the challenge was lost. Therefore, the process adopted involved a trial and error technique of decreasing the percentage value of mass reduction, until the software was able to solve and run without failure. Once the software had optimisation the package space, the results were viewed in a Material Mass Plot. From the plot, a Smoothed Mesh graphic body was created, which was overlaid onto the model and used as guidance to modify, clean geometry and cut mass from the original package space. Once this process had been completed, a second topology study was repeated on the new part, and the geometry was cleaned with a second graphic body. The process in total took three iterations to reach the target mass value as illustrated in Figure 7.



Figure 7: Topology Optimisation Iterations

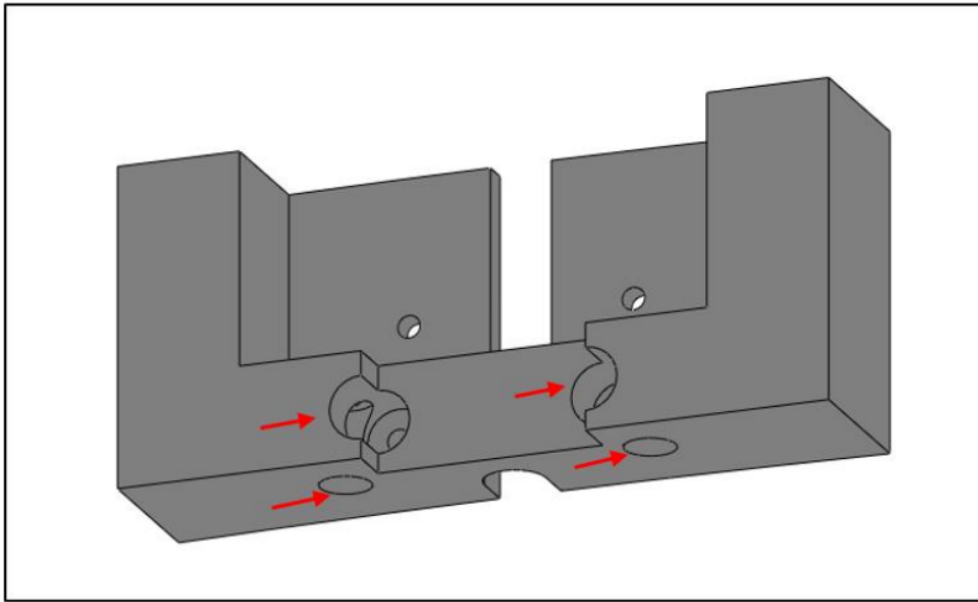
TOPOLOGY OPTIMISATION ITERATIONS			
ITERATION	PACKAGE SPACE & SMOOTHED MESH GRAPHIC	MATERIAL MASS PLOT	PARAMETERS
1			- Mesh: Draft Quality; Curvature; Max element 3 mm; Min element 1 mm 80% of package space removed - Optimised calculated element mass: 87 g - Preserved regions on loads faces - Solve time 10 mins - No. of Nodes: 20424 - No. of Elements: 99551
2			- Mesh: Draft Quality; Curvature; Max Element 3 mm; Min Element 1 mm 40% of adjusted package space removed - Optimised calculated element mass: 51 g - Solve time 6 mins - No. of Nodes: 6896 - No. of Elements: 30813
3			- Mesh: Draft Quality; Curvature; Max element 3 mm; Min element 1 mm 20% of adjusted package space removed - Optimised calculated element mass: 22 g - Solve time 1.2 mins - No. of Nodes: 2196 - No. of Elements: 9338



The package space geometry changes which made the greatest impact and allowed the software to solve and reach the target mass value, came with the realisation that by preserving the wall thickness on the holes, these preserved regions were also maintaining the depth of each hole. Therefore, model mass was being excluded from study which may have had no structural benefit. To overcome this, the original package space model geometry was modified as shown in Figure 8. The Extruded Cut features allowed adjustment to the depth of each hole, whilst still allowing the selection of the preserved regions feature for the hole wall thickness within the topology study.

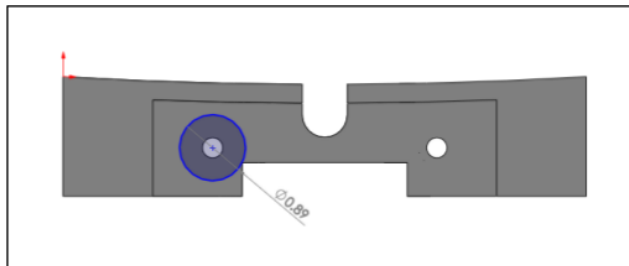


Figure 8: Package Space Geometry Modification for Hole Depths



Another important change to the model geometry which greatly affected the outcome was the relaxation to the minimum wall thickness on Hole Pattern E. As can be seen in Figure 9, the minimum wall thickness had already been breached from the initial package space. As this would have already structurally limited the maximum loading to the part by introducing a point of failure, the decision was made to maintain this dimension as the minimum wall thickness for Hole Pattern E.

Figure 9: Hole Pattern E. Minimum Wall Thickness Breach



After each iteration, care was taken to ensure that errors were not introduced through model geometry changes and remeshing. The SolidWorks' Geometry Analysis tool was utilised to check the model for potential topology problems with edges and faces, and did highlight a sliver edge which had been missed when cutting sections from the model after the second iteration. Also, as with all FE model analysis, mesh quality is paramount in obtaining accurate results. A quick way to determine the quality of the mesh is to view the Mesh Details from the Mesh Property Manager, as illustrated in

Figure 10. The values shown for the maximum Aspect Ratio should be low, and be between ten and thirty; the percentage of elements with an Aspect Ratio of less than three, should be low nineties for complex models, and upwards to ninety nine for simpler models; the percentage of elements with an Aspect Ratio of more than ten should be as close to zero as possible (Petrock, 2017).

1.

Figure 10. Mesh Detail Quality Check

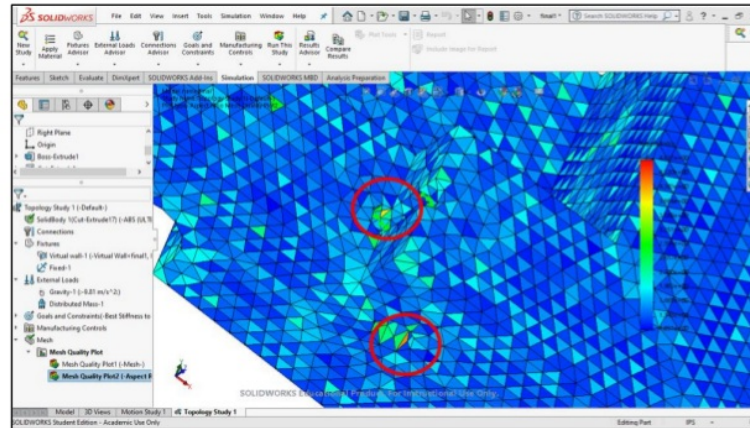
Mesh Details	
Study name	Topology Study 1 (-Default-)
Mesh type	Solid Mesh
Mesher Used	Curvature-based mesh
Jacobian points	4 points
Max Element Size	0.11811 in
Min Element Size	0.0393701 in
Mesh quality	Draft
Total nodes	2196
Total elements	9338
Maximum Aspect Ratio	17.69
Percentage of elements with Aspect Ratio < 3	98.9
Percentage of elements with Aspect Ratio > 10	0.0107
Time to complete mesh(hh:mm:ss)	00:00:03
Computer name	CURTIS-CLAN

In the event that the Mesh Details window highlights any problems with element Aspect Ratios, the Aspect Ratio Check feature can be used to identify regions which have elements that have been misshaped within the geometry. The Aspect Ratio Check looks for elements within the mesh that have edges that are not equal in length. Elements can be stretched or compressed to fit into curved geometry, thin features and sharp corners. The SolidWorks' feature identifies these problem elements, which can then be dealt with by altering the model geometry, or by refining the mesh in these areas using mesh control. Figure 11 illustrated the problem areas identified during one of the iterations before a Curvature-based mesh was used. Any errors in mesh quality may lead to high regions of stress, which would prevent the topology software from removing mass in these areas giving you an inaccurate result (Petrock, 2017).



10

Figure 11: Mesh Quality Plot –Aspect Ration Check



Once the target mass value had been reached through the optimisation process, a Topology Variable Stress Plot was created to examine the stress regions and their values (Figure 12). However, this plot only provides an estimate on how the model may behave, as the model could contain porous elements that are less rigid than fully dense elements. The plot shows that the yield point of the material has been exceeded and a peak Von Mises stress value of 37.2 MPa had been reached. However, these areas of high stress were located around the fixture points (Figure 13), which are infinitely stiff and provide meaningless stress. In real applications a body would be free to expand or contract under a loading environment, therefore, these values can be disregarded (Acin, 2015). The Topology Displacement and Strain Plots have been excluded from this report on the basis that no details or constraints were outlined within the GrabCAD challenge guidance.

11

Figure 12: Topology Variable Stress Plot

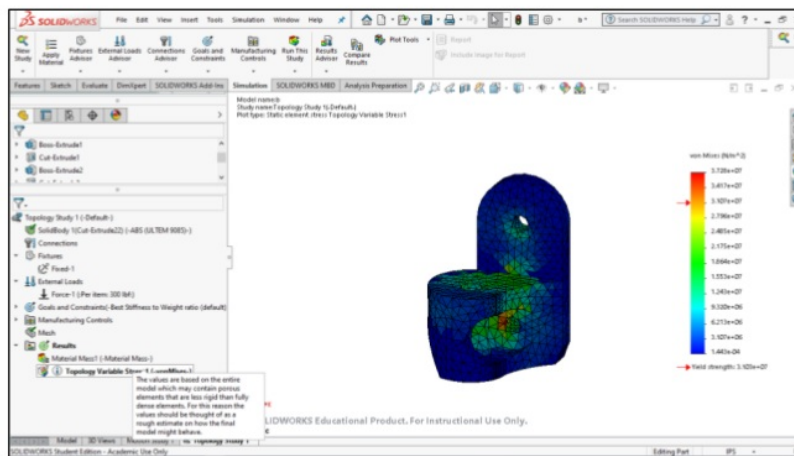
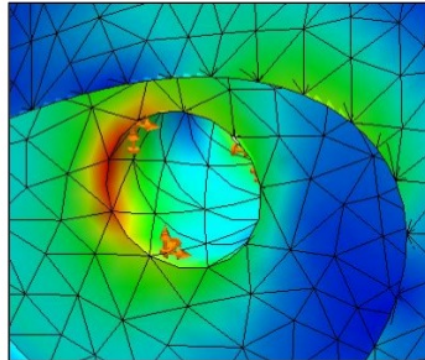
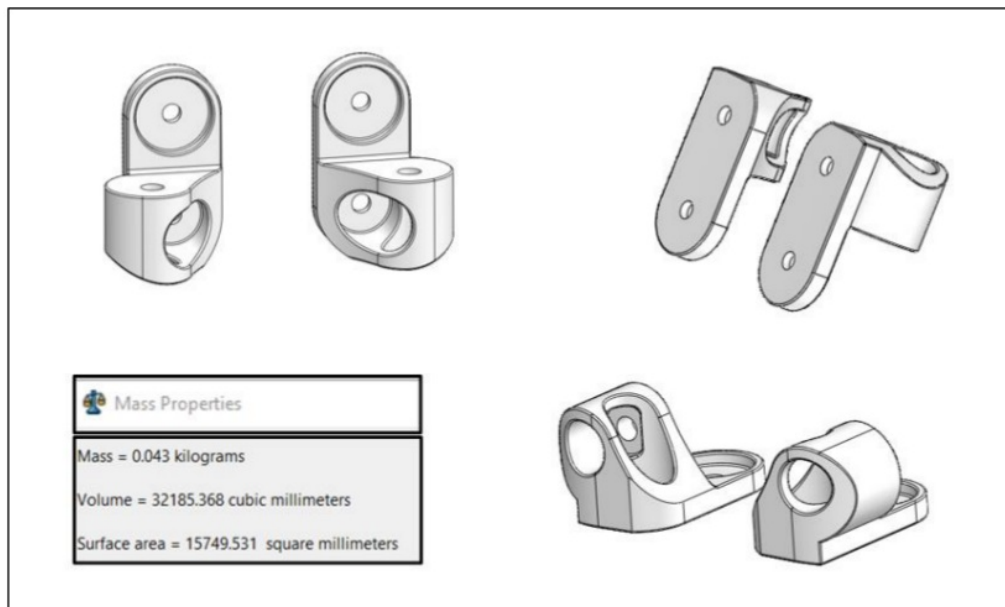


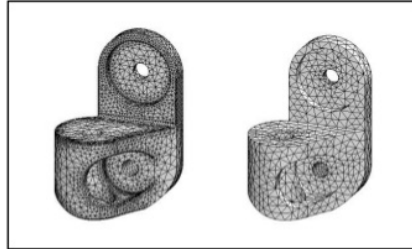
Figure 13: Peak Stress Regions Around Fixtures

Although it was not a requirement for this project, the decision was made to model the final part and perform a true SolidWorks Static Analysis Stress Plot as a comparison to the Topology Stress Plot. The final part can be seen in Figure 14, the mass from the package space has been removed following the direction from the SolidWorks Topology Optimisation algorithm, based on a best stiffness to weight ratio. The geometry has been cleansed and smoothed to remove sharp edges and fillets have been added to corners. The GrabCAD Challenge target weight was achieved and the total weight of the two paired parts was 0.043 Kg. The required wall thickness on the holes had been maintained, apart from the relaxation as aforementioned to Hole Pattern E.

Figure 14: Modelled Final Part

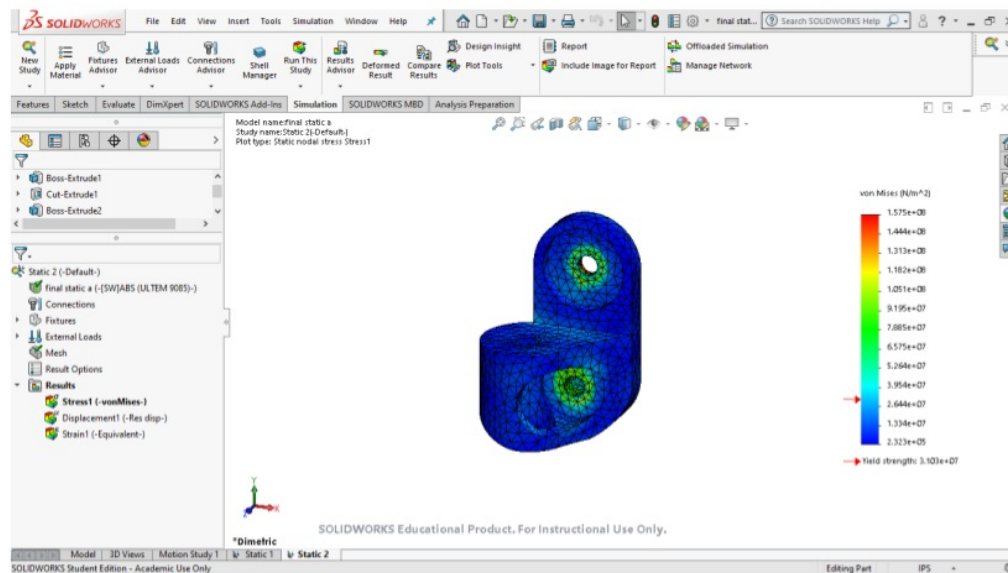
To perform the Linear Static Stress Analysis, the model was simplified and all fillets were suppressed. A Curvature based mesh with a maximum element size of 3 mm and minimum element size of 1 mm was applied to the FE model to mirror the Topology Stress Plot as shown in Figure 15.

Figure 15: Model Simplification



The peak Von Mises stress value on the Linear Static Analysis was far higher than that of the Topology plot and was 157.5 MPa. However, these peaks were once again at the fixture points and stress within the rest of the model was below that of the material yield point. The model could be further analysed within an assembly to remove the stiffness of the fixtures to reduce these high stress regions, however, for the purpose of this report, the part had met the challenge criteria and could be loaded accordingly.

Figure 16: Linear Static Stress Analysis



4.0 CRITICAL APPRAISAL

The SolidWorks Topology Optimisation process demonstrated within this report did provide a successful outcome with a suitably optimised and designed part, that met the GrabCAD Challenge's criteria. However, unfamiliarity with the software and not knowing its limitations, made this a very time-consuming project. An example of lessons learnt can be demonstrated by the attention to detail given to selecting, defining and applying complex fixtures and loads, with the aim to best simulate the actual loading environment within this analysis. The Bolt feature was later replaced for Fixed Geometry. The Simulation Wall and Gravity were deleted from the simulation tree. The Distributed Mass was replaced with a Force. As all boundary conditions were found to achieved very similar end results after progressing through the iterations, although the solution time increased greatly. To run an analysis on the original package space with a defined Force and Fixed Geometry, resulted in a solution time of 10 minutes. With a Simulation Wall, Gravity, Bolt Feature and a Distributed Mass, the solution time increase to 50 minutes. Keeping these boundary conditions simple, to quickly move through the process was the most efficient was to undertake this challenge. The final Topology iteration process illustrated in Figure 7 was a result of these lessons learnt.

Within the SolidWorks Topology Software there is an option to export a smoothed mesh solid body as a new part file. Unfortunately, this feature would not work and once the solid body was generated, SolidWorks would run a feature recognition process and fail. This could have been a lack of software knowledge, a hardware, or a software problem. However, as there was an option to continue with the topology optimisation using a graphic body and following the process previously outlined, this problem was disregarded. Although, exporting the solution as a solid body would have prevented the time-consuming model geometry changes at each iteration. This feature requires further evaluation into the benefits it may provide.

5.0 CONCLUSION AND RECOMMENDATIONS

The objective of this project has been successfully accomplished from the use of Topology Optimisation; the software was very useful in guiding the design process to meet the criteria specified within the GrabCAD Challenge. The finished design has attained the target weight and complied with all preserved regions set within the challenge. The finished part has been optimised to give a 'best stiffness to weight ratio', which would have provided the design with a strong structure to support the quantified load applied in the negative Z direction. The static analysis stress plot for the model illustrated that the loading is uniformly spread across the model and as a result of the applied load, the Von Mises stress is close to the yield point, however, it has not exceeding it. Research into other designs, namely the finalists within the competition, provided many similarities between the features

within these models and the design from this report. Two of the top five models had removed the unnecessary bridging material from the centre of the two supports, and two had very similar profiles, please refer to Appendix B.



To further conclude on the topology optimisation experience, the software has been found to be a very useful tool within the design process, but must be recognised that the analysis is only to be used as a guide. The topology Stress, displacement and strain plots are only an estimate and are not a true static analysis. The process is iterative, where assumptions, loads, fixtures and constraints can be defined and run, then re-evaluated, changed and run again until the required goal is reached.

The recommendations from this project would be to manufacture a prototype from the design and perform destructive testing such as fatigue and load testing, to confirm the parts suitability for the application.



REFERENCE LIST

- 3Dprinting.co.uk. (2020), 'Ultem 9085 3D Printing Material', [Online]. Available: <https://3dprinting.co.uk/3d-printing-material/ultem-9085/>, (Accessed: 7 January 2020).
- Acin, M. (2015), 'Stress singularities, stress concentrations and mesh convergence', [Online]. Available: <http://www.acin.net/2015/06/02/stress-singularities-stress-concentrations-and-mesh-convergence/>, (Accessed: 14 January 2020).
- Additivemanufacturingtoday.com. (2016), 'ULA Additive Manufacturing Rocket Hardware Challenge Winners Announced', [Online]. Available: <https://additivemanufacturingtoday.com/ula-additive-manufacturing-rocket-hardware-challenge-winners-announced>, (Accessed: 2 January 2020).
- Amiri, P. (2018), 'Curvature-Based Meshing'. [Online]. Available: <https://centralinnovation.com/technical-resources/learn/curvature-based-meshing/>, (Accessed: 11 January 2020).
- Engineersrule.com. (2020), 'Topology Optimisation', [Online]. Available: <https://www.engineersrule.com/topology-optimization-comes-solidworks/>, (Accessed: 5 January 2020).
- GrabCAD.com. (2016), '3-2-1 Liftoff! ULA Rocket Hardware Challenge', [Online]. Available: <https://grabcad.com/challenges/3-2-1-liftoff-ula-rocket-hardware-challenge>, (Accessed: 5 January 2020).
- Javelin-tech.com. (2018), 'SolidWorks Simulation-Bolt Connector', [Online]. Available: <https://www.javelin-tech.com/2015/01/solidworks-simulation-bolt-connector/>, (Accessed: 14 January 2020).
- Linkedin.com. (2020), 'Stratasys', [Online]. Available: <https://www.linkedin.com/company/stratasys/about/>, (Accessed: 2 January 2020).
- Lboro.ac.uk. (2020), 'Material Extrusion', [Online]. Available: <https://www.lboro.ac.uk/research/amrg/about/the7categoriesofadditivemanufacturing/materialextrusion/>, (Accessed: 12 January 2020).
- Marketwatch.com. (2016), 'ULA Awards \$10,000 to Winner for Designing Atlas V Flight Part', [Online]. Available: <https://www.marketwatch.com/press-release/united-launch-alliance-announces-winners-for-its-3-2-1-liftoff-ula-rocket-hardware-challenge-additive-manufacturing-competition-2016-12-15>, (Accessed: 2 January 2020).

NASA.gov. (2015), 'First Work Platform for NASA's Space Launch System Arrives at Kennedy Space Center', [Online]. Available: <https://www.nasa.gov/feature/first-work-platform-for-nasas-space-launch-system-arrives-at-kennedy-space-center>, (Accessed: 5 January 2020).

Palermo, E. (2013), 'Fused Deposition Modelling: Most Common 3D Printing Method', [Online]. Available: <https://www.livescience.com/39810-fused-deposition-modeling.html>, (Accessed: 7 January 2020).

Petrock, S. (2017), 'FEA Mesh Quality', [Online]. Available: <https://blogs.solidworks.com/tech/2017/08/good-fea-mesh-heres-answer-yes-no-need-know-meshing-infographic.html>, (Accessed: 11 January 2020).

SolidWorks.com. (2020a), 'Material Mass Plot', [Online]. Available: http://help.solidworks.com/2018/English/WhatsNew/c_material_mass_topology.htm, (Accessed: 5 January 2020).

SolidWorks.com. (2020b), 'Draft and High-Quality Mesh', [Online]. Available: https://help.solidworks.com/2020/English/WhatsNew/c_combine_draft_high_quality_mesh.htm, (Accessed: 11 January 2020).

Sundin, M. N., Tahir, M. M., Ramili, F. R., Shamsuddin, S. S. (2014), 'Topology Optimization in Automotive Brake Pedal Redesign', *International Journal of Engineering and Technology* (ISSN:0975-4024), vol. 6, no. 01, pp. 398-402, [Online]. Available: https://www.researchgate.net/publication/260449002_Topology_Optimization_in_Automotive_Brake_Pedal_Redesign, (Accessed: 14 January 2020).

Tms-scotland.co.uk. (2018), 'The Simulation Virtual Wall', [Online]. Available: <http://www.tms-scotland.co.uk/Article/SolidWorks/The-Simulation-Virtual-Wall/5741>, (Accessed: 14 January 2020).

APPENDIX A



ULTEM™ 9085

PRODUCTION-GRADE THERMOPLASTIC FOR
FORTUS 3D PRODUCTION SYSTEMS

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. This unique material's certifications make it an excellent choice for the commercial transportation industry – especially aerospace, marine and ground vehicles. Combined with a Fortus® 3D Production System, 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.

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

THERMAL PROPERTIES²	TEST METHOD	ENGLISH	METRIC
Heat Deflection (HDT) @ 264 psi, 0.125" unannealed	ASTM D648	307°F	153°C
Glass Transition Temperature (Tg)	DSC (SYS5)	367°F	186°C
Coefficient of Thermal Expansion	ASTM E831	3.67x10 ⁻⁶ in/(in·°F)	65.27 µm/(m·°C)
Melting Point	-----	Not Applicable³	Not Applicable³



STRATASYS.COM



THE 3D PRINTING SOLUTIONS COMPANY

(3Dprinting.co.uk, 2020).



ULTEM™ 9085

PRODUCTION-GRADE THERMOPLASTIC FOR FORTUS 3D PRODUCTION SYSTEMS

At the core:

Advanced FDM Technology

Fortus systems are based on FDM® (fused deposition modeling) technology. FDM is the industry's leading additive manufacturing technology, and the only one that uses production-grade thermoplastics, enabling the most durable parts.

Fortus systems use a wide range of thermoplastics with advanced mechanical properties so your parts can endure high heat, caustic chemicals, sterilization and high-impact applications.

No special facilities needed

You can install a Fortus 3D Production System just about anywhere. No special venting is required because Fortus systems don't produce noxious fumes, chemicals or waste.

No special skills needed

Fortus 3D Production Systems are easy to operate and maintain compared to other additive fabrication systems because there are no messy powders to handle and contain. They're so simple, an operator can be trained to operate a Fortus system in less than 30 minutes.

Get your benchmark on the future of manufacturing

Fine details. Smooth surface finishes. Accuracy. Strength. The best way to see the advantages of a Fortus 3D Production System is to have your own part built on a Fortus system. Get your free part at: stratasys.com.

ELECTRICAL PROPERTIES	TEST METHOD	VALUE RANGE
Volume Resistivity	ASTM D257	4.9 x10 ¹⁴ - 8.2x10 ¹⁴ ohm-cm
Dielectric Constant	ASTM D150-98	3 - 3.2
Dissipation Factor	ASTM D150-98	.0026 - .0027
Dielectric Strength	ASTM D149-09, Method A	110 - 290 V/mil

OTHER ¹	TEST METHOD	VALUE
Specific Gravity	ASTM D792	1.34
Rockwell Hardness	ASTM D785	---
Flame Classification	UL94	V-0 (1.5 mm, 3 mm)
Oxygen Index	ASTM D2863	0.49
OSU Total Heat Release (2 min test, .060" thick)	FAR 25.853	16 kW min/m ²
UL File Number	-----	E345258
Outgassing		
Total Mass Loss (TML)	ASTM E595	0.41% (1.00% maximum)
Collected Volatile Condensable Material (CVCM)	ASTM E595	-0.1% (0.10% maximum)
Water Vapor Recovered (WVR)	ASTM E595	-0.37% (report)
Fungus Resistance (Method 508.6)	MIL-STD-810G	Passed
Burn Testing		
Horizontal Burn (15 sec)	14 CFR/FAR 25.853	Passed (0.060" thick)
Vertical Burn (60 sec)	14 CFR/FAR 25.853	Passed (0.060" thick)
Vertical Burn (12 sec)	14 CFR/FAR 25.853	Passed (0.060" thick)
45° Ignition	14 CFR/FAR 25.853	Passed (0.060" thick)
Heat Release	14 CFR/FAR 25.853	Passed (0.060" thick)
NBS Smoke Density (flaming)	ASTM F814/E862	Passed (0.060" thick)
NBS Smoke Density (non-flaming)	ASTM F814/E862	Passed (0.060" thick)

(3Dprinting.co.uk, 2020).



ULTEM™ 9085

PRODUCTION-GRADE THERMOPLASTIC FOR FORTUS 3D PRODUCTION SYSTEMS

SYSTEM AVAILABILITY	LAYER THICKNESS CAPABILITY	SUPPORT STRUCTURE	AVAILABLE COLORS
Fortus 400mc™ Fortus 450mc™ Fortus 900mc™	0.013 inch (0.330 mm) 0.010 inch (0.254 mm)	Breakaway	■ Tan (Natural) ■ Black

The performance characteristics of these materials may vary according to application, operating conditions, or end use. Each user is responsible for determining that the Stratasys material is safe, lawful, and technically suitable for the intended application, as well as for identifying the proper disposal (or recycling) method consistent with applicable environmental laws and regulations. Stratasys makes no warranties of any kind, express or implied, including, but not limited to, the warranties of merchantability, fitness for a particular use, or warranty against patent infringement.

The information presented in this document are typical values intended for reference and comparison purposes only. They should not be used for design specifications or quality control purposes. End-use material performance can be inspected (u/i) by, but not limited to, part design, end-use conditions, test conditions, color, etc. Actual values will vary with build conditions. Tested parts were built on Fortus 400mc @ 0.010" (0.254 mm) slice. Product specifications are subject to change without notice.

¹Build orientation is on side long edge.

²Literature value unless otherwise noted.

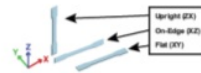
³Due to amorphous nature, material does not display a melting point.

⁴All Electrical Property values were generated from the average of test plaques built with default part density (solid). Test plaques were 4.0 x 4.0 x 0.1 inches (102 x 102 x 2.5 mm) and were built both in the flat and vertical orientation. The range of values is mostly the result of the difference in properties of test plaques built in the flat vs. vertical orientation.

XZ = X or "on edge"

XY = Y or "flat"

ZX = or "upright"



(3Dprinting.co.uk, 2020).

APPENDIX B



(GrabCAD.com, 2016).



(GrabCAD.com, 2016).



(GrabCAD.com, 2016).

Andrew Curtis 1706668 ULA Rocket Hardware Challenge

GRADEMARK REPORT

FINAL GRADE

GENERAL COMMENTS

Instructor

76/100

PAGE 1

PAGE 2



Comment 1

Good summary

PAGE 3

PAGE 4

PAGE 5



Comment 2

Good - by stating this you can refer to solve times later in the report.



Comment 3

Good explanation of Topology

PAGE 6

PAGE 7



Comment 4

Figure 3?

PAGE 8

QM

1. Secondary(+)

Good secondary research

PAGE 9

PAGE 10

QM

1. Secondary(+)

Good secondary research



Comment 5

Figure 5?

PAGE 11

PAGE 12

PAGE 13



Comment 6

Good explanation of the process that you have undertaken. An iterative approach is often required.

PAGE 14



Comment 7

Why are you only considering one side in this analysis?

PAGE 15



Comment 8

Good. You may need to modify your designable and your non-designable material regions.

PAGE 16

QM

1. Secondary(+)

Good secondary research



Comment 9

good



Comment 10

Ratio



Comment 11

Good. Unrealistically stiff boundary conditions can lead to very high stresses that would not be seen in reality.



Comment 12

Good analysis - although, as you say, this was not necessary.



Comment 13

Good comparison of your design with the designs that were chose by GrabCAD - it would be interesting to know if these winning designs used topology optimisation.



Comment 14

Good - numerical tools such as this are used to complement physical prototypes - not replace them.



Comment 15

Watch your word count.

ANDREW CURTIS

75.70

RESEARCH-INFORMED LITERATURE



15

KNOWLEDGE AND UNDERSTANDING



14.6

ANALYSIS



24

PRACT. APP. & DEPLOYMENT



15.6

SKILLS FOR PROF. PRACT.



6.5