



University of Wales Trinity
Saint David - School of
Engineering

Major project –
MCR front upright
development –
M6X00417

Ifan Hopkins – 1701085

4/05/20

Declaration of authenticity

This thesis is the result of the author's original research. It has been composed by the author and has not been previously submitted for examination which has led to the award of a degree.

Signed: Ifan Hopkins

Date: 6/03/20

Acknowledgements

I wish to express my deepest gratitude to Kelvin Lake for his continued support and guidance throughout the project and for his help learning the VR&D Design Studio for GENESIS software. Further, I would like to thank Tim Tudor for his support despite not being my supervisor. Thanks to David Cooper for his assistance whilst planning manufacture of the uprights. Thanks to the MSport Eng Team for allowing me to remove and measure the upright. Finally, thanks to my fellow student Michael Anderson for taking the time to 3D print models of the original and new upright design.

Abstract

This project aimed to improve performance of the MSport Eng race team's MCR car, by reducing mass through the development of the front suspension upright. The objective was to be able to manufacture the new upright design in-house. A target of 0.60kg weight saving was proposed, whilst maintaining or improving each suspension attachment points stiffness.

FEA studies were performed on the original upright to identify areas of the design which could be improved. Topology optimization was used to generate potential designs which would inform the new upright design.

Eight alternative designs were developed, each being lighter than the original upright design. The design chosen achieved a weight saving of approximately 0.59kg (23.4%) compared to the original upright, achieving the target saving. Most of the suspension attachment points had higher stiffness than the original upright, however, the upper and lower wishbone and steering arm were slightly less rigid in certain directions. The new upright design was a simple geometry that could be easily manufactured at the university using the CNC milling machine.

1 Contents

Declaration of authenticity.....	2
Acknowledgements.....	2
Abstract	2
1 Contents.....	3
2 Introduction	- 1 -
3 Aims and objectives.....	- 3 -
4 Timing	- 4 -
4.1 Project proposal Gantt chart	- 4 -
4.2 Final Gantt chart prior to COVID-19 pandemic.....	- 5 -
4.3 Final Gantt chart following COVID-19 pandemic	- 7 -
5 Background research	- 9 -
5.1 Finite Element Analysis	- 9 -
5.1.1 Finite Element Analysis meshing.....	- 12 -
5.1.2 Linear static analysis.....	- 15 -
5.2 Topology Optimization	- 18 -
5.3 Stress and Strain	- 21 -
5.4 Manufacturing methods.....	- 22 -
5.4.1 Forging	- 22 -
5.4.2 Casting	- 23 -
5.4.3 Additive manufacturing	- 24 -
5.4.4 CNC Machining	- 25 -
5.5 Material selection	- 27 -
5.6 Katwa's Dissertation.....	- 30 -
6 Methodology.....	- 31 -
6.1 Creating CAD of original upright	- 31 -

6.2	Attachment points	- 32 -
6.3	Topology optimization.....	- 33 -
6.4	SolidWorks FEA	- 34 -
6.5	Design Studio FEA	- 34 -
7	Results.....	- 36 -
7.1	Topology optimization.....	- 36 -
8	Analysis	- 43 -
8.1	Original upright FEA	- 43 -
8.1.1	SolidWorks FEA using bonded contact set.....	- 43 -
8.1.2	SolidWorks FEA using no-penetration contact set	- 44 -
8.2	New designs.....	- 47 -
8.2.1	SolidWorks FEA results	- 47 -
8.2.2	Design Studio Stiffnesses	- 50 -
8.2.3	Design 1 – Initial design	- 51 -
8.2.4	Design 2 – Improving upper and lower wishbone stiffness.....	- 54 -
8.2.5	Design 3 – Improving upper and lower wishbone and steering arm stiffness.....	- 57 -
8.2.6	Design 4 – Reducing weight and improving stiffness	- 59 -
8.2.7	Design 5 – Improving upper and lower wishbone stiffness	- 60 -
8.2.8	Design 6 – Improving steering arm stiffness.....	- 61 -
8.2.9	Design 7 – Alternate design language	- 62 -
8.2.10	Design 8 – Best design	- 63 -
8.3	Scaled model FEA.....	- 65 -
8.3.1	Original upright with scaled loads	- 66 -
8.3.2	New upright with scaled loads	- 67 -
8.3.3	Maximum load before yield for the original upright design.....	- 68 -
8.3.4	Maximum load before yield for the new upright design.....	- 69 -
8.3.5	6061 and 6082 aluminium scaled model stiffnesses.....	- 70 -
8.4	SolidWorks and Design Studio results comparison.....	- 72 -

8.4.1	Elemental values	- 76 -
8.4.2	Nodal values.....	- 78 -
9	Pandemic Impact.....	- 80 -
9.1	Manufacture of original and new design	- 80 -
9.2	Lack of facilities.....	- 81 -
10	Further work.....	- 82 -
11	Conclusion	- 83 -
12	References.....	- 84 -
13	List of figures	- 89 -
14	List of plates	- 92 -
15	List of tables	- 93 -
16	Appendix	- 95 -
16.1	Calculation of maximum lateral upright forces	- 95 -
16.1.1	Upper wishbone	- 95 -
16.1.2	Lower wishbone	- 96 -
16.1.3	Acceleration (Hub).....	- 96 -
16.1.4	Braking.....	- 96 -
16.2	Convergence and mesh independence study.....	- 97 -
16.2.1	Standard mesh	- 98 -
16.2.2	Curvature-based mesh	- 102 -
16.3	H-refinement mesh	- 106 -
16.4	SolidWorks FEA – Original and new design comparison	- 109 -
16.4.1	Design 1	- 109 -
16.4.2	Design 2	- 110 -
16.4.3	Design 2 updated.....	- 111 -
16.4.4	Design 3	- 112 -
16.4.5	Design 3 updated.....	- 113 -
16.4.6	Design 4	- 114 -

16.4.7	Design 5	- 115 -
16.4.8	Design 6	- 116 -
16.4.9	Design 7	- 117 -
16.4.10	Design 8.....	- 118 -
16.5	Stress distributions for original and final upright design	- 119 -
16.5.1	Acceleration load.....	- 119 -
16.5.2	Upper wishbone	- 120 -
16.5.3	Lower wishbone	- 121 -
16.5.4	Steering.....	- 122 -
16.5.5	Braking.....	- 123 -
16.6	3D printed models of original and new upright.....	- 124 -
16.7	Upright CNC quotation	- 125 -
16.8	6082-T6 Aluminium order	- 126 -
16.9	Supervision logs with Kelvin Lake	- 127 -
16.9.1	15/10/19	- 127 -
16.9.2	24/10/19	- 128 -
16.9.3	01/11/19	- 129 -
16.9.4	04/11/19	- 130 -
16.9.5	5/11/19	- 131 -
16.9.6	22/11/19	- 132 -
16.9.7	24/01/20	- 133 -
16.9.8	31/01/20	- 134 -
16.9.9	Unlogged meetings.....	- 135 -

2 Introduction

University of Wales Trinity Saint David Swansea compete in the Sports 2000 Duratec racing championship, often obtaining great results. The university use an MCR S2 race car; a full-bodied, open-cockpit, rear engine race car (see *Plate 2-1*). The championship has various regulations; therefore, many modifications are prohibited. Only OEM parts can be used in the Ford DHE 420 engine and it must have a power output of approximately 200hp.

'Minimum weight must be the weight of the car in the condition at which it crosses the finishing line, or at any time during the competition and/or practice. 521Kg without the driver.'
(SRCC SPORTS 2000, 2019)

Regulations to the chassis are unrestricted.

'Unrestricted except that the use of carbon fibre composite structural materials is prohibited.'
(SRCC SPORTS 2000, 2019)

For this reason, modifications are limited to aerodynamic, chassis upgrades and weight reduction. At Silverstone International circuit (October 5th/6th 2019) Tim Tudor finished 8th in qualifying with a time of 1.07.059, only 1.284 seconds off the leader. Since the competition is so fierce small modifications may be the difference in first or second place.



Plate 2-1 - MCR S2 race car

The MCR uses a box tubular space frame chassis with double wishbone suspension front and rear (see *Plate 2-2*). The upright connects the ball joints, wishbones, pushrods and tie rods to the wheel. It is a key component and must withstand all forces subjected to it. The upright is subjected to various forces from the tyres, brake calipers and suspension, therefore, it is an important part of the

suspension geometry. If the upright has excessive displacement when subjected to these forces it could have a negative effect on the handling dynamics of the car. Excessive displacement could lead to a change in camber and toe meaning a change in normal handling balance. Later versions of the MCR S2 have a redesigned front and rear upright, which appears to have been topologically optimized. This suggests that the current MCR uprights are over-engineered and heavier than needed. Also, the current upright does not appear to have been optimized. Since aerodynamic and chassis upgrades will be expensive, the aim of this project is to reduce weight of the car by developing the front upright. Currently, the MCR car weighs approximately 527.5kg, therefore, 6.5kg could be saved. Due to the limited development time, the OEM hub bearing and stub axle will be used.



Plate 2-2 - MCR S2 front suspension geometry

A previous student, Krishna Katwa, completed a topology study of the front MCR upright during his final major project (2018-2019) with a goal of minimising mass and maintaining stiffness.

‘With a Topology study, you can set a design goal to find the best stiffness to weight ratio, minimize the mass, or reduce the maximum displacement of a component.’

(Solidworks, 2018)

Continuing work Katwa completed last year, an optimum front suspension upright will be designed and manufactured for the MCR car. By using Katwa’s topology study, and studies conducted during this project, an improved upright design can be developed.

Wheel load data will be obtained for the MCR car, and maximum lateral loads on the upright calculated. These loads will be used to test the current and new upright designs using practical testing and Computer Aided Engineering (CAE). Finite Element Analysis (FEA) will be used to validate the new upright design based off the topology study.

3 Aims and objectives

Using Katwa's topology study and several conducted by the author, a number of optimized CAD designs for the upright will be produced. Stress analysis using FEA will be performed on the designs to ensure they are strong enough and will not deform during operation. Wheel loads will be provided for the MCR car from simulator data. The best design will be chosen, and a prototype manufactured in the university. This prototype will be tested in the workshop.

The aim is to design a new upright which is 0.6kg lighter than the original upright, whilst maintaining the stiffness of each suspension attachment point. By reducing the unsprung mass, the suspension will be more subtle, since less inertia forces need to be overcome to compress the dampers. Also, the change in damper oil temperature will be reduced meaning more consistent damping.

The aim is to manufacture the best design in the university and install to the MCR car to improve performance. To ensure the uprights can be manufactured in the university, this project will consider different manufacturing methods. The MSport Eng race team have a limited budget, therefore, costing will be considered during development. The type of material will also be considered when designing the upright as it must be affordable and manufacturable.

The author aims to gain a good understanding of Computer Aided Engineering (CAE) from the project.

4 Timing

For effective time management, a Gantt chart will be used during the project. It ensures that enough work will be completed each week to achieve the final goal. Deadlines will be set for each task, allowing progress to be reviewed throughout the project.

Weekly meetings with Kelvin Lake will be arranged, and tasks set each week (see *Appendix 16.9*).

Guidance will be provided by Kelvin Lake throughout the project.

4.1 Project proposal Gantt chart

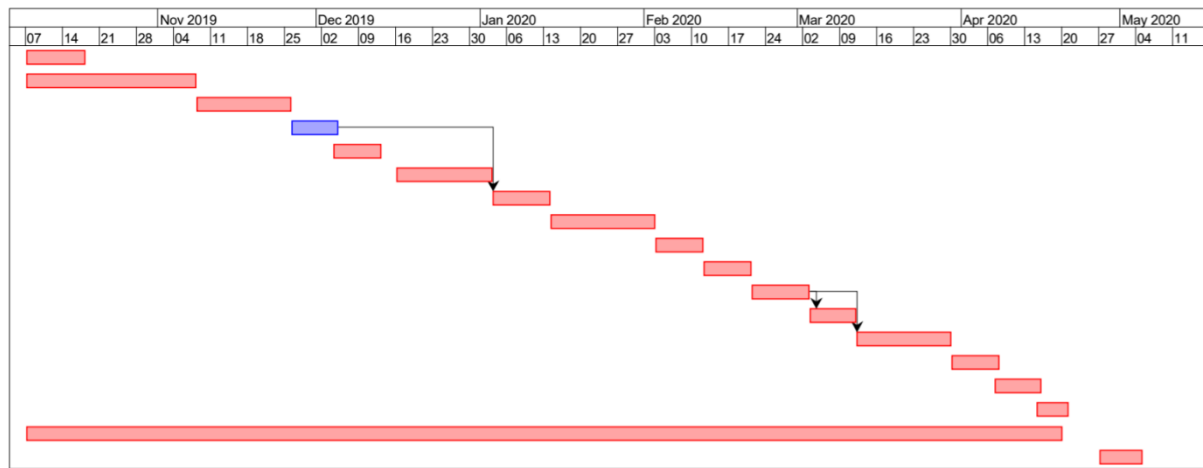


Figure 4-1 - Project proposal Gantt chart

Table 4-1 - Project proposal list of tasks and timings

		Name	Duration	Start	Finish	Predecessors
1		Project proposal	9 days	07/10/19 09:00	18/10/19 09:00	
2		Background research	24 days	07/10/19 09:00	08/11/19 09:00	
3		Analyse Katwa's topology study	12 days	08/11/19 09:00	26/11/19 09:00	
4		FEA on original upright	7 days	26/11/19 09:00	05/12/19 09:00	
5		Interim viva	7 days	04/12/19 09:00	13/12/19 09:00	
6		Christmas vacation	14 days	16/12/19 09:00	03/01/20 09:00	
7		Analyse FEA results for original upright	7 days	03/01/20 09:00	14/01/20 09:00	4
8		Christmas vacation	14 days	14/01/20 09:00	03/02/20 09:00	
9		Test and develop Katwa's Design Studio models	7 days	03/02/20 09:00	12/02/20 09:00	
10		CAD new design based off Katwa's topology and original upright FEA	7 days	12/02/20 09:00	21/02/20 09:00	
11		FEA on new upright	7 days	21/02/20 09:00	03/03/20 09:00	
12		Analyse FEA. Compare results for original and new upright	7 days	03/03/20 09:00	12/03/20 09:00	11
13		Manufacture new design	12 days	12/03/20 09:00	30/03/20 09:00	11
14		Test new design	7 days	30/03/20 09:00	08/04/20 09:00	
15		Analyse results	7 days	07/04/20 09:00	16/04/20 09:00	
16		Compare original to new upright	4 days	15/04/20 09:00	21/04/20 09:00	
17		Final report	140 days	07/10/19 09:00	20/04/20 09:00	
18		Final viva and poster	6 days	27/04/20 09:00	05/05/20 09:00	

Figure 4-1 and Table 4-1 show the Gantt chart created for the project proposal. It was proposed that the CAD model of the original upright and Design Studio for GENESIS topology optimization files would be supplied by Katwa. It was also proposed that the CAD of the original upright would be used for FEA, and that the optimization files would be developed and analysed.

Katwa was contacted on 15/10/19, however, he no longer had either file. The university's IT service desk could not retrieve the files either as they had not been saved locally on university computers. Therefore, a new CAD model of the original upright needed to be created, and only screenshots from Katwa's topology were supplied.

4.2 Final Gantt chart prior to COVID-19 pandemic

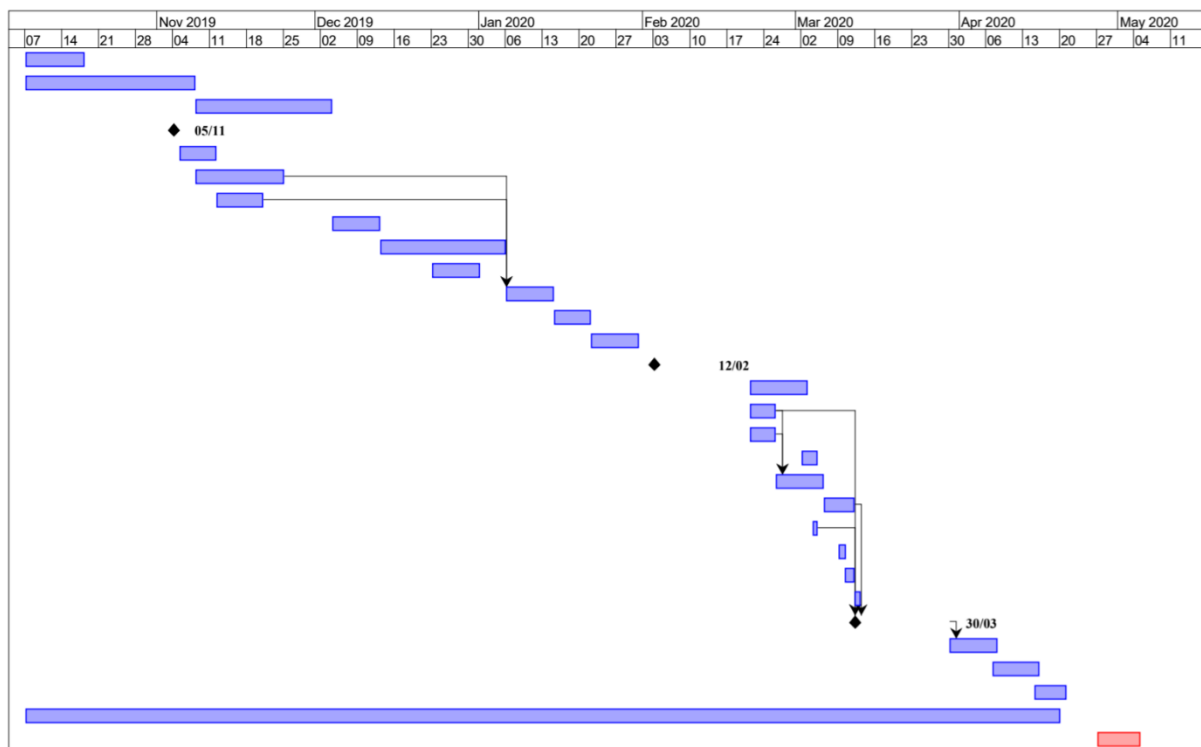


Figure 4-2 - Final Gantt chart prior to COVID-19 pandemic

Figure 4-2 shows the final Gantt chart prior to the COVID-19 pandemic. Milestones were added during the project which signifies a major progress point.

'A project milestone is a task of zero duration that shows an important achievement in a project.'
(Bunner, 2017)

The milestones within the project were:

- Create CAD model of original upright
- Create CAD model of new designs based off FEA and topology optimization results
- Manufacture scaled model of original upright and new upright

Table 4-2 - Final list of tasks and timings prior to COVID-19 pandemic

		Name	Duration	Start	Finish	Predecessors
1		Project proposal	9 days	07/10/19 09:00	18/10/19 09:00	
2		Background research	24 days	07/10/19 09:00	08/11/19 09:00	
3		Literature review	18 days	08/11/19 09:00	04/12/19 09:00	
4		Create original upright in CAD	1 day	04/11/19 09:00	05/11/19 09:00	
5		Analyse Krishna's topology Study	5 days	05/11/19 09:00	12/11/19 09:00	
6		FEA upright on Design Studio	11 days	08/11/19 09:00	25/11/19 09:00	
7		FEA upright on SolidWorks	7 days	12/11/19 09:00	21/11/19 09:00	
8		Interim viva	7 days	04/12/19 09:00	13/12/19 09:00	
9		Christmas vacation	16 days	13/12/19 09:00	06/01/20 09:00	
10		Topology optimization in SolidWorks	7 days	23/12/19 09:00	01/01/20 09:00	
11		Analyse FEA results for original upright	7 days	06/01/20 09:00	15/01/20 09:00	7,6
12		Compare Design Studio and SolidWorks FEA results	5 days	15/01/20 09:00	22/01/20 09:00	
13		Manufacturing technique research	7 days	22/01/20 09:00	31/01/20 09:00	
14		CAD new design based off FEA and topology results	7 days	03/02/20 09:00	12/02/20 09:00	
15		Mesh independence study	7 days	21/02/20 09:00	03/03/20 09:00	
16		FEA new upright on Design Studio	3 days	21/02/20 09:00	26/02/20 09:00	
17		FEA new upright on SolidWorks	3 days	21/02/20 09:00	26/02/20 09:00	
18		Calculate stiffnesses for each upright design	3 days	02/03/20 09:00	05/03/20 09:00	
19		Analyse FEA results for new uprights. Compare to original design. Choose best design	7 days	26/02/20 09:00	06/03/20 09:00	16,17
20		FEA scaled original and new upright designs using 6082-T6 aluminium	4 days	06/03/20 09:00	12/03/20 09:00	
21		Purchase materials (6082-T6 Aluminium)	1 day	04/03/20 09:00	05/03/20 09:00	
22		Manufacture 3D printed models of original and new upright design	2 days	08/03/20 09:00	10/03/20 17:00	
23		Calculate maximum load before yield for scaled original and new upright design	2 days	10/03/20 09:00	12/03/20 09:00	
24		Calculate effect of fillets on new designs stiffness	1 day	12/03/20 09:00	13/03/20 09:00	
25		Manufacture scaled model of original and new design	12 days	12/03/20 09:00	30/03/20 09:00	16,21,20
26		Practical testing on original and new design	7 days	30/03/20 09:00	08/04/20 09:00	25
27		Analyse results, comparing FEA and practical testing	7 days	07/04/20 09:00	16/04/20 09:00	
28		Compare original to new upright. Measure success	4 days	15/04/20 09:00	21/04/20 09:00	
29		Final report	140 days	07/10/19 09:00	20/04/20 09:00	
30		Final viva and poster	6 days	27/04/20 09:00	05/05/20 09:00	

Table 4-2 shows the lists of tasks and their start and finish date. Predecessors were added to some tasks which indicate to complete that task, another task requires completion first. For example, to analyse FEA results for original upright, first FEA needs to be conducted on the upright using SolidWorks and Design Studio.

4.3 Final Gantt chart following COVID-19 pandemic

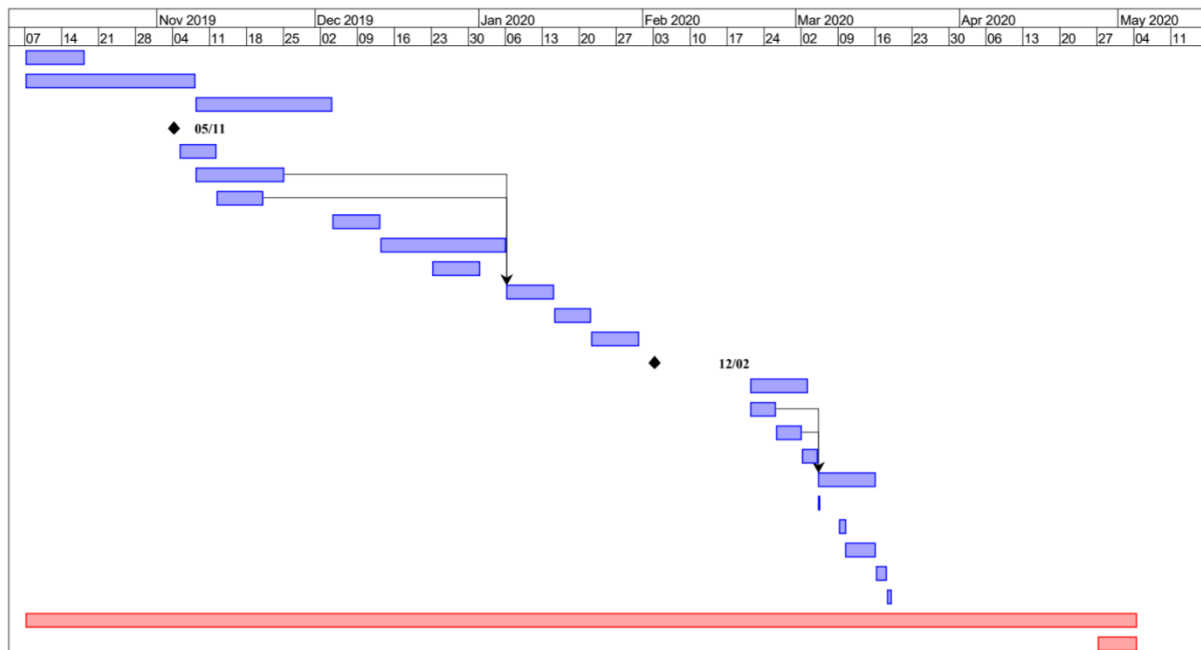


Figure 4-3 - Final Gantt chart following the COVID-19 pandemic

Table 4-3 - Final list of tasks and timings following COVID-19 pandemic

		Name	Duration	Start	Finish	Predecessors
1		Project proposal	9 days	07/10/19 09:00	18/10/19 09:00	
2		Background research	24 days	07/10/19 09:00	08/11/19 09:00	
3		Literature review	18 days	08/11/19 09:00	04/12/19 09:00	
4		Create original upright in CAD	1 day	04/11/19 09:00	05/11/19 09:00	
5		Analyse Krishna's topology Study	5 days	05/11/19 09:00	12/11/19 09:00	
6		FEA upright on Design Studio	11 days	08/11/19 09:00	25/11/19 09:00	
7		FEA upright on SolidWorks	7 days	12/11/19 09:00	21/11/19 09:00	
8		Interim viva	7 days	04/12/19 09:00	13/12/19 09:00	
9		Christmas vacation	16 days	13/12/19 09:00	06/01/20 09:00	
10		Topology optimization in SolidWorks	7 days	23/12/19 09:00	01/01/20 09:00	
11		Analyse FEA results for original upright	7 days	06/01/20 09:00	15/01/20 09:00	7,6
12		Compare Design Studio and SolidWorks FEA results	5 days	15/01/20 09:00	22/01/20 09:00	
13		Manufacturing technique research	7 days	22/01/20 09:00	31/01/20 09:00	
14		CAD new designs based off FEA and topology results	7 days	03/02/20 09:00	12/02/20 09:00	
15		Mesh independence study	7 days	21/02/20 09:00	03/03/20 09:00	
16		FEA new upright designs on Design Studio	3 days	21/02/20 09:00	26/02/20 09:00	
17		FEA new upright designs on SolidWorks	3 days	26/02/20 09:00	02/03/20 09:00	
18		Calculate stiffnesses for each upright design	3 days	02/03/20 09:00	05/03/20 09:00	
19		Analyse FEA results for new uprights. Compare to original designs. Choose best design	7 days	05/03/20 09:00	16/03/20 09:00	16;17
20		Purchase materials (6082-T6 Aluminium)	0.875 days	05/03/20 09:00	05/03/20 17:00	
21		Manufacture 3D printed models of original and new upright design	2 days	08/03/20 09:00	10/03/20 17:00	
22		FEA scaled original and new upright design using 6082-T6 aluminium	4 days	10/03/20 09:00	16/03/20 09:00	
23		Calculate maximum load before yield for scaled original and new upright design	2 days	16/03/20 09:00	18/03/20 09:00	
24		Calculate effect of fillets on new designs stiffness	1 day	18/03/20 09:00	19/03/20 09:00	
25		Final report	150.875 days	07/10/19 09:00	04/05/20 17:00	
26		Final viva and poster	5.875 days	27/04/20 09:00	04/05/20 17:00	

Figure 4-3 and Table 4-3 shows the final Gantt chart and tasks which were completed during the project. Milestones include:

- Creating CAD model of the original upright
- Creating CAD model of new designs.

Due to the COVID-19 pandemic, *'Practical testing on original and new design'*, *'Analyse results, comparing FEA and practical testing'* and *'Compare original to new upright. Measure success'* were not able to be completed.

Practical testing was going to be conducted by hanging known weights off each suspension attachment point of the original and new upright design, and displacement measured using a DTI gauge. The stiffness of each attachment point could then be calculated for both uprights. Comparing the stiffnesses calculated using Design Studio and practical testing, the same increase in stiffness for each attachment point would be expected.

Conducting no practical testing had a large impact on the project as it meant that the FEA results could not be verified, meaning the validity of the results was unknown. Despite the FEA results showing the correct trends, it is unknown whether the solver results and reality correlate. FEA makes various assumptions during a study, therefore, the solver results and reality may have large discrepancies in results. Had practical testing been achieved, the FEA results could have been more reliable, and the conclusions made during this project been further supported by physical tests. See *Chapter 9* for more information on the COVID-19 pandemic impact.

Since practical testing could not be achieved, SolidWorks and Design Studio results were compared to determine if both solvers led to similar results. Despite not being as robust as verifying with practical testing, it increased confidence in the results.

5 Background research

5.1 Finite Element Analysis

Linear static Finite Element Analysis (FEA) was used to test the original upright design and new designs. FEA is a useful tool used during the project to highlight areas that could be improved of each design. Background research was conducted on FEA to gain a better understanding of the tool. This meant its use during the project was more effective.

FEA is a modelling technique which was developed from the numerical technique Finite Element Method (FEM). FEA can be used to simulate various physical phenomena. FEM was developed by Alexander Hrennikoff and Richard Courant with a publication of papers in the 1940s. Different approaches were used by both; however, they shared an essential characteristic;

‘mesh discretization of a continuous domain into a set of discrete sub-domains, usually called elements.’
(Crahmaliuc, 2019)

In the early sixties the aviation and aerospace industry adopted the use FEA. NASA developed the earliest commercially available software, NASTRAN. Both SolidWorks and Design Studio are based off the NASTRAN software. Today, FEA is used in all sectors of industry.

According to Reif et al. (2018), firstly a structural problem is defined within a geometry. The geometry is discretized (meshed) into finite elements. This involves replacing the complex body with a number of smaller simpler elements, each permanently connected to each other at their corners (‘nodes’).

Mirzendehtdel and Suresh (2017), state that an element stiffness matrix is constructed for each element; a global stiffness matrix is then assembled from these element matrices. To find the unknown field, the resulting system of equations are solved. Finally, to compute other derived fields, post-processing is carried out.

Figure 5-1 shows a typical FEA workflow, showing the progression from the beginning stage, structural problem to the final stage, post-processing.

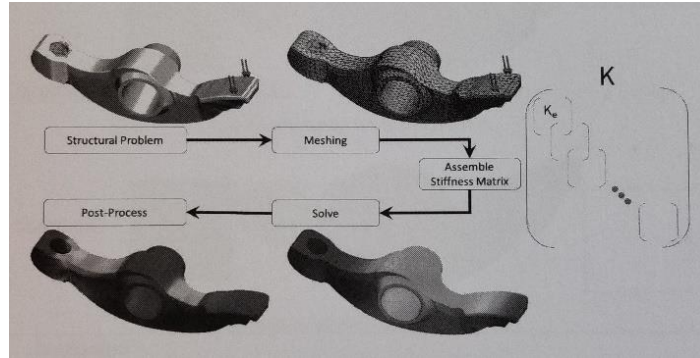


Figure 5-1 - Typical finite element analysis workflow (Mirzendehtel & Suresh, 2017)

Once the mesh has been produced, a stiffness matrix must be calculated at each node in order to calculate the unknown.

$$Ku = f$$

Where:

K = Resulting matrix

u = Unknown nodal field distribution

f = Global force vector

'The number of rows (or columns) in K is referred as the degrees of freedom (DOF) for the problem and it can vary from a few thousand, to millions, depending on the density of the mesh.'

(Mirzendehtel & Suresh, 2017, p. 22)

As explained by Weck and Kim (2004), the resulting matrix, K , is described as the property of the physical problem for example the stiffness of the upright. The unknown nodal field distribution, u , is described as the behaviour (the unknown) for example the displacement of the upright. The global force vector, f , is described as the action for example the force.

The equation above is solved by either a direct or an iterative solver. Methods for solving the stiffness matrix include, Direct method, Variational method and Weighted Residual method.

According to Akin (2009), the Galerkin method and the Method of Least Squares are the two most common Weighted Residual methods.

Displacements are calculated at each node, therefore, the greater the number of elements, the more nodes, meaning a higher accuracy of approximation. With a larger number of elements, more Gauss points are present, which are used to calculate stresses, resulting in greater accuracy.

Figure 5-2 represents how element size affects results, as the element size decreases, the accuracy increases.

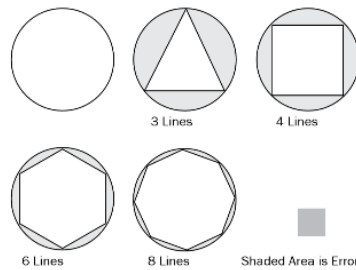


Figure 5-2 - Number of straight lines is equivalent to mesh size in FEA (Altair University, 2015)

FEA expresses the physical problem as a mathematical model, which leads to differential equations. FEA is used to solve this mathematical model. For accurate results it is imperative that the mathematical model completely defines the physical phenomena.

There are two different approaches to solve a problem; continuous approach and discrete approach. A discrete problem has a finite number of Degrees Of Freedom (DOF), whereas a continuous problem has an infinite number of DOF. All real-life problems are continuous, and all problems solved using FEM follow the discrete approach.

‘Meshing (nodes and elements) is nothing but the discretization of a continuous system with infinite degrees of freedom to a finite degrees of freedom.’

(Altair University, 2015)

According to Zienkiewicz et al. (2013), for a discrete problem, a finite number of well-defined components can be used to obtain an adequate model. The problems can be defined using differential equations or equivalent statements. Different methods of discretization have been proposed in order to solve these continuous problems, however, all are approximations.

FEA is used to evaluate designs and can lead to better designs created in a shorter time. FEA allows engineers to manufacture fewer physical prototypes and optimize the design within the design space.

‘The analysis process helps to identify good new designs and can be used to improve a design with respect to performance and cost.’

(Bathe, 2016, p. 12)

Areas of potential failure of a design can be highlighted more efficiently using FEA compared to physical testing.

'It can also be difficult to interrogate the results of physical testing.'

(Revellat, 2019)

5.1.1 Finite Element Analysis meshing

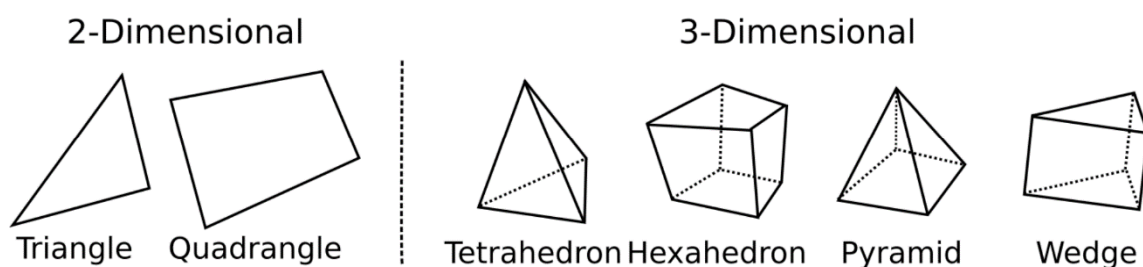


Figure 5-3 - Common element types used in FEA (SimScale, n.d.)

Figure 5-3 shows some of the element types used in FEA. Often tetrahedral elements are used for three-dimensional geometries; however, each type have their advantages. Higher order elements result in higher accuracy but require much higher computational power.

Element	Name	Dimension	# Nodes	Interpolation
	Tetrahedral	3D	4	Linear
	Hexahedral	3D	8	Linear
	Quadratic tetrahedral	3D	10	Quadratic
	Quadratic hexahedral	2D	20	Quadratic

Figure 5-4 - Common element types used in FEA (Mirzendehtdel & Suresh, 2017)

The SolidWorks mesher was used for both SolidWorks and Design Studio studies. For three-dimensional geometries SolidWorks 'draft quality' mesh generates linear tetrahedral elements each with four nodes. For three-dimensional geometries the 'high quality' mesh generates quadratic tetrahedral elements each with ten nodes, improving the accuracy of approximation (see Figure 5-4).

'A linear tetrahedral element is defined by four corner nodes connected by six straight edges. A parabolic tetrahedral element is defined by four corner nodes, six mid-side nodes, and six edges.'

(SolidWorks, n.d.)

The 'high quality' mesh represents curved geometry more accurately than 'draft quality' mesh, therefore, for accurate studies a 'high quality' mesh was used (see *Figure 5-5*).

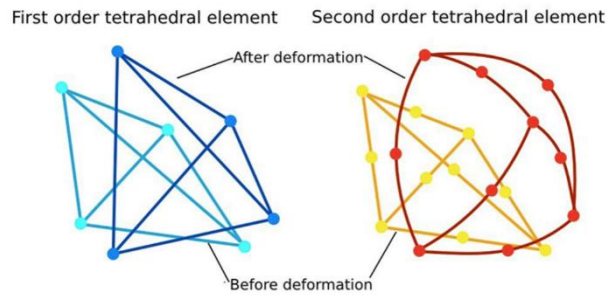


Figure 5-5 - Deformation of a first order and second order tetrahedral element (Murad, 2019)

SolidWorks has the option of 'standard mesh' or 'curvature-based mesh'. The 'standard mesh' uses the unstructured Delaunay method. The faces of the three-dimensional geometry are meshed first with triangles, and the software attempts to mesh with equilateral triangles of the specified element size.

'It makes use of Voronoi-Delaunay triangulation, which tries to maximize the smallest angle of all triangles, while the geometry is being split into element triangles or tetrahedrons.'

(Hawk Ridge Systems, 2013)

'Tolerance' is used to knit surface meshes into a solid, and this tolerance defaults at 5% of the global element size. Following this, the volume is filled with tetrahedrons. Since the mesher endeavours to maximise the smallest angle of the triangles, this reduces the number of elements with a high aspect ratio, which can have a negative effect on results.

'...the elements should be relatively "round" in shape, because elements with large or small angles can degrade the quality of the numerical solution.'

(Shewchuk, 1999)

The 'curvature-based mesh' creates smaller elements in high-curvature regions of the geometry and allows for multi-threaded meshing for assemblies. The 'standard mesh' only supports multi-core volume meshing, therefore, the 'curvature-based mesh' is often quicker to mesh. As explained by

Hawk Ridge Systems (2013), the 'curvature-based mesh' determines the element size mathematically by taking into consideration the user specified maximum and minimum element size and the minimum number of elements that fit into a hypothetical circle. SolidWorks default value for the minimum number of elements in a circle is 8, and this was used during the project. The mesher will create 8 elements within the circle, provided that the value of L is within the specified maximum and minimum element size (see *Figure 5-6*).

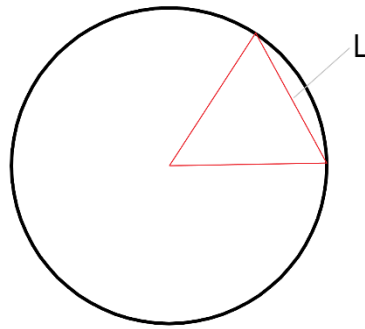


Figure 5-6 - Curvature based mesh element generation

Both types of meshes have advantages and disadvantages, therefore, both were used during the project.

5.1.2 Linear static analysis

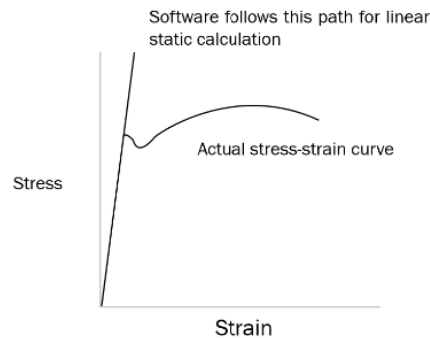


Figure 5-7 - Stress-Strain path for linear-static FE solver (Altair University, 2015)

Linear static analysis was used when testing the uprights. Altair University (2015) explains that to satisfy the linear condition; from base to deformed state the FE solver will always follow a straight line. The equation for the straight line seen in *Figure 5-7* is:

$$\sigma = \varepsilon E$$

Where:

σ = Stress

ε = Strain

E = Young's Modulus

After crossing the yield point for the material, the solver continues to follow the linear line, despite the material following the non-linear curve. After crossing the ultimate stress point the material will break into two parts, however, the FE solver will not show this failure. Linear deformation will be assumed when performing FEA on the upright, since plastic deformation cannot occur. This method is quick and cheap.

For static analysis the FE solver must fulfil two conditions at each node:

- *The force must be static i.e there is no variation with respect to time.*
- *An equilibrium condition must apply i.e the sum of forces must equal zero, the sum of moments must equal zero*

(Altair University, 2015)

When performing FEA on the upright a static condition will be assumed. This assumes that the rigidity of the material is constant, that small deformations occur, and that the material obeys Hooke's law.

Hooke's law states that for small deformations, the displacement of an object is directly proportional to the force applied:

$$Kx = F$$

Where:

K = Resulting matrix (stiffness matrix)

x = Displacement

F = Load

Linear static analysis follows the principle of Hooke's law.

There are two types of FE solvers; direct solvers and linear solvers.

For a Direct FE solver, the resulting matrix is solved directly. A simple and efficient way would be to inverse the resulting matrix, K , and multiply with the global force vector, f . Often this is not the approach that is taken.

$$u = K^{-1}f$$

'More commonly adopted approaches to directly calculate the solution include Gaussian elimination, LU, Cholesky, and QR decompositions.'

(Harish, 2019)

For an Iterative FE solver, the resulting matrix is very large, therefore, finding the inverse of the matrix is time consuming. An iterative solver will start with an initial assumption for the solution and the solution is then iterated upon until an 'exact' solution is reached.

Different models have been produced to describe the design process. They are categorized by descriptive models and prescriptive models.

Cross (2008) explains how French's model is a descriptive model which is solution focused, since it identifies the significance of generating a solution concept early in the design process (see *Figure 5-9*). The solution concept is then analysed and evaluated, and refinements made following feedback. Circles represent different stages of the design process and the rectangles represent different activities.

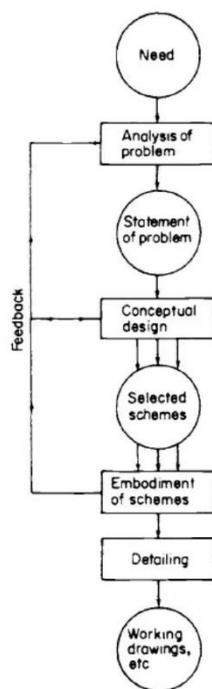


Figure 5-9 - French's model of the design process (Cross, 2008)

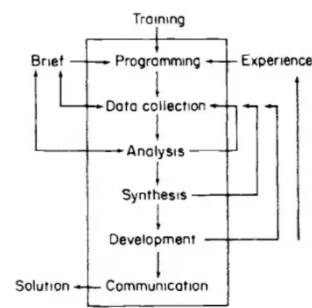


Figure 5-8 - Archer's model of the design process (Cross, 2008)

Archer's model is a prescriptive model which aims to encourage a more efficient design process (see *Figure 5-8*). Archer's model emphasises the need for analytical work to be carried out before the design process.

'Many of these prescriptive models have emphasised the need for more analytical work to precede the generation of solution concepts.' (Cross, 2008)

This project aims to move from a descriptive to a prescriptive design model. By analysing the current upright using FEA and topology optimization, more efficient designs can be developed following the analytical work. This will ensure that no important elements of the design are overlooked.

5.2 Topology Optimization

Alongside FEA, topology optimization was used during the project to aid the design process for the new upright. Research was conducted into topology optimization to gain a better understanding of how it is used. Structural topology optimization was used in this project.

Topology optimization is a computer software which uses an algorithmic process to optimises a design within a domain based on a set of constraints.

'Topology optimization is an algorithmic process that reveals the most efficient design based on a set of constraints or characteristics, often by removing material from the design.'

(Autodesk, n.d.)

It optimizes the design by removing any material from the domain that is not needed and optimizes the distribution of material which has not been removed.

'A typical topology optimization problem can be stated as follows: Given a design domain, and a valid (structural, thermal, fluid, ...) problem, find an optimized design within the design domain, that minimizes an objective (such as volume), such that certain performance constraints (such as stiffness, strength, ...) and manufacturing constraints (such as minimum feature size) are met.'

(Mirzendehtdel & Suresh, 2017, p. 33)

Initially the domain is meshed and inputted into the topology study where parameters are entered for optimization. FEA plays a central role in topology optimization, as linear elastic FEA is executed during the optimization process to determine stress on each element. Inefficient, low stressed material is removed iteratively during the optimization. A highly efficient and customized FEA engine, therefore, can significantly accelerate topology optimization.

Topology optimization is used in the early stages of a component development process.

Mirzendehtdel and Suresh (2017), describe optimization as *'an essential tool for early stage design.'*

It is used to produce an acceptable initial design, which can then be developed using FEA and shape optimization.

'Topology optimization is usually employed in order to achieve an acceptable initial layout of the structure, which is then refined with a shape optimization tool.'

(Gandomi, et al., 2013)

Previously an engineer would have to use judgement alone to design a component. With topology optimization the engineer can input data into the software to create an optimized design within a short period of time. Topology optimization often creates unique designs that an engineer may not be able to envisage, therefore, reducing the design development time, saving money (see *Figure 5-12*). Often components that have not been topologically optimized use more material, leading to greater material costs. They are also heavier than needed, resulting in higher loads on the component. The optimization results can be verified and developed using FEA.

According to Mirzendehtdel and Suresh (2017), the two most common methods to parameterize the topology include density-based methods and level-set methods. Each method uses a different sensitivity analysis.

SolidWorks uses the density-based approach, also known as Solid Isotropic Material with Penalization (SIMP). Mirzendehtdel and Suresh (2017), explain how each finite element is assigned a fictitious pseudo-density variable between zero and one, representing the state of void and solid. Using the densities the static problem is solved. The densities are updated. To reach the desired output, the pseudo-densities are optimized. *Figure 5-10* shows the workflow process for the SIMP approach.

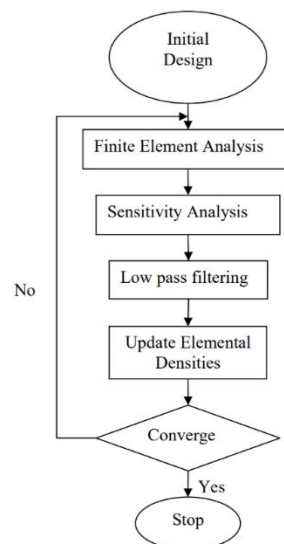


*Figure 5-10 - SIMP flow chart
(Aremu, et al., 2010)*

Checkerboarding can occur to an optimized output, where elements alternate from a solid to a void (see Figure 5-11). To minimise the chance of checkerboarding during optimization studies, a fine mesh was used with 'high quality' elements.

'Furthermore, higher order elements and mesh refinement could also mitigate the checkerboard problem.'

(Larsson, 2016)



Figure 5-11 - Checkerboarding effect (Larsson, 2016)

Continuous variables must be used to represent the material distribution problem. Intermediate values of the pseudo-densities represent fictitious materials.

'In general, the optimal solution of problems involves large grey areas of intermediate densities in the structural domain. Such solutions are not meaningful when we are looking for the topology of a given material, and not meaningful when considering the use of different materials within the design space.'

(Altair Univeristy, 2018)

Therefore, a penalization technique must be used. A value of 3 is typically used for three-dimensional problems.



Figure 5-12 - Upright designed using AMendate optimization software for additive manufacturing (AMFG, 2019)

5.3 Stress and Strain

Stress is described as the force divided by a cross-sectional area of an object. Strain is described as the amount of deformation in the direction of the applied stress divided by its original length. When a 3-dimensional body is subjected to loads, it sees normal stresses and shear stresses. Normal stress occurs when a body is subjected to an axial force:

$$\sigma = \frac{F}{A}$$

Shear stress occurs when a body is subjected to a force parallel to the face:

$$\tau_{xy} = \frac{F_x}{A}$$

Out of the five main theories of elastic failure, the shear strain energy per unit volume (Maxwell-Huber-von Mises) is

‘widely regarded as the most reliable basis for design, particularly when dealing with ductile materials.’
(Hearn, 1997, p. 404)

Since aluminium is a ductile material, this theory was used when developing new upright designs.

‘This theory states that failure occurs when the maximum shear strain energy component in the complex stress system is equal to that at the yield point in the tensile test...’
(Hearn, 1997, p. 404)

Stress is described as a tensor. Von Mises stress was developed to determine whether a structure has started to yield. It is a scalar theoretical unit for the combination of stress components.

‘The stresses calculated at any point can be mathematically written into a scalar quantity known as von Mises stress, which can then be compared with experimentally observed yield points.’
(Harish, 2020)

For the FEA solves, von Mises stress was used as opposed to principal stress, due to the fact it represents all components of stress.

‘Von Mises Stress is preferred for linear static analysis of ductile materials such as steel.’
(Value Design Consulting, n.d.)

Despite being the preferred type of stress to use for linear static analysis, von Mises does have its drawbacks. Von Mises does not show if the stress is direct stress or shear stress dominated. In addition, principal stress investigations should have been conducted as it better represents stress flow.

'If we only plot von Mises stress, then we miss two important factors and cannot tell whether the stress state is tensile or compressive.' (Abbey, 2016)

5.4 Manufacturing methods

The suspension upright is a vital component of the vehicle, holding the front wheel and allowing it to pivot. It connects the wheel to the suspension components and chassis.

'If any item can be called more important than others in the suspension, the upright has a good claim. At its most advanced it is a marvellous piece of design having to cope with a singularly complicated mixture of loads, stresses and strains of a very high order, the rear obviously having an even tougher time than the front.'

(Staniforth, 1994)

According to Staniforth (2006), the two main criteria for an upright are that;

- *The top and bottom pivots do not contact the wheel rim*
- *The further outboard they can be placed, the lower the leverage of the wheel against the links*

Too much deflection of the upright can lead to vague steering feel, therefore, it is imperative that it is designed to its optimum. A limited number of uprights will be produced for this project, as the university MSport Eng team only need one set for the car. The number of production units will be considered when deciding on a manufacturing method.

5.4.1 Forging

According to FIA (n.d.), there are three different variations of forging; impression die forging, open die forging and cold forging. Most forging is executed with the material at a high temperature, of up to 1250°C, however, a part can also be forged with cold material.

For impression die forging (closed die forging) two dies with the same profile as the desired part are used to pound or press the metal. Hydraulic presses, mechanical presses or hammers are used. This method is suitable for parts of a few hundred grams up to about 25,000 kilograms. More complex shapes with closer tolerances can be produced using this method compared to open-die forging, as the contours of the dies limit the flow of metal. Alloy can be forged using this method. Forged suspension uprights are manufactured for the motorsport industry using this method.

Open die forging is done using two flat dies, containing no impressions of the desired part. It can produce parts of a few hundred grams up to 150,000 kilograms. This process progressively shapes the part to the desired geometry. This method would not be suitable for the upright as the geometry is too complex.

Since forging requires very high tooling costs, it is only efficient for a large amount of production units. Only one or two sets will be produced by the university, therefore, it will not be used to manufacture the MCR upright.

5.4.2 Casting

Casting is a manufacturing technique used for many years. Liquefied material is poured into the cavity of a mold and the material is left to solidify. Advancements in technology have led to various types of casting methods. Cast ductile iron is often used for automotive uprights.

Once the material has solidified it is removed from the mold and is often subjected to various finishing treatments.

According to ThomasNet (n.d.), sand casting is the most widely used method, it uses expendable sand molds, often silica sand (SiO_2), to form the part. The sand is packed around a replica of the part, and the mold is split along a parting line into two parts; the cope (upper) and the drag (lower). The sand is resilient to high temperatures and allows for some flexibility as the part cools and contracts. After the sand is compacted, replica part removed, the mold will have a cavity of the part. Often lubrication is applied to the inside of the mold cavity to increase ease of removal of the part.

Advantages of sand casting include:

- Relatively inexpensive production costs
- Ability to produce parts from ferrous and non-ferrous materials
- Low cost for post casting tooling

Disadvantages include:

- Often this method has a low production rate since the mold needs to be removed and destroyed
- Part tend to have rough surface finish

Investment casting uses polymer wax molds. It can be used to create intricate designs with excellent dimensional accuracy. Unlike other casting processes alloy material can be cast using this method. Casting was not used to manufacture the upright as it is expensive when producing low quantities of components.

5.4.3 Additive manufacturing

According to GE Additive (n.d.), additive manufacturing is a new process which uses CAD software or 3D object scanners to direct hardware to deposit material, in successive superfine layers, into precise shapes. Traditional manufacturing methods often require material to be removed through machining, milling or other means. Additive manufacturing creates an object by adding material, therefore, reducing waste.

Fuse Deposition Modelling (FDM) is a method used for plastic material extrusion process. Stratasys originally patented this method. The filament is heated and drawn through a nozzle, with each successive layer bonding to the previous layer of melted or partially melted material.

Powder Bed Fusion (PDF) is a manufacturing method to produce metal bodies. Direct Metal Laser Sintering (DMLS) sinters the powder, layer by layer to melt and bond the material together. Electron Beam Melting (EBM) uses an electron beam projected by an electron gun to fuse the material together. It extracts the electron from a tungsten filament in a vacuum. It can be used with metals and alloys.

‘Electron beam melting (EBM), methods require a vacuum but can be used with metals and alloys in the creation of functional parts.’

(Loughborough University, n.d.)

Despite additive manufacturing allowing flexibility for intricate designs, it was not used due to its high cost.



Figure 5-13 - Topologically optimized Lotus Elise S1 upright designed by Fransisco, manufactured by Carbon Performance (Peters, 2019)

Figure 5-13 shows a motorsport upright for the Lotus Elise S1 designed by Fransisco. The design is very intricate and led to a 25% weight saving over the OEM upright. It would not be manufacturable with any method other than additive manufacturing. This upright is manufactured by Carbon Performance additive manufacturing. Despite additive manufacturing allowing flexibility for intricate designs, it was not used due to its high cost.

5.4.4 CNC Machining

A billet is described as a solid block of material. A Computer Numerical Control (CNC) machine removes material from the billet to produce the final component. The CNC machines can produce parts with very high dimensional accuracy in short lead times. The bearing surfaces of the upright need very high dimensional accuracy. This method is also

'cost effective in low quantities'.

(Mason, 2015)

Only 1 or 2 sets of the uprights will need manufacturing initially, therefore, some of the other manufacturing methods will not be viable. Some intricate designs cannot be machined from a billet; however, the simplicity of the upright's geometry enables machining. The university have a 3-axis

CNC milling machine; therefore, this method is an obvious choice for the upright as it can be manufactured in-house.

CNC machining was chosen as the manufacturing method due to its high accuracy, short lead time and cost effectiveness for 2-4 units. David Cooper, the CAM lab technician, was met on several occasions, and the upright model was run in the Computer Aided Manufacturing (CAM) software. The first cutting operation, machining from the rear of the upright, would take around 3 hours. To produce one whole upright, it would take around 8 hours.

'3-axis machines including CNC milling machines, which cost around \$40 per hour to run...'

(Prototech Asia, 2019)

The running costs for the 3-axis machine would be approximately £32 an hour, therefore, running the machine for one upright would cost approximately £250. This excludes material and tooling costs, since the milling drill bits will need replacing over time. See *Appendix 16.8* for material cost and *Appendix 16.7* for a CNC quotation of the upright.

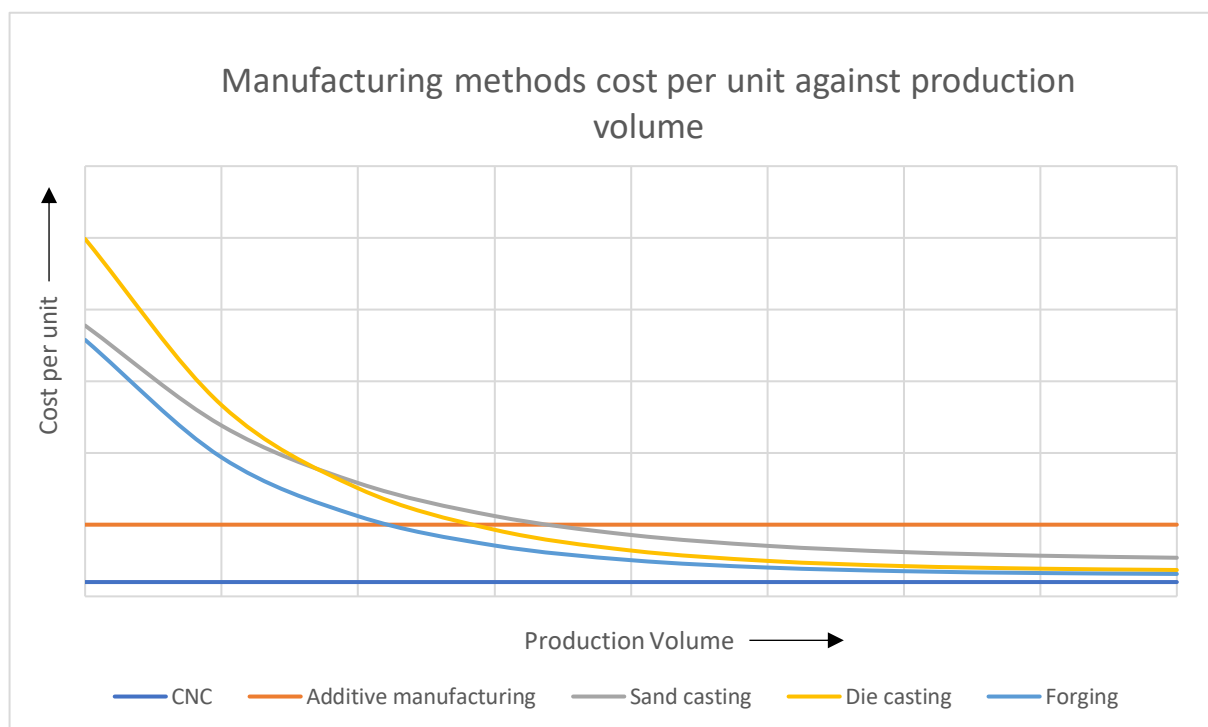


Figure 5-14 - Cost per unit against production volume for different manufacturing methods

Figure 5-14 shows the trends for cost per unit against production volume for the different manufacturing methods. The graph was created with data collected from (Alteams, 2019), (Manufacturing.net, 2018), (Engineers Edge, n.d.) and (Simmons, 2000).

The graph shows how CNC machining and additive manufacturing's cost per unit remain constant as production volume increases. The sand casting, die casting and forging's cost per unit decreases as production volume increases. At very high production volumes CNC and additive manufacturing are not viable. For the purpose of the graph the casting and forging cost per unit data was added from very low production volumes. In reality, a few hundred units are the minimum annual production volumes to make these methods feasible.

Since only 1-2 sets of uprights will be produced for the MSport Eng team, CNC machining is clearly the best manufacturing method to use for this project.

5.5 Material selection

Table 5-1 - Potential upright materials specifications (ASM Aerospace Specification Metals, n.d.)

Material	Ultimate Tensile Strength (MPa)	Tensile Yield Strength (MPa)	Density (g/cm ³)	Elastic Modulus (GPa)	Poisson ratio
6061-T6 Aluminium	310	276	2.70	68.9	0.330
6082-T6 Aluminium	330	270	2.70	70.0	0.330
7075-T6 Aluminium	572	503	2.81	71.7	0.330
Titanium 6Al-4V (Grade 5), Annealed	950	880	4.43	113.8	0.342
Titanium 6Al-4V (Grade 5), STA	1170	1100	4.43	114.0	0.330
AISI Type 410 Stainless Steel	1085	1005	7.80	200.0	0.300
AISI 4130 Steel	670	435	7.85	205	0.290
Ductile Iron Grade 80-55-06	552	379	7.10	170	0.310

Table 5-1 shows specifications for potential materials for the upright. The Ultimate Tensile Strength (UTS) (R_m) is the maximum stress a standardized material sample can withstand. UTS is calculated by dividing the cross-sectional area by the stress placed on the material (see Figure 5-15). The 0.2% yield strength is the amount of stress a material can withstand before plastic deformation occurs.

'The 0.2% yield strength ($R_{p0.2}$) is the amount of tensile stress that causes a lasting (plastic) strain of 0.2% in a material.' (Reif, et al., 2018, p. 210)

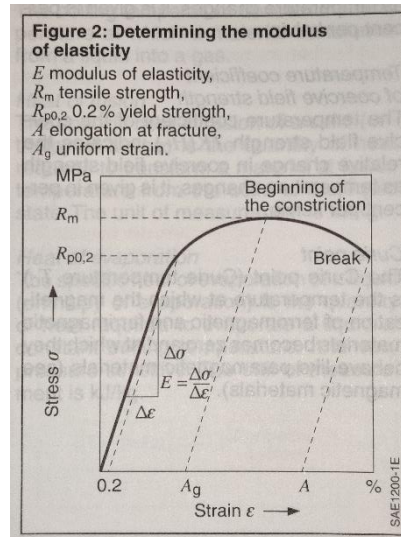


Figure 5-15 - Material stress-strain curve
 (Reif, et al., 2018)

The elastic modulus (E) is the ratio of stress to the corresponding strain within the plastic deformation region. Elastic modulus describes the material's resilience to deformation.

$$E = \frac{\delta\sigma}{\delta\epsilon}$$

Poisson's ratio (ν) is the ratio between the longitudinal strain and resulting transverse strain for a body under tension or compression.

$$\nu = \frac{\epsilon_{lateral}}{\epsilon_{axial}}$$

6061-T6 Aluminium is used for two components of the original upright. Both the steering arm and upright are made from 6061.

'The elemental composition of 6061 aluminium alloy is as follows: 0.6% Si, 1.0% Mg, 0.2% Cr, 0.28% Cu, and 97.9% Al.'
 (Cavallo, n.d.)

The first number '6' describes how the alloy contains magnesium and silicon. 'T6' describes how the aluminium has been heat-treated and then artificially aged. The material is strong due to its medium fatigue strength, yet light weight due to its low density. It is one of the cheapest heat-treated materials.

'It is also fairly malleable, and can be worked into many configurations, while still maintaining a very rigid strength.'
 (AWC, 2016)

It is used extensively in the motorsport and automotive industry.

'6061 is commonly employed as a construction material, in particular in the automotive industry.'
(Clinton Aluminium, 2017)

7075-T6 Aluminium contains a higher percentage of Zinc which makes it one of the hardest and highest strength alloys available.

'Its nominal chemical breakdown is 90.0% Al, 5.6% Zn, 2.5%Mg, 0.23%Cr, and 1.6% Cu.'
(Cavallo, n.d.)

Since it has a higher copper content than 6061 it has inferior corrosion resistance. It is often used for parts subjected to high stress. If the alloy is annealed, it is highly formable and may be flash or spot-welded. The strength is increased considerably by heat-treating. Its UTS is 84.5% higher than 6061-T6 aluminium yet is only 4% more dense. 7075-T6 is approximately 54% more expensive than 6061-T6 aluminium (see *Plate 5-1*).

2" x 3" Aluminum Rectangle Bar 6061-T6511-Extruded - Part #: 1180



QTY **\$122.84 ea.**
\$122.84 for 1

Dimensions
Thickness: 2" Width: 3"

See all available dimensions for this product

48.0" - \$122.84 ea.

Create Custom Cut

Add To Cart

Add to Favorites Add to Project List

Overview:
[See Aluminum Tolerances](#)
Aluminum Rectangle Bar 6061-T6511 exhibits above average corrosion resistance, good machinability, high strength, light weight, good weldability.
[Read More Specifications](#)

****Special Buy** 2" x 3" Aluminum Rectangle Bar 7075-T651-Cold Finish** - Part #: 10000044



QTY **\$188.78 ea.**
\$188.78 for 1

Dimensions
Availability: Limited To Stock On Hand Thickness: 2" Width: 3"

See all available dimensions for this product

48.0" - \$188.78 ea.

Add To Cart

Add to Favorites Add to Project List

Overview:
[See Aluminum Tolerances](#)
Aluminum Rectangle Bar 7075-T651 has high strength, light weight, good fatigue resistance, average corrosion resistance, and average machinability.
[Read More Specifications](#)

*Plate 5-1 - 6061 and 7075 aluminium material costs
(Onlinemetals.com, n.d.)*

Titanium 6Al-4V (Grade 5) Annealed has a UTS 3.06 greater than 6061-T6 aluminium. The density of 6Al-4V is 64% greater than 6061-T6. Since titanium is stronger than aluminium, a lighter upright could be produced. Annealing is a heat treatment process used to reduce hardness and increase ductility of a material.

‘Annealing is often performed after a material has undergone a hardening or cold working process to prevent it from brittle failure or to make it more formable for subsequent operations.’
(Metal Supermarkets, 2018)

Titanium is difficult to machine due to its low thermal conductivity, therefore, the CNC machines in the university may not be able to produce a titanium upright.

Another form of titanium 6Al-4V (Grade 5) STA (Solution Treated and Aged) further increases the UTS to 3.77 larger than 6061-T6. This material in *Table 5-1* is treated at 900-955°C and aged at 540°C.

AISI Type 410 stainless steel could be another material to consider with a high UTS of 1085GPa, however, like titanium it is an expensive material, and it is difficult to cut accurately with the CNC machine.

Many production vehicles have uprights produced from cast ductile iron. Iron has a high density of 7.1g/cm³ with a relatively low UTS of 552MPa.

7075-T6 aluminium was chosen for the new upright design due to its high strength-to-weight ratio. Comparing the SN curve for 7075 and 6061 showed how 7075 had higher fatigue strength until failure resulting in higher durability. Despite having a much higher UTS it is only 4% more dense and still retains good machinability.

5.6 Katwa's Dissertation

In 2019 Katwa completed a topology optimization using VR&D Genesis software. The Team MCR CAD model of the upright was used during FEA. Katwa completed FEA on the upright and compared the results to a practical test conducted in the workshop using a twist rig. Krishna designed a new upright based on the topology study, however, the design could be improved. The study will be used in order to support findings in this report, and to further optimize the upright design.

6 Methodology

6.1 Creating CAD of original upright

The CAD model of the original upright was going to be supplied from Katwa, however, he did not have access to the CAD model, therefore, a new one needed to be produced. The upright was removed from the chassis and measured on 4/11/19 (see *Plate 6-1*). Measurements were recorded and an accurate CAD model was created in SolidWorks (see *Figure 6-1*). The original upright consisted of three components, the upright, steering arm and brake bracket. The upright and steering arm are made from 6061-T6 aluminium, and the brake bracket from mild steel. Each component is bolted using three M8 bolts.

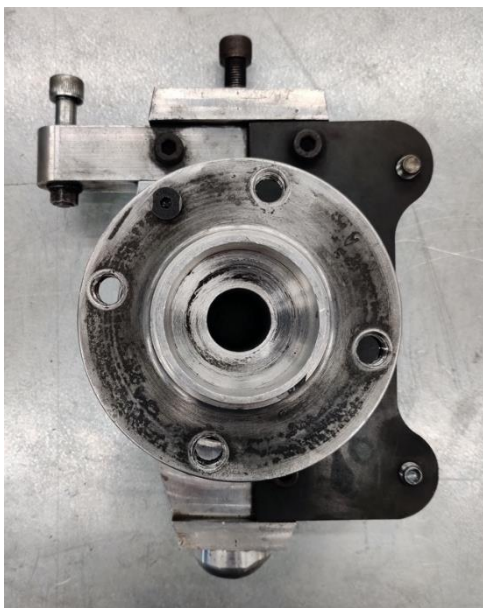


Plate 6-1 - MCR front upright

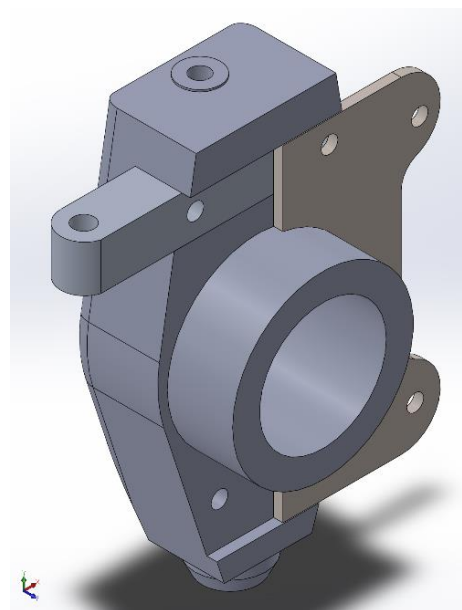


Figure 6-1 - CAD model of MCR front upright

6.2 Attachment points

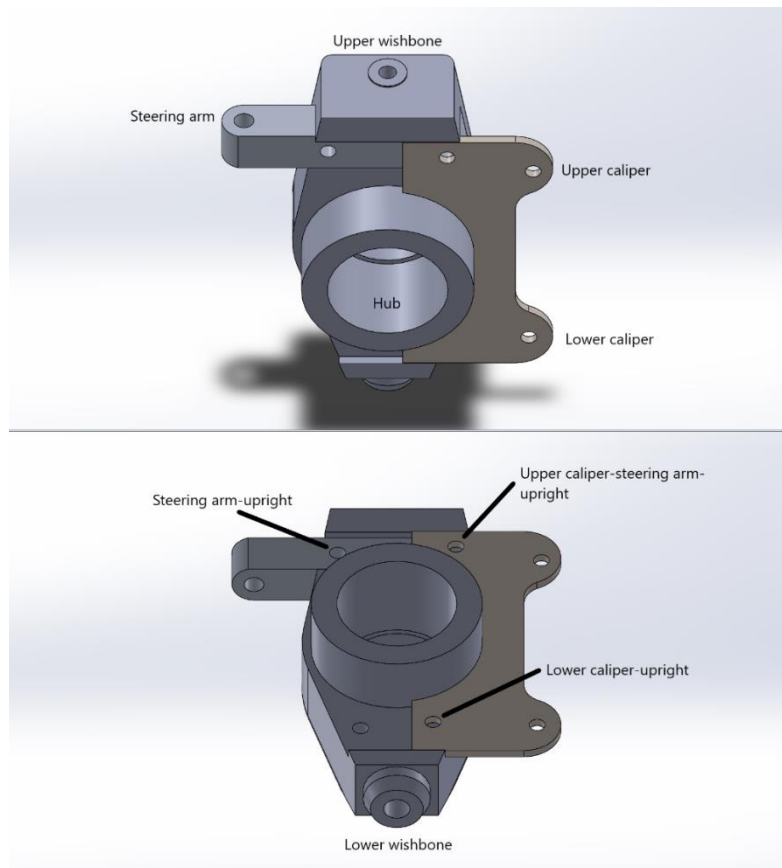


Figure 6-2 - Locations of the attachment points

Figure 6-2 shows the CAD model created of the original upright and each of the attachment points. For each loadcase constraints were applied at the upper and lower wishbone, steering arm and hub. It is however noted that constraints and loads to an attachment point cannot be applied simultaneously, otherwise the solver will fail.

Table 6-1 - Maximum lateral loads on each attachment point

Attachment point	Maximum load (N)	Direction
Upper wishbone	2696.1	Z
Lower wishbone	7279.4	-Z
Hub	6145.8	X
Steering	150	Z
Braking	7357.8 (total)	Y

Table 6-1 shows the maximum lateral load and their direction on each attachment point. The calculations for these loads can be found in *Appendix 16.1*.

6.3 Topology optimization

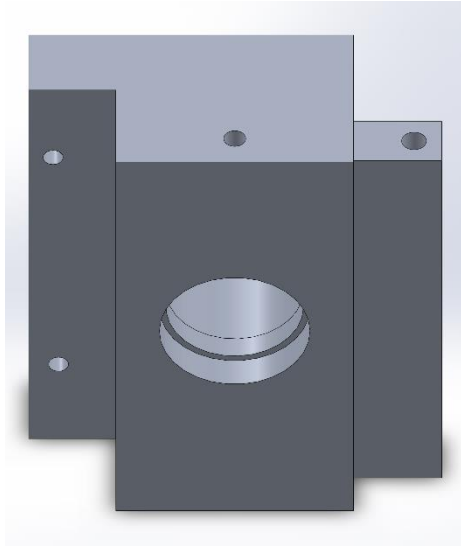


Figure 6-3 - Domain used for SolidWorks topology studies

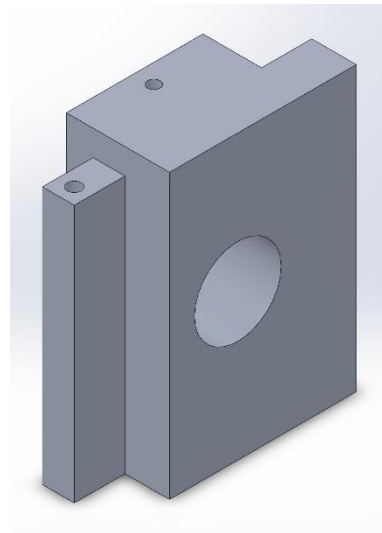


Figure 6-4 - Domain used for SolidWorks topology studies

Figure 6-3 and 6-4 show the domain used for the topology optimization studies conducted in SolidWorks. This domain was created using the 'defeature' tool, which simplified the original upright into a rectangular shape with maximum dimensions. To improve the topology results, the domain could have been refined and increased in size, however, Katwa had already conducted a topology using a larger domain. The domain was made from 7075-T6 aluminium, the desired material for the new upright designs, and had a mass of 6019 grams.

For each study the fixtures and loads were defined. Preserved regions were defined for each attachment point, to include in the new design. Finally, goals and constraints were defined.

6.4 SolidWorks FEA

SolidWorks was used alongside Design Studio for FEA throughout the project. Since the model had been created in SolidWorks, the first study was conducted using SolidWorks. Material properties had to be defined for each component. Following this, 'connections' were defined. Contact sets were used as opposed to component contacts, since this would lead to faster solution times. Contact sets define the type of contact using faces, edges or vertices. Component contact, as the name suggests, uses components to define the contact, meaning every node on each component will be checked for contact.

'Component Contacts are easy to set up but lead to additional solver time.'

(Warren, 2019)

To reduce solve time the 'bonded' contact set was primarily used over 'no penetration', see *Chapter 8.5*. 'Fixtures' were then defined, and the fixed constraint used on each suspension attachment point; upper wishbone, lower wishbone, hub bearing and steering arm. For more accurate results the fixed constraint would have not been used. The fixed constraint assumes the point is infinitely stiff which is incorrect. 'External loads' were defined and the maximum loads during operation used for each attachment point (see *Table 6-1*). Finally, a mesh was created, and the study solved.

6.5 Design Studio FEA

Meshing had to be produced in SolidWorks, and the mesh exported from SolidWorks as a '.NASTRAN' file. The meshed upright was then imported into Design Studio where Rigid Body Element 2 (RBE2) had to be created at each attachment point (see *Figure 6-5*). An independent grid (node) had to be created in the centre of the attachment point, which was linked to the nodes at the attachment point face. The RBE2 is described as multi point constraints and in the case of the upright simulated the effect of a bolt connector.

'A rigid element is a link from one node to one or more other nodes, where the motion of the node(s) is governed by the "degrees of freedom" you choose to connect.'

(McKenzie, n.d.)

Grid component sets were defined for each loadcase, which defined the constraints. Like in SolidWorks, fixed constraints were applied, therefore, the RBE2 independent nodes were constrained in six degrees of freedom. Static loads were applied for each loadcase. Finally, each loadcase was defined by choosing the grid component set, static load and the data to plot. When calculating the stiffnesses in Design Studio, 1000N load was chosen since it was below the yield and simplified calculating the stiffnesses using Hooke's law.

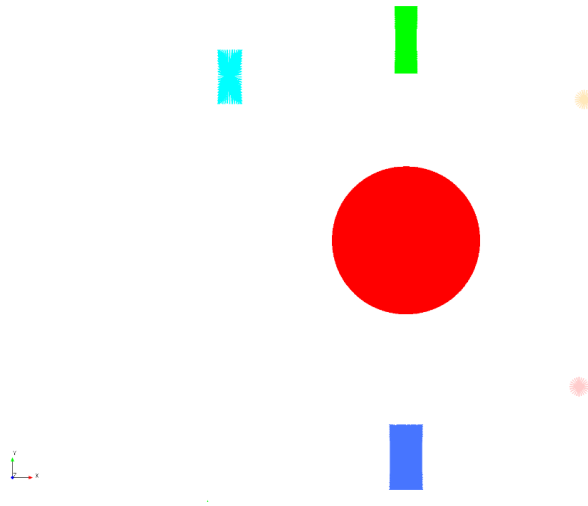


Figure 6-5 - RBE2 elements created for the new upright design

Stiffnesses at each attachment point were measured using Design Studio, since the displacement at RBE2 independent nodes could be shown (unlike in SolidWorks). SolidWorks could only show the displacement at the attachment point surfaces. RBE2 elements add stiffness unlike the RBE3 elements, therefore, this suggests improvements to the accuracy could have been made using RBE3.

'RBE2 elements artificially add stiffness by constraining the system to follow a one to one linear displacement and/or rotational relationship between the connected nodes.'

(Batchu, 2015)

The RBE3 element does not artificially add stiffness, as it is not a rigid element.

'The motion at the 'dependent' node is a weighted average of the motion at the 'independent' nodes.'

(Batchu, 2015)

7 Results

7.1 Topology optimization

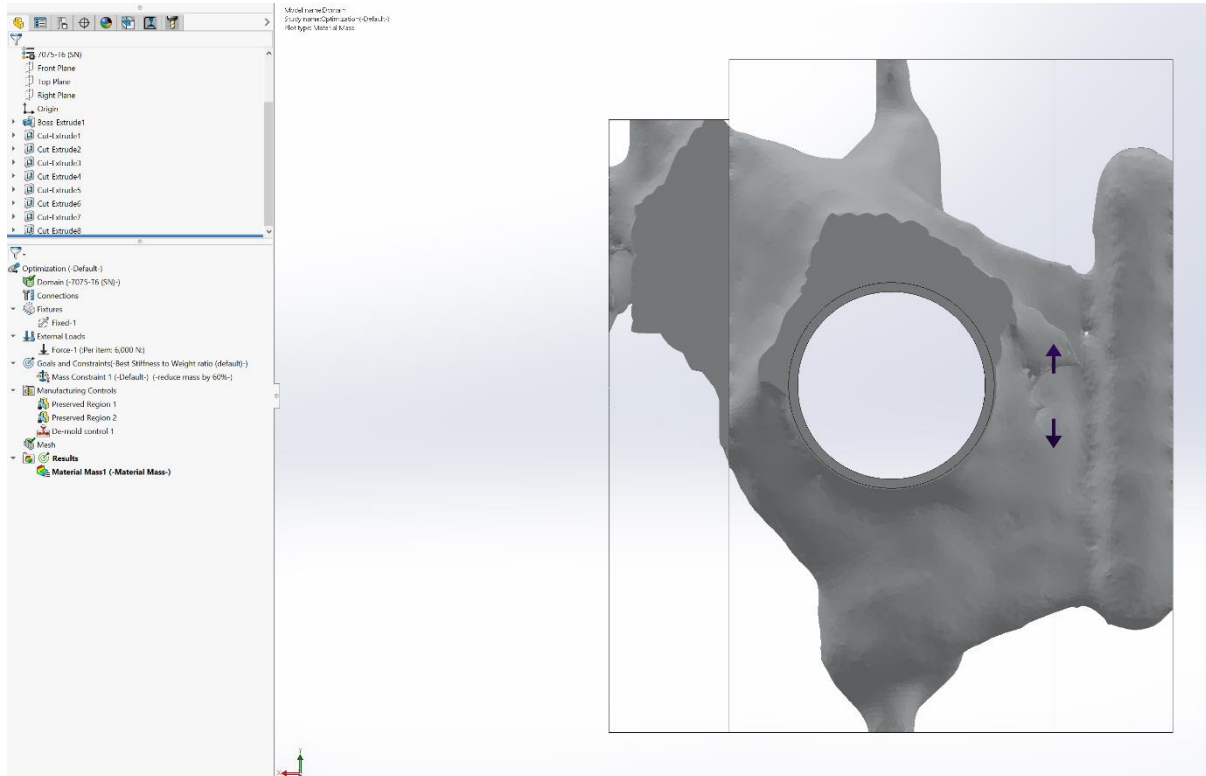


Figure 7-1 - First topology optimization study

The first topology was constrained at the upper, lower wishbone attachment point and the steering arm attachment. A load of 6000N was applied at the hub bearing in the -X direction (see Figure 7-1). The goal of the optimization was best stiffness to weight ratio, with a mass constraint, reducing mass by 60%. A de-mold control was defined so that the shape of the design would be manufacturable. 'Mid-plane (both directions)' was applied parallel to the Y axis, meaning no voids would be created along the X and Z axis.

'Imposing this constraint will ensure that there would be no cavities along the prescribed direction. This can be particularly useful for casting.'

(Mirzendehtdel & Suresh, 2017, p. 106)

By using this constraint, the manufacturing process for machining can be simplified, resulting in reduced labour costs. Each of the suspension attachment points were selected as preserved regions, therefore, would be included in the new design.

The optimized design weighed 2406 grams, a decrease of over 60% compared to the mass of the domain. A fine curvature-based mesh was used (see *Figure 7-2*), and the solution took 11 hours, 29 minutes and 3 seconds.

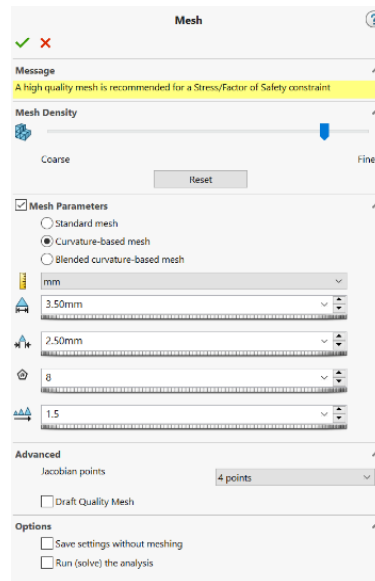


Figure 7-2 - Mesh used for the first, second and third topology study

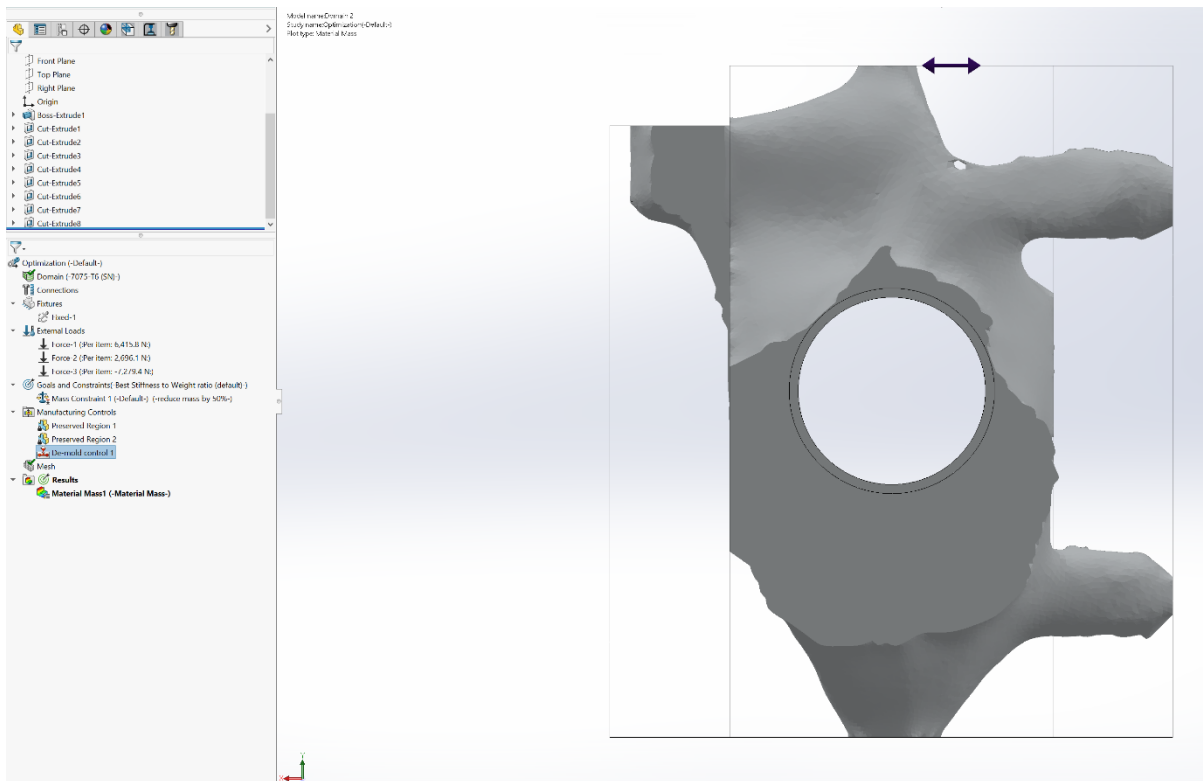


Figure 7-3 - Second topology optimization study

The second topology study conducted had a fixed constraint at the steering arm attachment and on the cylindrical surface behind the hole for the hub bearing. Maximum loads during operation were applied at the upper and lower wishbone attachment point and at the hub bearing attachment point. The goal of the optimization was best stiffness to weight ratio, with a mass constraint, reducing mass by 50%. 'Mid-plane (both directions)' was applied parallel to the X axis (see *Figure 7-3*). The same fine mesh was used as the first study (see *Figure 7-2*) and the solution took 7 hours, 5 minutes and 50 seconds. The optimized design weighed 3007 grams, so required further weight reduction.

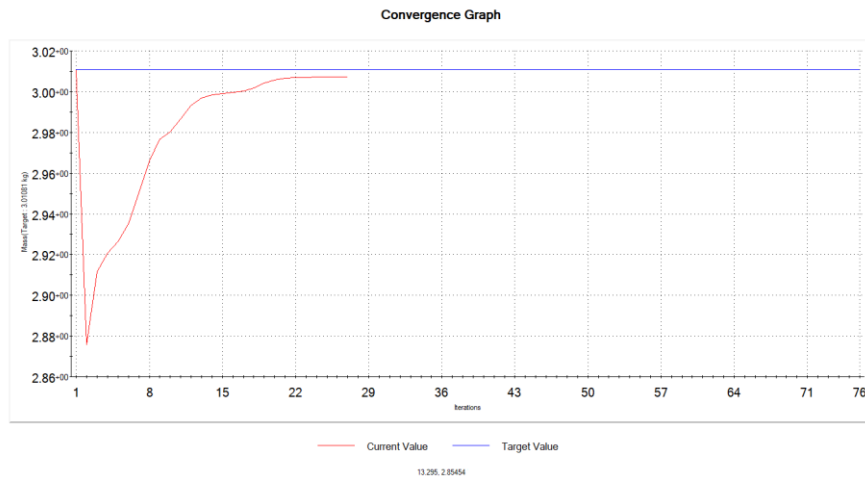


Figure 7-4 - Convergence graph for the second topology optimization study

Figure 7-4 shows the convergence graph for the second study. Mass target on the Y axis is plotted against design iterations on the X axis. The solver stops when the graph has reached the target value and has converged.

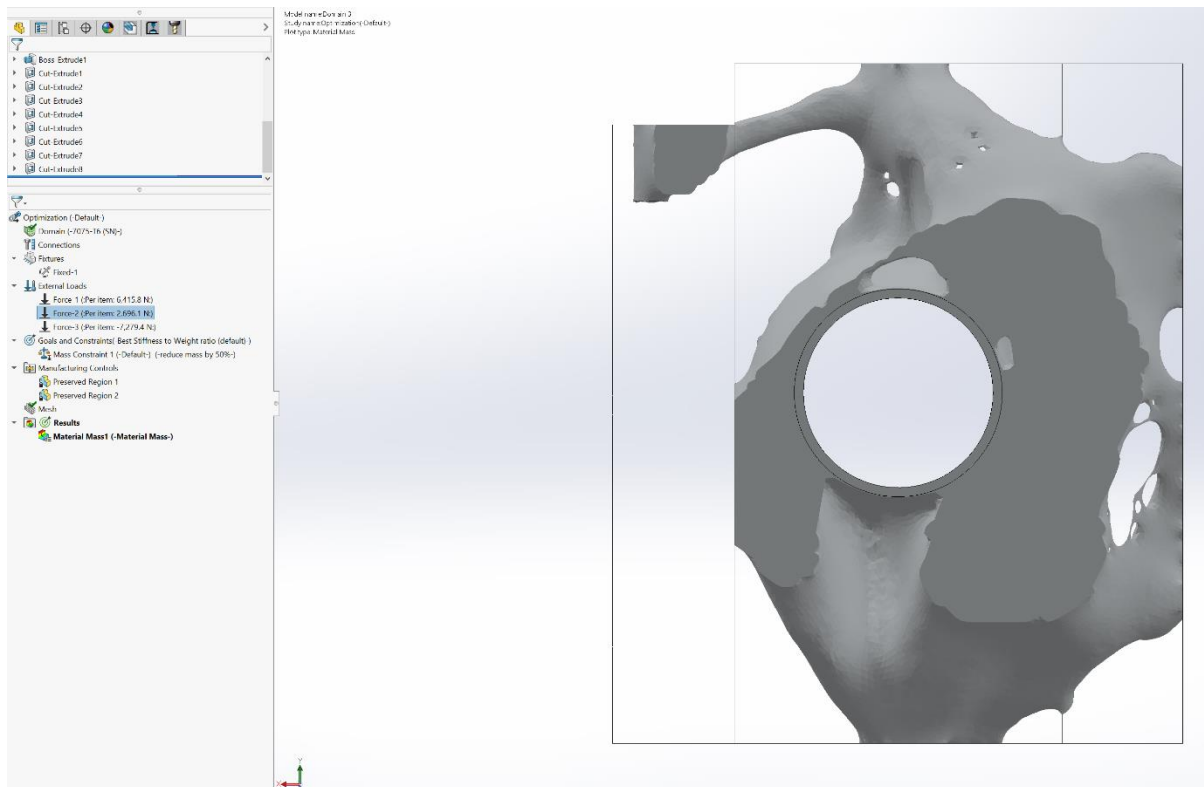


Figure 7-5 - Third topology optimization study

The third topology study conducted had the same fixed constraints as the second study, with the addition of a fixed constraint on the brake caliper attachments (see *Figure 7-5*). The same loads, goal and constraints were used as the second study. No de-mold control was applied to this study, therefore, the upright design created is an organic shape, and would be difficult to manufacture due to the voids. Despite not being a manufacturable design using CNC, aspects of the design were used to inform the final designs. The same fine mesh was used as the first study (see *Figure 7-2*) and the solution took 8 hours, 28 minutes and 15 seconds. The optimized design weighed 3005 grams, so required further weight reduction.

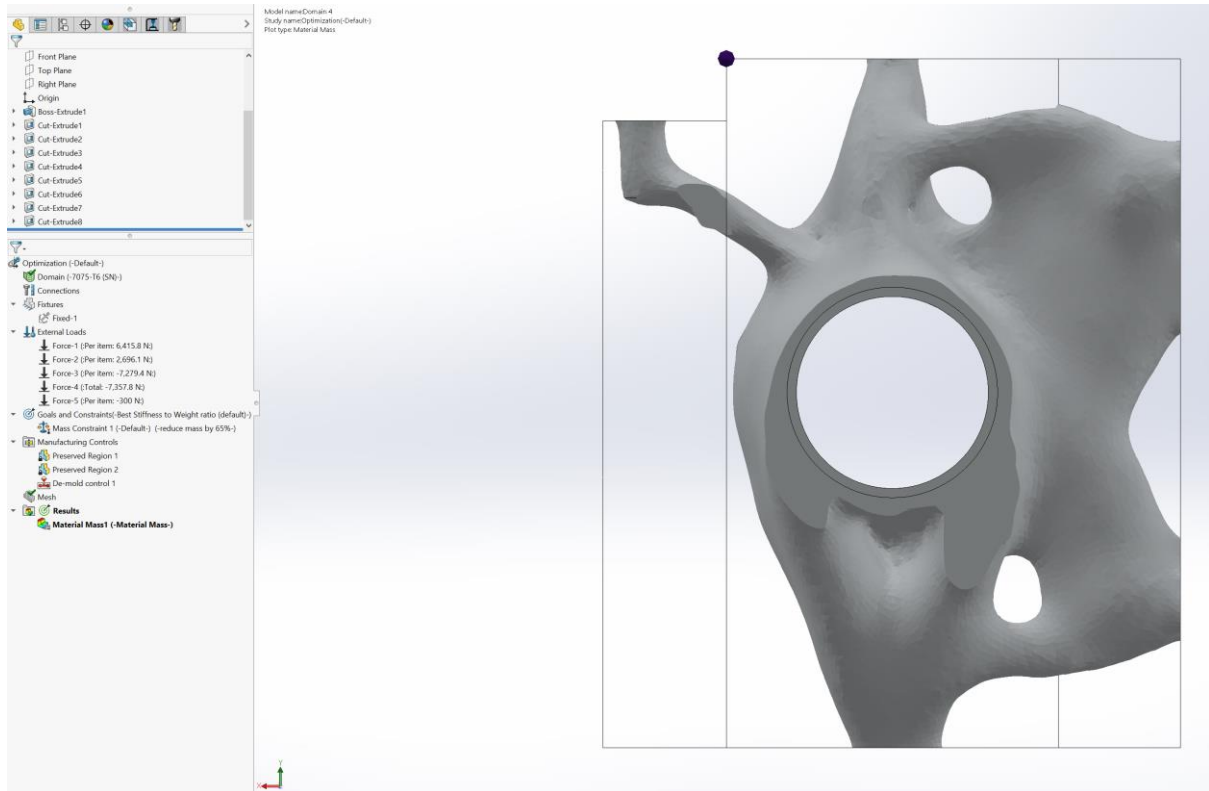


Figure 7-6 - Final topology optimization study

The final topology study had a fixed constraint on the cylindrical surface behind the hole for the hub bearing (see Figure 7-6). Every attachment point had a load applied, however, 300N was now assumed for the steering arm. The goal was best stiffness to weight ratio, and a mass constraint was applied, reducing mass by 65%. A 'Mid-plane (both directions)' de-mold control was applied parallel to the Z-axis. A standard mesh was used with maximum element size of 4mm and Jacobian points selected as 4 nodes. The solve took 1 hour, 12 minutes and 49 seconds, and the optimised upright weighed 2105 grams.

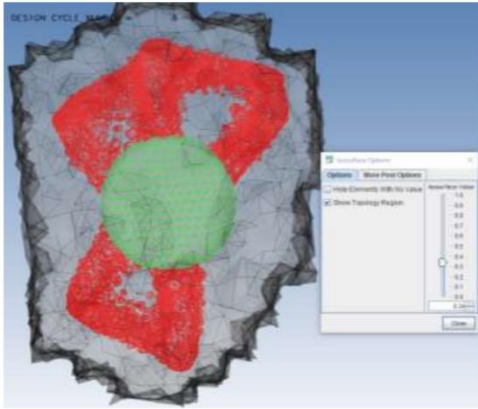


Figure 7-7 - Katwa's topology optimization results with low isosurface value

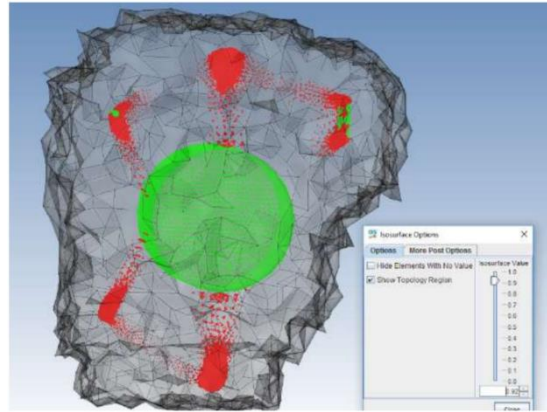


Figure 7-8 - Katwa's topology optimization results with high isosurface value

Katwa's topology study results can be seen in Figure 7-7 and 7-8. The green areas show non-designable regions and the red show important material. Figure 7-8 uses a high isosurface value, showing only the most important material, and Figure 7-7 uses a lower isosurface value.

8 Analysis

8.1 Original upright FEA

FEA was originally performed in SolidWorks for each loadcase for the original upright. Results were collected for two types of contact sets; bonded and no-penetration. Design Studio was also used to test the original upright; however, the global bonded contact set was used.

8.1.1 SolidWorks FEA using bonded contact set

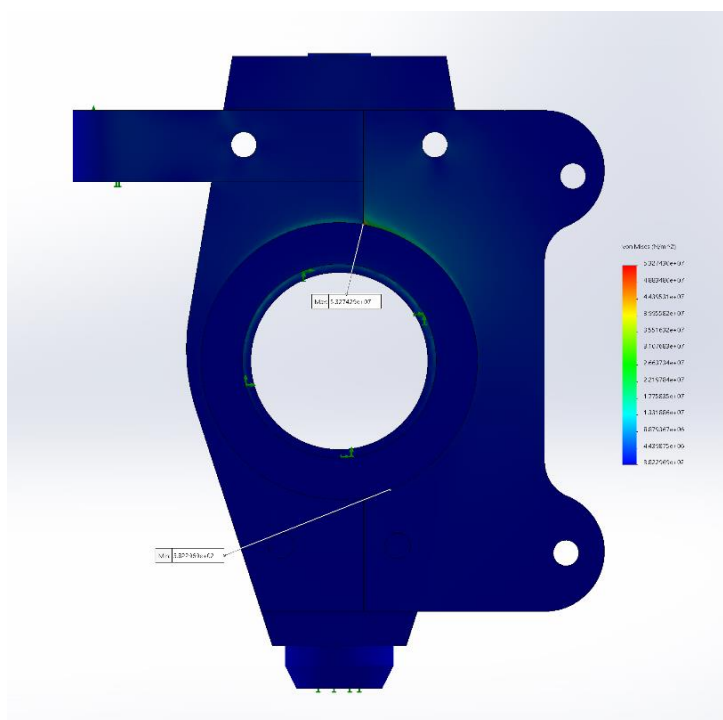


Figure 8-1 - von Mises stress for the top wishbone loadcase using bonded contact set

Figure 8-1 shows the von Mises stress for maximum top wishbone load using the bonded contact set. There is a stress concentration at the point where the brake bracket and upright contact. According to CAD2M (2016), the bonded contact set treats each component as welded, therefore, the whole model acts as one solid body.

Although this is not the correct way of representing the uprights, it was used throughout later analysis. Since the displacements were small, contact sets wouldn't have had a large effect on results.

8.1.2 SolidWorks FEA using no-penetration contact set

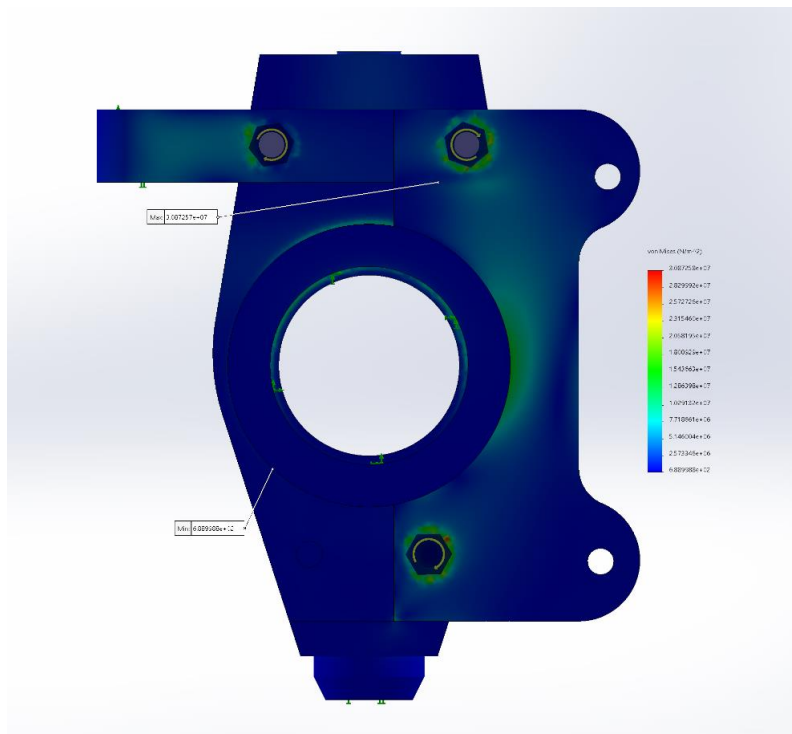


Figure 8-2 - von Mises stress for the top wishbone loadcase using no penetration contact set

Figure 8-2 shows the von Mises stress using the no-penetration contact set. According to GoEngineer (2017), the no-penetration contact set allows two or more faces to touch and develop contact pressures/forces, however, the components cannot flow into each other.

This type of contact set is the more accurate representation for the original upright. Connectors had to be defined to connect each component and a torque of 15Nm was assumed for the bolts. This relationship took longer to solve than the bonded contact set, therefore, the bonded contact set was used for the rest of the project. With a coarse standard mesh of 2.4mm element size, the no penetration contact set was taking approximately 30 minutes for each solve. Had more computational power been available this contact set would have been used throughout the analysis.

Table 8-1 - FEA results for each loadcase using the bonded and no penetration contact set

	Load (N)	Von mises (N/m ²)	Displacement (mm)	Strain	Solve time	
BONDED						
Top Wishbone	2696.1	4.828E+07	1.105E-02	1.608E-04	1min 01sec	(Z Direction)
Bottom Wishbone	-7279.4	1.509E+08	5.256E-02	7.383E-04	1min 01sec	(Z Direction)
Acceleration	6415.8	3.919E+07	2.342E-02	6.330E-04	1min 01sec	(X Direction)
Braking Force	7357.8 (total)	8.631E+07	1.978E-02	3.341E-04	4min 26sec	(Y Direction)
Steering Load	150	6.864E+06	5.109E-03	5.659E-05	1min 01sec	(-Z Direction)
Multiple loads	top, bottom, steering	1.883E+08	5.329E-02	7.316E-04	3min 28sec	
NO PENETRATION						
Top Wishbone	2696.1	3.087E+07	2.489E-02	2.201E-04	24min 10 sec	(Z Direction)
Bottom Wishbone	-7279.4	8.248E+07	5.991E-02	7.229E-04	20min 14sec	(Z Direction)
Acceleration	6415.8	6.468E+07	3.491E-02	8.286E-04	46min 31sec	(X Direction)
Braking Force	7357.8 (total)	2.433E+08	3.528E-02	7.373E-04	29min 19sec	(Y Direction)
Steering Load	150	6.981E+06	6.015E-03	7.339E-05	27min 27sec	(-Z Direction)
Multiple loads	top, bottom, steering	7.141E+07	6.151E-02	7.440E-04	2hours 03min 26sec	

Table 8-1 shows the results comparing bonded and no penetration contact set. Despite using the same standard mesh there is a substantial difference between the results. The stresses and strains show no clear trend; however, the displacements of the no penetration contact set were larger. On average the displacements were 55.2% larger. On average, for the no penetration contact set the solve times were 26.4 times larger than the bonded studies. Many studies needed to be conducted in a short space of time the bonded contact set was chosen for the rest of the original upright's studies.

Since the bonded contact set was used for the original upright's studies, when comparing the original and new upright design, the lower value of displacement was used. The stiffness of the original upright was overestimated, and the new designs would need to surpass this stiffness.

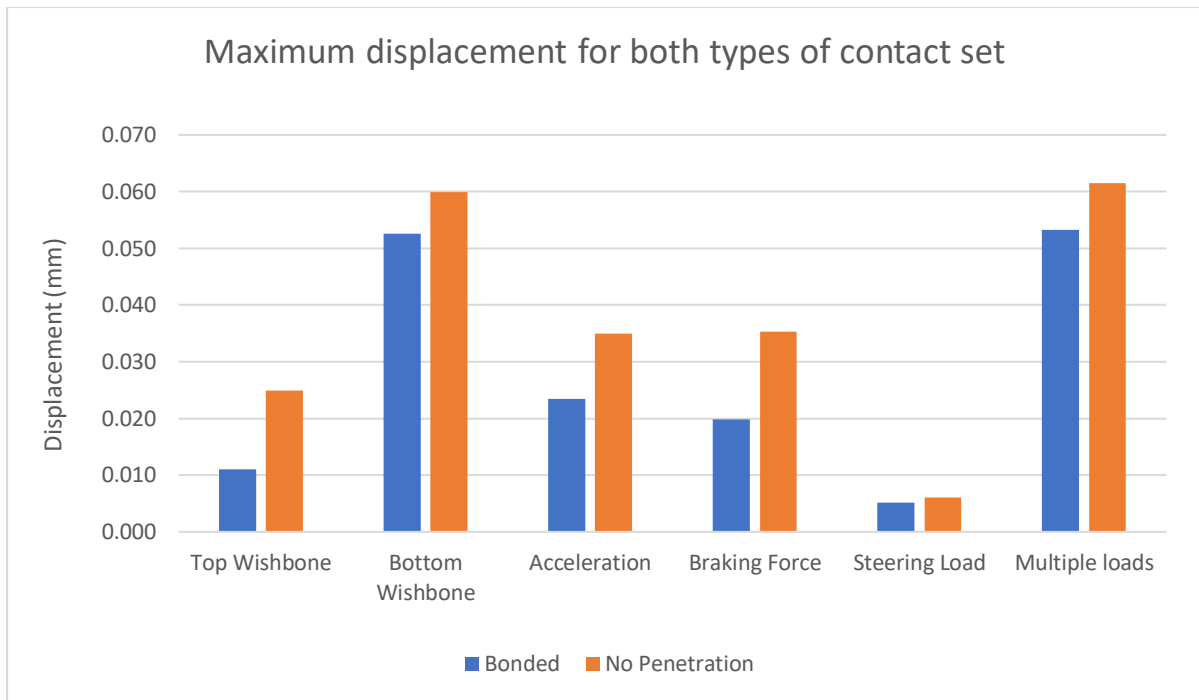


Figure 8-3 - Maximum displacement for each loadcase using the bonded and no penetration contact set

Figure 8-3 shows the displacements of each loadcase for bonded and no penetration contact set. The no penetration contact set has consistently larger displacements. The multiple loads study was fixed at the hub and the top wishbone, bottom wishbone and steering load applied.

8.2 New designs

Table 8-2 - Weight and weight saving of original and new designs

	<i>Weight (g)</i>	<i>Weight saving (g)</i>
<i>Original design</i>	2527	N/A
<i>Design 1</i>	1776	751
<i>Design 2</i>	1928	599
<i>Design 2 updated</i>	1941	586
<i>Design 3</i>	1836	691
<i>Design 3 updated</i>	1831	696
<i>Design 4</i>	1782	745
<i>Design 5</i>	1960	567
<i>Design 6</i>	1943	584
<i>Design 7</i>	2073	454
<i>Design 8</i>	1935	592

Table 8-2 shows the weight of each new design, and the weight saving achieved. The topology and original design's FEA results were used to develop the new designs.

8.2.1 SolidWorks FEA results

The original design and each new design were tested in SolidWorks using two different meshes to ensure the same trends would be obtained. A curvature-based mesh was initially used with parabolic elements with maximum size of 3mm and minimum of 0.6mm. The studies were then repeated using a curvature-based mesh with parabolic elements with maximum size of 1.5mm and minimum of 0.3mm. The same trends were seen for both meshes; however, the results were slightly different. See *Appendix 16.4* for each design's results using both meshes.

Table 8-3 - Von Mises stress for each loadcase for each new design

	Von mises (N/m ²)				
	Top wishbone	Bottom wishbone	Acceleration	Braking	Combo
Design 1	2.05E+07	6.64E+07	5.92E+07	1.33E+08	6.63E+07
Design 2	1.91E+07	6.40E+07	5.69E+07	6.36E+07	6.42E+07
Design 2 (U)	1.92E+07	6.20E+07	5.78E+07	6.27E+07	6.22E+07
Design 3	2.01E+07	7.22E+07	6.19E+07	5.48E+07	7.24E+07
Design 3 (U)	2.02E+07	7.28E+07	5.90E+07	5.46E+07	7.31E+07
Design 4	2.31E+07	7.21E+07	1.28E+08	5.51E+07	7.21E+07
Design 5	1.87E+07	6.07E+07	5.51E+07	6.10E+07	6.11E+07
Design 6	2.11E+07	6.94E+07	5.55E+07	4.37E+07	6.98E+07
Design 7	1.91E+07	6.50E+07	5.37E+07	4.76E+07	6.54E+07
Design 8	1.97E+07	7.85E+07	5.10E+07	4.70E+07	6.69E+07
Maximum value:	Design 4	Design 8	Design 4	Design 1	Design 3 (U)
Minimum value:	Design 5	Design 5	Design 8	Design 6	Design 5

Table 8-3 shows the von Mises stress for each new design. Five loadcases were tested. A formula was written to display which design had the maximum and minimum value for each loadcase. The objective of the new design was to minimise stress. Table 8-3 suggests that design 5 was the optimum design for reducing stress, with the minimum value of stress for the top wishbone load of 2696.1N, bottom wishbone load of 7279.4N and combination of loads. Design 4 was the worst in terms of stress, as it had the maximum value for the top wishbone load of 2696.1N and acceleration load of 6145.8N.

Table 8-4 - Displacement for each loadcase for each new design

	Displacement (mm)				
	<i>Top wishbone</i>	<i>Bottom wishbone</i>	<i>Acceleration</i>	<i>Braking</i>	<i>Combo</i>
Design 1	0.0117	0.0445	0.0245	0.0108	0.0445
Design 2	0.0116	0.0392	0.0243	0.0546	0.0394
Design 2 (U)	0.0114	0.0389	0.0238	0.0518	0.0388
Design 3	0.0118	0.0460	0.0268	0.0485	0.0463
Design 3 (U)	0.0118	0.0462	0.0266	0.0490	0.0464
Design 4	0.0131	0.0467	0.0276	0.0487	0.0469
Design 5	0.0115	0.0368	0.0239	0.0517	0.0370
Design 6	0.0117	0.0424	0.0250	0.0303	0.0427
Design 7	0.0114	0.0397	0.0228	0.0366	0.0398
Design 8	0.0120	0.0408	0.0243	0.0376	0.0410
<i>Maximum value:</i>	<i>Design 4</i>	<i>Design 4</i>	<i>Design 4</i>	<i>Design 2</i>	<i>Design 4</i>
<i>Minimum value:</i>	<i>Design 7</i>	<i>Design 5</i>	<i>Design 7</i>	<i>Design 1</i>	<i>Design 5</i>

Table 8-4 shows the displacements for each design and each loadcase. The table suggests that design 5 or 7 is the best, since both designs have the lowest displacement for two loadcases. Design 4 has the highest displacement for every loadcase apart from braking.

Table 8-5 - Strain for each loadcase for each new design

	Strain				
	<i>Top wishbone</i>	<i>Bottom wishbone</i>	<i>Acceleration</i>	<i>Braking</i>	<i>Combo</i>
Design 1	2.47E-04	7.12E-04	8.31E-04	1.41E-03	7.12E-04
Design 2	2.38E-04	7.39E-04	8.38E-04	7.10E-04	7.40E-04
Design 2 (U)	2.50E-04	7.99E-04	8.65E-04	6.67E-04	8.06E-04
Design 3	2.45E-04	8.66E-04	8.90E-04	5.96E-04	8.71E-04
Design 3 (U)	2.50E-04	7.44E-04	8.74E-04	5.74E-04	7.51E-04
Design 4	3.13E-04	9.06E-04	1.44E-03	5.65E-04	9.08E-04
Design 5	2.34E-04	7.83E-04	8.29E-04	6.65E-04	7.90E-04
Design 6	2.62E-04	8.74E-04	7.88E-04	4.61E-04	8.78E-04
Design 7	2.31E-04	7.98E-04	8.24E-04	5.09E-04	8.11E-04
Design 8	2.56E-04	1.03E-04	6.30E-04	5.01E-04	8.90E-04
<i>Maximum value:</i>	<i>Design 4</i>	<i>Design 4</i>	<i>Design 4</i>	<i>Design 1</i>	<i>Design 4</i>
<i>Minimum value:</i>	<i>Design 7</i>	<i>Design 8</i>	<i>Design 8</i>	<i>Design 6</i>	<i>Design 1</i>

Table 8-5 shows the strains for each design and each loadcase. The table suggests that design 8 is the best design, with the minimum strain for bottom wishbone load of 7279.4N and acceleration load of 6145.8N.

8.2.2 Design Studio Stiffnesses

Stiffnesses of each attachment point were calculated in Design Studio for the original upright. Design Studio was used as it is more accurate than the 'black box' FE solver contained in SolidWorks.

'Firstly, virtually fully automatic "Black Box" FEM contained in all CAD programmes (computer-aided design) for rough calculations by the design engineer (e.g. in designing a bumper) and, secondly, the use reserved for specialists of special FEM programs (e.g. in body calculations, in axle development or in driving dynamics).'

(Reif, et al., 2018, p. 194)

The new designs aimed to maintain these stiffnesses. Eight different designs were developed using the topology optimization and FEA results. Design 8 was determined as the best design and selected as the final upright design. A 5mm draft mesh was used in Design Studio, however, using Design Studio's 'modify elements' feature, the element size was halved to ensure the same trends were seen with a smaller mesh. On average the smaller mesh led to 14% smaller stiffnesses than that of the 5mm draft mesh.

8.2.3 Design 1 – Initial design

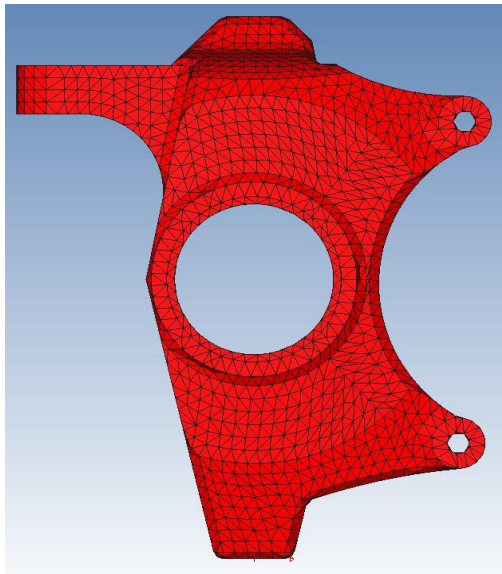


Figure 8-4 - Design 1 front

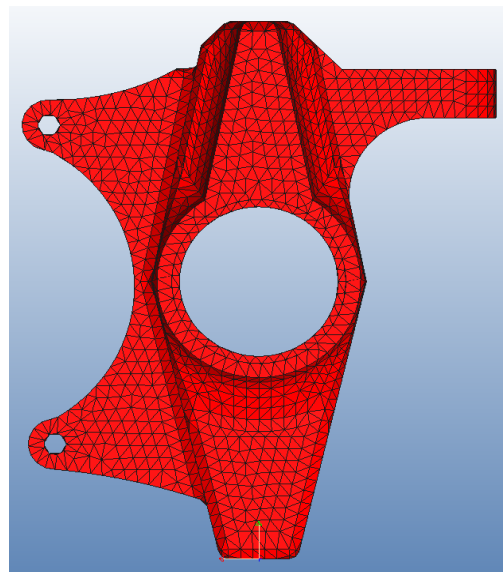


Figure 8-5 - Design 1 rear

Figure 8-4 and 8-5 show the first new design developed. It was decided to make the new upright out of a single piece of 7075-T6 aluminium as large stress concentrations were seen on edges of the original design with component contact. An FEA study conducted on a one-piece and two-piece prosthetic abutment for external hex butt-joint connection implant, show that under loading the single-piece abutments had lower von Mises stresses at the head and first two threads of the screw. Under preload, the transition between the body and head of the screw was higher for the two-piece abutment.

'The one-piece mini-conical abutment (M2) presented a more homogeneous behaviour of stress distribution'
(Moreira, et al., 2013)

Fatigue cracks mostly occur at areas with stress concentrations; therefore, it is important to eliminate from the design.

'The stress at the edge of a hole, or at a notch root, has a higher value than the remote stress. This phenomenon is called 'stress concentration'. Fatigue cracks mostly initiate at the sites of stress concentrations.'
(Murakami, 2019)

Fracture accidents are caused by fatigue. Investigations indicated that nearly all fractures originated from sites of stress concentrations, therefore, it was aimed to eliminate these from the new design.

'It has been said that 80%–90% of fracture accidents are caused by fatigue. Investigation indicates that almost 100% of these fractures start from the sites of stress concentrations at structural discontinuities such as holes, notches, shoulders, cracks, defects, and scratches.'

(Murakami, 2019)

Katwa's topology study and the final SolidWorks topology were used to create the shape of the brake support area (see *Figure 8-6*). Since the Young's modulus of steel is around 2.9 times greater than aluminium, the thickness of the brake support had to be thicker than that of the original design's steel bracket. Stress concentrations are located at areas of abrupt change in geometry in a load path, therefore sharp corners were avoided during the design process.

'They will be located in the small radii and sharp corners that are in a load path.'

(Thomas, n.d.)



Figure 8-6 - Design 1 and final topology overlay

Both topology studies were also used to inform the design of the steering arm support area. *Figure 8-6* shows how the steering arm was created from the final SolidWorks topology. It can be seen how the shape of the bottom wishbone area was chosen.

Table 8-6 - Original design and design 1 attachment point stiffnesses

Old Upright					New upright 1		
Attachement Point	Stiffness (N/m)				Stiffness (N/m)		
	X	Y	Z		X	Y	Z
Upper caliper-tierod-upright	1.558E+09	7.986E+08	2.696E+08		N/A	N/A	N/A
Lower caliper-upright	1.944E+09	2.697E+09	1.369E+09		N/A	N/A	N/A
Tierod-upright	2.555E+09	1.059E+09	7.810E+08		N/A	N/A	N/A
Lower upright	2.166E+09	2.943E+09	1.385E+09		N/A	N/A	N/A
Upper caliper	1.899E+08	1.109E+08	6.323E+06		2.899E+08	1.169E+08	2.358E+07
Lower caliper	2.305E+08	8.913E+07	3.133E+06		3.258E+08	1.183E+08	2.267E+07
Upper wishbone	6.826E+08	1.037E+09	1.873E+08		5.696E+08	7.409E+08	3.418E+08
Lower wishbone	4.206E+08	1.968E+09	1.764E+08		3.627E+08	1.189E+09	2.599E+08
Tierod	2.254E+08	2.690E+07	3.316E+07		3.936E+08	5.436E+07	3.826E+07
Hub	4.125E+08	2.184E+09	7.944E+08		7.152E+08	3.723E+09	1.237E+09

Table 8-6 compares the original and new design's stiffnesses in each direction. The cells highlighted in green are the higher value of stiffness. It shows how the new design had higher stiffnesses for each attachment point in each direction, apart from the upper and lower wishbone in X and Y direction. The original upright included four extra attachment points since it was a three-piece construction. As supported by Moreira et al. (2013), the three-piece construction is an inferior design.

8.2.4 Design 2 – Improving upper and lower wishbone stiffness

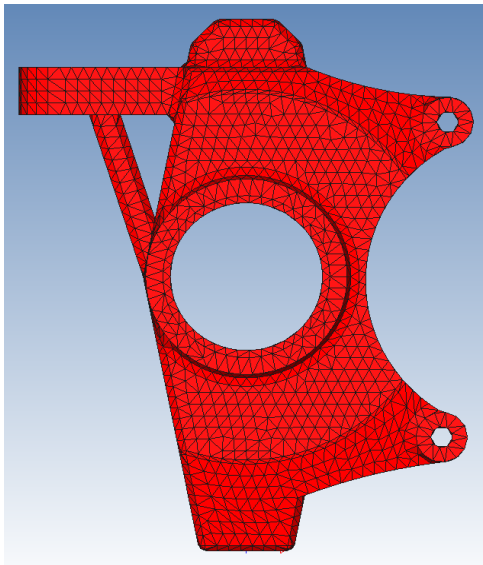


Figure 8-7 - Design 2 front

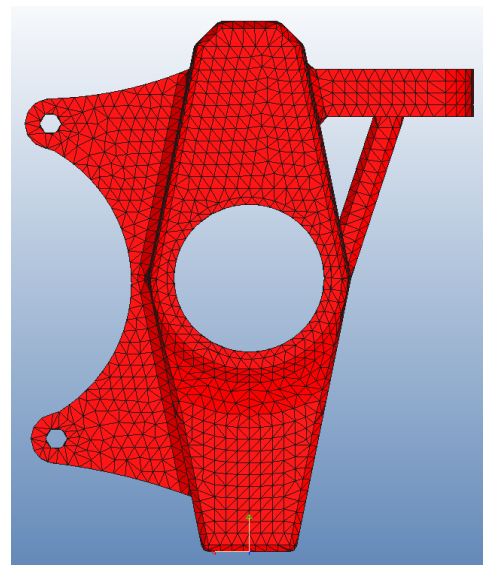


Figure 8-8 - Design 2 rear

The aim of this design was to improve on the upper and lower wishbone stiffnesses of design 1. This was done by increasing the width of the upright's lower wishbone area (see *Figure 8-7* and *8-8*). To improve the upper wishbone stiffness, the sweep cuts at the rear of design 1 were removed (see *Figure 8-5*). This increased stiffness in each direction; however, it was still inferior to the original design. To reduce mass, the large filleted edge below the steering arm from design 1 was replaced by a thin support coming from the centre of the upright to centre of steering arm. The first and second topology study showed that material was required in this area (see *Figure 8-9* and *8-10*). This steering arm support type also showed greater similarities to the final topology study (see *Figure 8-10*), and to Katwa's study, therefore, this support was used over the first design's support. Overall, the mass of this design increased by 152g compared to design 1, however, the stiffness of the lower wishbone in X direction increased and surpassed the original's design. Unfortunately, the stiffness of the steering arm in Z direction decreased and were subsequently worse than the original upright.

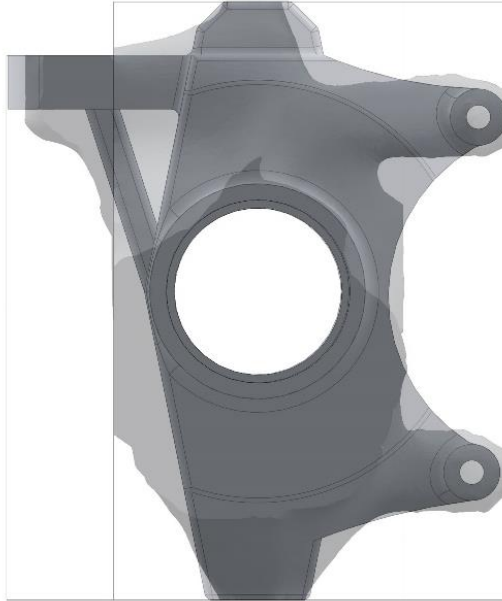


Figure 8-9 - Design 2 and second topology overlay



Figure 8-10 - Design 2 and final topology overlay

Table 8-7 - Original design and design 2 attachment point stiffnesses

Attachement Point	Old Upright				New upright 2		
	Stiffness (N/m)				Stiffness (N/m)		
	X	Y	Z		X	Y	Z
Upper caliper-tierod-upright	1.558E+09	7.986E+08	2.696E+08		N/A	N/A	N/A
Lower caliper-upright	1.944E+09	2.697E+09	1.369E+09		N/A	N/A	N/A
Tierod-upright	2.555E+09	1.059E+09	7.810E+08		N/A	N/A	N/A
Lower upright	2.166E+09	2.943E+09	1.385E+09		N/A	N/A	N/A
Upper caliper	1.899E+08	1.109E+08	6.323E+06		2.913E+08	1.123E+08	2.203E+07
Lower caliper	2.305E+08	8.913E+07	3.133E+06		3.432E+08	1.308E+08	2.376E+07
Upper wishbone	6.826E+08	1.037E+09	1.873E+08		6.059E+08	8.674E+08	3.539E+08
Lower wishbone	4.206E+08	1.968E+09	1.764E+08		4.224E+08	1.471E+09	2.880E+08
Tierod	2.254E+08	2.690E+07	3.316E+07		4.094E+08	6.184E+07	2.802E+07
Hub	4.125E+08	2.184E+09	7.944E+08		6.842E+08	3.940E+09	1.234E+09

Table 8-7 shows the stiffnesses for each attachment point in each direction compared to the original upright. The stiffness of the lower wishbone in X direction was improved over design 1 and was higher than the original designs. The highest forces through the upper and lower wishbones were in the Z direction, therefore, these stiffnesses were the most important. Unfortunately, the steering arm in Z direction had a lower stiffness than design 1 and the original designs. Despite this, it was a lighter way of supporting the steering arm compared to design 1.

8.2.5 Design 3 – Improving upper and lower wishbone and steering arm stiffness

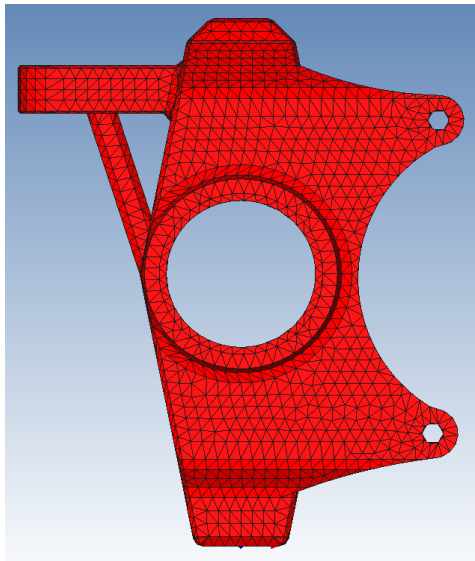


Figure 8-11 - Design 3 front

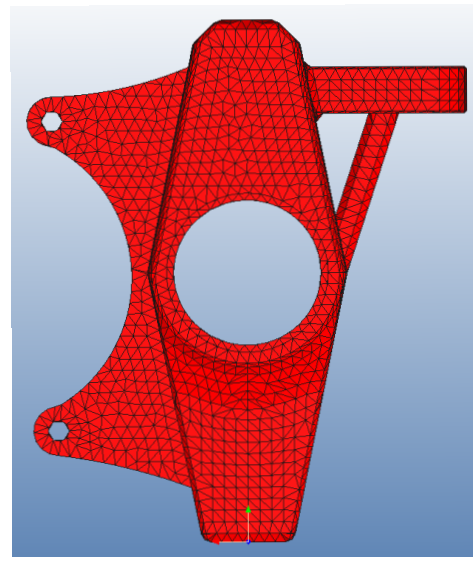


Figure 8-12 - Design 3 rear

For design 3, the revolved cut was removed which had reduced material from the brake support area (see *Figure 8-11*). A chamfer was added from the top edge of the upright to the top edge of the brake support area which aimed to improve the upper wishbone stiffness. To improve the lower wishbone stiffness the revolved cut at the rear of the upright below the hub bearing hole was modified to remove less material. The 3mm fillets remained on the steering arm support area, and where the steering arm intersected the upright. 1mm fillets were added to the steering arm support area, as stress concentrations appeared here under acceleration loading. Weight decreased by 92g compared to the second design, however, the lower wishbone stiffness in X direction was lower than the original designs. Also, each of the stiffnesses inferior to the original designs were lower than that of design 2.

Table 8-8 - Original design and design 3 attachment point stiffnesses

Old Upright					New upright 3		
Attachement Point	Stiffness (N/m)				Stiffness (N/m)		
	X	Y	Z		X	Y	Z
Upper caliper-tierod-upright	1.558E+09	7.986E+08	2.696E+08		N/A	N/A	N/A
Lower caliper-upright	1.944E+09	2.697E+09	1.369E+09		N/A	N/A	N/A
Tierod-upright	2.555E+09	1.059E+09	7.810E+08		N/A	N/A	N/A
Lower upright	2.166E+09	2.943E+09	1.385E+09		N/A	N/A	N/A
Upper caliper	1.899E+08	1.109E+08	6.323E+06		2.541E+08	1.124E+08	1.595E+07
Lower caliper	2.305E+08	8.913E+07	3.133E+06		3.064E+08	1.331E+08	1.746E+07
Upper wishbone	6.826E+08	1.037E+09	1.873E+08		5.981E+08	7.993E+08	3.272E+08
Lower wishbone	4.206E+08	1.968E+09	1.764E+08		3.891E+08	1.493E+09	2.455E+08
Tierod	2.254E+08	2.690E+07	3.316E+07		4.416E+08	6.804E+07	2.920E+07
Hub	4.125E+08	2.184E+09	7.944E+08		6.316E+08	3.730E+09	1.119E+09

Table 8-8 shows how the stiffnesses exhibit the same trend as design 2. Despite having more material at the upper and lower wishbone attachment point area, the stiffnesses were worse than design 2.

8.2.6 Design 4 – Reducing weight and improving stiffness

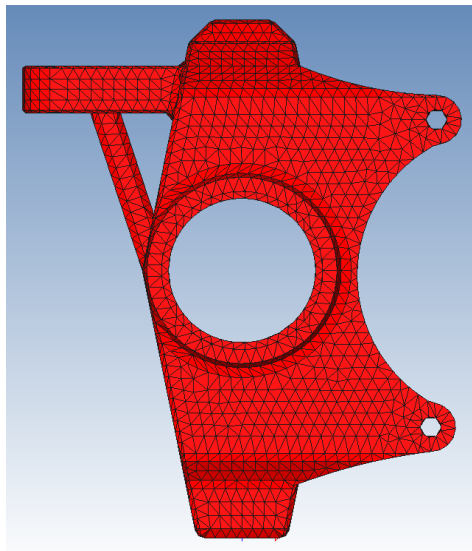


Figure 8-13 - Design 4 front

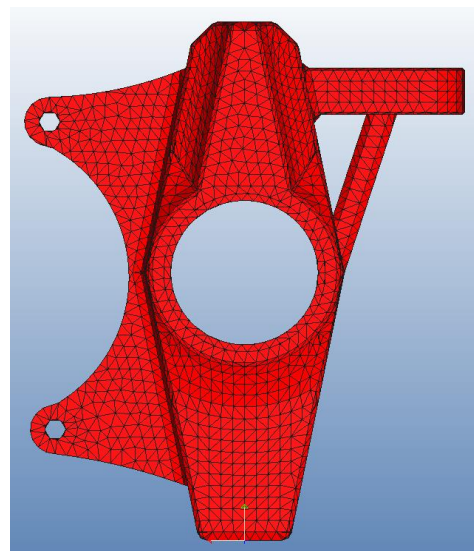


Figure 8-14 - Design 4 rear

This design was much like design 3, however, sweep cuts had been added to the rear of the design to remove mass (see *Figure 8-14*). Unfortunately, this further lowered the stiffnesses of the design, and from the FEA testing it was clear this was the worst design (see *Table 8-9*).

Table 8-9 - Original design and design 4 attachment point stiffnesses

Old Upright					New upright 4		
Attachement Point	Stiffness (N/m)				Stiffness (N/m)		
	X	Y	Z		X	Y	Z
Upper caliper-tierod-upright	1.558E+09	7.986E+08	2.696E+08		N/A	N/A	N/A
Lower caliper-upright	1.944E+09	2.697E+09	1.369E+09		N/A	N/A	N/A
Tierod-upright	2.555E+09	1.059E+09	7.810E+08		N/A	N/A	N/A
Lower upright	2.166E+09	2.943E+09	1.385E+09		N/A	N/A	N/A
Upper caliper	1.899E+08	1.109E+08	6.323E+06		2.762E+08	1.150E+08	1.740E+07
Lower caliper	2.305E+08	8.913E+07	3.133E+06		3.230E+08	1.456E+08	2.070E+07
Upper wishbone	6.826E+08	1.037E+09	1.873E+08		5.511E+08	7.256E+08	3.004E+08
Lower wishbone	4.206E+08	1.968E+09	1.764E+08		3.833E+08	1.463E+09	2.402E+08
Tierod	2.254E+08	2.690E+07	3.316E+07		4.342E+08	6.064E+07	2.659E+07
Hub	4.125E+08	2.184E+09	7.944E+08		6.128E+08	3.644E+09	1.073E+09

8.2.7 Design 5 – Improving upper and lower wishbone stiffness

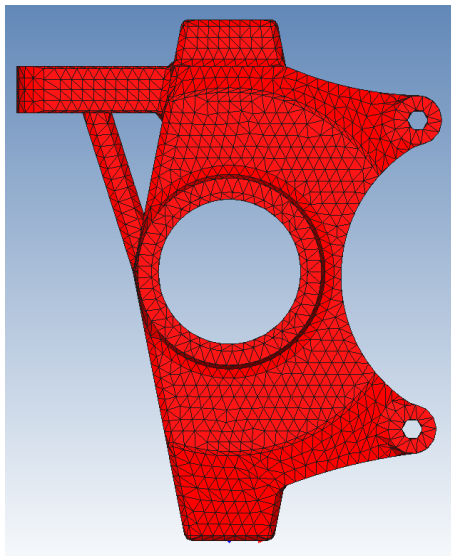


Figure 8-16 - Design 5 front

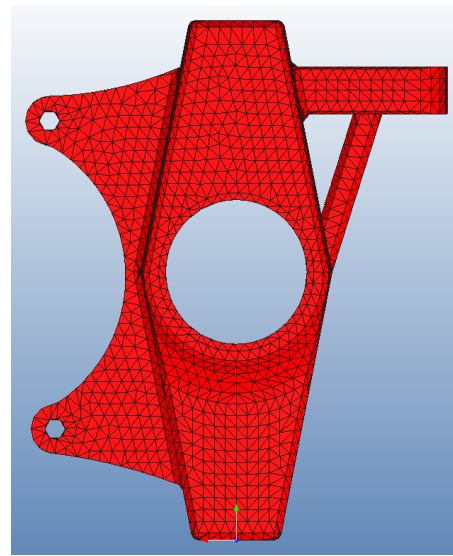


Figure 8-15 - Design 5 back

Design 5 brought back the revolved cut on the uprights front face which removed material from the brake support area and lower wishbone from design 2 (see *Figure 8-16*). No chamfer was added from the top edge of the upright and brake support since this had not improved design 3 and 4. Compared to each previous design, the upper wishbone area changed with the removal of chamfers around both edges parallel to the Z axis. From the FEA testing, design 5 had the lowest von Mises stress for the upper and lower wishbone and combination of loads. Four of the stiffnesses were inferior to the original design (see *Table 8-10*). The steering arm stiffness in Z direction could be improved by increasing the material on the support area, which led to design 6.

Table 8-10 - Original design and design 5 attachment point stiffnesses

Old Upright					New upright 5		
Attachement Point	Stiffness (N/m)				Stiffness (N/m)		
	X	Y	Z		X	Y	Z
Upper caliper-tierod-upright	1.558E+09	7.986E+08	2.696E+08		N/A	N/A	N/A
Lower caliper-upright	1.944E+09	2.697E+09	1.369E+09		N/A	N/A	N/A
Tierod-upright	2.555E+09	1.059E+09	7.810E+08		N/A	N/A	N/A
Lower upright	2.166E+09	2.943E+09	1.385E+09		N/A	N/A	N/A
Upper caliper	1.899E+08	1.109E+08	6.323E+06		3.039E+08	1.119E+08	2.182E+07
Lower caliper	2.305E+08	8.913E+07	3.133E+06		3.435E+08	1.308E+08	2.327E+07
Upper wishbone	6.826E+08	1.037E+09	1.873E+08		5.972E+08	8.343E+08	3.480E+08
Lower wishbone	4.206E+08	1.968E+09	1.764E+08		4.316E+08	1.579E+09	2.963E+08
Tierod	2.254E+08	2.690E+07	3.316E+07		4.077E+08	6.121E+07	2.754E+07
Hub	4.125E+08	2.184E+09	7.944E+08		6.880E+08	3.979E+09	1.263E+09

8.2.8 Design 6 – Improving steering arm stiffness

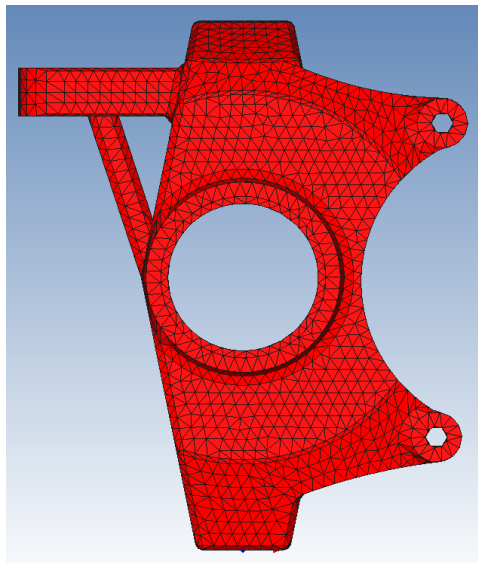


Figure 8-17 - Design 6 front

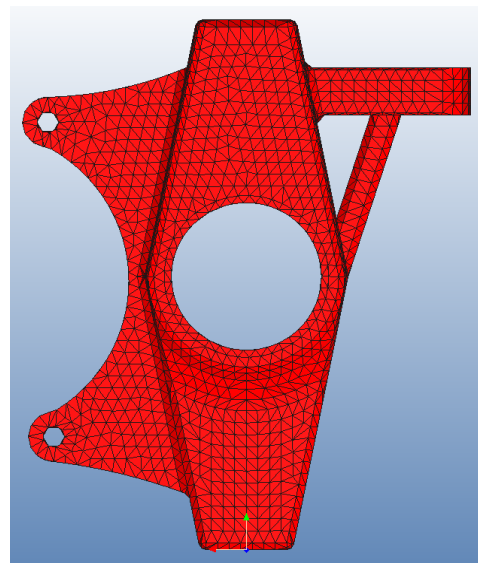


Figure 8-18 - Design 6 back

Figure 8-17 and 8-18 show design 6, which increased the material at the upper and lower wishbone by increasing the width. A large chamfer was added from the top of the upright to the brake support area. Compared to design 5, each of the stiffnesses inferior to the original upright were improved, however, the upper caliper in Y direction was now inferior (see Table 8-11).

Table 8-11 - Original design and design 6 attachment point stiffnesses

Old Upright					New upright 6		
Attachement Point	Stiffness (N/m)				Stiffness (N/m)		
	X	Y	Z		X	Y	Z
Upper caliper-tierod-upright	1.558E+09	7.986E+08	2.696E+08		N/A	N/A	N/A
Lower caliper-upright	1.944E+09	2.697E+09	1.369E+09		N/A	N/A	N/A
Tierod-upright	2.555E+09	1.059E+09	7.810E+08		N/A	N/A	N/A
Lower upright	2.166E+09	2.943E+09	1.385E+09		N/A	N/A	N/A
Upper caliper	1.899E+08	1.109E+08	6.323E+06		3.100E+08	5.666E+07	2.179E+07
Lower caliper	2.305E+08	8.913E+07	3.133E+06		3.587E+08	1.311E+08	2.346E+07
Upper wishbone	6.826E+08	1.037E+09	1.873E+08		6.297E+08	8.529E+08	3.774E+08
Lower wishbone	4.206E+08	1.968E+09	1.764E+08		4.316E+08	1.578E+09	2.960E+08
Tierod	2.254E+08	2.690E+07	3.316E+07		4.656E+08	6.893E+07	2.972E+07
Hub	4.125E+08	2.184E+09	7.944E+08		7.008E+08	4.036E+09	1.272E+09

8.2.9 Design 7 – Alternate design language

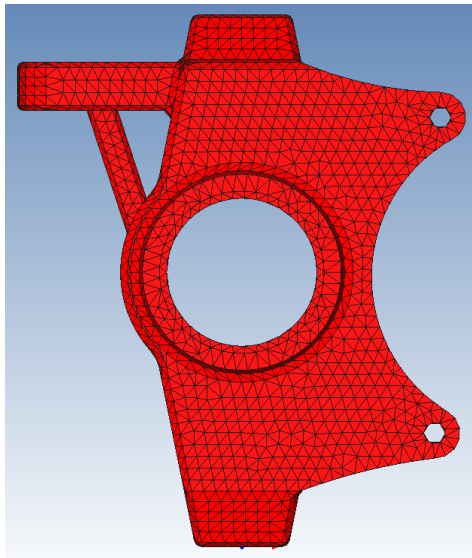


Figure 8-19 - Design 7 front

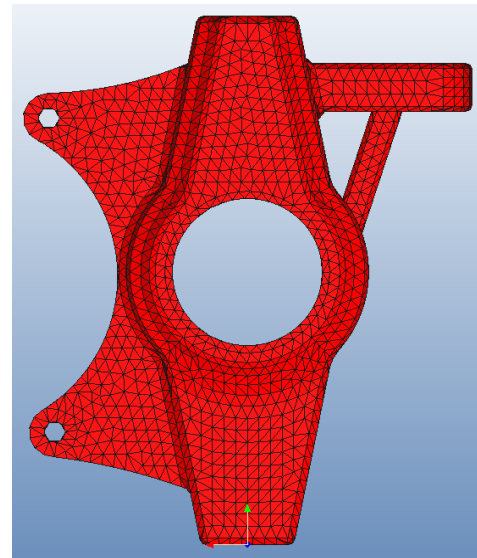


Figure 8-20 - Design 7 back

The design language was changed for design 7 (see *Figure 8-19* and *8-20*). Thinner columns coming from the centre of the design to the upper and lower wishbone attachment point were used, as the final topology included this approach. At 2073 grams design 7 was the heaviest. The brake support area shape had to be changed as the centre of the design was thicker than the previous designs. The upper caliper stiffness in the Y direction and steering arm stiffness in Z direction was improved compared to design 6. A larger 2mm radius was added to the steering arm and the stiffness was similar the original upright (see *Table 8-12*).

Table 8-12 - Original design and design 7 attachment point stiffnesses

Old Upright					New upright 7		
Attachement Point	Stiffness (N/m)				Stiffness (N/m)		
	X	Y	Z		X	Y	Z
Upper caliper-tierod-upright	1.558E+09	7.986E+08	2.696E+08		N/A	N/A	N/A
Lower caliper-upright	1.944E+09	2.697E+09	1.369E+09		N/A	N/A	N/A
Tierod-upright	2.555E+09	1.059E+09	7.810E+08		N/A	N/A	N/A
Lower upright	2.166E+09	2.943E+09	1.385E+09		N/A	N/A	N/A
Upper caliper	1.899E+08	1.109E+08	6.323E+06		3.108E+08	1.490E+08	2.065E+07
Lower caliper	2.305E+08	8.913E+07	3.133E+06		3.850E+08	1.967E+08	2.519E+07
Upper wishbone	6.826E+08	1.037E+09	1.873E+08		6.176E+08	8.230E+08	3.414E+08
Lower wishbone	4.206E+08	1.968E+09	1.764E+08		4.395E+08	1.616E+09	2.738E+08
Tierod	2.254E+08	2.690E+07	3.316E+07		4.500E+08	7.949E+07	3.154E+07
Hub	4.125E+08	2.184E+09	7.944E+08		6.980E+08	3.895E+09	1.205E+09

8.2.10 Design 8 – Best design

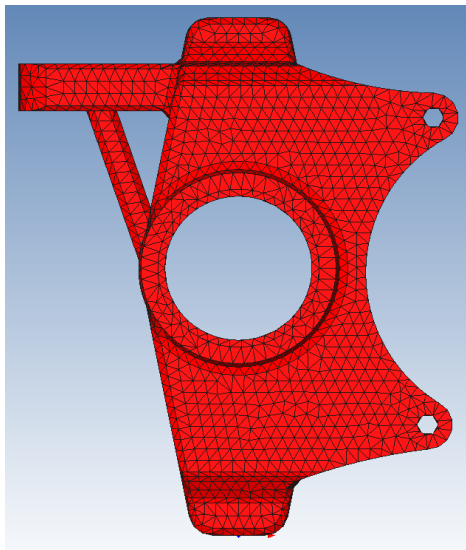


Figure 8-21 - Design 8 front

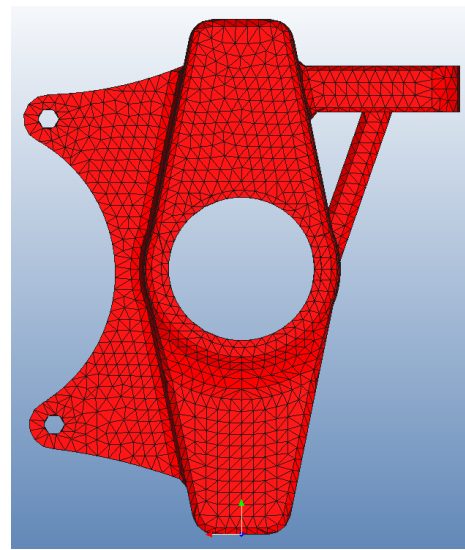


Figure 8-22 - Design 8 rear

Design 8 was selected as the best design (see *Figure 8-21* and *8-22*). Only 4 of the 18 stiffnesses were inferior to the original design. The weight was 1935 grams, meaning a weight saving of approximately 592 grams. The steering arm stiffness in Z direction was approximately 11% less compared to the original design (see *Table 8-13*). The stiffness could be improved by increasing the size of the steering support arm. Design 7 had a steering arm support 0.1mm wider and had increased stiffness. As with design 7, the brake support shape was changed, with a larger radius included between the top and bottom caliper mount. 10mm radii were included on the corners of the upright, parallel to the Z axis.

Table 8-13 - Original design and design 8 attachment point stiffnesses

Old Upright					New upright 8		
Attachment Point	Stiffness (N/m)				Stiffness (N/m)		
	X	Y	Z		X	Y	Z
Upper caliper-tierod-upright	1.558E+09	7.986E+08	2.696E+08		N/A	N/A	N/A
Lower caliper-upright	1.944E+09	2.697E+09	1.369E+09		N/A	N/A	N/A
Tierod-upright	2.555E+09	1.059E+09	7.810E+08		N/A	N/A	N/A
Lower upright	2.166E+09	2.943E+09	1.385E+09		N/A	N/A	N/A
Upper caliper	1.899E+08	1.109E+08	6.323E+06		3.072E+08	1.558E+08	2.178E+07
Lower caliper	2.305E+08	8.913E+07	3.133E+06		3.619E+08	1.898E+08	2.416E+07
Upper wishbone	6.826E+08	1.037E+09	1.873E+08		8.094E+08	3.285E+08	5.924E+08
Lower wishbone	4.206E+08	1.968E+09	1.764E+08		4.201E+08	1.575E+09	2.675E+08
Tierod	2.254E+08	2.690E+07	3.316E+07		4.351E+08	7.294E+07	2.964E+07
Hub	4.125E+08	2.184E+09	7.944E+08		6.664E+08	3.808E+09	1.187E+09



Figure 8-23 - Design 8 and second topology overlay



Figure 8-24 - Design 8 and final topology overlay

Figure 8-23 and 8-24 show the second and final topology study overlaid with this design. It shows how the second and final study was used to inform the brake bracket support area and the lower wishbone area. The final topology study (see Figure 8-24) and Katwa's study was also used to inform the steering arm support, despite a steeper angle being used.

Table 8-14 - Original design and design 8 with small radii removed attachment point stiffnesses

Attachement Point	Old Upright				New upright 8 - No fillets		
	Stiffness (N/m)				Stiffness (N/m)		
	X	Y	Z		X	Y	Z
Upper caliper-tierod-upright	1.558E+09	7.986E+08	2.696E+08		N/A	N/A	N/A
Lower caliper-upright	1.944E+09	2.697E+09	1.369E+09		N/A	N/A	N/A
Tierod-upright	2.555E+09	1.059E+09	7.810E+08		N/A	N/A	N/A
Lower upright	2.166E+09	2.943E+09	1.385E+09		N/A	N/A	N/A
Upper caliper	1.899E+08	1.109E+08	6.323E+06		3.050E+08	1.428E+08	1.880E+07
Lower caliper	2.305E+08	8.913E+07	3.133E+06		3.707E+08	1.759E+08	2.127E+07
Upper wishbone	6.826E+08	1.037E+09	1.873E+08		5.889E+08	8.030E+08	3.216E+08
Lower wishbone	4.206E+08	1.968E+09	1.764E+08		4.169E+08	1.593E+09	2.603E+08
Tierod	2.254E+08	2.690E+07	3.316E+07		4.293E+08	6.935E+07	2.712E+07
Hub	4.125E+08	2.184E+09	7.944E+08		6.644E+08	3.806E+09	1.162E+09

Design 8 was chosen as the final design but to manufacture, the smallest radii had to be removed as the 3-axis CNC machine at the university cannot cut any radii smaller than 3mm. Stiffnesses were tested for design 8 excluding the fillets with a radius smaller than 3mm (on the steering arm). The stiffness in the Y direction for the upper and lower wishbone were improved by removing the 2mm steering arm radii (see Table 8-14). Each of the other stiffnesses inferior to the original design were reduced compared to those with the smaller radii. However, removing the 2mm radii caused the stiffness of the upper wishbone in X direction to become inferior to the original upright.

8.3 Scaled model FEA

In order to conduct practical tests, 40% scale physical models were planned to be manufactured for the original and new upright design. It was planned that known weights would be hung off each of the attachment points and the displacements measured with a DTI gauge. The results from the practical tests would have been used to determine the model's stiffness. The change in stiffness for the original physical scaled model and new physical scaled models would have been compared to the FEA results. The difference in stiffness of the scaled models and FEA results should have been the same. The FEA results could have been compared to ensure correlation, and to validate the results.

FEA was conducted using scale CAD models and scaled loads, ready to compare to practical testing. FEA was also conducted with these models to determine the maximum load the models could withstand without plastic deformation. This minimised the risk of plastic deformation of the physical models when testing. For each study in *Section 8.3* a 0.8mm standard mesh with 'high quality' elements were used, and 4 Jacobian points selected.

8.3.1 Original upright with scaled loads

Table 8-15 - 6061 and 6082 scaled original upright with scaled loads

40% scale model from 6061-T6 aluminium			5.515E+07 yield
Loadcase	Stress (N/m ²)	Displacement (mm)	Strain
Top wishbone	5.459E+07	0.0117	5.645E-04
Bottom wishbone	2.043E+08	0.0567	2.000E-03
Acceleration	8.238E+07	0.0244	1.074E-03
Braking	2.266E+08	0.0498	2.476E-03
Steering	1.969E+07	0.0051	1.912E-04
Combo	2.074E+08	0.0576	2.031E-03
40% scale model from 6082-T6 aluminium			2.700E+08 yield
Loadcase	Stress (N/m ²)	Displacement (mm)	Strain
Top wishbone	5.459E+07	0.0116	5.564E-04
Bottom wishbone	2.043E+08	0.0559	1.971E-03
Acceleration	8.238E+07	0.0241	1.059E-03
Braking	2.266E+08	0.0491	2.441E-03
Steering	1.969E+07	0.0050	1.885E-04
Combo	2.074E+08	0.0568	2.002E-03
Combo - Steering, top, bottom wishbone loads			

Maximum operational loads for each attachment point were scaled to 40% original size, and the scaled original upright was tested using the 6061-T6 aluminium and the 6082-T6 aluminium (proposed for manufacture). As seen in *Table 8-15*, the von Mises stresses exceeded yield for the 6061-T6 scaled model for the bottom wishbone, acceleration, braking and combination loads. Interestingly, it was predicted that the same results would be observed for the full-size models with maximum operational loads and the scaled model with scaled loads, however, this was not observed. On average the upright made from 6061-T6 has 1.4% larger displacements and strains compared to the upright made from 6082-T6.

8.3.2 New upright with scaled loads

Table 8-16 - 7075 and 6082 scaled new upright with scaled loads

40% scale model from 7075-T6 aluminium			5.050E+08 yield
Loadcase	Stress (N/m ²)	Displacement (mm)	Strain
Top wishbone	5.624E+07	0.0125	6.643E-04
Bottom wishbone	2.440E+08	0.0430	1.802E-03
Acceleration	1.522E+08	0.0248	1.301E-03
Braking	1.189E+08	0.0379	1.256E-03
Steering	2.964E+07	0.0084	2.225E-04
Combo	2.435E+08	0.0429	1.848E-03
40% scale model from 6082-T6 aluminium			2.700E+08 yield
Loadcase	Stress (N/m ²)	Displacement (mm)	Strain
Top wishbone	5.666E+07	0.0128	6.826E-04
Bottom wishbone	2.439E+08	0.0443	1.907E-03
Acceleration	1.522E+08	0.0255	1.338E-03
Braking	1.189E+08	0.0390	1.292E-03
Steering	2.964E+07	0.0086	2.288E-04
Combo	2.435E+08	0.0441	1.901E-03
Combo - Steering, top, bottom wishbone loads			

Table 8-16 shows how the results of the new upright with scaled loads was similar for both the 7075-T6 model and 6082-T6 model. The results show how using 6082-T6 aluminium for testing instead of the 7075-T6 proposed for the manufactured upright will not have a large influence on results. On average the upright made from 6082-T6 had 2.7% larger displacements and 3.3% larger strains.

8.3.3 Maximum load before yield for the original upright design

Table 8-17 - 6061 and 6082 scaled original upright maximum loads before yield

40% scale model from 6061-T6 aluminium			5.515E+07 yield	
Loadcase	Stress (N/m ²)	Displacement (mm)	Strain	Max load before yeild (N)
Top wishbone	5.062E+07	0.0109	5.234E-04	1000
Bottom wishbone	4.912E+07	0.0136	4.807E-04	700
Acceleration	5.457E+07	0.0162	7.114E-04	1700
Braking	5.384E+07	0.0119	5.901E-04	700
Steering	3.281E+07	0.0085	3.187E-04	100
40% scale model from 6082-T6 aluminium			2.700E+08 yield	
Loadcase	Stress (N/m ²)	Displacement (mm)	Strain	Max load before yeild (N)
Top wishbone	2.632E+08	0.0557	2.683E-03	5200
Bottom wishbone	2.667E+08	0.0730	2.572E-03	3800
Acceleration	2.632E+08	0.0768	3.383E-03	8200
Braking	2.694E+08	0.0583	2.902E-03	3500
Steering	2.626E+08	0.0672	2.513E-03	800

Table 8-17 shows the maximum load before yield for the 40% scale original upright made from 6061-T6 and 6082-T6. Studies were conducted to ensure the models wouldn't yield during practical testing. The results show how the 6082-T6 material can withstand considerably higher loads before yield, due to its higher yield point.

8.3.4 Maximum load before yield for the new upright design

Table 8-18 - 7075 and 6082 scaled new upright maximum loads before yield

40% scale model from 7075-T6 aluminium			5.050E+08 yield	
Loadcase	Stress (N/m ²)	Displacement (mm)	Strain	Max load before yeild (N)
Top wishbone	5.007E+08	0.1108	5.914E-03	9600
Bottom wishbone	5.012E+08	0.0884	3.702E-03	6100
Acceleration	5.014E+08	0.0817	4.286E-03	8100
Braking	5.011E+08	0.1597	5.293E-03	12400
Steering	4.941E+08	0.1392	3.708E-03	1000
40% scale model from 6082-T6 aluminium			2.700E+08 yield	
Loadcase	Stress (N/m ²)	Displacement (mm)	Strain	Max load before yeild (N)
Top wishbone	2.627E+08	0.0594	3.165E-03	5000
Bottom wishbone	2.628E+08	0.0477	2.054E-03	3200
Acceleration	2.662E+08	0.0446	2.340E-03	4300
Braking	2.627E+08	0.0861	2.854E-03	6500
Steering	2.470E+08	0.0716	1.907E-03	500

Table 8-18 shows maximum load before yield for new design made from 7075-T6 and 6082-T6 aluminium. Due to its higher yield, the 7075-T6 model could withstand higher load before yield. By calculating the maximum load before yield, a suitable load could be applied to the physical model resulting in a high elastic displacement. A small load on the physical model would result in a very small displacement which would be hard to measure. According to Mitutoyo (2016), the error when measuring below 1mm with a 0.001mm graduation DTI gauge is approximately 0.005mm. The value of error of the DTI gauge would have a large effect on the results since displacements are so small.

To minimise errors in the practical tests, significant loads would have been used to create measurable displacements. For the 7075-T6 model with top wishbone loadcase a load of 950kg would have been the optimum, as it is just before yield, however, for physical testing this would have be impractical. This highlights the importance and benefits of FEA and CAE, as tests can be conducted that would otherwise be impractical and expensive.

8.3.5 6061 and 6082 aluminium scaled model stiffnesses

8.3.5.1 Original design

Table 8-19 - 6061 and 6082 scaled original upright attachment point stiffnesses

<i>40% scale model from 6061-T6 aluminium</i>	
Attachement	Stiffness (N/m²)
Top wishbone	1.055E+08
Bottom wishbone	6.487E+07
Hub	1.769E+08
Upper caliper	3.103E+07
Lower caliper	3.250E+07
Steering	1.335E+07
<i>40% scale model from 6082-T6 aluminium</i>	
Attachement	Stiffness (N/m²)
Top wishbone	1.070E+08
Bottom wishbone	6.582E+07
Hub	1.794E+08
Upper caliper	3.152E+07
Lower caliper	3.295E+07
Steering	1.355E+07

The stiffness of each attachment point was calculated using the displacements and loads from *Table 8-17*. The probe tool in SolidWorks was used to calculate displacement at the attachment points, and the displacement was displayed at the same nodes for the 6061-T6 model and 6082-T6 model. The results in *Table 8-19* show using the 6082-T6 aluminium for manufacture would have a small effect on the stiffness, therefore, would be a suitable material for the manufacture of the scaled models. On average the 6082-T6 material had 1.4% larger stiffnesses. For the validation of the results, the difference in these stiffnesses and those in *Table 8-20* would be compared to the difference of the physical tests.

8.3.5.2 New design

Table 8-20 - 7075 and 6082 scaled new upright attachment point stiffnesses

40% scale model from 7075-T6 aluminium	
Attachement	Stiffness (N/m ²)
Top wishbone	1.034E+08
Bottom wishbone	9.002E+07
Hub	1.544E+08
Upper caliper	4.203E+07
Lower caliper	5.053E+07
Steering	8.137E+06
40% scale model from 6082-T6 aluminium	
Attachement	Stiffness (N/m ²)
Top wishbone	1.005E+08
Bottom wishbone	8.758E+07
Hub	1.501E+08
Upper caliper	4.086E+07
Lower caliper	4.911E+07
Steering	7.913E+06

The stiffness of each attachment point was calculated using the displacements and loads from *Table 8-18*. The difference in Young's modulus of 7075-T6 and 6082-T6 used for the new design is larger than the difference of 6061-T6 and 7075-T6 of the original design (see *Table 8-20*). This resulted in slightly larger differences in stiffnesses. On average the stiffnesses are 2.8% larger for the 7075-T6 model compared to the 6082-T6 model.

8.4 SolidWorks and Design Studio results comparison

SolidWorks and Design Studio solver results were compared to determine how the different solver algorithms would affect results. The final design was tested using the same loads and mesh.

The standard mesh was selected with maximum element size of 2.5mm, and parabolic tetrahedral elements used. The maximum operational loads at each attachment point were used for both studies. A mesh control of 1.1mm was defined (see *Figure 8-25*) and applied at areas of stress concentrations.

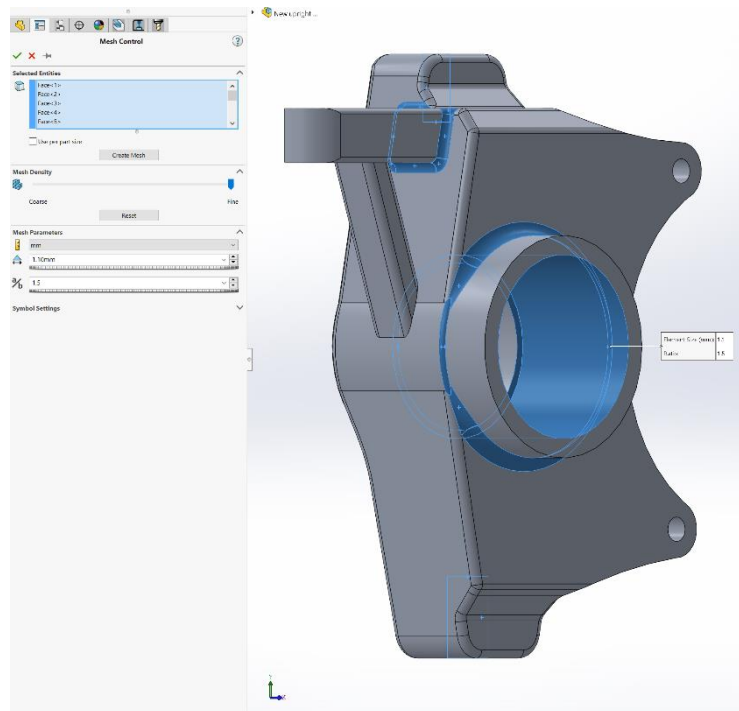


Figure 8-25 - Location of the mesh controls added to the model

A standard mesh with 0.7mm, 0.8mm, 0.9mm, 1.2mm and 1.5mm element sizes were tested prior to this mesh in Design Studio, however, all these studies failed due to lack of computational power (drive space was too low). A study with 0.7mm maximum element size was going to be conducted since this was the optimum element size found in the mesh independence study (see *Appendix 16.2*).

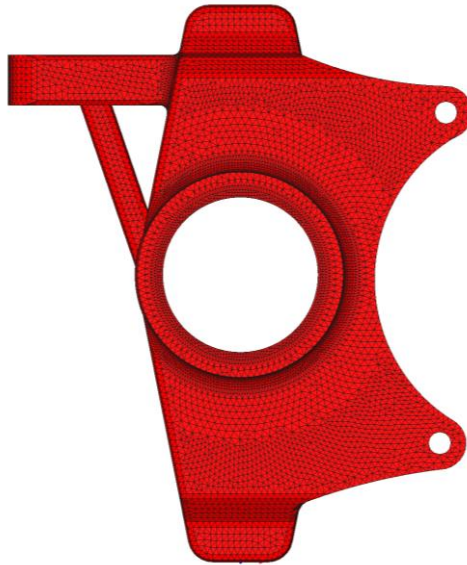


Figure 8-26 - 2.5mm mesh used with 1.1mm mesh controls

Figure 8-26 shows the mesh used for both studies. To improve the accuracy of results, areas with high stress had a mesh control with 1.1mm elements. Areas of the upright with small elements from the H-refinement studies was used to inform this mesh (see *Appendix 16.3*). Figure 8-27 shows the elemental stress for the bottom wishbone load. A high area of stress can be seen at the inner hub edge; therefore, a mesh control was applied at this location. Before the mesh control was defined there were neighbouring elements with large differences in stress which should be avoided.

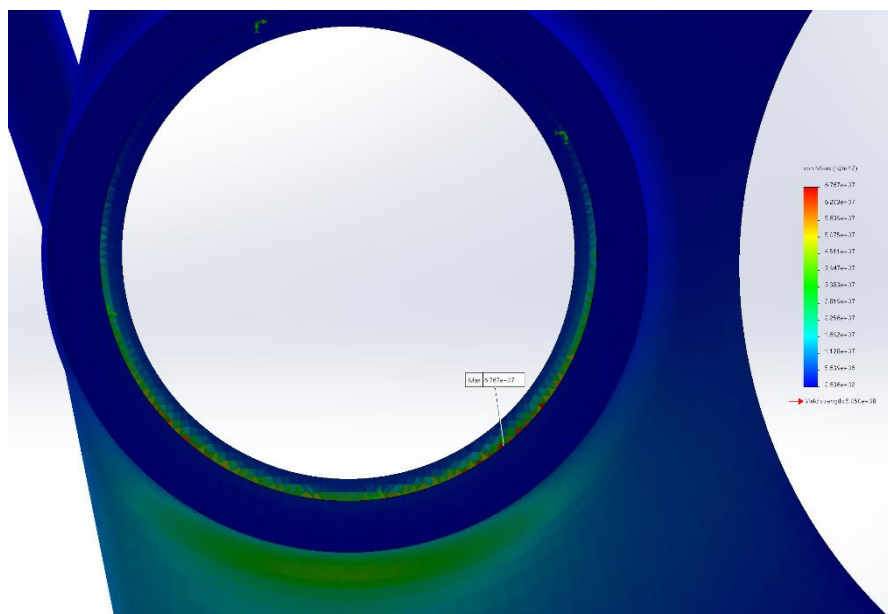


Figure 8-27 - Elemental von Mises stress for bottom wishbone load

Mesh Details	
Study name	Top wishbone (-Default-)
Mesh type	Solid Mesh
Mesher Used	Standard mesh
Automatic Transition	Off
Include Mesh Auto Loops	Off
Jacobian points	4 points
Mesh Control	Defined
Element size	2.5 mm
Tolerance	0.125 mm
Mesh quality	High
Total nodes	835940
Total elements	585252
Maximum Aspect Ratio	1,048
Percentage of elements with Aspect Ratio < 3	99.7
Percentage of elements with Aspect Ratio > 10	0.00171
% of distorted elements (Jacobian)	0
Time to complete mesh(hh:mm:ss)	00:00:29
Computer name	DES

Figure 8-28 - Mesh details

Figure 8-28 shows the mesh details from SolidWorks. The number of elements used was 585,252, however, the mesh independence study indicated that 10,000,000 elements should have been used. This was not possible as a mesh with more than 600,000 elements would fail using the technology available. The maximum aspect ratio for the mesh was 1,048 which is very high.

‘A general rule of thumb from the Simulation Training manual is to keep the aspect ratio below 50 in areas where stress is of crucial importance.’

(Hawk Ridge Systems, 2013)

Since the computational power was not available this mesh was used to compare SolidWorks and Design Studio. For a good mesh, a value of around 15 would be expected for maximum aspect ratio. Petrock (2017), states how depending on complexity the maximum aspect ratio should be between 10 - 30. A value of 99.7% was obtained for the percentage of elements with aspect ratio less than three, which indicate a good mesh. The percentage of elements with aspect ratio less than three...

‘Should be in the low 90s for more complicated models upwards of 99 for simpler models’

(Petrock, 2017)

Finally, since the maximum aspect ratio was so high the percentage of elements with aspect ratio over 10 was 0.00171%. This value should be as low as possible, ideally zero. Had more time, and computational power been available this mesh would have been improved, and an element size of 0.7mm used.

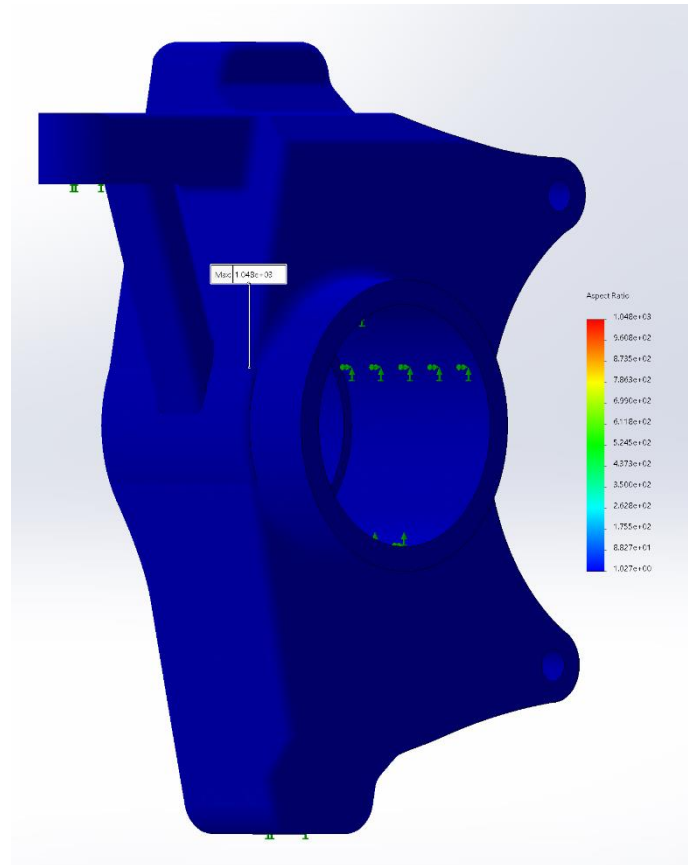


Figure 8-29 - Aspect ratio plot with location of element with maximum aspect ratio

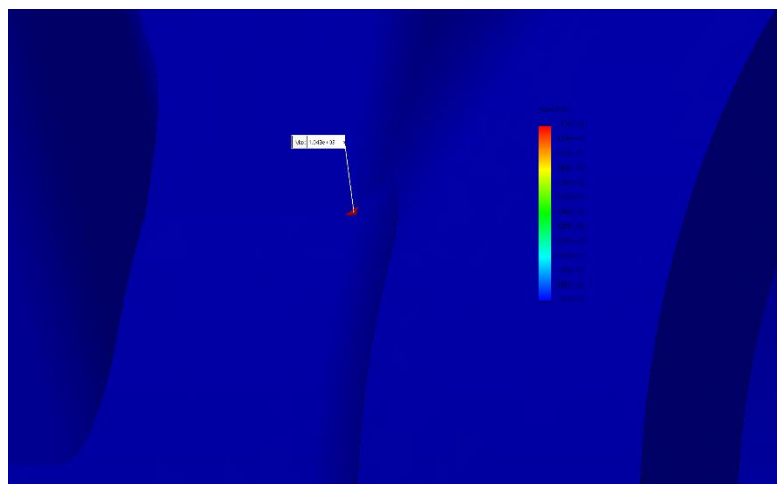


Figure 8-30 - Location of element with maximum aspect ratio

Figure 8-29 and 8-30 show the location of the maximum aspect ratio. Since it was not in a high stressed area it would have had minimal effect on the results.

8.4.1 Elemental values

Design Studio defaults by displaying the elemental values. SolidWorks uses elemental values as default for strain, and there is no option to change the displacement. According to Michalski (2019), to obtain a stiffness matrix, the FE solver uses numerical integration over the element volume, and it does this at the Gauss points. The stresses are calculated at the Gauss points, where they are most accurate, and then are extrapolated to the rest of the element.

'In this method, each element individually looks at the stresses at its nodes from the Gaussian points. The stress at the element is the average of the stresses seen at its corresponding nodes.'
(3DVision, 2008)

Table 8-21 - Elemental FEA results for SolidWorks and Design Studio

SolidWorks			
Loadcase	Von mises (N/m ²)	Displacements (mm)	Strain
Top wishbone	2.317E+07	1.199E-02	2.853E-04
Bottom wishbone	6.767E+07	4.080E-02	8.333E-04
Acceleration	5.561E+07	2.436E-02	6.849E-04
Steering	1.040E+07	1.350E-02	1.281E-04
Upper caliper	3.767E+07	3.715E-02	4.639E-04
Lower caliper	3.414E+07	3.034E-02	4.204E-04
Design Studio			
Loadcase	Von mises (N/m ²)	Displacements (mm)	Strain
Top wishbone	2.328E+07	1.146E-02	2.881E-04
Bottom wishbone	6.820E+07	3.860E-02	8.415E-04
Acceleration	5.097E+07	2.248E-02	6.303E-04
Steering	1.033E+07	1.279E-02	1.274E-04
Upper caliper	2.872E+07	3.229E-02	3.553E-04
Lower caliper	2.736E+07	2.609E-02	3.387E-04
Element values			
Loadcase	Difference (%)	Difference (%)	Difference (%)
Top wishbone	0.5	4.6	1.0
Bottom wishbone	0.8	5.7	1.0
Acceleration	9.1	8.4	8.7
Steering	0.7	5.6	0.5
Upper caliper	31.2	15.1	30.6
Lower caliper	24.8	16.3	24.1

Table 8-21 shows the results for SolidWorks and Design Studio. For stress, the smallest difference in SolidWorks and Design Studio results was 0.5% and the largest 31.2%. The smallest difference in displacement was 4.6% and the largest 16.3%. The smallest difference in strain was 0.5% and the largest 30.6%. Overall, both solvers showed similar trends, however, the results differ substantially.

This can be explained as both solvers will use different algorithms for calculations, and different constraints were defined. Both models were constrained in 6 D.O.F, however, Design Studio was using an RBE2.

8.4.2 Nodal values

In SolidWorks stress defaults to nodal values. Nodal values are described as the averaged value at each node. If elements sharing a node are considered, the nodal stresses are obtained from each elements Gaussian points.

'The value shown at the node is the average of the stresses from the gaussian points of each element that it belongs to.' (3DVision, 2008)

Table 8-22 - Nodal FEA results for SolidWorks and Design Studio

SolidWorks			
Loadcase	Von mises (N/m ²)	Displacements (mm)	Strain
Top wishbone	2.380E+07	N/A	2.931E-04
Bottom wishbone	8.584E+07	N/A	1.057E-03
Acceleration	6.091E+07	N/A	7.501E-04
Steering	1.216E+07	N/A	1.498E-04
Upper caliper	4.605E+07	N/A	5.671E-04
Lower caliper	4.116E+07	N/A	5.068E-04
Design Studio			
Loadcase	Von mises (N/m ²)	Displacements (mm)	Strain
Top wishbone	1.260E+07	1.157E-02	1.556E-04
Bottom wishbone	4.403E+07	3.890E-02	5.438E-04
Acceleration	3.973E+07	2.263E-02	4.918E-04
Steering	9.565E+06	1.293E-02	1.181E-04
Upper caliper	2.810E+07	3.270E-02	3.476E-04
Lower caliper	2.558E+07	2.638E-02	3.165E-04
Node values			
Loadcase	Difference (%)	Difference (%)	Difference (%)
Top wishbone	88.9	N/A	88.4
Bottom wishbone	95.0	N/A	94.4
Acceleration	53.3	N/A	52.5
Steering	27.1	N/A	26.9
Upper caliper	63.9	N/A	63.1
Lower caliper	60.9	N/A	60.1

Table 8-22 shows the results obtained from both solvers. The difference in solvers is larger for the nodal values, and this is to be expected as averaging has occurred, and different methods are likely employed by both solvers.

The elemental values should be the preferred method for post-processing, as areas of the mesh that require refining are more visible. As nodal values are averaged, they can disguise areas of weakness in a mesh, for example, elements with singularities.

'Such averaging will disguise any mesh effects that need to be addressed for accurate solutions.'
(Harish, 2019)

On the other hand, nodal values are normally used in post-processing.

'Normally, contour results in the Mechanical application are displayed as averaged results.'
(Simutech, n.d.)

Had the results been validated, the FEA results could have been proved to show the correct results meaning nodal values could have been used. Since nodal values are averaged results, the FEA requires conformation before use. In industry FEA results are validated. The post-processing looks more effective using nodal values, it therefore is used in industry. The elemental values were used during the project since no validation was carried out.

The elemental and nodal values should be similar, and a large difference in results highlight the fact that the mesh is too coarse. For SolidWorks the nodal values of stress were on average 16% larger than the elemental values. For Design Studio the nodal values of stress were on average 31% larger than the elemental values. This suggest that both studies should have had a finer mesh, however, with the limited computational power available this was not possible.

9 Pandemic Impact

9.1 Manufacture of original and new design

To ensure validity of the FEA results, scaled models of the original and new upright design were going to be manufactured using the 3-axis CNC machine and practical testing conducted on the models to determine stiffness. The results from the practical testing and FEA were going to be compared to determine whether both results correlated. This would ensure that the FEA results were valid and accurate, since various assumptions are made during FEA. According to Broekaart (2015), the assumptions for linear static analysis include constant material production grade, material is assumed to be linear, forces are being applied slowly, contact doesn't change with time and small deformations. If the upright were to be used on the MCR, then prototypes would have to be manufactured and tested.

'FEA does not eliminate the need for prototypes, but it can shorten the process.'

(Thornton, 2011)

A scale of 40% was chosen for the models, and 6082-T6 aluminium was chosen for its low cost and its similar Young's Modulus to 6061 and 7075 aluminium (see *Table 9-1*). Since 6061-T6, 6082-T6 and 7075-T6 aluminium have similar Young's modulus values, by using 6082 for practical testing the change in stiffness would have been negligible (see *Table 9-1*).

A 114.3mm (4.5") square bar with length 175mm was purchased from '1stChoiceMetals.co.uk' on 4/03/20 (see *Appendix 16.8* for invoice). The company offered cutting, and approximately 5mm of material was lost with each cut. Two pieces with approximately 54mm length were cut for the uprights and two pieces with approximately 24mm lengths cut for the CNC jigs. Since the material was cut by the material supplier the aluminium billets were ready for machining. On the 5/03/20 both the CAD files for the scaled original and new upright design were sent to Cooper for manufacture. Meetings were organized with Cooper to check on the status of the machining, see (*Appendix 16.9*). On the 20/03/20 all university facilities closed due to COVID-19; therefore, the models could not be machined, and practical testing could not be conducted. This had a large impact on the project, as all the FEA results could not be verified to ensure they were valid.

Table 9-1 - Young's Modulus of different aluminium grades
(Thyssenkrupp, n.d.)

<i>Aluminium alloy</i>	<i>Young's Modulus (GPa)</i>
1050	71.0
2014	71.0
3103	69.5
6061	70.0
6082	70.0
7075	73.1

9.2 Lack of facilities

Despite having the required software for the project, the computational power available following lockdown inhibited further FEA studies. Approximately 15 hours of FEA solves were attempted only to fail due to lack of drive space. Studies on SolidWorks and Design Studio using accurate meshes, with element size of 0.7mm, were unable to be conducted due to a shortage of drive space. Studies using 0.8mm, 1.2mm and 1.5mm element size also all failed in Design Studio.

Following the closure of the university, productivity of the project decreased as less resources and technology were available. A variety of types of references were aimed to be cited during the report, however, with the lack of library resources available most sources were found online.

Due to the duration of lockdown being unknown, leaving Swansea while travel was still allowed seemed wise. This move disrupted the flow of the project. Despite best efforts, having five adults working from home impacted the project due to limited space, resources and technology. The online support from the supervisor was invaluable, however, support and discussion with fellow students was missed.

10 Further work

More design development would have been carried out had a longer timescale been available. The aim was to create an upright design with each attachment point as stiff as the originals. Most attachment points were stiffer than the original design, however, further work could have improved each point. Further weight reduction could have been achieved with more development time. More time would have been spent analysing the best material choice for the new design. Despite being considerably cheaper, 6082-T6 didn't have a large effect on the displacements and strains compared to 7075-T6 aluminium.

A topology study using Design Studio would be conducted, and a larger domain created, which may have allowed for different designs. Maximum lateral loads during operation were calculated for the upright, however, maximum longitudinal loads could have also been calculated. Investigating what combination of loads would be observed during operation would have been beneficial in further optimizing the upright. Non-linear analysis would have been performed, to observe where the upright would fail under different loading. This could have been used alongside linear analysis to further improve the design.

Alongside more development time, physical models could have been produced. Practical testing using these physical models could have been conducted to verify the FEA results obtained in the project. Verification would mean more confidence in the results.

'Errors are inevitable in finite element models - the more complex the model, the higher the probability that they will occur. The important thing is to put a process in place to catch them before they do any damage - the verification process.'

(Veikos, 2016)

FEA is only a support activity, and if the upright failed during operation this could endanger the driver's life, meaning practical testing is imperative to ensure that the new upright is of high quality and is safe.

'Although a most exciting field of activity, engineering analysis is clearly only a support activity in the larger field of engineering design.'

(Bathe, 2016, p. 12)

11 Conclusion

Overall, the project was a success. Despite having been affected by the COVID-19 pandemic, most of the projects aims were achieved. A new front upright was developed for the MCR race-car resulting in a weight saving of approximately 23.4%. The aim of reducing the current uprights weight by 0.6kg was achieved with a weight saving of approximately 592g.

Most attachment points surpassed the original uprights stiffnesses in each direction. The original upright had not been optimized, suggesting that each attachment points stiffness had not been considered during development. Therefore, had more development time been available, the stiffnesses required for each attachment point could have been calculated. This would mean of the new design could be proved to be sufficiently strong. Further work could have increased the stiffness of each attachment points. The weight of the final design could have been further reduced. Overall, the stress distributions are better on the new upright design, meaning it would last longer before failure (see *Appendix 16.5*).

The aim of developing a new upright which could be manufactured in the university was achieved, and despite using expensive materials, overall, the cost of the uprights would be relatively low. For two uprights and materials, Xometry quoted approximately £1170 (see *Appendix 16.7*). Considering a weight saving of over 1.18kg could be achieved for the car, this upgrade is worthwhile. Also, since the CNC machine was available at the university, only the materials would have to be purchased.

A successful new design for the MCR front upright was produced. The new design was a simple design, which included design features found in the topology studies. A greater understanding of the development process has been gained from this project, and further knowledge of CAE has been acquired from both theoretical and physical work undertaken during this project.

12 References

- Moreira, W. et al., 2013. *A three-dimensional finite element study on the stress distribution pattern of two prosthetic abutments for external hexagon implants*, Rockville: Pub Med.
- 3DVision, 2008. *Nodal versus Elemental Stresses*. [Online]
Available at: <https://www.cati.com/blog/2008/04/nodal-versus-elemental-stresses/>
- Abbey, T., 2016. *Stress in FEA: Part 3*. [Online]
Available at: <https://www.digitalengineering247.com/article/stress-in-fea-part-3>
- Akin, J. E., 2009. *Finite Element Analysis Concepts via Solidworks*. [Online]
Available at: https://www.clear.rice.edu/mech403/HelpFiles/FEAC_final.pdf
- Altair Univeristy, 2018. *Practical Aspects of Structural Optimization*. Troy: Altair Univeristy.
- Altair University, 2015. Practical Aspects of Finite Element Simulation . In: *A Study Guide*. s.l.:s.n., pp. 16-17.
- Alteams, 2019. *Aluminium casting method comparison – Pros and cons of different casting methods for aluminium parts*. [Online]
Available at: <https://www.alteams.com/aluminium-casting/>
- AMFG, 2019. *Expert Interview: AMendate on its Automated Topology Optimisation Software for 3D Printing*. [Online]
Available at: <https://amfg.ai/2019/02/06/amendate-topology-optimisation-software-for-3d-printing-expert-interview/>
- Aremu, et al., 2010. *Suitability of SIMP and BESO Topology*. [Online]
Available at: <https://sffsymposium.engr.utexas.edu/Manuscripts/2010/2010-57-Aremu.pdf>
- ASM Aerospace Specification Metals, n.d. *Technical Data Sheet*. [Online]
Available at: <http://www.aerospacemetals.com/aluminum-distributor.html#tech>
- Autodesk, n.d. *Topology Optimization*. [Online]
Available at: <https://www.autodesk.com/solutions/topology-optimization>
- AWC, 2016. *6061-T6 or 7075-T6: Pros, Cons, and Decisions*. [Online]
Available at: <https://americanweaponscomponents.com/6061-t6-or-7075-t6-pros-cons-and-decisions>
- Batchu, S., 2015. *RBE2 vs RBE3*. [Online]
Available at: <https://www.stressebook.com/rbe2-vs-rbe3/>
- Bathe, K.-J., 2016. *Finite Element Procedures*. 2nd ed. s.l.:s.n.

Broekaart, D., 2015. *Why should you consider performing non-linear finite element analysis?*. [Online]
 Available at: <https://info.simuleon.com/blog/why-should-you-consider-performing-non-linear-finite-element-analysis>

Bunner, A., 2017. *What Are Project Milestones and How Do We Define Them?*. [Online]
 Available at: <https://www.clarizen.com/what-are-project-milestones/>

CAD2M, 2016. *SOLIDWORKS Simulation: Get connected with Contacts*. [Online]
 Available at: <https://blogs.solidworks.com/tech/2016/11/solidworks-simulation-get-connected-contacts.html>

Cavallo, C., n.d. *6061 Aluminum vs. 7075 Aluminum - Differences in Properties, Strength and Uses*. [Online]
 Available at: <https://www.thomasnet.com/articles/metals-metal-products/6061-aluminum-vs-7075-aluminum/>

Clinton Aluminium, 2017. *6061 VS. 7075*. [Online]
 Available at: <https://www.clintonaluminum.com/6061-vs-7075/>

Comsol, 2017. *The Finite Element Method (FEM)*. [Online]
 Available at: <https://uk.comsol.com/multiphysics/finite-element-method>

Crahmaliuc, R., 2019. *75 Years of the Finite Element Method (FEM)*. [Online]
 Available at: <https://www.simscale.com/blog/2015/11/75-years-of-the-finite-element-method-fem/>

Cross, N., 2008. *Engineering Design Methods*. 4th ed. s.l.:Wiley.

Engineers Edge, n.d. *Casting processes*. [Online]
 Available at: https://www.engineersedge.com/casting_design.htm

FIA, n.d. *Types of Forging Processes*. [Online]
 Available at: <https://www.forging.org/types-of-forging-processes>

Gandomi, A. H., Yang, X.-S., Talatahari, S. & Alavi, A. H., 2013. *Metaheuristic Applications in Structures and Infrastructures*. 1 ed. s.l.:Elsevier.

GE Additive, n.d. *What is Additive Manufacturing?*. [Online]
 Available at: <https://www.ge.com/additive/additive-manufacturing>

GoEngineer, 2017. *SOLIDWORKS Simulation: An Explanation of the Global Contact Option*. [Online]
 Available at: <https://blogs.solidworks.com/tech/2017/08/solidworks-simulation-explanation-global-contact-option.html>

Harish, A., 2019. *How to Calculate Stress and Strain with FEM Software?*. [Online]
 Available at: <https://www.simscale.com/blog/2017/04/stress-and-strain/>

Harish, A., 2019. *How to Choose a Solver for FEM Problems: Direct or Iterative?*. [Online]
Available at: <https://www.simscale.com/blog/2016/08/how-to-choose-solvers-for-fem/>

Harish, A., 2020. *What is Convergence in Finite Element Analysis?*. [Online]
Available at: <https://www.simscale.com/blog/2017/01/convergence-finite-element-analysis/>

Harish, A., 2020. *What Is the Meaning of the Von Mises Stress and the Yield Criterion?*. [Online]
Available at: <https://www.simscale.com/blog/2017/04/von-mises-stress/>

Hawk Ridge Systems, 2013. *ADAPTIVE MESHING IN SOLIDWORKS SIMULATION*. [Online]
Available at: <https://hawkridgesys.com/blog/simulation-adaptive-meshing>

Hawk Ridge Systems, 2013. *Curvature based mesh advantages in SolidWorks Simulation*. [Online]
Available at: <https://hawkridgesys.com/blog/curvature-based-mesh-advantages>

Hearn, E. J., 1997. *Mechanics of materials 1*. 3 ed. Oxford: Butterworth Heinemann.

Larsson, R., 2016. *Methodology for Topology and Shape Optimization: Application to a Rear Lower Control Arm*. [Online]
Available at: [https://www.chalmers.se/SiteCollectionDocuments/Produkt-%20och%20produktionsutveckling/Nationell%20kompetensarena%20kring%20produktoptimering/Methodology for Topology and Shape Optimization report.pdf](https://www.chalmers.se/SiteCollectionDocuments/Produkt-%20och%20produktionsutveckling/Nationell%20kompetensarena%20kring%20produktoptimering/Methodology%20for%20Topology%20and%20Shape%20Optimization%20report.pdf)

Loughborough University, n.d. *Additive Manufacturing Research Group*. [Online]
Available at:
<https://www.lboro.ac.uk/research/amrg/about/the7categoriesofadditivemanufacturing/powderbedfusion/>

Manufacturing.net, 2018. *Comparing Metal Parts Manufacturing Methods*. [Online]
Available at: <https://www.manufacturing.net/operations/article/13196732/comparing-metal-parts-manufacturing-methods>

Mason, G., 2015. *CPM Industries' Blog*. [Online]
Available at: <http://info.cpm-industries.com/blog/bid/283266/Cast-vs-Billet>

McKenzie, V., n.d. *RBE2 vs RBE3 Rigid Elements in Nastran*. [Online]
Available at: <https://www.endurasim.com.au/rbe2-vs-rbe3-rigid-elements-nastran/>

Metal Supermarkets, 2018. *What is Annealing?*, s.l.: s.n.

Michalski, J., 2019. *What is Gauss point in FEM?*. [Online]
Available at: <https://www.quora.com/What-is-Gauss-point-in-FEM>

Mirzendehdel, A. M. & Suresh, K., 2017. *A Hands-on Introduction to Topology Optimization*. 1st ed. s.l.:CreateSpace Independent Publishing Platform.

Mitutoyo, 2016. *Quick guide to precision measuring instruments*, s.l.: s.n.

MSport Eng, n.d. *Gallery*. [Online]

Available at: http://www.msport-eng.com/?page_id=511

Murad, J., 2019. *Errors in FEA and Understanding Singularities (Beginners' Guide)*. [Online]

Available at: <https://www.simscale.com/blog/2016/06/errors-in-fea-and-singularities/>

Murakami, Y., 2019. *Metal Fatigue*. 2 ed. s.l.:s.n.

OnlineMetals.com, n.d. *Aluminum Rectangle Bar 7075 T6/T651 Cold Finish*. [Online]

Available at: <https://www.onlinemetals.com/en/buy/aluminum-rectangle-bar-7075-t6-t651-cold-finish>

Peters, G., 2018. *Twitter*. [Online]

Available at: https://twitter.com/Fisco_GP/status/1054357698068860933/photo/1

Peters, G., 2019. s.l.: s.n.

Petrock, S., 2017. *Is This a Good FEA Mesh?*. [Online]

Available at: <https://blogs.solidworks.com/tech/2017/08/good-fea-mesh-heres-answer-yes-no-need-know-meshing-infographic.html>

Prototech Asia, 2019. *How Much Does CNC Machining Cost: The Important Things to Know*. [Online]

Available at: <https://prototechasia.com/en/how-much-does-cnc-machining-cost>

Reif, K., Dietsche, K.-H. & E. A., 2018. *Bosch Automotiv Handbuch*. 10th ed. Karlsruhe: Robert Bosch.

Revellat, H., 2019. *FEA: A tool to manage risk in product innovation*. [Online]

Available at: <https://www.pddinnovation.com/fea-a-tool-to-manage-risk-in-product-innovation/>

Shewchuk, J. R., 1999. *Lecture Notes on Delaunay Mesh Generation*. [Online]

Available at: <http://web.mit.edu/ehliu/Public/ProjectX/Summer2005/delnotes.pdf>

Simmons, J. M., 2000. *Designing for investment casting*. [Online]

Available at: <https://www.machinedesign.com/learning-resources/basics-of-design/article/21832490/designing-for-investment-casting>

SimScale, n.d. *What is a Mesh?*. [Online]

Available at:

<https://www.simscale.com/docs/content/simwiki/preprocessing/whatisamesh.html#id1>

Simutech, n.d. *ANSYS WORKBENCH MECHANICAL: AVERAGED VS. UNAVERAGED CONTOUR RESULTS*.

[Online]

Available at: <https://www.simutechgroup.com/tips-and-tricks/fea-articles/93-fea-tips-tricks-pinball-contour-results>

Solidworks, 2018. *Topology Study*. [Online]

Available at: http://help.solidworks.com/2018/english/WhatsNew/c_generative_design_study.htm

SolidWorks, n.d. *Solid Mesh*. [Online]

Available at:

https://help.solidworks.com/2020/English/SolidWorks/cworks/c_Solid_Mesh.htm?verRedirect=1

SRCC SPORTS 2000, 2019. *Regulations*. [Online]

Available at: <https://www.sports2000.co.uk/wp-content/uploads/2019/03/SRCC-SPORTS-2000-DURATEC-CHAMPIONSHIP-REGS-2019-PUBLISHED-COPY-WEB-VERSION.pdf>

Staniforth, A., 1994. *Competition Car Suspension - Design, Construction, Tuning*. 2nd ed. s.l.:s.n.

Staniforth, A., 2006. *Competition car suspension*. 4 ed. s.l.:s.n.

ThomasNet, n.d. *Types of Casting Processes*. [Online]

Available at: <https://www.thomasnet.com/articles/custom-manufacturing-fabricating/types-of-casting-processes/>

Thomas, S., n.d. *Stress Concentrations: How to Identify and Reduce Them in Your Designs*. [Online]

Available at: <https://www.fictiv.com/hwg/design/stress-concentrations-how-to-identify-and-reduce-them-in-your-designs>

Thornton, J., 2011. *FEA and the Question of Credibility*. [Online]

Available at: <https://www.asme.org/topics-resources/content/fea-and-the-question-of-credibility>

Thyssenkrupp, n.d. *Properties of aluminium*. [Online]

Available at: <https://www.thyssenkrupp-materials.co.uk/properties-of-aluminium>

Value Design Consulting, n.d. *Stress Theories*. [Online]

Available at: <http://www.value-design-consulting.co.uk/stress-theories.html>

Veikos, N. M., 2016. *Verifying Your Finite Element Analysis Results*. [Online]

Available at:

<https://www.engineering.com/DesignSoftware/DesignSoftwareArticles/ArticleID/11356/Verifying-Your-Finite-Element-Analysis-Results.aspx>

Warren, R., 2019. *Component Contact or Contact Set? Which one makes sense for you?*. [Online]

Available at: <https://www.cati.com/blog/2019/09/component-contact-or-contact-set-which-one-makes-sense-for-you/>

Weck, P. O. d. & Kim, D. I. Y., 2004. *Finite Element Method*, s.l.: s.n.

XCEED Engineering, 2017. *Finite Element Analysis Convergence and Mesh Independence*. [Online]

Available at: <https://www.xceed-eng.com/finite-element-analysis-convergence-and-mesh-independence/>

Zienkiewicz, O., Taylor, R. & Zhu, J. Z., 2013. *The Finite Element Method: Its Basis and Fundamentals*.

6th ed. s.l.:Butterworth-Heinemann.

13 List of figures

Figure 4-1 - Project proposal Gantt chart	- 4 -
Figure 4-2 - Final Gantt chart prior to COVID-19 pandemic	- 5 -
Figure 4-3 - Final Gantt chart following the COVID-19 pandemic.....	- 7 -
Figure 5-1 - Typical finite element analysis workflow (Mirzendehtdel & Suresh, 2017)	- 10 -
Figure 5-2 - Number of straight lines is equivalent to mesh size in FEA (Altair University, 2015) ..	- 11 -
Figure 5-3 - Common element types used in FEA (SimScale, n.d.).....	- 12 -
Figure 5-4 - Common element types used in FEA (Mirzendehtdel & Suresh, 2017)	- 12 -
Figure 5-5 - Deformation of a first order and second order tetrahedral element (Murad, 2019)...	- 13 -
Figure 5-6 - Curvature based mesh element generation.....	- 14 -
Figure 5-7 - Stress-Strain path for linear-static FE solver (Altair University, 2015).....	- 15 -
Figure 5-8 - Archer's model of the design process (Cross, 2008).....	- 17 -
Figure 5-9 - French's model of the design process (Cross, 2008).....	- 17 -
Figure 5-10 - SIMP flow chart (Aremu, et al., 2010).....	- 19 -
Figure 5-11 - Checkerboarding effect (Larsson, 2016)	- 20 -
Figure 5-12 - Upright designed using AMendate optimization software for additive manufacturing (AMFG, 2019)	- 20 -
Figure 5-13 - Topologically optimized Lotus Elise S1 upright designed by Fransisco, manufactured by Carbon Performance (Peters, 2019).....	- 25 -
Figure 5-14 - Cost per unit against production volume for different manufacturing methods.....	- 26 -
Figure 5-15 - Material stress-strain curve (Reif, et al., 2018)	- 28 -
Figure 6-1 - CAD model of MCR front upright.....	- 31 -
Figure 6-2 - Locations of the attachment points.....	- 32 -
Figure 6-3 - Domain used for SolidWorks topology studies	- 33 -
Figure 6-4 - Domain used for SolidWorks topology studies	- 33 -
Figure 6-5 - RBE2 elements created for the new upright design	- 35 -
Figure 7-1 - First topology optimization study	- 36 -
Figure 7-2 - Mesh used for the first, second and third topology study	- 37 -
Figure 7-3 - Second topology optimization study	- 38 -
Figure 7-4 - Convergence graph for the second topology optimization study	- 39 -
Figure 7-5 - Third topology optimization study.....	- 40 -
Figure 7-6 - Final topology optimization study	- 41 -

Figure 7-7 - Katwa's topology optimization results with low isosurface value.....	- 42 -
Figure 7-8 - Katwa's topology optimization results with high isosurface value.....	- 42 -
Figure 8-1 - Von Mises stress for the top wishbone loadcase using bonded contact set	- 43 -
Figure 8-2 - Von Mises stress for the top wishbone loadcase using no penetration contact set	- 44 -
Figure 8-3 - Maximum displacement for each loadcase using the bonded and no penetration contact set.....	- 46 -
Figure 8-4 - Design 1 front	- 51 -
Figure 8-5 - Design 1 rear.....	- 51 -
Figure 8-6 - Design 1 and final topology overlay.....	- 52 -
Figure 8-7 - Design 2 front	- 54 -
Figure 8-8 - Design 2 rear.....	- 54 -
Figure 8-9 - Design 2 and second topology overlay	- 55 -
Figure 8-10 - Design 2 and final topology overlay.....	- 55 -
Figure 8-11 - Design 3 front	- 57 -
Figure 8-12 - Design 3 rear.....	- 57 -
Figure 8-13 - Design 4 front	- 59 -
Figure 8-14 - Design 4 rear.....	- 59 -
Figure 8-15 - Design 5 back.....	- 60 -
Figure 8-16 - Design 5 front	- 60 -
Figure 8-17 - Design 6 front	- 61 -
Figure 8-18 - Design 6 back.....	- 61 -
Figure 8-19 - Design 7 front	- 62 -
Figure 8-20 - Design 7 back.....	- 62 -
Figure 8-21 - Design 8 front	- 63 -
Figure 8-22 - Design 8 rear.....	- 63 -
Figure 8-23 - Design 8 and second topology overlay	- 64 -
Figure 8-24 - Design 8 and final topology overlay.....	- 64 -
Figure 8-25 - Location of the mesh controls added to the model.....	- 72 -
Figure 8-26 - 2.5mm mesh used with 1.1mm mesh controls	- 73 -
Figure 8-27 - Elemental von Mises stress for bottom wishbone load	- 73 -
Figure 8-28 - Mesh details	- 74 -
Figure 8-29 - Aspect ratio plot with location of element with maximum aspect ratio	- 75 -
Figure 8-30 - Location of element with maximum aspect ratio.....	- 75 -
Figure 16-1 - Calculation of upper wishbone maximum lateral load	- 95 -

Figure 16-2 - Calculation of lower wishbone maximum lateral load	- 96 -
Figure 16-3 - Solve time against number of elements for the standard mesh	- 99 -
Figure 16-4 - Maximum von Mises stress against number of elements for the standard mesh ...	- 100 -
Figure 16-5 - Maximum displacement against number of elements for the standard mesh	- 100 -
Figure 16-6 - Maximum strain against number of elements for the standard mesh	- 101 -
Figure 16-7 - Solve time against number of elements for the curvature-based mesh	- 102 -
Figure 16-8 - Maximum von Mises Stress against number of elements for the curvature-based mesh solve time.....	- 103 -
Figure 16-9 - Maximum displacement against number of elements for the curvature-based mesh	- 104 -
Figure 16-10 - Maximum strain against number of elements for the curvature-based mesh	- 105 -
Figure 16-11 - H-refinement settings used in SolidWorks	- 106 -
Figure 16-12 - Steering arm elemental stress	- 107 -
Figure 16-13 - Upper caliper mesh refinement.....	- 108 -
Figure 16-14 – Von Mises stress under acceleration load of 6145.8N for the original and new upright design	- 119 -
Figure 16-15 - Von Mises stress under upper wishbone load of 2696.1N for the original and new upright design	- 120 -
Figure 16-16 - Von Mises stress under lower wishbone load of 7279.4N for the original and new upright	- 121 -
Figure 16-17 - Von Mises stress under steering load of 250N for the original and new upright...	- 122 -
Figure 16-18 - Von Mises stress under braking load of 7357.8N for the original and new upright-	- 123 -

14 List of plates

Plate 2-1 - MCR S2 race car	- 1 -
Plate 2-2 - MCR S2 front suspension geometry	- 2 -
Plate 5-1 - 6061 and 7075 aluminium material costs (Onlinemetals.com, n.d.).....	- 29 -
Plate 6-1 - MCR front upright.....	- 31 -
Plate 16-1 - 3D printed models of the original and new upright design (front)	- 124 -
Plate 16-2 - 3D printed models of the original and new upright design (rear).....	- 124 -
Plate 16-3 - CNC quotation from Xometry for design 3.....	- 125 -
Plate 16-4 - 6082-T6 aluminium invoice	- 126 -

15 List of tables

Table 4-1 - Project proposal list of tasks and timings.....	- 4 -
Table 4-2 - Final list of tasks and timings prior to COVID-19 pandemic	- 6 -
Table 4-3 - Final list of tasks and timings following COVID-19 pandemic.....	- 7 -
Table 5-1 - Potential upright materials specifications (ASM Aerospace Specification Metals, n.d.)	- 27 -
Table 6-1 - Maximum lateral loads on each attachment point.....	- 32 -
Table 8-1 - FEA results for each loadcase using the bonded and no penetration contact set	- 45 -
Table 8-2 - Weight and weight saving of original and new designs	- 47 -
Table 8-3 - Von Mises stress for each loadcase for each new design	- 48 -
Table 8-4 - Displacement for each loadcase for each new design	- 49 -
Table 8-5 - Strain for each loadcase for each new design	- 49 -
Table 8-6 - Original design and design 1 attachment point stiffnesses.....	- 53 -
Table 8-7 - Original design and design 2 attachment point stiffnesses.....	- 56 -
Table 8-8 - Original design and design 3 attachment point stiffnesses.....	- 58 -
Table 8-9 - Original design and design 4 attachment point stiffnesses.....	- 59 -
Table 8-10 - Original design and design 5 attachment point stiffnesses.....	- 60 -
Table 8-11 - Original design and design 6 attachment point stiffnesses.....	- 61 -
Table 8-12 - Original design and design 7 attachment point stiffnesses.....	- 62 -
Table 8-13 - Original design and design 8 attachment point stiffnesses.....	- 63 -
Table 8-14 - Original design and design 8 with small radii removed attachment point stiffnesses	- 64 -
Table 8-15 - 6061 and 6082 scaled original upright with scaled loads.....	- 66 -
Table 8-16 - 7075 and 6082 scaled new upright with scaled loads.....	- 67 -
Table 8-17 - 6061 and 6082 scaled original upright maximum loads before yield	- 68 -
Table 8-18 - 7075 and 6082 scaled new upright maximum loads before yield	- 69 -
Table 8-19 - 6061 and 6082 scaled original upright attachment point stiffnesses	- 70 -
Table 8-20 - 7075 and 6082 scaled new upright attachment point stiffnesses	- 71 -
Table 8-21 - Elemental FEA results for SolidWorks and Design Studio	- 76 -
Table 8-22 - Nodal FEA results for SolidWorks and Design Studio.....	- 78 -
Table 9-1 - Young's Modulus of different aluminium grades (Thyssenkrupp, n.d.)	- 81 -
Table 16-1 - Specifications of the computer used.....	- 97 -
Table 16-2 - FEA results for the standard mesh as element size decreases	- 98 -
Table 16-3 - FEA results for the curvature-based mesh as element size decreases	- 102 -

Table 16-4 - FEA results using the H-refinement mesh	- 107 -
Table 16-5 - FEA results for design 1	- 109 -
Table 16-6 - FEA results for design 2	- 110 -
Table 16-7 - FEA results for design 2 updated	- 111 -
Table 16-8 - FEA results for design 3	- 112 -
Table 16-9 - FEA results for design 3 updated	- 113 -
Table 16-10 - FEA results for design 4	- 114 -
Table 16-11 - FEA results for design 5	- 115 -
Table 16-12 - FEA results for design 6	- 116 -
Table 16-13 - FEA results for design 7	- 117 -
Table 16-14 - FEA results for design 8	- 118 -

16 Appendix

16.1 Calculation of maximum lateral upright forces

16.1.1 Upper wishbone

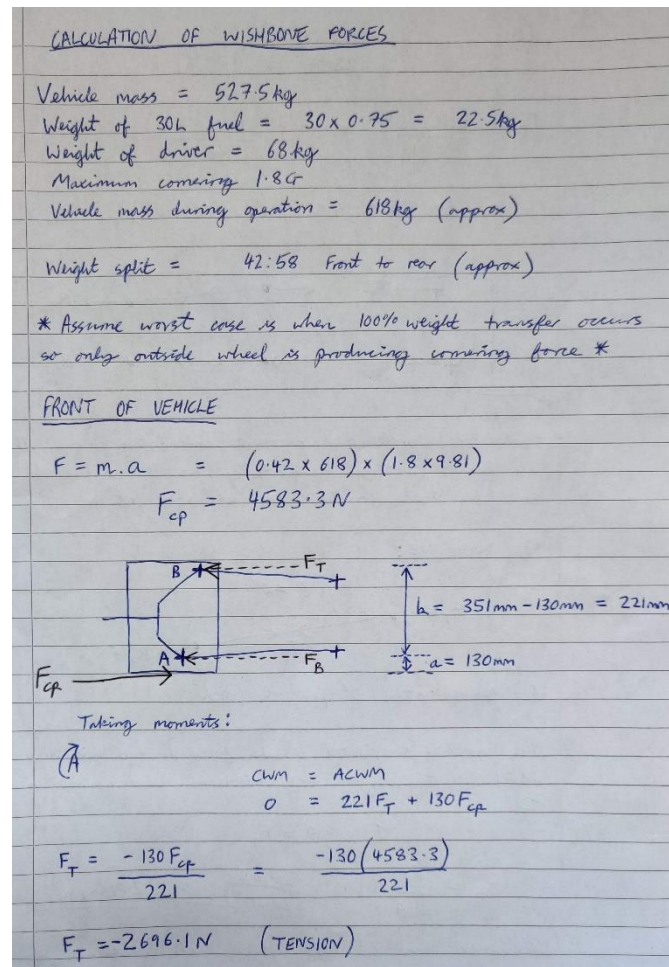


Figure 16-1 - Calculation of upper wishbone maximum lateral load

16.1.2 Lower wishbone

$\vec{B}$
 $CWM = ACWM$
 $221 F_B = 351 F_{cp}$
 $F_B = \frac{351 F_{cp}}{221} = \frac{351 (4583.3)}{221}$
 $F_B = 7279.4 \text{ N (COMPRESSION)}$

The diagram illustrates a lower wishbone assembly. A vertical line represents the wishbone, with a horizontal line representing the control arm. Forces are labeled as follows: F_{cp} (cornering force) acts horizontally to the right at the bottom of the wishbone. F_B (bush force) acts horizontally to the left at the top of the wishbone. F_T (tire force) acts horizontally to the right at the top of the wishbone. T_B (bush torque) acts horizontally to the right at the bottom of the wishbone. T_T (tire torque) acts horizontally to the left at the top of the wishbone. R_{Tn} (tire normal reaction) acts horizontally to the right at the top of the wishbone. R_{Bn} (bush normal reaction) acts horizontally to the left at the bottom of the wishbone.

Figure 16-2 - Calculation of lower wishbone maximum lateral load

16.1.3 Acceleration (Hub)

Mass of vehicle with driver 600kg approx.

1.09 G Force long during Silverstone race (acceleration at start)

$$\begin{aligned}
 \text{Acceleration} &= 1.09 \times 9.81 = 10.693 \text{ m/s}^2 \\
 +\text{Force} &= 600 \times 10.693 = 6415.8 \text{ N}
 \end{aligned}$$

16.1.4 Braking

-1.25 G Force long during Silverstone race (deceleration)

$$\begin{aligned}
 \text{Acceleration} &= -1.25 \times 9.81 = -12.263 \text{ m/s}^2 \\
 \text{Force} &= 600 \times -12.263 = -7357.8 \text{ N}
 \end{aligned}$$

16.2 Convergence and mesh independence study

A mesh independence study was conducted to determine the optimum element size for the studies. As the element size decreases the solution begins to converge. Since FEA is a balance between accuracy and time, it is important to determine the soonest point where the results are no longer affected.

‘Mesh convergence determines how many elements are required in a model to ensure that the results of an analysis are not affected by changing the size of the mesh. System response (stress, deformation) will converge to a repeatable solution with decreasing element size.’

(XCEED Engineering, 2017)

An element size of 0.7mm was determined to be the best for the standard mesh, and an element size of 0.80mm / 0.64mm for the curvature-based mesh. For both types of mesh the results were collected for the top wishbone load using the original upright.

Table 16-1 - Specifications of the computer used

<i>Computer specifications:</i>	
Processor	Ryzen 2700X 8 core 3.7GHz - 4.3GHz
Graphics card	GTX 1060 6GB DDR5
RAM	32GB DDR4 2933MHz

16.2.1 Standard mesh

Table 16-2 - FEA results for the standard mesh as element size decreases

Standard mesh					
Max element size	Von mises stress (N/m ²)	Displacement (mm)	Strain	Solve time (sec)	Number of elements
2.5	2.702E+07	0.04024	3.165E-04	43	393,590
2.4	2.702E+07	0.04024	3.165E-04	42	393,590
2.3	2.812E+07	0.04023	2.974E-04	45	424,263
2.2	2.685E+07	0.04023	3.416E-04	51	480,228
2.1	2.722E+07	0.04026	3.364E-04	60	540,248
2.00	2.711E+07	0.04028	3.357E-04	70	627,156
1.90	2.803E+07	0.04034	3.391E-04	84	732,230
1.80	2.744E+07	0.04034	3.509E-04	97	834,957
1.70	2.781E+07	0.04034	3.479E-04	118	978,804
1.60	2.901E+07	0.04039	3.783E-04	146	1,171,415
1.50	2.939E+07	0.04041	3.803E-04	175	1,382,070
1.40	3.060E+07	0.04042	4.005E-04	218	1,675,217
1.30	3.117E+07	0.04042	3.960E-04	275	2,046,888
1.20	3.337E+07	0.04044	4.111E-04	350	2,477,381
1.10	3.270E+07	0.04048	4.381E-04	473	3,135,623
1.00	3.486E+07	0.0405	4.490E-04	628	3,986,970
0.90	3.536E+07	0.04052	4.713E-04	905	5,268,457
0.80	3.923E+07	0.04052	4.993E-04	1364	7,092,473
0.70	4.150E+07	0.04052	5.505E-04	2160	9,984,679
0.60	4.248E+07	0.04057	5.765E-04	3846	14,694,431
0.50	Failed (not enough RAM)				

Table 16-2 shows the results for the top wishbone load tested using the original upright. The computer specifications can be seen in Table 16-1.

The study with an element size of 0.5mm failed due to a shortage of RAM. The element size had the smallest effect on the displacement, and the largest effect on strain. Interestingly, from 2.5mm to 2.4mm element size, the number of elements did not increase. The range of stresses obtained was 1.6×10^7 N/m². The actual stress will be somewhere within the range; therefore, the range is also the absolute uncertainty. The range of displacements was 3.3×10^{-4} mm. The range of strains obtained was 2.6×10^{-4} .

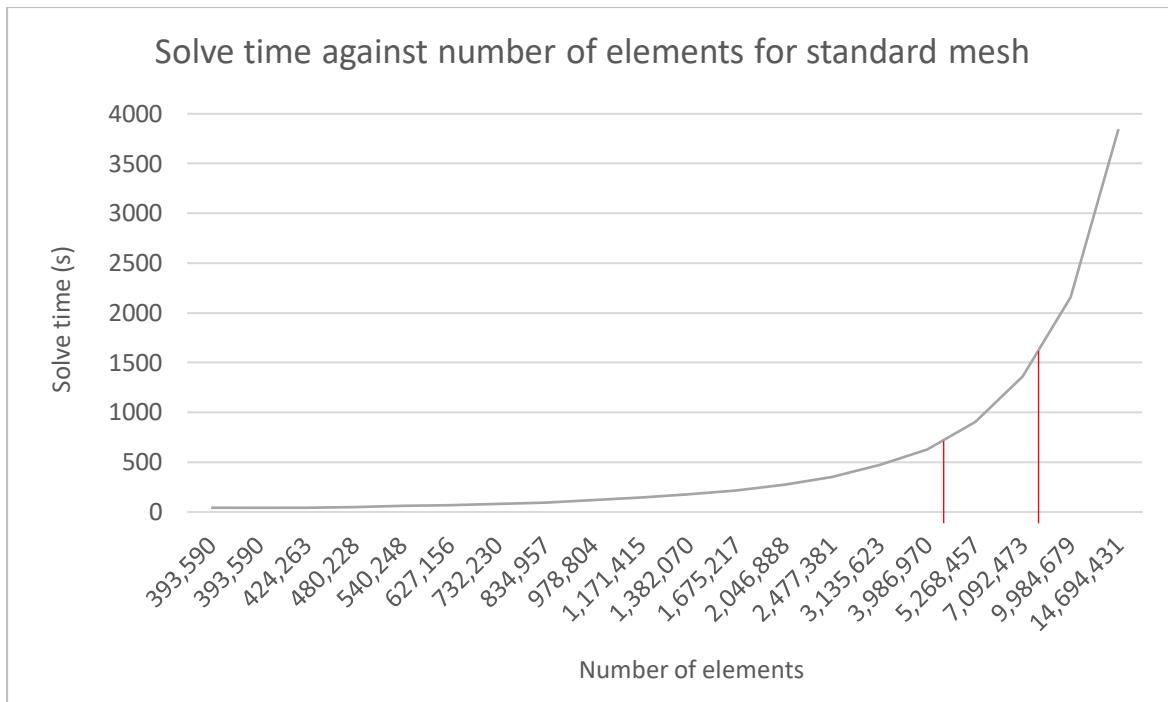


Figure 16-3 - Solve time against number of elements for the standard mesh

Figure 16-3 shows the solve time against number of elements. The increase in solve time is initially gradual as number of elements increases. From around 3,000,000 elements, the increase in solve time climbs rapidly. The increase in results is described as an exponential trend. The element sizes between the vertical red lines is the optimum element size.

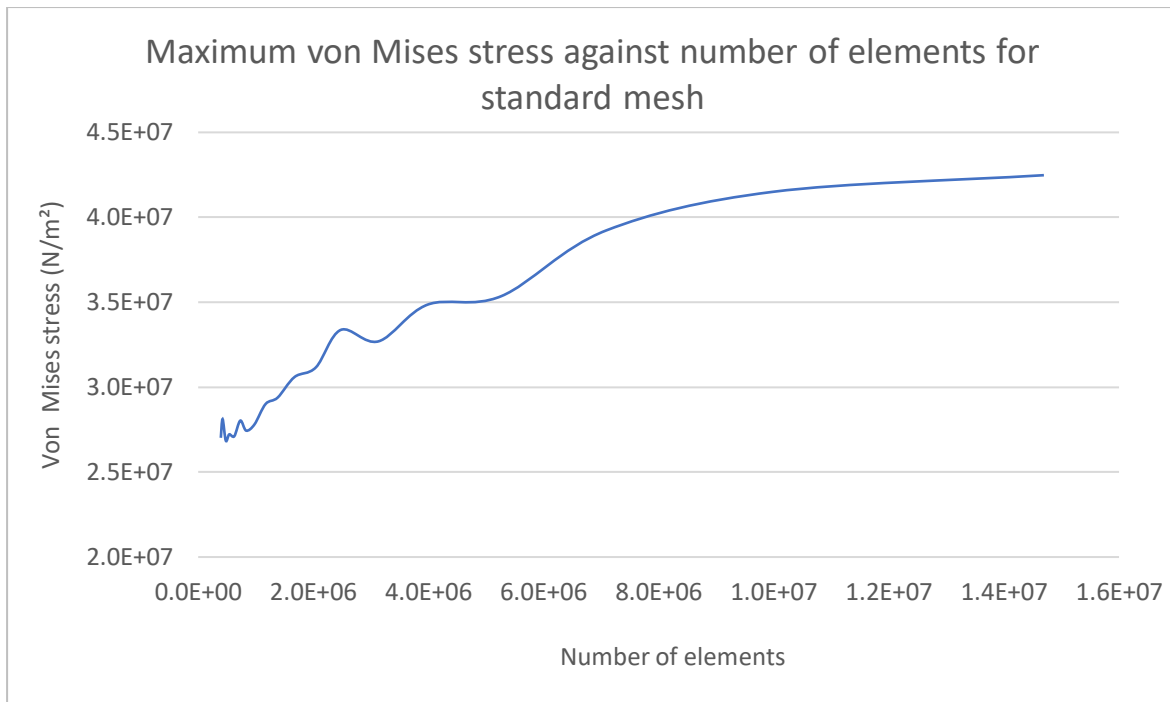


Figure 16-4 - Maximum von Mises stress against number of elements for the standard mesh

Figure 16-4 shows the von Mises stress against the number of elements. From 393,590 to 7,092,473 elements the increase in stress is 1.2×10^7 N/m². From 7,092,473 to 14,694,431 elements the increase in stress is 3.3×10^6 N/m². The results appear to converge at around 1.0×10^7 elements.

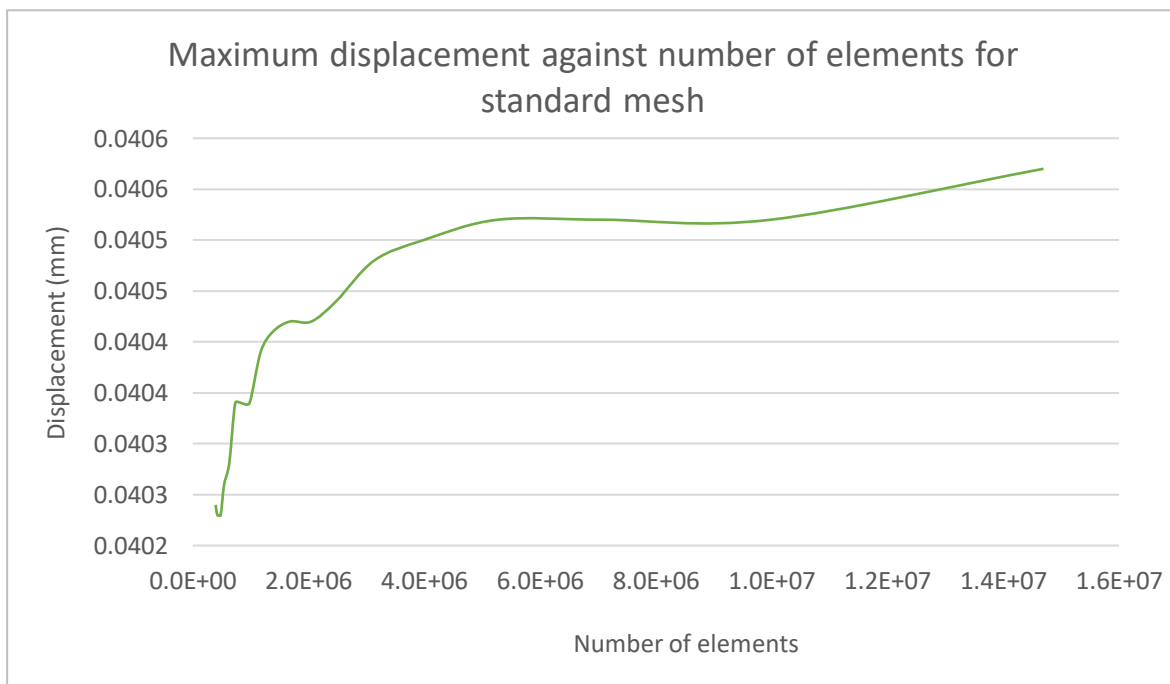


Figure 16-5 - Maximum displacement against number of elements for the standard mesh

Figure 16-5 shows the displacement against number of elements. From 393,590 to 5,268,457 the results increase by 2.8×10^{-4} mm. From 5,268,457 to 14,694,431 the results increase by 5×10^{-5} mm. The results show how displacement appears to converge at around 6.0×10^6 elements.

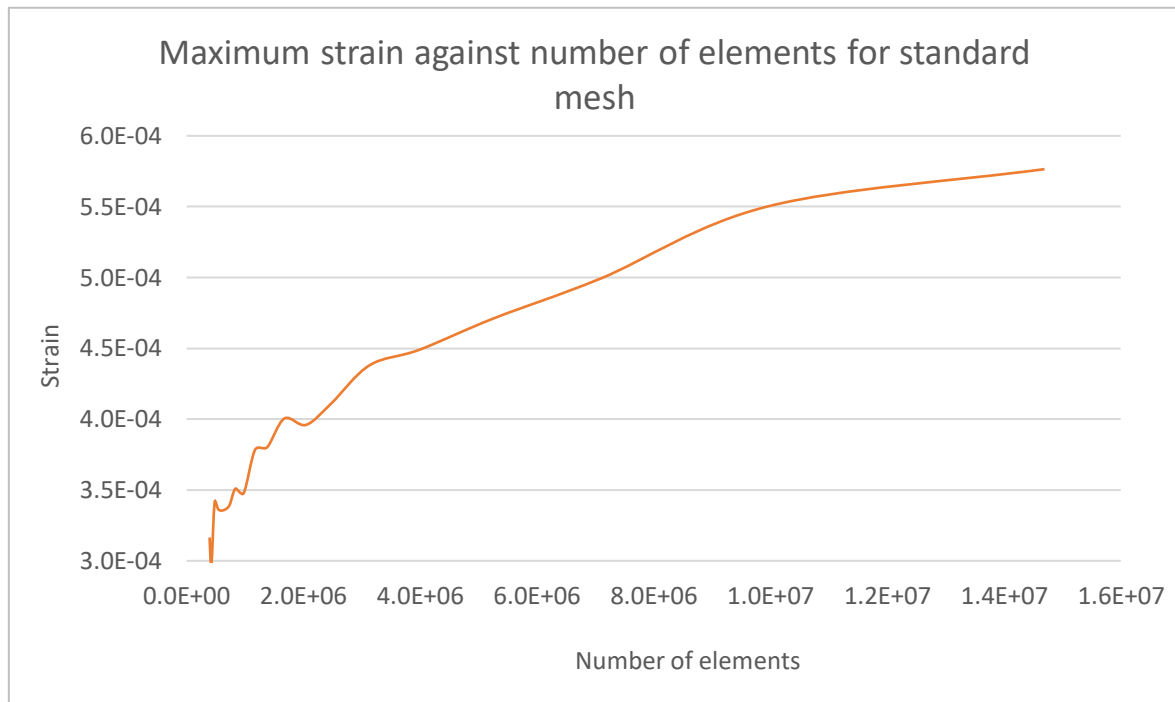


Figure 16-6 - Maximum strain against number of elements for the standard mesh

Figure 16-6 shows strain against number of elements. The results do not show a clear sign of converging, however, from around 1.0×10^7 the increase in strain as element size increases is at its smallest.

Using the standard mesh, Figure 16-3, Figure 16-4 and Figure 16-5 concur, with convergence occurring at around 1.0×10^7 elements. The results show that a standard mesh with elements size 0.70mm is the optimum for the study.

16.2.2 Curvature-based mesh

Table 16-3 - FEA results for the curvature-based mesh as element size decreases

Curvature-based mesh					
Max / Min element size	Von mises stress (N/m ²)	Displacement (mm)	Strain	Solve time (sec)	Number of elements
2.50 / 2.00	3.431E+07	0.04027	4.115E-04	56	513,202
2.40 / 1.92	6.658E+07	0.04028	4.610E-04	54	513,663
2.30 / 1.84	5.564E+07	0.04028	3.911E-04	58	537,487
2.20 / 1.76	5.850E+07	0.04031	4.160E-04	72	614,990
2.10 / 1.68	6.754E+07	0.04032	4.670E-04	80	685,099
2.00 / 1.60	6.247E+07	0.04033	4.328E-04	90	756,346
1.90 / 1.52	3.634E+07	0.04039	4.318E-04	106	876,877
1.80 / 1.44	3.900E+07	0.04039	4.664E-04	118	991,830
1.70 / 1.36	3.621E+07	0.04040	4.346E-04	146	1,157,037
1.60 / 1.28	3.913E+07	0.04042	4.716E-04	185	1,431,938
1.50 / 1.20	3.458E+07	0.04043	4.351E-04	223	1,685,398
1.40 / 1.12	3.928E+07	0.04044	4.559E-04	281	2,040,706
1.30 / 1.04	4.093E+07	0.04046	4.779E-04	383	2,619,169
1.20 / 0.96	8.220E+07	0.04048	5.433E-04	478	3,141,472
1.10 / 0.88	8.494E+07	0.04049	5.495E-04	610	3,824,995
1.00 / 0.80	9.011E+07	0.04051	5.905E-04	796	4,507,866
0.90 / 0.72	5.035E+07	0.04053	5.781E-04	1095	5,856,912
0.80 / 0.64	4.815E+07	0.04055	6.085E-04	1743	8,307,324
0.70 / 0.56	5.526E+07	0.04056	6.187E-04	2805	11,246,153
0.60 / 0.48	Failed (not enough RAM)				

Table 16-3 shows the results for the curvature-based mesh as maximum and minimum element size decreases. The element size had the largest effect on von Mises stress, and the smallest effect on displacement. The range of stress obtained was 2.1×10^7 N/m². The range of displacements was 2.9×10^{-4} mm. The range of strains was 2.1×10^{-4} .

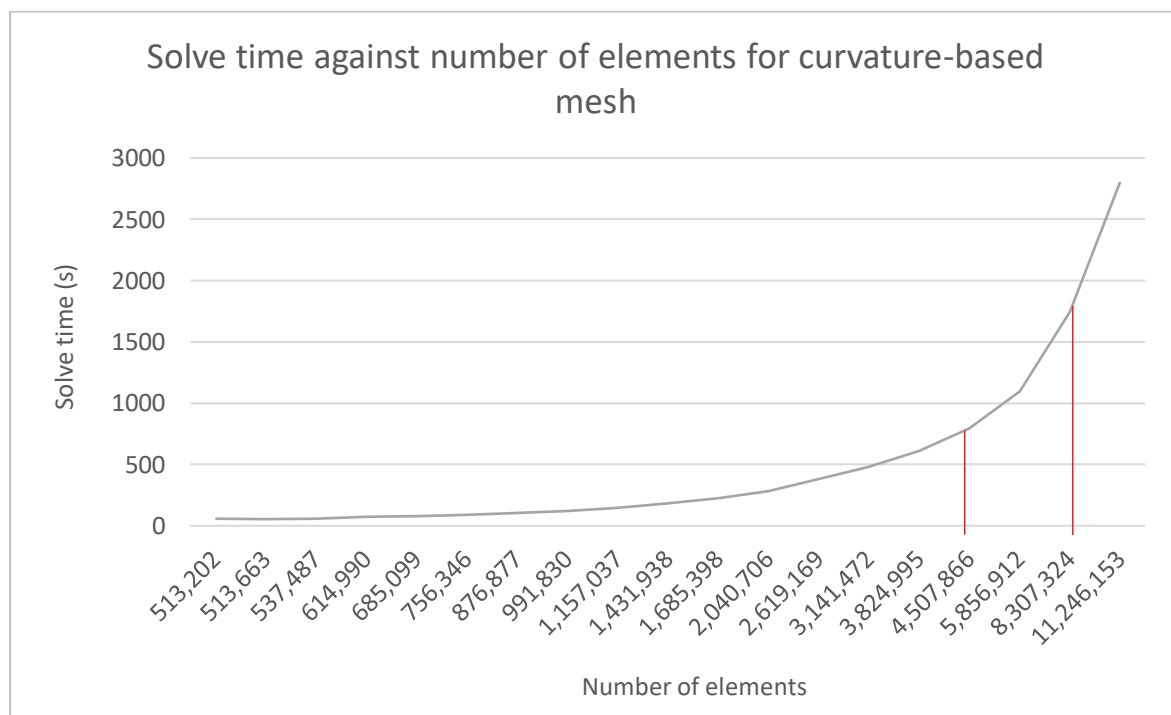


Figure 16-7 - Solve time against number of elements for the curvature-based mesh

Figure 16-7 shows how solve time increases exponentially against number of elements. The graph shows how the optimum number of elements is between the vertical red lines.

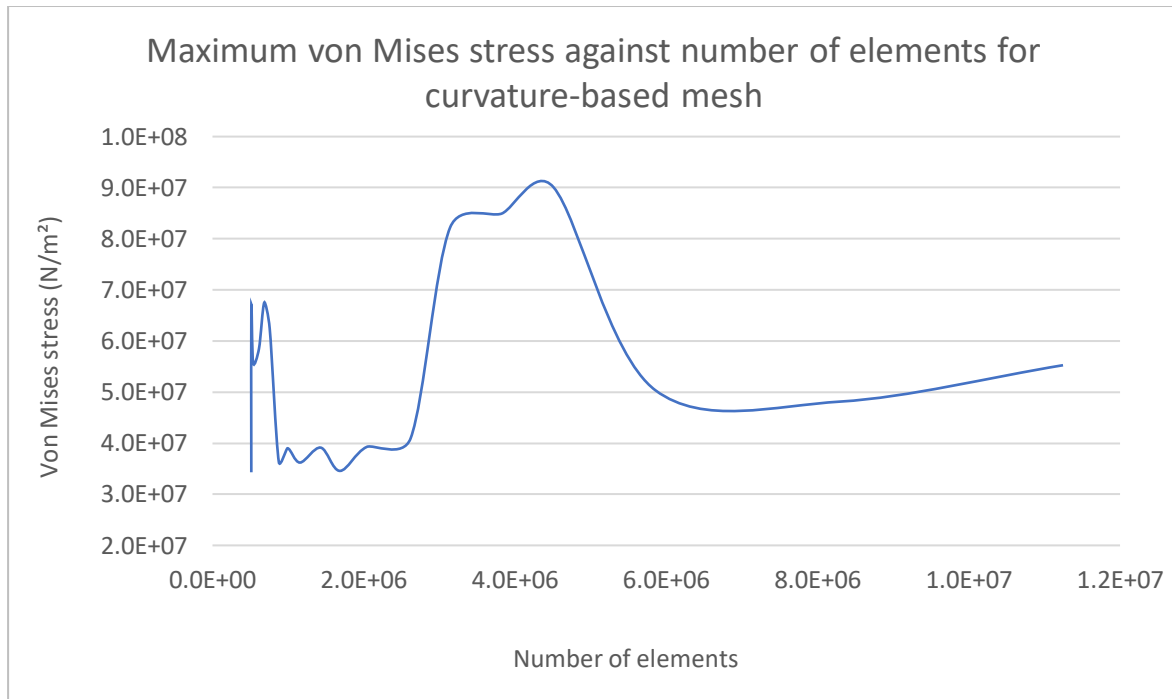


Figure 16-8 - Maximum von Mises Stress against number of elements for the curvature-based mesh solve time

Figure 16-8 shows von Mises stress against number of elements. The graph shows an interesting trend highlighting the difference element size can make to the results. The results do not appear to converge, however, the change in stress as element size increases reduces after around 8.0×10^6 elements.

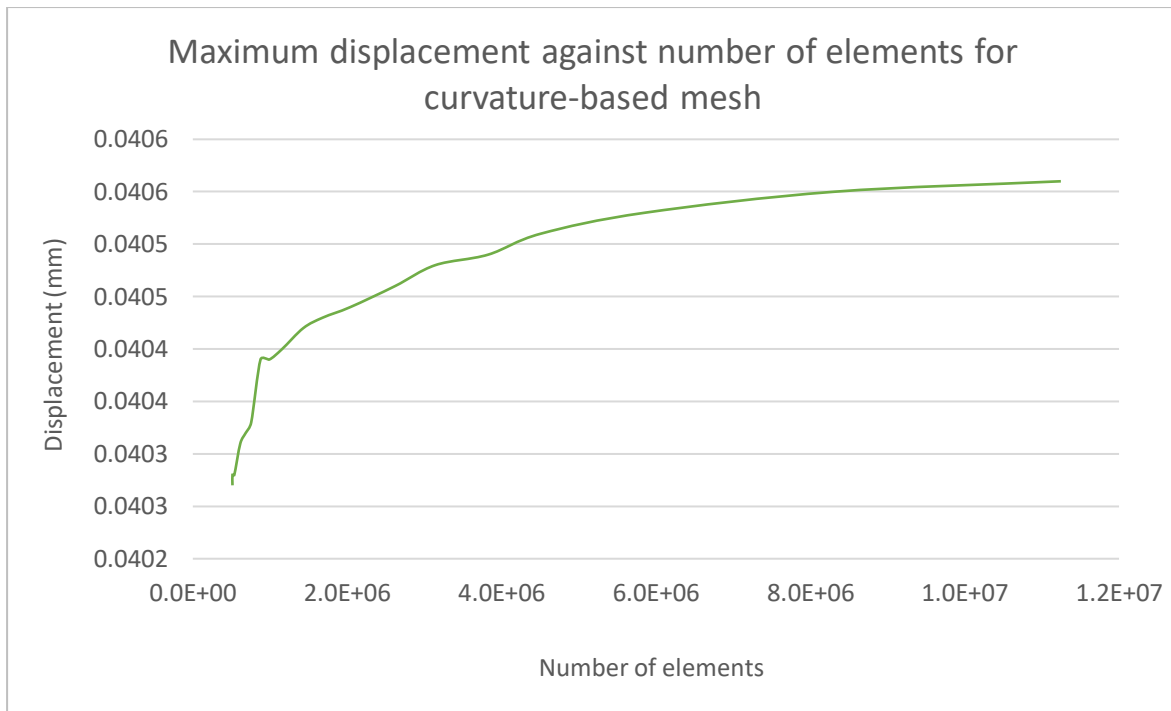


Figure 16-9 - Maximum displacement against number of elements for the curvature-based mesh

Figure 16-9 shows displacement against number of elements. The displacement increases as number of elements increases; a logarithmic trend is observed. The results start to converge at around 8.0×10^6 elements.

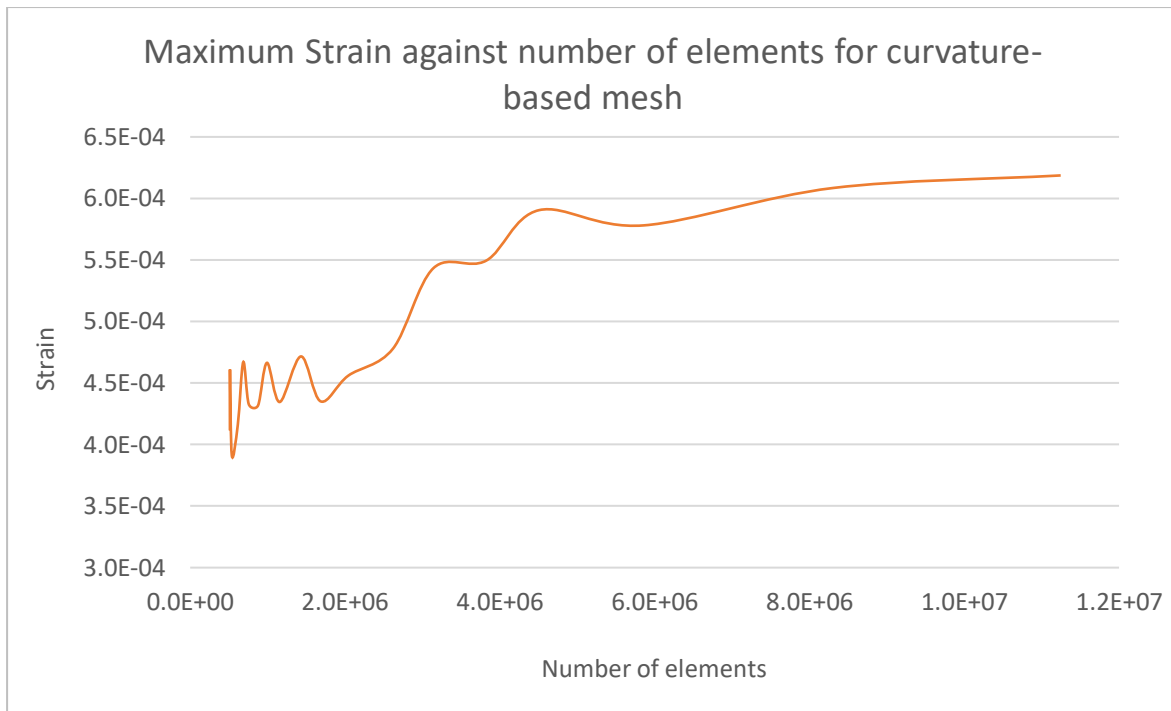


Figure 16-10 - Maximum strain against number of elements for the curvature-based mesh

Figure 16-10 shows strain against number of elements. Initially the results do not show any clear trend, however, after around 5.0×10^6 elements a gradual increase in strain is observed. The results appear to converge at around 8.0×10^6 elements.

Using the curvature-based mesh, all the graphs apart from von Mises stress concur, with convergence occurring at around 8.0×10^6 elements. The results show that a curvature-based mesh with elements size 0.80mm / 0.64mm is the optimum for the study.

16.3 H-refinement mesh

Adaptive meshing was used in SolidWorks to optimize the mesh. This tool refines the mesh by iteratively varying the mesh without user intervention until the desired accuracy is achieved. There are two types of mesh refinements; H-refinement, which reduces element size and P-refinement, which increases element order. SolidWorks H-refinement was used to optimise the mesh as it represents curved geometry more accurately than P-refinement.

'However, it is important to distinguish between the geometric effect and mesh convergence. Particularly when meshing a curved surface using straight (or linear) elements, which will require more elements (or otherwise mesh refinement) to capture the boundary exactly.'

(Harish, 2020)

According to Hawk Ridge Systems (2013), the software achieves the target accuracy by measuring the change in strain energy and once two values differ by less than the stated value iterations stop. A target accuracy of 98% was chosen for the study, therefore, when the change in strain energy between iterations was below 2% the solver would terminate (see *Figure 16-11*). The accuracy bias controls whether the software iterates to achieve accurate peak stress, in areas of stress concentrations or whether it achieves accurate global results. The accuracy bias was left as default. Mesh coarsening allows the solver to coarsen the mesh in areas that do not require a fine mesh. This was deselected as a base mesh element size of 2.5mm was used.

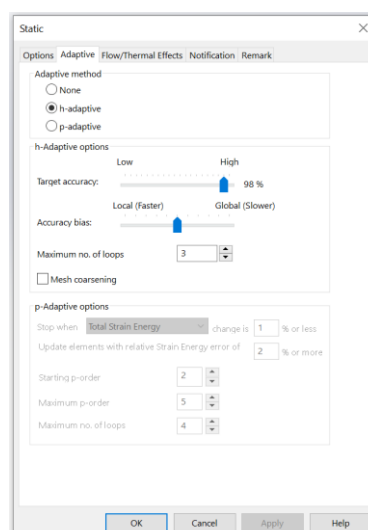


Figure 16-11 - H-refinement settings used in SolidWorks

Table 16-4 - FEA results using the H-refinement mesh

Loadcase	Von Mises (N/m ²)	Displacement (mm)	Strain	Number of Elements	Solve time (s)
Upper wishbone	1.22E+08	1.21E-02	1.50E-03	3,703,005	270
Lower wishbone	1.51E+09	4.11E-02	1.86E-02	1,181,980	81
Acceleration	1.25E+09	2.45E-02	1.53E-02	912,922	66
Upper caliper	4.65E+07	3.72E-02	5.72E-04	750,927	54
Lower caliper	4.16E+07	3.04E-02	5.12E-04	612,996	43
Steering	1.49E+07	1.35E-02	1.84E-04	752,526	54
Combination	1.97E+09	4.13E-02	2.43E-02	7,435,942	677
(Elemental values)					

Table 16-4 shows the results obtained from the H-refinement studies. The same settings in Figure 16-11 were used for each solve. Despite the results converging, the overall stress distribution on the models did not appear to be accurate possibly because of the large changes in element size. Singularities were found during upper wishbone, acceleration and combination of loads studies. Singularities are points in the model which approach values of infinity.

'Singularities can be confusing because they cause an accuracy problem inside the model, which implies a problem with visualization because singularities extend the range of the stresses.'
(Murad, 2019)

Since the peak stresses were so high, the stress distribution was not accurately represented as smaller stresses appeared negligible. Despite this, each solve did inform the accurate mesh used to compare SolidWorks and Design Studio results in Chapter 8.4. Figure 16-12 shows the elemental stress during the steering arm loadcase and shows the smaller elements at high stress areas.

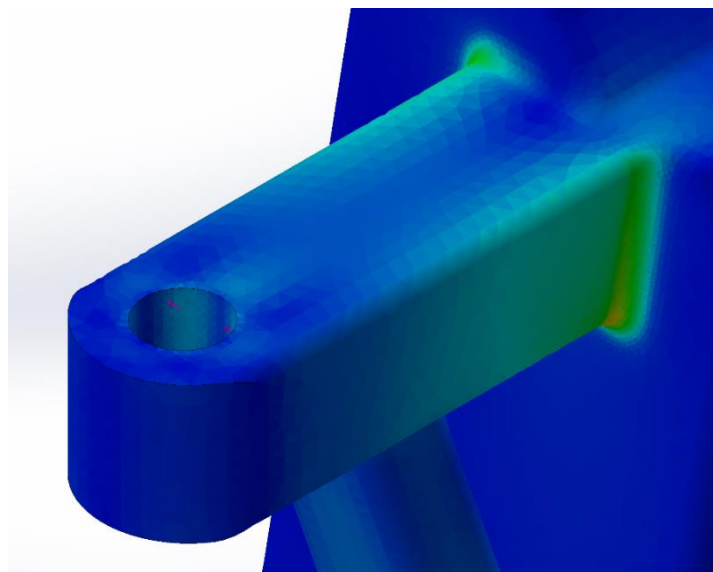


Figure 16-12 - Steering arm elemental stress

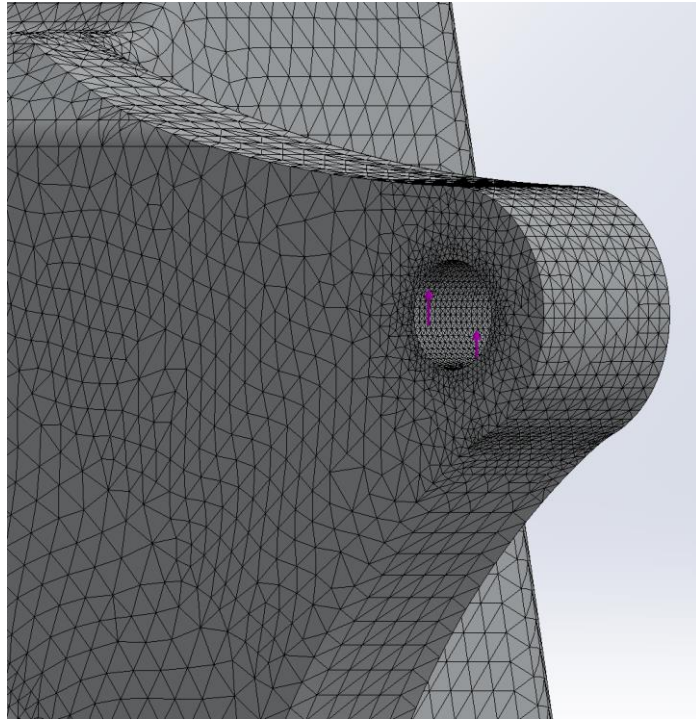


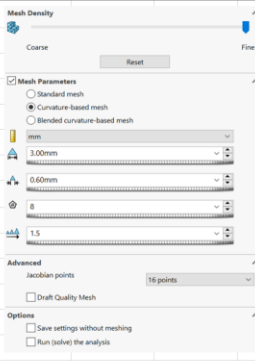
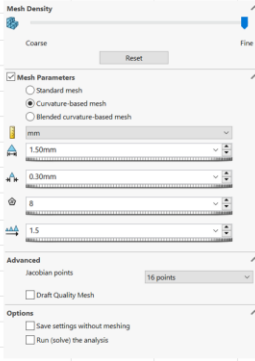
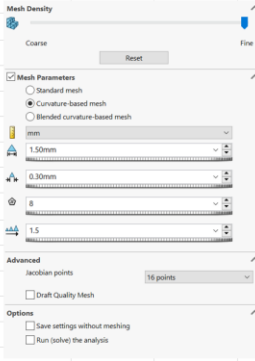
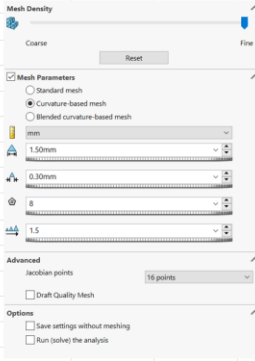
Figure 16-13 - Upper caliper mesh refinement

16.4 SolidWorks FEA – Original and new design comparison

The FEA results of the original design and new designs were compared to determine the performance of the new designs. After conducting the FEA for design 1 the results were analysed; the succeeding designs were developed, aiming to improve on the previous designs performance. The cells highlighted in green show the higher performance values (see *Table 16-5 to 16-14*).

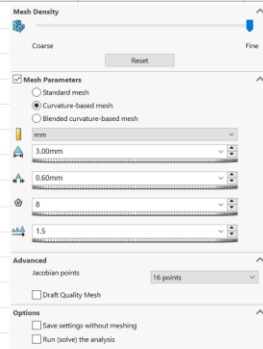
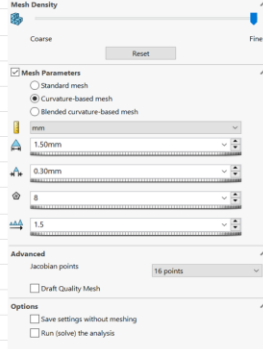
16.4.1 Design 1

Table 16-5 - FEA results for design 1

	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 1	Top wishbone	1.098E+07	0.0117	1.252E-04	
	Bottom wishbone	3.905E+07	0.0442	4.051E-04	
	Acceleration	7.421E+07	0.0244	4.399E-04	
	Braking	1.254E+08	0.1075	1.300E-03	
	Combo	3.916E+07	0.0443	4.061E-04	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.160E+07	0.0111	1.801E-04	
	Bottom wishbone	1.713E+08	0.0525	6.618E-04	
	Acceleration	3.863E+07	0.0234	6.527E-04	
	Braking	8.735E+08	0.0198	3.287E-04	
	Combo	1.988E+08	0.0533	6.814E-04	
		yield - 5.050E+08			
New design 1	Top wishbone	2.045E+07	0.0117	2.470E-04	
	Bottom wishbone	6.642E+07	0.0445	7.123E-04	
	Acceleration	5.924E+07	0.0245	8.311E-04	
	Braking	1.326E+08	0.0108	1.406E-03	
	Combo	6.631E+07	0.0445	7.120E-04	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.494E+07	0.0111	2.512E-04	
	Bottom wishbone	2.061E+08	0.0528	9.675E-04	
	Acceleration	4.749E+07	0.0235	7.580E-04	
	Braking	8.245E+07	0.0169	3.156E-04	
	Combo	2.396E+08	0.0536	9.983E-04	

16.4.2 Design 2

Table 16-6 - FEA results for design 2

	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 2	Top wishbone	1.236E+07	0.0115	1.245E-04	
	Bottom wishbone	3.712E+07	0.0390	3.892E-04	
	Acceleration	4.374E+07	0.0242	3.904E-04	
	Braking	6.118E+07	0.0545	6.502E-04	
	Combo	3.731E+07	0.0392	3.915E-04	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.160E+07	0.0111	1.801E-04	
	Bottom wishbone	1.713E+08	0.0525	6.618E-04	
	Acceleration	3.863E+07	0.0234	6.527E-04	
	Braking	8.735E+08	0.0198	3.287E-04	
	Combo	1.988E+08	0.0533	6.814E-04	
New design 2	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	1.906E+07	0.0116	2.379E-04	
	Bottom wishbone	6.404E+07	0.0392	7.389E-04	
	Acceleration	5.685E+07	0.0243	8.382E-04	
	Braking	6.357E+07	0.0546	7.103E-04	
	Combo	6.423E+07	0.0394	7.400E-04	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.494E+07	0.0111	2.512E-04	
	Bottom wishbone	2.061E+08	0.0528	9.675E-04	
	Acceleration	4.749E+07	0.0235	7.580E-04	
	Braking	8.245E+07	0.0169	3.156E-04	
	Combo	2.396E+08	0.0536	9.983E-04	

16.4.3 Design 2 updated

Table 16-7 - FEA results for design 2 updated

New design 2 updated	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain
	Top wishbone	1.228E+07	0.0114	1.212E-04
	Bottom wishbone	3.765E+07	0.0384	3.862E-04
	Acceleration	4.275E+07	0.0237	4.785E-04
	Braking	6.083E+07	0.0516	5.995E-04
Combo	3.688E+07	0.0386	3.902E-04	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain
	Top wishbone	5.160E+07	0.0111	1.801E-04
	Bottom wishbone	1.713E+08	0.0525	6.618E-04
	Acceleration	3.863E+07	0.0234	6.527E-04
	Braking	8.735E+08	0.0198	3.287E-04
Combo	1.988E+08	0.0533	6.814E-04	
New design 2 updated	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain
	Top wishbone	1.919E+07	0.0114	2.496E-04
	Bottom wishbone	6.204E+07	0.0389	7.993E-04
	Acceleration	5.778E+07	0.0238	8.648E-04
	Braking	6.269E+07	0.0518	6.674E-04
Combo	6.219E+07	0.0388	8.059E-04	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain
	Top wishbone	5.494E+07	0.0111	2.512E-04
	Bottom wishbone	2.061E+08	0.0528	9.675E-04
	Acceleration	4.749E+07	0.0235	7.580E-04
	Braking	8.245E+07	0.0169	3.156E-04
Combo	2.396E+08	0.0536	9.983E-04	

Mesh Density

Coarse

Reset

☒ Mesh Parameters

☐ Standard mesh

☒ Curvature-based mesh

☐ Blended curvature-based mesh

min

1.00mm

max

0.60mm

points

8

Advanced

Jacobian points

16 points

Options

☐ Draft Quality Mesh

☐ Save settings without meshing

☐ Run (solve) the analysis

Mesh Density

Coarse

Reset

☒ Mesh Parameters

☐ Standard mesh

☒ Curvature-based mesh

☐ Blended curvature-based mesh

min

1.50mm

max

0.30mm

points

8

Advanced

Jacobian points

16 points

Options

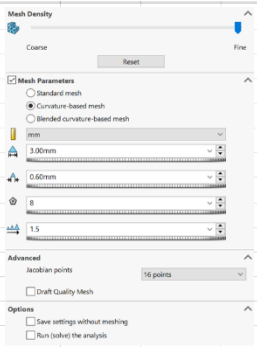
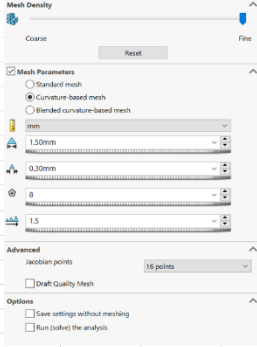
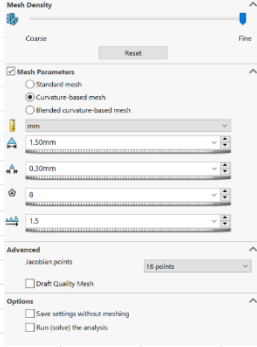
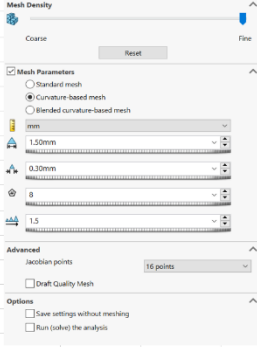
☐ Draft Quality Mesh

☐ Save settings without meshing

☐ Run (solve) the analysis

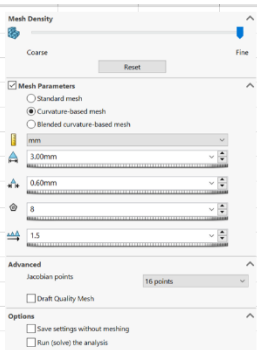
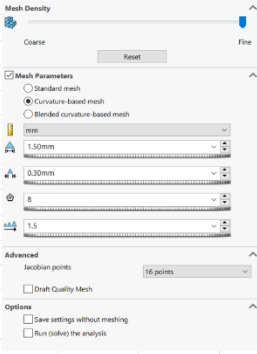
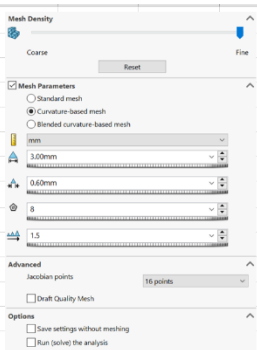
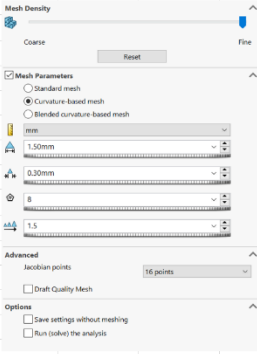
16.4.4 Design 3

Table 16-8 - FEA results for design 3

	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 3	Top wishbone	1.175E+07	0.0117	1.209E-04	
	Bottom wishbone	3.897E+07	0.0458	4.338E-04	
	Acceleration	4.032E+07	0.0266	4.074E-04	
	Braking	5.283E+07	0.0484	5.394E-04	
	Combo	3.918E+07	0.0460	4.375E-04	
	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
Original design (Bonded contact set)	Top wishbone	5.160E+07	0.0111	1.801E-04	
	Bottom wishbone	1.713E+08	0.0525	6.618E-04	
	Acceleration	3.863E+07	0.0234	6.527E-04	
	Braking	8.735E+08	0.0198	3.287E-04	
	Combo	1.988E+08	0.0533	6.814E-04	
	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 3	Top wishbone	2.007E+07	0.0118	2.446E-04	
	Bottom wishbone	7.215E+07	0.0460	8.656E-04	
	Acceleration	6.185E+07	0.0268	8.900E-04	
	Braking	5.480E+07	0.0485	5.963E-04	
	Combo	7.239E+07	0.0463	8.713E-04	
	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
Original design (Bonded contact set)	Top wishbone	5.494E+07	0.0111	2.512E-04	
	Bottom wishbone	2.061E+08	0.0528	9.675E-04	
	Acceleration	4.749E+07	0.0235	7.580E-04	
	Braking	8.245E+07	0.0169	3.156E-04	
	Combo	2.396E+08	0.0536	9.983E-04	

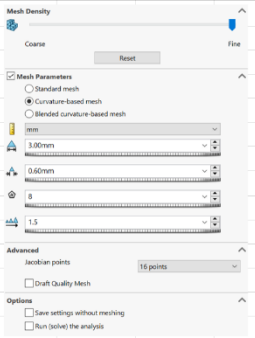
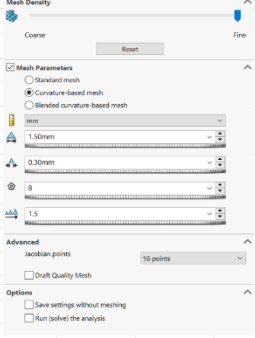
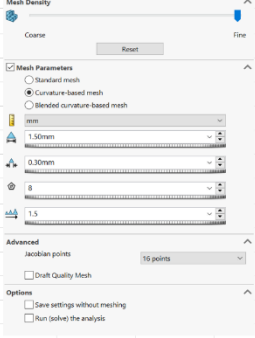
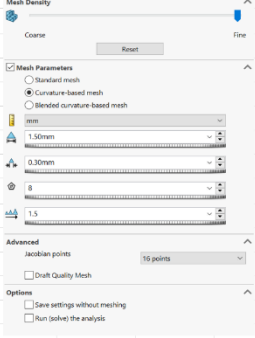
16.4.5 Design 3 updated

Table 16-9 - FEA results for design 3 updated

	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 3 updated	Top wishbone	1.203E+07	0.0117	1.345E-04	
	Bottom wishbone	4.033E+07	0.0459	4.317E-04	
	Acceleration	4.325E+07	0.0265	4.929E-04	
	Braking	5.236E+07	0.0489	5.222E-04	
	Combo	4.053E+07	0.0461	4.353E-04	
Original design (Bonded contact set)	Top wishbone	5.160E+07	0.0111	1.801E-04	
	Bottom wishbone	1.713E+08	0.0525	6.618E-04	
	Acceleration	3.863E+07	0.0234	6.527E-04	
	Braking	8.735E+08	0.0198	3.287E-04	
	Combo	1.988E+08	0.0533	6.814E-04	
New design 3 updated	Top wishbone	2.023E+07	0.0118	2.499E-04	
	Bottom wishbone	7.275E+07	0.0462	7.443E-04	
	Acceleration	5.903E+07	0.0266	8.744E-04	
	Braking	5.458E+07	0.0490	5.743E-04	
	Combo	7.310E+07	0.0464	7.512E-04	
Original design (Bonded contact set)	Top wishbone	5.494E+07	0.0111	2.512E-04	
	Bottom wishbone	2.061E+08	0.0528	9.675E-04	
	Acceleration	4.749E+07	0.0235	7.580E-04	
	Braking	8.245E+07	0.0169	3.156E-04	
	Combo	2.396E+08	0.0536	9.983E-04	

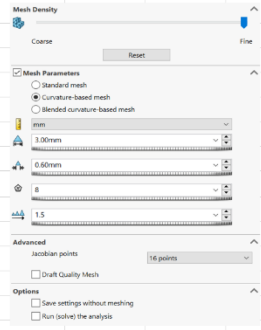
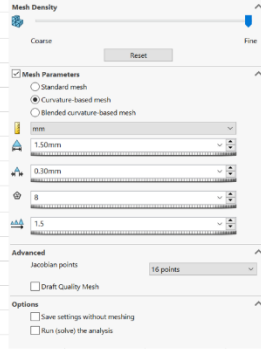
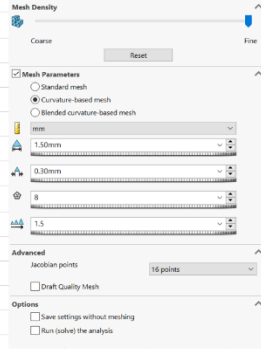
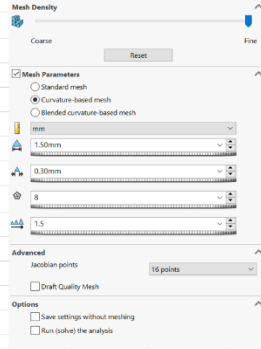
16.4.6 Design 4

Table 16-10 - FEA results for design 4

	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 4	Top wishbone	1.214E+07	0.0130	1.560E-04	
	Bottom wishbone	4.008E-07	0.0465	4.320E-04	
	Acceleration	1.275E+08	0.0276	1.372E-03	
	Braking	5.221E+07	0.0486	5.117E-04	
	Combo	4.120E+07	0.0466	4.730E-04	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.160E+07	0.0111	1.801E-04	
	Bottom wishbone	1.713E+08	0.0525	6.618E-04	
	Acceleration	3.863E+07	0.0234	6.527E-04	
	Braking	8.735E+08	0.0198	3.287E-04	
	Combo	1.988E+08	0.0533	6.814E-04	
New design 4	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	2.306E+07	0.0131	3.128E-04	
	Bottom wishbone	7.206E+07	0.0467	9.061E-04	
	Acceleration	1.284E+08	0.0276	1.440E-03	
	Braking	5.506E+07	0.0487	5.654E-04	
	Combo	7.214E+07	0.0469	9.075E-04	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.494E+07	0.0111	2.512E-04	
	Bottom wishbone	2.061E+08	0.0528	9.675E-04	
	Acceleration	4.749E+07	0.0235	7.580E-04	
	Braking	8.245E+07	0.0169	3.156E-04	
	Combo	2.396E+08	0.0536	9.983E-04	

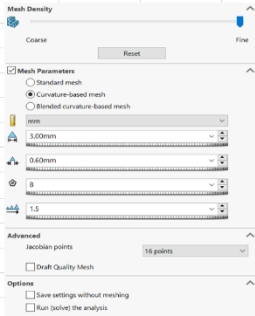
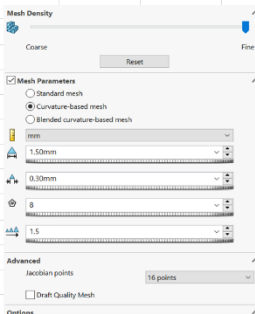
16.4.7 Design 5

Table 16-11 - FEA results for design 5

	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 5	Top wishbone	1.171E+07	0.0114	1.207E-04	
	Bottom wishbone	3.730E+07	0.0366	3.883E-04	
	Acceleration	4.153E+07	0.0238	3.848E-04	
	Braking	6.133E+07	0.0516	5.962E-04	
	Combo	3.750E+07	0.0367	3.943E-04	
	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
Original design (Bonded contact set)	Top wishbone	5.160E+07	0.0111	1.801E-04	
	Bottom wishbone	1.713E+08	0.0525	6.618E-04	
	Acceleration	3.863E+07	0.0234	6.527E-04	
	Braking	8.735E+08	0.0198	3.287E-04	
	Combo	1.988E+08	0.0533	6.814E-04	
	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 5	Top wishbone	1.867E+07	0.0115	2.336E-04	
	Bottom wishbone	6.066E+07	0.0368	7.831E-04	
	Acceleration	5.513E+07	0.0239	8.291E-04	
	Braking	6.102E+07	0.0517	6.648E-04	
	Combo	6.110E+07	0.0370	7.898E-04	
	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
Original design (Bonded contact set)	Top wishbone	5.494E+07	0.0111	2.512E-04	
	Bottom wishbone	2.061E+08	0.0528	9.675E-04	
	Acceleration	4.749E+07	0.0235	7.580E-04	
	Braking	8.245E+07	0.0169	3.156E-04	
	Combo	2.396E+08	0.0536	9.983E-04	

16.4.8 Design 6

Table 16-12 - FEA results for design 6

	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 6	Top wishbone	1.204E+07	0.0116	1.337E-04	
	Bottom wishbone	4.088E+07	0.0421	4.356E-04	
	Acceleration	3.363E+07	0.0249	4.296E-04	
	Braking	4.288E+07	0.0302	4.330E-04	
	Combo	4.115E+07	0.0424	4.397E-04	
Original design (Banded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.160E+07	0.0111	1.801E-04	
	Bottom wishbone	1.713E+08	0.0525	6.618E-04	
	Acceleration	3.863E+07	0.0234	6.527E-04	
	Braking	8.735E+08	0.0198	3.287E-04	
	Combo	1.988E+08	0.0533	6.814E-04	
New design 6	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	2.110E+07	0.0117	2.617E-04	
	Bottom wishbone	6.936E+07	0.0424	8.739E-04	
	Acceleration	5.554E+07	0.0250	7.880E-04	
	Braking	4.373E+07	0.0303	4.613E-04	
	Combo	6.981E+07	0.0427	8.776E-04	
Original design (Banded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.494E+07	0.0111	2.512E-04	
	Bottom wishbone	2.061E+08	0.0528	9.675E-04	
	Acceleration	4.749E+07	0.0235	7.580E-04	
	Braking	8.245E+07	0.0169	3.156E-04	
	Combo	2.396E+08	0.0536	9.983E-04	

16.4.9 Design 7

Table 16-13 - FEA results for design 7

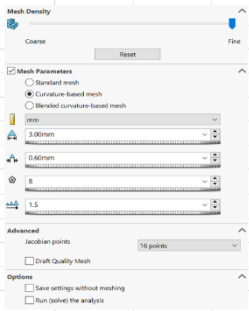
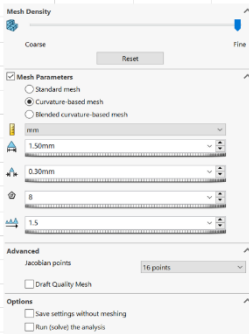
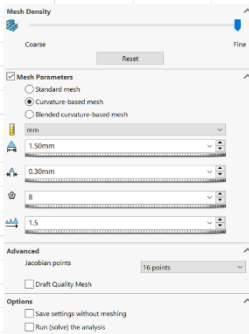
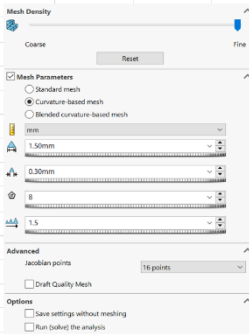
	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 7	Top wishbone	1.159E+07	0.0114	1.275E-04	
	Bottom wishbone	3.940E+07	0.0394	4.153E-04	
	Acceleration	3.334E+07	0.0227	3.546E-04	
	Braking	4.346E+07	0.0365	4.663E-04	
	Combo	3.960E+07	0.0396	4.172E-04	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.160E+07	0.0111	1.801E-04	
	Bottom wishbone	1.713E+08	0.0525	6.618E-04	
	Acceleration	3.863E+07	0.0234	6.527E-04	
	Braking	8.735E+08	0.0198	3.287E-04	
	Combo	1.988E+08	0.0533	6.814E-04	
New design 7	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	1.913E+07	0.0114	2.310E-04	
	Bottom wishbone	6.501E+07	0.0397	7.978E-04	
	Acceleration	5.373E+07	0.0228	8.244E-04	
	Braking	4.759E+07	0.0366	5.085E-04	
Original design (Bonded contact set)	Combo	6.537E+07	0.0398	8.107E-04	
	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.494E+07	0.0111	2.512E-04	
	Bottom wishbone	2.061E+08	0.0528	9.675E-04	
	Acceleration	4.749E+07	0.0235	7.580E-04	
	Braking	8.245E+07	0.0169	3.156E-04	
	Combo	2.396E+08	0.0536	9.983E-04	

Table 16-14 - FEA results for design 8

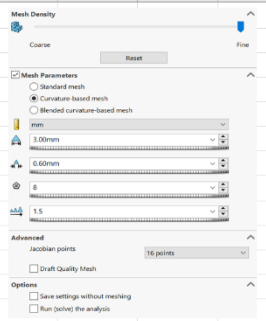
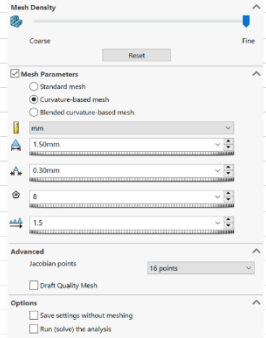
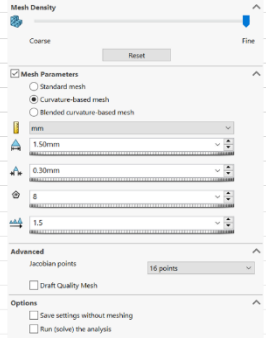
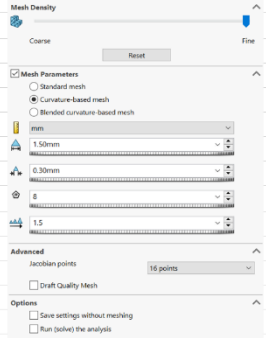
	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
New design 8	Top wishbone	1.163E+07	0.0119	1.286E-04	
	Bottom wishbone	4.092E+07	0.0405	4.234E-04	
	Acceleration	3.703E+07	0.0242	3.914E-04	
	Braking	4.324E+07	0.0374	4.540E-04	
	Combo	4.115E+07	0.4307	4.240E-02	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.160E+07	0.0111	1.801E-04	
	Bottom wishbone	1.713E+08	0.0525	6.618E-04	
	Acceleration	3.863E+07	0.0234	6.527E-04	
	Braking	8.735E+08	0.0198	3.287E-04	
	Combo	1.988E+08	0.0533	6.814E-04	
New design 8	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	1.973E+07	0.0120	2.558E-04	
	Bottom wishbone	7.846E+07	0.0408	1.030E-04	
	Acceleration	5.100E+07	0.0243	6.296E-04	
	Braking	4.704E+07	0.0376	5.007E-04	
	Combo	6.690E+07	0.0410	8.901E-04	
Original design (Bonded contact set)	Loadcase	Von mises (N/m ²)	Displacement (mm)	Strain	
	Top wishbone	5.494E+07	0.0111	2.512E-04	
	Bottom wishbone	2.061E+08	0.0528	9.675E-04	
	Acceleration	4.749E+07	0.0235	7.580E-04	
	Braking	8.245E+07	0.0169	3.156E-04	
	Combo	2.396E+08	0.0536	9.983E-04	

Table 16-14 shows how using the 1.5mm / 0.3mm element size mesh on average, for each loadcase the new upright design had lower stress and strain. On average, for each attachment point, the final design had 52% lower stress, 2% larger displacement and 29% lower strain compared to the original.

16.5 Stress distributions for original and final upright design

Each of the results show scaled displacements for better representation of how the uprights would deform. The same mesh could not be used as the original upright failed when using the 2.5mm element size with 1.1mm refinements, therefore, the all the elements are 2.5mm for the original upright.

16.5.1 Acceleration load

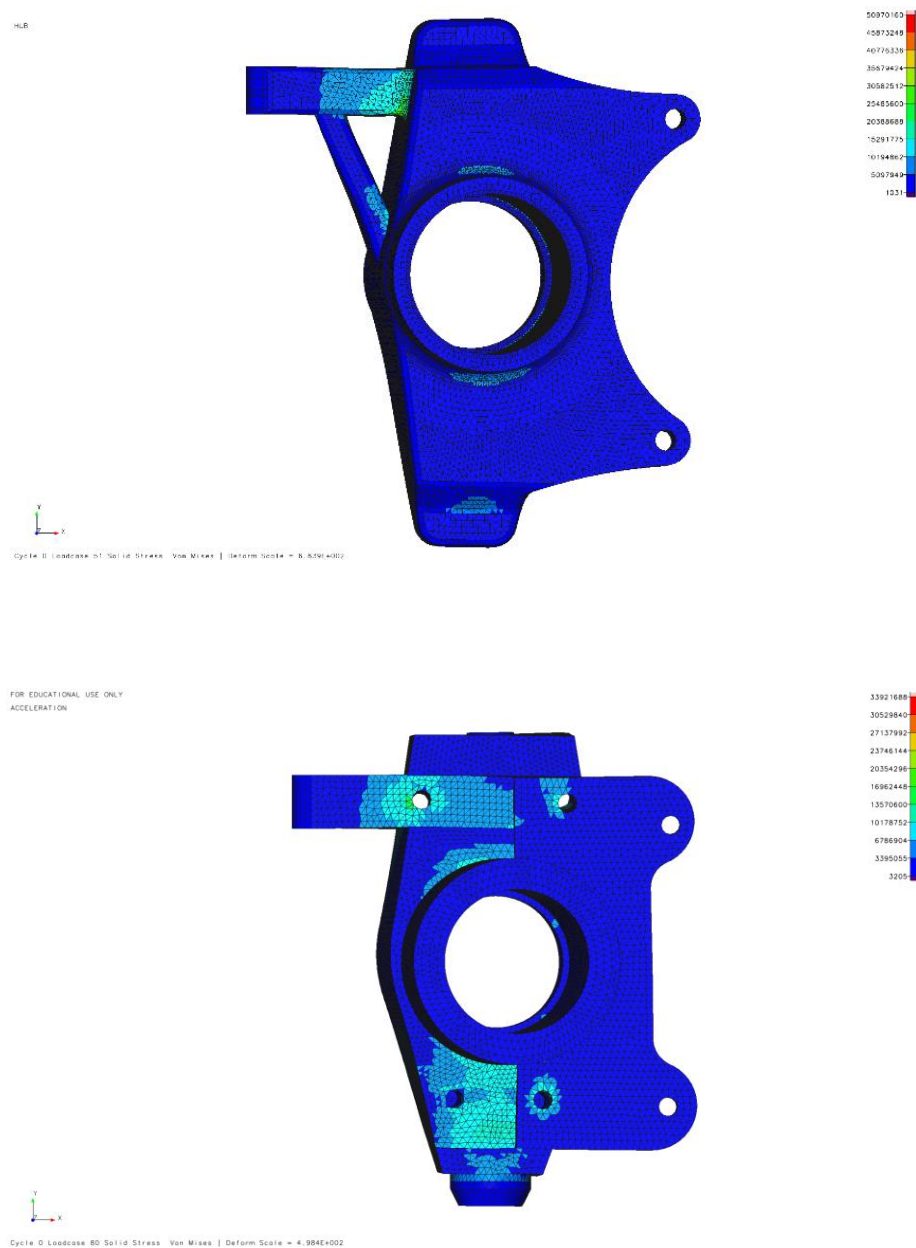


Figure 16-14 – Von Mises stress under acceleration load of 6145.8N for the original and new upright design

16.5.2 Upper wishbone

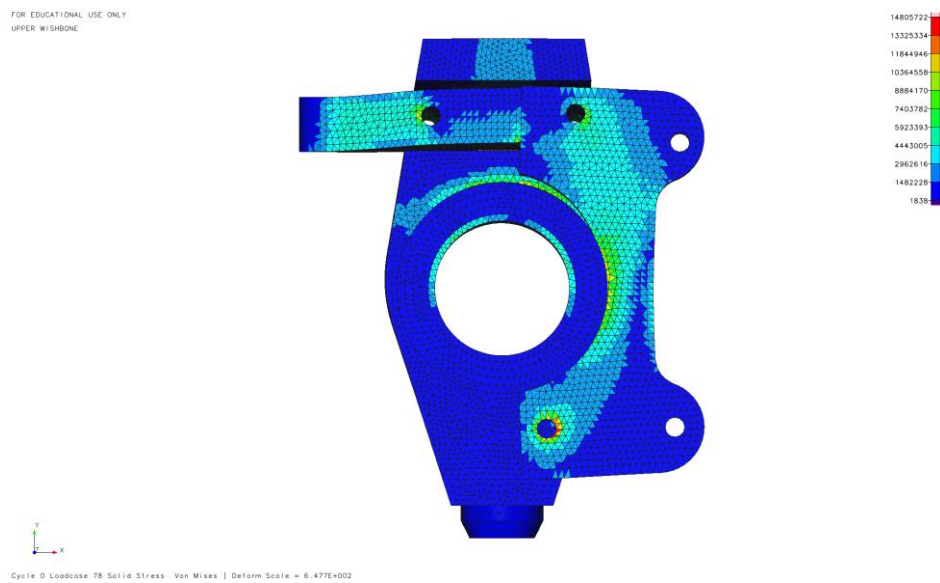
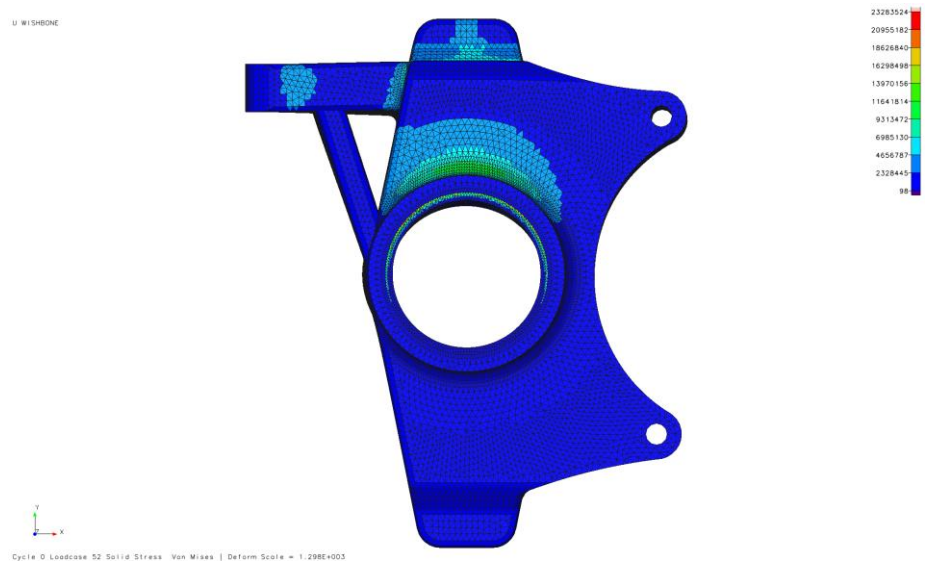


Figure 16-15 - Von Mises stress under upper wishbone load of 2696.1N for the original and new upright design

16.5.3 Lower wishbone

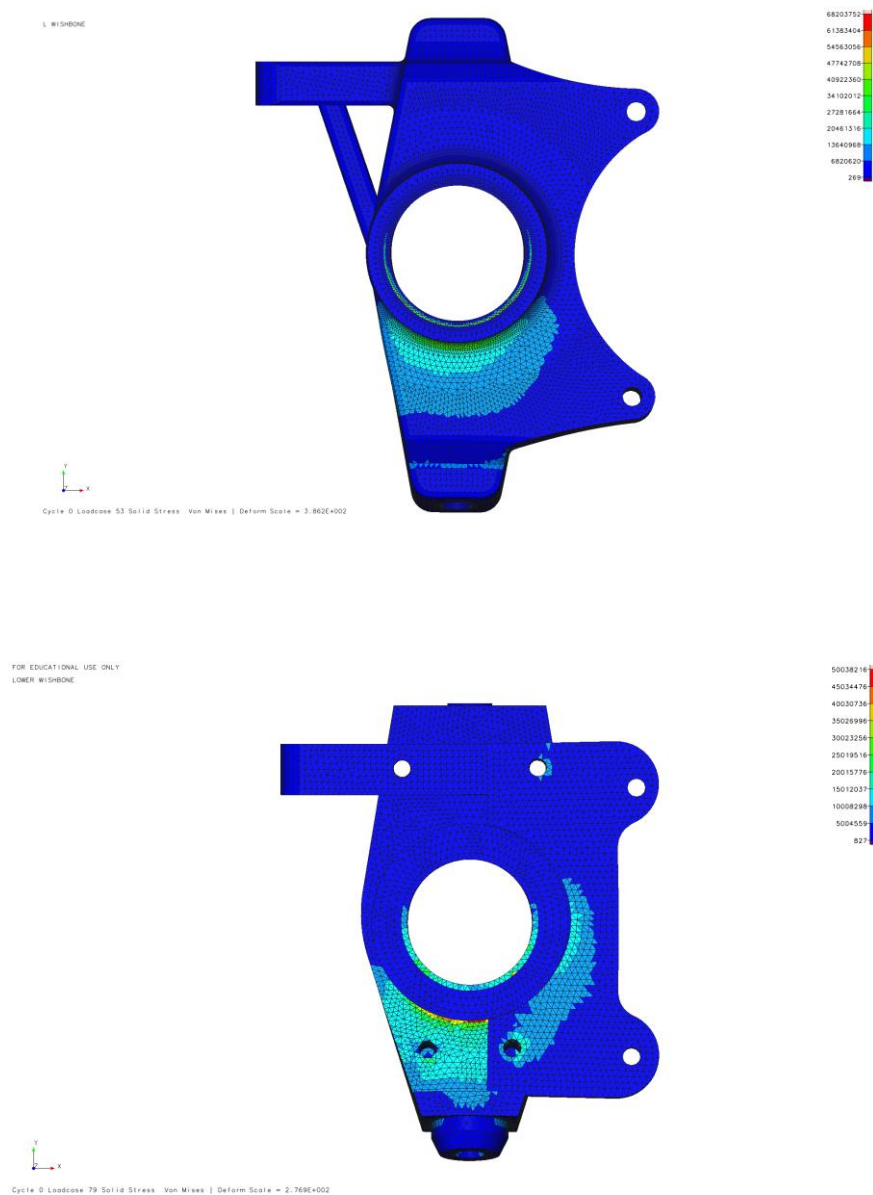


Figure 16-16 - Von Mises stress under lower wishbone load of 7279.4N for the original and new upright

16.5.4 Steering

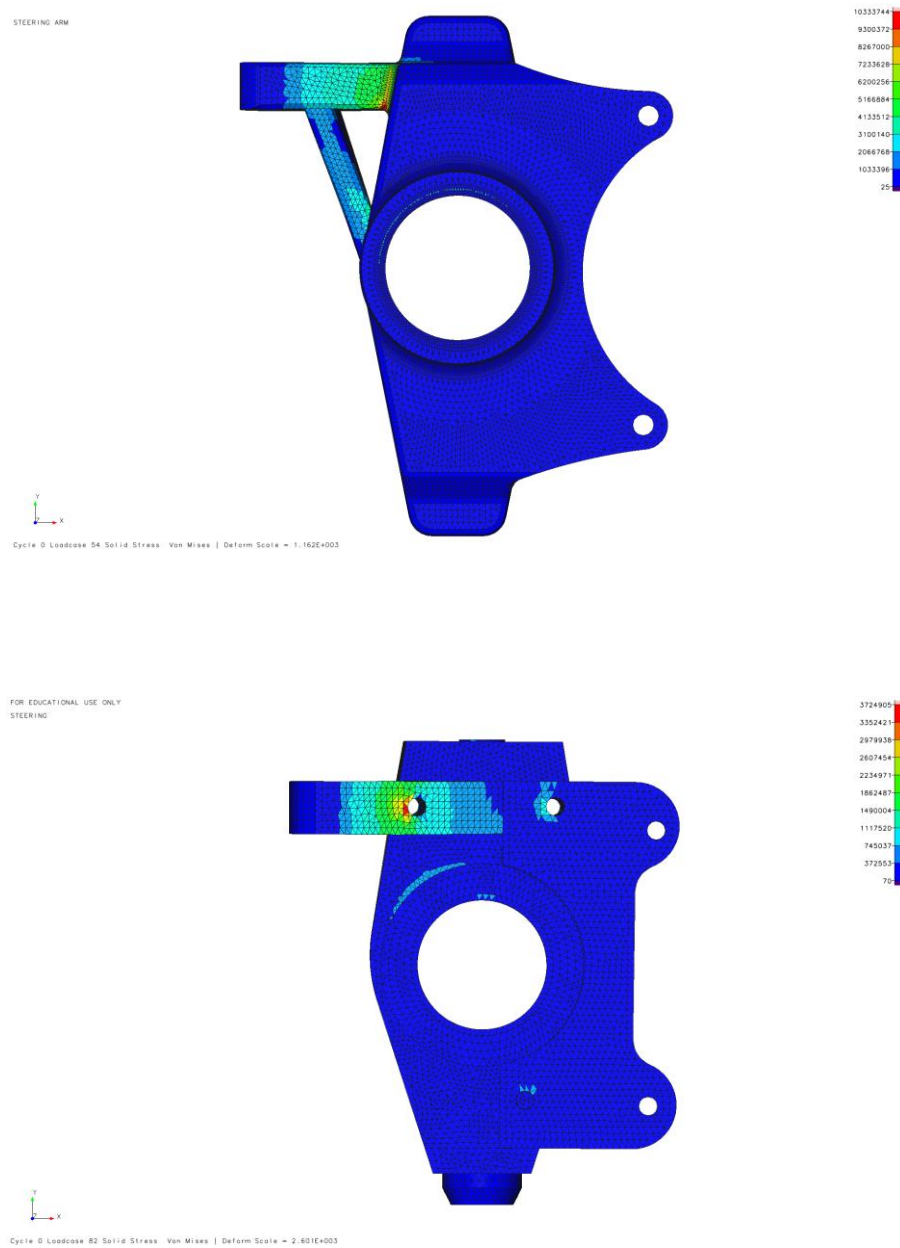


Figure 16-17 - Von Mises stress under steering load of 250N for the original and new upright

16.5.5 Braking

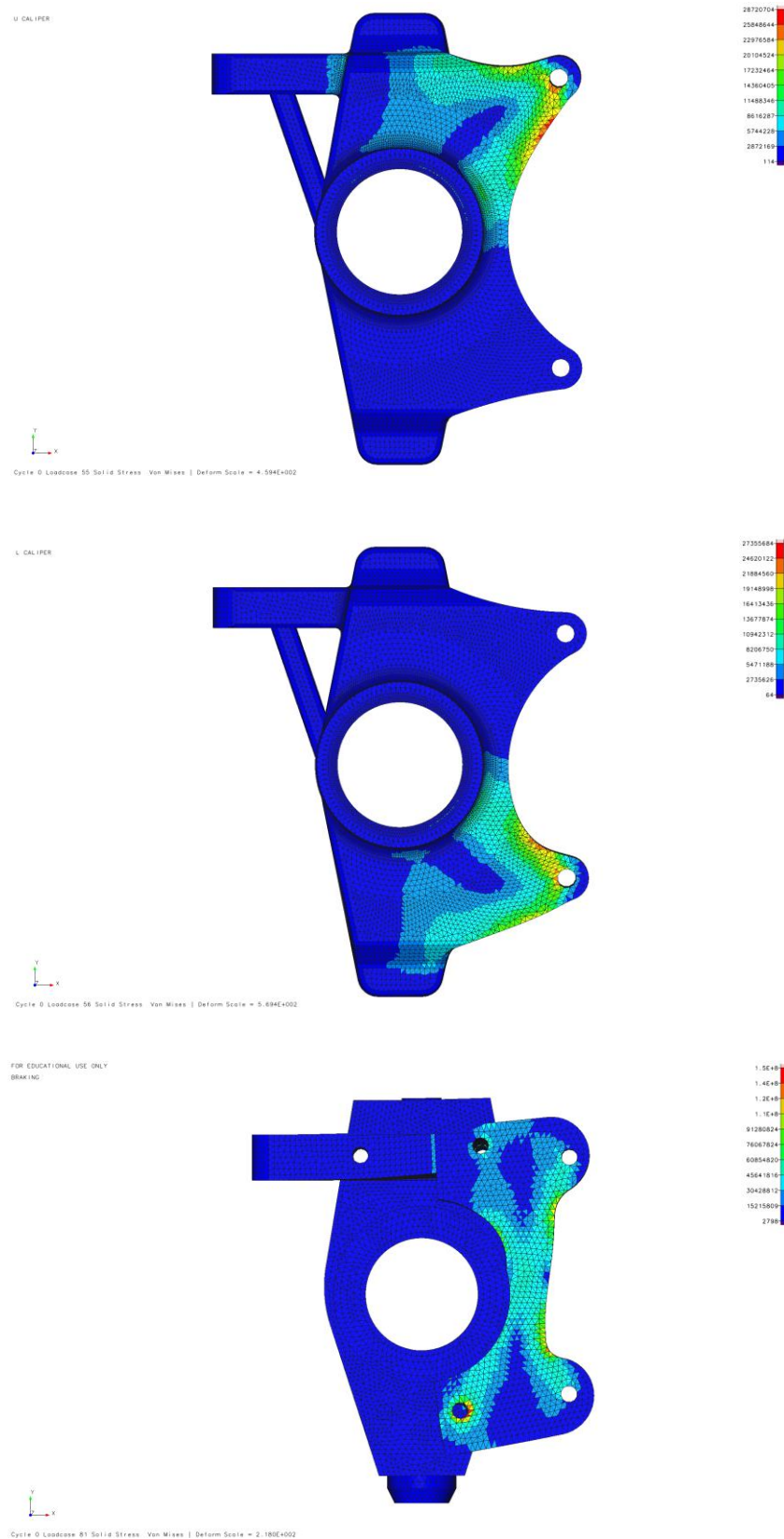


Figure 16-18 - Von Mises stress under braking load of 7357.8N for the original and new upright

16.6 3D printed models of original and new upright



Plate 16-1 - 3D printed models of the original and new upright design (front)



Plate 16-2 - 3D printed models of the original and new upright design (rear)

Plate 16-1 and *16-2* show the 3D models printed by Michael Anderson. The CAD models were saved as a '.STL' file and sent to Michael Anderson for production. The models were 40% scale, to give the audience a visual representation of the original and new upright during the final viva presentation.

16.7 Upright CNC quotation

Enquiries > Enquiry 84323-216 > Part: New upright design Updated 3.SLDPRT

New upright design Updated 3.SLDPRT

Quantity: 1

BACK SAVE € 674.06 Delivery 05.04.2025 Est. Unit

MANUFACTURING PROCESS

*Process: **CNC Machining**

MATERIAL

*Material: **Aluminium EN AW-7075 / 7A18X / AlZnMgCu**

COMPLICATIONS

*Finish: **Standard**

Threads and Tapped Holes: Specify the number of standard threads in this part.

Tolerance: **±0.025mm** Units: **1**

Tolerances will be tied to Standard in accordance with ISO 2768 unless otherwise specified.

Surface Roughness: **Standard (R1.2 um Ra)** Surface Finish: ☐ Yes ☒ No

Surface roughness will be tied to Standard unless otherwise specified.

PART MARKING

☐ Bag and Tag

☐ Preparing

☐ Laser Mark

Notes:

DFM Feedback

Drawing requested
The features requested require additional location information or details.

Recommendation:
Upload a drawing or image identifying additional features or custom requests in the drawings section.

Drawings

PDF | DWG | JPG | JPEG | PNG

UPLOAD FILE

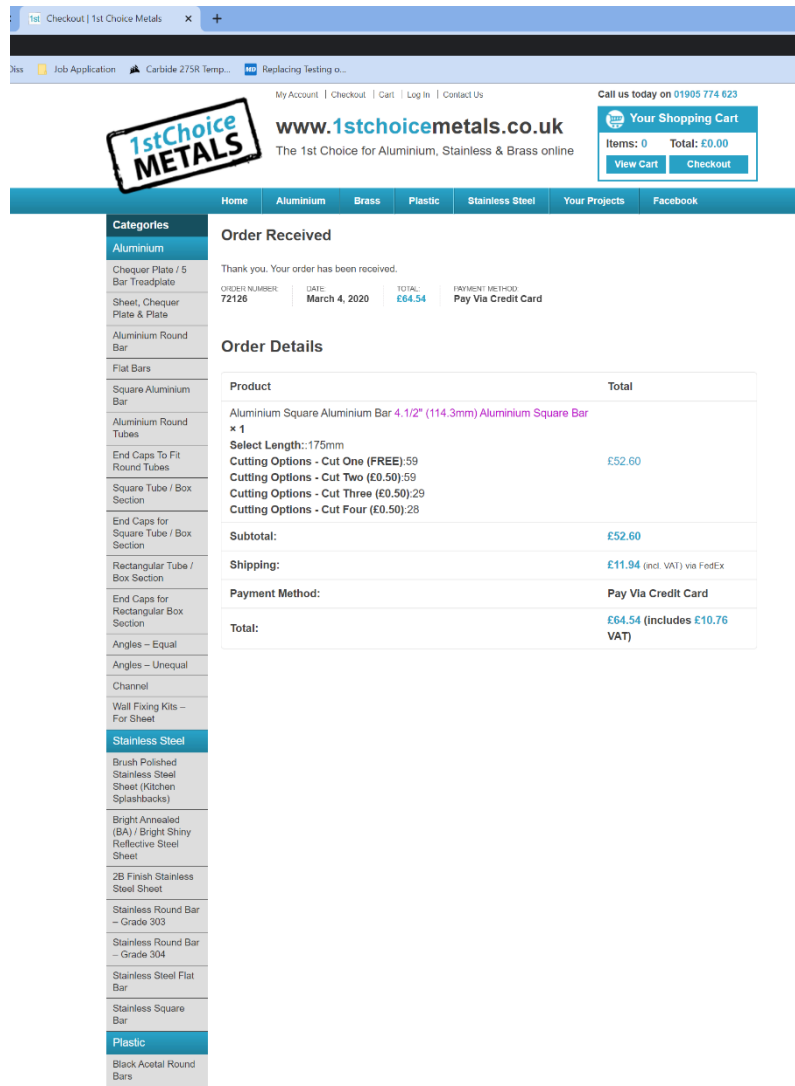
Max file size: 10MB

Wir verwenden Cookies, um unsere Website für Sie beutzerfreundlicher zu gestalten. Weitere Informationen

COOKIES AKZEPTIEREN

Plate 16-3 - CNC quotation from Xometry for design 3

16.8 6082-T6 Aluminium order



1st Choice METALS

www.1stchoicemetals.co.uk
The 1st Choice for Aluminium, Stainless & Brass online

Call us today on 01905 774 623

Your Shopping Cart
Items: 0 Total: £0.00
[View Cart](#) [Checkout](#)

Order Received
Thank you. Your order has been received.

ORDER NUMBER	DATE	TOTAL	PAYMENT METHOD
72126	March 4, 2020	£64.54	Pay Via Credit Card

Order Details

Product	Total
Aluminium Square Aluminium Bar 4.1/2" (114.3mm) Aluminium Square Bar x 1 Select Length: 175mm	
Cutting Options - Cut One (FREE): 59	£52.60
Cutting Options - Cut Two (£0.50): 59	
Cutting Options - Cut Three (£0.50): 29	
Cutting Options - Cut Four (£0.50): 28	
Subtotal:	£52.60
Shipping:	£11.94 (incl. VAT) via FedEx
Payment Method:	Pay Via Credit Card
Total:	£64.54 (includes £10.76 VAT)

Categories

- Aluminium**
 - Chequer Plate / 5 Bar Treadplate
 - Sheet, Chequer Plate & Plate
 - Aluminium Round Bar
 - Flat Bars
 - Square Aluminium Bar
 - Aluminium Round Tubes
 - End Caps To Fit Round Tubes
 - Square Tube / Box Section
 - End Caps for Square Tube / Box Section
 - Rectangular Tube / Box Section
 - End Caps for Rectangular Box Section
 - Angles – Equal
 - Angles – Unequal
 - Channel
 - Wall Fixing Kits – For Sheet
- Stainless Steel**
 - Brush Polished Stainless Steel Sheet (Kitchen Splashbacks)
 - Bright Annealed (BA) / Bright Shiny Reflective Steel Sheet
 - 2B Finish Stainless Steel Sheet
 - Stainless Round Bar – Grade 303
 - Stainless Round Bar – Grade 304
 - Stainless Steel Flat Bar
 - Stainless Square Bar
- Plastic**
 - Black Acetal Round Bars

Plate 16-4 - 6082-T6 aluminium invoice

Required material:

Full size models:

250mm x 200mm x 100mm – UPRIGHT (x2)

250mm x 200mm x 40mm – JIG (x2)

40% scale models:

100mm x 80mm x 40mm – Upright (x2)

100mm x 80mm x 16mm – Jig (x2)

One square bar (100 x 100) with length 112mm

16.9 Supervision logs with Kelvin Lake

16.9.1 15/10/19



Supervision Form	
MODULE: Dissertation	
ASSIGNMENT: Development of the MCR suspension upright.	
STUDENT: Ifan Hopkins	SUPERVISOR: Kelvin Lake
DATE: 15/10/19	Length:
Objective of the meeting	
To discuss the aims of the project	
Work Submitted by the student	
- none	
Issues discussed at the meeting	
- 1. A topology study of the front MCR suspension upright was completed in 2018/2019. This project will take the results of this study and develop and test the upright. Ifan has received and reviewed the project from last year. Revisit this taking into account what was discussed today. 2. Try to contact Krish to obtain the following a. His Design Studio files. b. His CAD and FEA models of the current MCR upright. 3. If the models in 2 can be obtained, these will be used in your project. If not, you can take the design input from Design Studio from the images in the final report that was submitted and create the CAD of the current design by taking measurements from the car. 4. Create a Gantt chart. Add your deliverable dates to this. This is a live document. Create a contents page for your report. 5. Potentially 2 or 3 designs could be produced virtually and compared to the current design, before manufacturing and testing your suggested design. 6. Research topology and uprights. Download and read the structural optimisation book from Altair. Speak to TT re suspension upright design.	
Student action BEFORE next meeting	
1, 2, 4, 6.	
Date of next meeting: 24/10/19 - 1pm	



Supervision Form	
MODULE: Dissertation	
ASSIGNMENT: Development of the MCR suspension upright.	
STUDENT: Ifan Hopkins	SUPERVISOR: Kelvin Lake
DATE: 24/10/19	Length: 15mins
Objective of the meeting	
To discuss the aims of the project	
Work Submitted by the student	
- none	
Issues discussed at the meeting	
- 1. IH has contacted Krish and received his CAD files. However these appear to be corrupt, so IH will obtain these from Ainsley on the MCR team. Alternatively, the MCR is due to race this weekend for the last time this season. After that the upright can be removed and used to create a CAD file if necessary. IH to contact Krish for the relevant DS files. If these have been deleted, is a back-up available - speak to IT. 2. Create a Gantt chart. Add your deliverable dates to this. This is a live document. Create a contents page for your report. 3. Research topology and uprights. The structural optimisation book and the FEA book have been downloaded. From Altair, IH is still in the process of reading these. Finish these off. In addition TT has recommended 3 books on uprights from the library. These have been reserved by IH.	
Student action BEFORE next meeting	
1, 2, 3	
Date of next meeting: 1/11/19 - 9am - NOTE THIS IS NOW FRIDAY.	



Supervision Form	
MODULE: Dissertation	
ASSIGNMENT: Development of the MCR suspension upright.	
STUDENT: Ifan Hopkins	SUPERVISOR: Kelvin Lake
DATE: 01/11/19	Length:
Objective of the meeting	
To discuss the aims of the project	
Work Submitted by the student	
- none	
Issues discussed at the meeting	
1. IH has started to research Topology (Mirzendehe and Suresh) 2. The DS file is not available - check with IT to see if there is a back-up of Krishs work on about the 07/05/19. Either the DS file, or the CAD of the package space. 3. CAD has been obtained of the current upright. Create you FE model of this. 4. A paper has been found that calculates the forces seen by an upright on a formula SAE car. Apply this methodology, along with the Motec data to calculate the forces seen by the current upright. Speak to TT to help you to do this. 5. A contents page has been generated - this is very optimistic, however the design methodology is understood. Progress on the tasks in this methodology will be monitored throughout the project.	
Student action BEFORE next meeting	
2, 3, 4	
Date of next meeting: 04/11/19	



Supervision Form	
MODULE: Dissertation	
ASSIGNMENT: Development of the MCR suspension upright.	
STUDENT: Ifan Hopkins	SUPERVISOR: Kelvin Lake
DATE: 04/11/19 - Major project week	Length: 10 mins
Objective of the meeting	
To discuss the aims of the project	
Work Submitted by the student	
- none	
Issues discussed at the meeting	
- 1. The DS file is not available - check with IT to see if there is a back-up of Krishs work on about the 07/05/19. Either the DS file, or the CAD of the package space. 2. CAD has been obtained of the current upright. However this is incorrect. IH to recreate this by taking measurements of the upright that is now off of the car.. 3. A paper has been found that calculates the forces seen by an upright on a formula SAE car. Apply this methodology, along with the Motec data to calculate the forces seen by the current upright. IH has mailed TT for help doing this and is waiting for a reply.	
Student action BEFORE next meeting	
1, 2, 3	
Date of next meeting: 5/11/19	



Supervision Form	
MODULE: Dissertation	
ASSIGNMENT: Development of the MCR suspension upright.	
STUDENT: Ifan Hopkins	SUPERVISOR: Kelvin Lake
DATE: 05/11/19 - Major project week	Length: 20 mins
Objective of the meeting	
To discuss the aims of the project	
Work Submitted by the student	
- none	
Issues discussed at the meeting	
- 1. IH has spoken to IT regarding getting the back-up from onedrive. Krish needs to confirm that he is happy for this from a GDPR point of view. 2. CAD has been produced of the existing part. Thus it is possible to build an FE model and measure the stiffness at each attachment point. This will be done in design studio. 3. DS tutorials have been sent to IH, complete these. This will help you to understand topology and introduce you to DS. We can then set up your FE model in DS on friday. 4. TT has mailed IH to discuss how to calculate the forces going through the upright. Complete this. 5. Continue your research into uprights.	
Student action BEFORE next meeting	
1, 3, 4	
Date of next meeting: 8/11/19 - 9am (IQ006)	



Supervision Form	
MODULE: Dissertation	
ASSIGNMENT: Development of the MCR suspension upright.	
STUDENT: Ifan Hopkins	SUPERVISOR: Kelvin Lake
DATE: 22/11/19	Length: 30 mins
Objective of the meeting	
To discuss the aims of the project	
Work Submitted by the student	
- none	
Issues discussed at the meeting	
- 1. Build your 27 loadcases in DS - work out your stiffness in each direction - we can review this next week. 2. Research different uprights.	
Student action BEFORE next meeting	
1, 2	
Date of next meeting: 29/11/19 - 9am (IQ006)	



Supervision Form	
MODULE: Dissertation	
ASSIGNMENT: Development of the MCR suspension upright.	
STUDENT: Ifan Hopkins	SUPERVISOR: Kelvin Lake
DATE: 24/01/2020	Length: 30 mins
Objective of the meeting	
To discuss the aims and the progress of the project	
Work Submitted by the student	
- none	
Issues discussed at the meeting	
- 1. A mesh convergence study has been carried out - repeat this, looking at the stress a couple of elements away from your BCs as this may be artificially pushing up your results. 2. A topology run has been set up in Solidworks with your 'operational loads' - repeat this is Design Studio to compare the results. 3. Bring along your new design next week so we can discuss how the topology runs have informed your design. 4. The manufacturing process has not been considered in your new design - this will be the next step.	
Student action BEFORE next meeting	
1, 2, 3	
Date of next meeting: 31/01/2020- 9am	



Supervision Form	
MODULE: Dissertation	
ASSIGNMENT: Development of the MCR suspension upright.	
STUDENT: Ifan Hopkins	SUPERVISOR: Kelvin Lake
DATE: 24/01/2020	Length: 30 mins
Objective of the meeting	
To discuss the aims and the progress of the project	
Work Submitted by the student	
- none	
Issues discussed at the meeting	
- 1. The new design has less stress and smaller displacements for all the assessed loadcases other than the acceleration loadcase. It is ~1kg lighter. 2. The topology result is significantly different to the Design Studio result. 3. The mesh convergence study is still not showing convergence. 4. Plot a graph of volumes v cost. This graph will have a line for each of your manufacturing processes. 5. Put your new design into Design Studio and assess the stiffness at each of the attachment points and then compare these to the stiffness of the same attachment points in a one page table that can be used in your report. 6. Alu 7075 T6 was used. The volume needed is approx 230mm x 200mm x 81mm. Cost this up and see where this can be sourced from so that this can be machined. At the same time see if any material is available in the workshop and assess the performance of this with you new design in your FE model.	
Student action BEFORE next meeting	
4, 5, 6	
Date of next meeting: 07/02/2020	

16.9.9 Unlogged meetings

8/11/2019 - Design Studio tutorial with Kelvin Lake for approximately 30 minutes

15/11/2019 - Tried meshing the CAD model in SolidWorks and exporting as a '.NASTRAN' file to import into Design Studio with Kelvin Lake. There were errors whilst trying with Kelvin but managed to complete later in the day

22/11/2019 - Tutorial with Kelvin Lake setting up RBE2 features again and defining the static loads and boundary conditions. After the session all the work was completed, but had a 'fatal error'

25/11/2019 - Met with Kelvin Lake for approximately 20 mins as the FEA solve had a 'fatal error' – The independent and dependent nodes of the RBE2 features were constrained. Was sorted and had a running solve

17/12/2019 – Meeting with Kelvin Lake

7/02/20 – Meeting with Kelvin Lake

14/02/20 – Meeting with Kelvin Lake

21/02/20 - Material discussion with Kelvin and Tim – It was decided to buy cheaper grade Aluminium, since E is similar, stiffnesses shouldn't be affected. Research will be conducted, and materials purchased.

11/02/20 - Meeting with David Cooper discussed cost of new design 1 – Approximately 16 hours running on the 3-axis CNC machine and approximately £1000 materials (for a single upright and jig material)

21/02/20 - Discussed with David Cooper running scaled models using any grade aluminium since stiffness shouldn't change

4/03/20 - Met with David Cooper and asked if 6082-T6 aluminium would be adequate. Also found out 3mm radius is the smallest cutting size possible for the tool.

10/03/20 - Asked David Cooper whether a session could be booked to manufacture the uprights since the sourced material was delivered, was told to send the CAD and joined queue

27/03/20 - Microsoft Teams call with Kelvin Lake. Discussed including a chapter on pandemic impact and further work. Gantt chart section was discussed and was decided to include 'Proposed Gantt', 'Gantt prior to COVID-19' and 'Gantt following COVID-19'

3/04/20 - Microsoft Teams call with Kelvin Lake. Asked Kelvin about his meetings log, however, not all were recorded due to the pandemic. Asked Kelvin whether elemental or nodal FEA values should be favoured

24/03/20 – Discussed feedback on major project report with Kelvin Lake.