



UNIVERSITY OF WALES TRINITY SAINT
DAVID

SCHOOL OF ENGINEERING

ADVANCED COMPUTATIONAL
METHODS:
MOTION SIMULATION AND FINITE
ELEMENT ANALYSIS

M6X00408

MAXIMILIAN HART
1700625

24-01-20

SOLIDWORKS FILES ONEDRIVE LINK:

https://uowtsd-my.sharepoint.com/:f:/g/personal/1700625_student_uwtsd_ac_uk/EpEac3L7nXJNrx77nKWWZwBSKQzdBcbhfTrqT5LkrsMPA?e=4vhBDU

Table of Contents

1	Introduction	4
2	Background Research	4
3	Modelling and Assembly	5
3.1	Second Year Group Project CAD	5
3.2	Remodelling of Components	5
3.3	Final Assembly	7
3.4	Global Variables and Equations	7
3.5	Material Selection.....	8
4	Motion Simulation	8
4.1	Initial Motion Simulation Conditions.....	9
4.2	Importing Motion Loads	9
5	Finite Element Analysis.....	10
6	Results - Optimisation of Upper Wishbone	11
6.1	Original Design (First Iteration)	11
6.2	Second Design Iteration.....	13
6.3	Third Design Iteration	15
6.4	Fourth Design Iteration	17
6.5	Fifth Design Iteration	19
6.6	Final Design (Sixth Iteration)	21
7	Analysis and Evaluation	23
7.1	Results.....	23
7.2	Simulation Improvements	24
7.3	Conclusion	24
8	Referenced Work	25

Table of Figures

Figure 1 - Second Year MCR Rolling Chassis Model	5
Figure 2 - Modelling of Upper Wishbone	5
Figure 3 - Logically Organised Feature Tree	6
Figure 4 - Final CAD Assembly	7
Figure 5 - Global Variables Used	7
Figure 6 - Von Mises Stress for Each Scenario	9
Figure 7 - Original Design CAD	11
Figure 8 - Original Design Von Mises Stress	11
Figure 9 - Original Design Displacement	12
Figure 10 - Iteration 2 CAD	13
Figure 11 - Iteration 2 Von Mises Stress	13
Figure 12 - Iteration 2 Displacement.....	14
Figure 13 - Iteration 3 CAD	15
Figure 14 - Iteration 3 Von Mises Stress	15
Figure 15 - Iteration 3 Displacement.....	16
Figure 16 - Iteration 4 CAD	17
Figure 17 - Iteration 4 Von Mises Stress	17
Figure 18 - Iteration 4 Displacement.....	18
Figure 19 - Iteration 5 CAD	19
Figure 20 - Iteration 5 Von Mises Stress	19
Figure 21 - Iteration 5 Displacement.....	20
Figure 22 - Final Design CAD	21
Figure 23 - Final Design Von Mises Stress	21
Figure 24 - Final Design Displacement	22
Figure 25 - Von Mises Stress Against Deflection.....	23
Figure 26 - Von Mises Stress Against Mass	23

Table of Tables

Table 1 - Materials Chosen.....	8
Table 2 - Original Design FEA Data	12
Table 3 - Iteration 2 FEA Data	14
Table 4 - Iteration 3 FEA Data	16
Table 5 - Iteration 4 FEA Data	18
Table 6 - Iteration 5 FEA Data	20
Table 7 - Final Design FEA Data	22

1 Introduction

For this report, the author shall analyse the upper wishbone within the double wishbone pushrod suspension system of the Sports 2000 MCR race car when hitting a track-side kerb. This component and assembly was chosen as it supports the author's studies, with a suspension-based dissertation and interest in vehicle dynamics.

Initially, the author conducted a critical analysis of the suspension system moving over a conventional trackside kerb; however, this did not place the component under any substantial stress. The component could have been optimised from here by maintaining strength and rigidity and reducing mass. As this report aims to analyse computer aided design (CAD) and finite element analysis (FEA) methods, it was decided that the assembly would be modelled to hit the kerb from the other direction. This produced much higher stresses, above the yield of the chosen materials. Although arguably unrealistic, this allowed the author to explore and further understand the processes of strengthening a part as well as reducing mass. From the initial wishbone, new designs were created and tested in a series of iterations to optimise the part.

2 Background Research

CAD modelling is an efficient way of analysing the properties and characteristics of a component, allowing various modifications to be made without the time or expense of producing a real-life version and physically testing it (Tovey, 2002). This is particularly important in an automotive application where components tend to be large, and often made of metal, thus making production expensive and time consuming (Bodein *et al.*, 2013).

FEA is a crucial tool for an engineer and is becoming more commonly used in the modern world. Software such as SolidWorks can be used to analyse how a component is affected by forces acting upon it and where over-stress or failure may be apparent (Rieg *et al.*, 2014). Use of FEA software allows multiple design iterations to be tested and analysed within a relatively short amount of time. Without computer aid, an engineer would have to produce physical models and test each one. If the test proved fatal to the component, a whole new model would have to be produced and re-tested, this is evidently not cost or time-efficient (Morris, 2008).

3 Modelling and Assembly

3.1 Second Year Group Project CAD

For a previous project (second year group project), the author had led a group of students who produced a CAD model of the MCR car seen below.

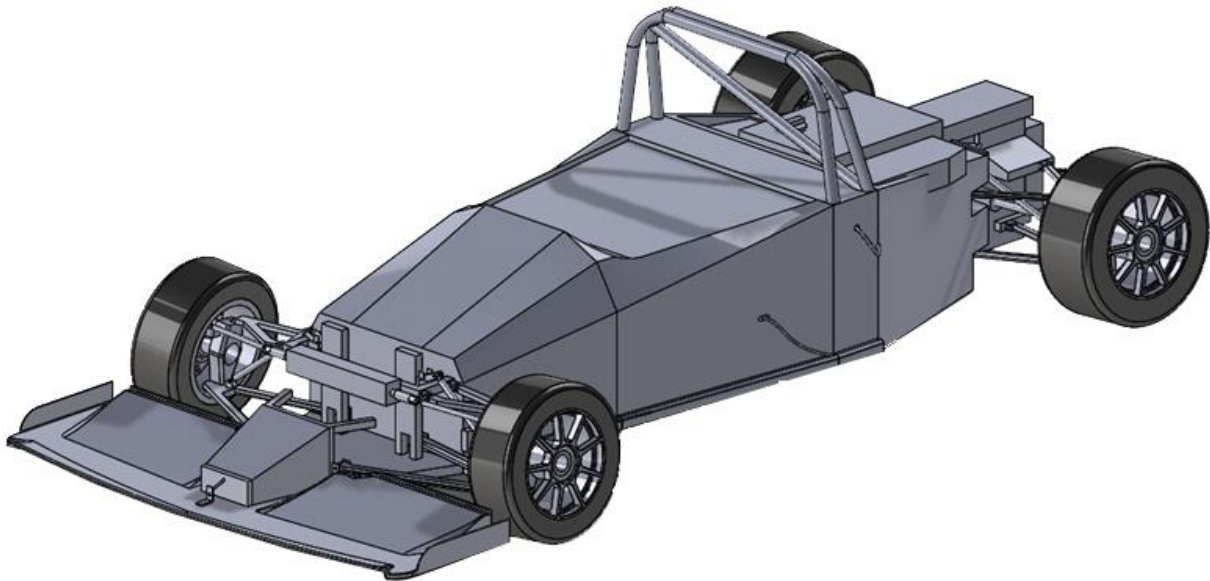


*Figure 1 - Second Year MCR Rolling Chassis Model
(Hart et al., 2018)*

For this assembly, the author produced the wheels, tyres, upright, various sections of the chassis and significantly contributed to building the overall assembly. This assembly was accurately measured and produced to fit bodywork over, so detail and mechanical fastenings are lacking in some areas. This will be addressed by the author.

3.2 Remodelling of Components

The author re-measured all components used for this report and checked against the existing cad, altering errors where necessary. All of the assembly apart from the front left corner was then removed, as was all unnecessary for this task. The author then re-modelled from scratch the upper wishbone as this was the component being analysed. Once accurately sketched, the shape was extruded by 15.5mm. Fillets and mounting holes were then added.

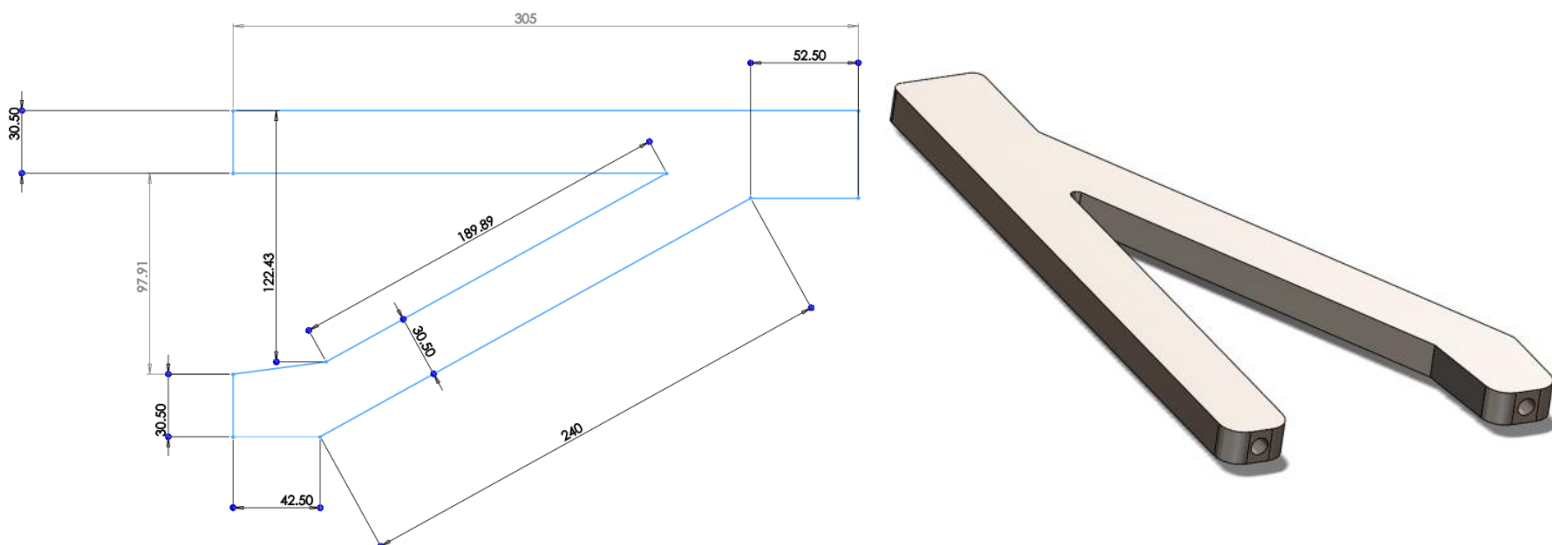


Figure 2 - Modelling of Upper Wishbone

One significant inaccuracy in the original CAD model was the lack of damper. This was simplistically modelled by the author, allowing an appropriate face upon which the damping characteristics can be modelled within the motion simulation.

Upon reassembling the model, the author incorporated as many mechanical mates and rigid body groups as possible to reduce mate redundancies further into the report. This shall be further explained in section 4.

For the efficient running of a large CAD assembly it is good practice to ensure the feature tree is neatly and logically organised. This will reduce computational strain and allow the user to find specific parts easily if they need to be re-named or edited (Lombard, 2013).

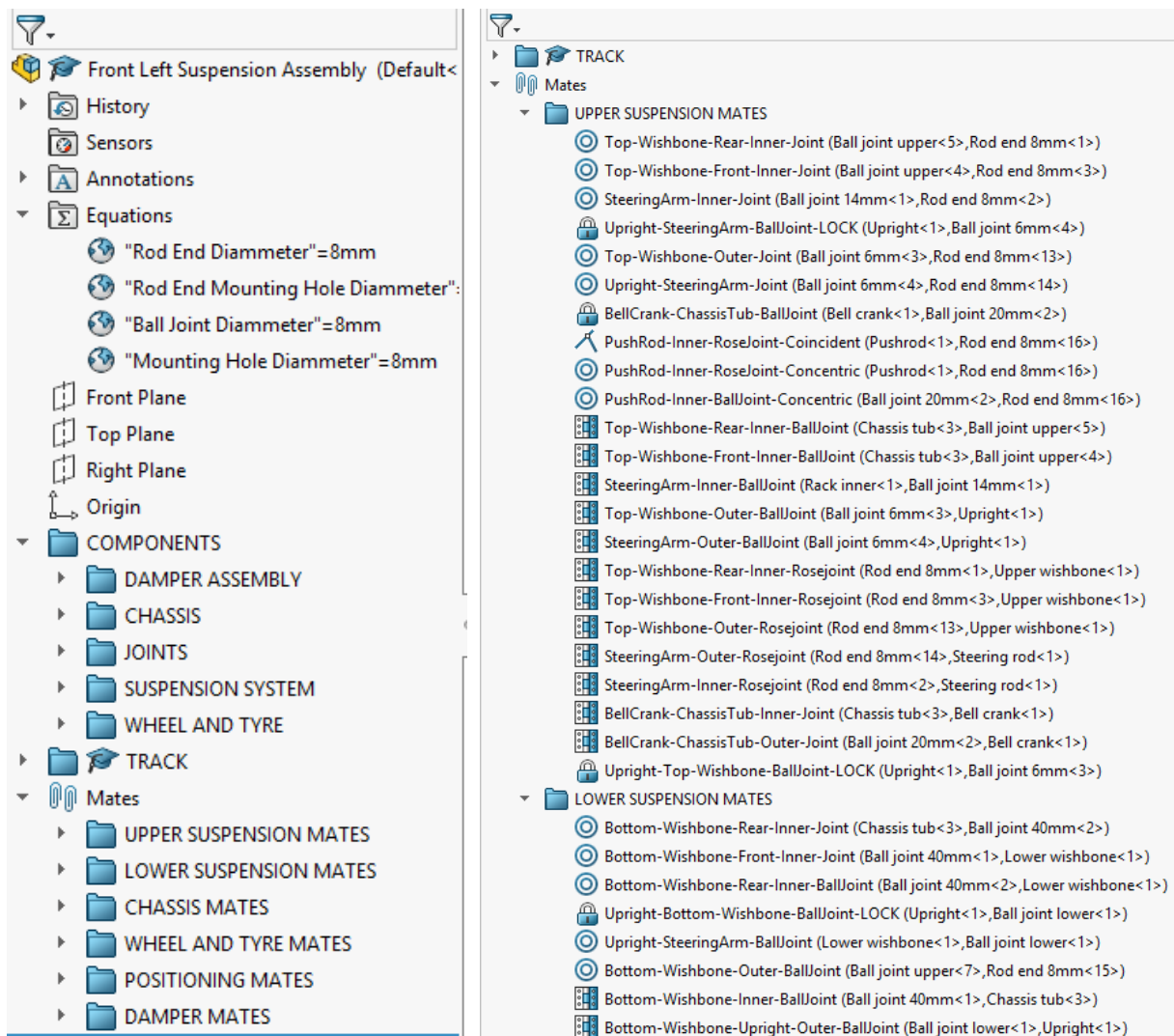


Figure 3 - Logically Organised Feature Tree

3.3 Final Assembly

Below the finished assembly, with all components and reversed track, can be seen in place.

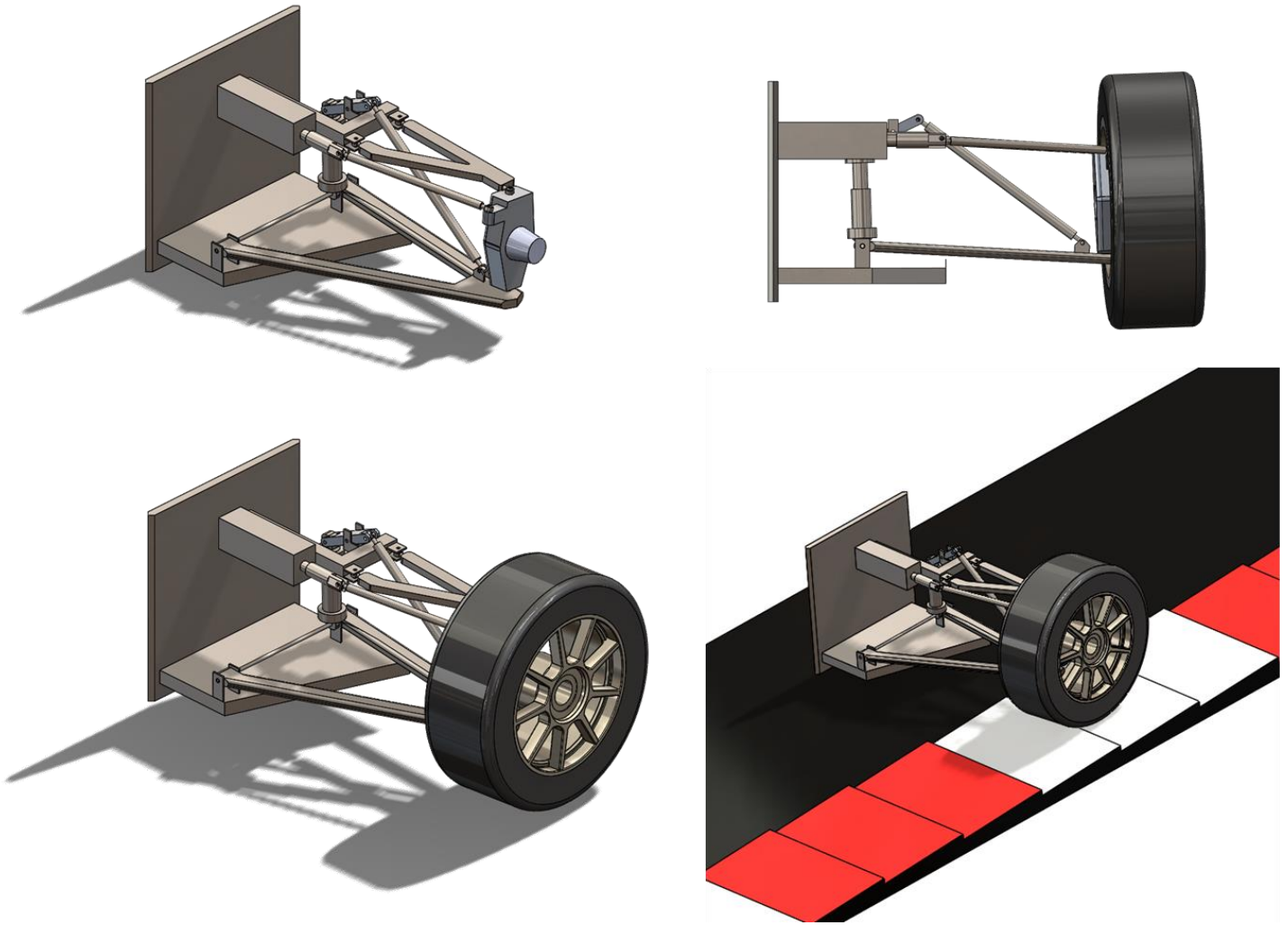


Figure 4 - Final CAD Assembly

3.4 Global Variables and Equations

Within a large assembly it is good practice to use global variable equations. This way, if one element of a component is changed, the corresponding components also change. For this assembly, a series of global variables relating to the ball and rose joints, and corresponding mounting holes were constructed. By altering the text file that assigns these variables, every sketch within the assembly that includes the variable will be altered. These can be interlinked within the assembly using equations (Ohara, 1984; Bodein *et al.*, 2014).

Name	Value / Equation	Evaluates to		Comments
☐ Global Variables				
"Rod End Diameter"	= 8mm	8mm	☒	
"Rod End Mounting Hole Diameter"	= 8mm	8mm	☒	
"Ball Joint Diameter"	= 8mm	8mm	☒	
"Mounting Hole Diameter"	= 8mm	8mm	☒	
Add global variable				

Figure 5 - Global Variables Used

3.5 Material Selection

To enhance the accuracy of the simulation, the author assigned each component a material:

Table 1 - Materials Chosen

Component	Material	Source
Wheel	Magnesium alloy	(O.Z Racing, 2018)
Tyre	Silicone rubber	(Korunovic <i>et al.</i> , 2014)
Track	Silicone rubber	
Rose joints	Alloy steel	(Levesley <i>et al.</i> , 2003)
Ball joints	Alloy steel	
Push rod	Alloy steel	
Steering rod	Alloy steel	
Damper	Alloy steel	
Chassis and side plate	Alloy steel	
Steering rack	Alloy steel	
Upper wishbone	Alloy steel	
Lower wishbone	Alloy steel	
Upright	Aluminium 1060 Alloy	(Levesley <i>et al.</i> , 2003)
Bell crank	Aluminium 1060 Alloy	

Due to some unclear information on what each component was made of, and the somewhat limited range of materials available within SolidWorks, materials were chosen that most closely resembled the actual thing, to the best of the author's knowledge.

4 Motion Simulation

To accurately model the forces on the wishbone, the author created a motion simulation to animate the wheel traveling over the kerb, whilst analysing and logging the loads on selected components.

The components were assembled using mates, which perform the job of mechanical fastenings. SolidWorks analyses these mates to understand how each component interacts; mates that are not integral to the running of the simulation are labelled redundant. Upon first assembling the components, around 70 redundant mates were found. The author replaced various mates with mechanical alternatives, such as hinge mates, and included rigid bodies to reduce this to 32 redundancies. This is still a high number; however it was decided the simulation was running correctly and accurately so this was not an issue. More mechanical mates, specifically within the rose joints, could be used to reduce redundancies for a better simulation in the future.

4.1 Initial Motion Simulation Conditions

Upon importing the assembly into the motion simulation, the author set a series of initial conditions.

- Downwards acceleration of 9.81m/s^2 added to apply gravitational force to the simulation.
- Linear motor was added to the assembly to move along the track at 10m/s (22.36mph).
- Solid bodies were set between the tyre and the road as the initial reaction force occurs here.
- A spring was placed between the chassis floor and top wishbone using correct MCR front spring rate of 39053 N/m (Hart, 2019).
- A damping force was placed on the damper using correct MCR front damping rate of 6254N/m/s (Hart, 2019).
- Load bearing faces were set within the ball joint mates on each of the upper wishbone joints.

4.2 Importing Motion Loads

The author viewed the animation to find where the wheel hits the kerb, and where it lands after the impact. The frames of these times were found, the upper wishbone selected, and motion loads were exported from the motion simulation into the component file. This was usually around 25-30 scenarios. From these, the author analysed the Von Mises stress and displacement to find the scenario with maximum of each, and analysed data from said scenario (J. Trevelyan, 1986).

The loads on the pushrod were analysed from the initial motion study and compared to MoTeC pushrod load data for validation that the simulation was producing reasonable forces. The loads matched and thus were validated and could be trusted.

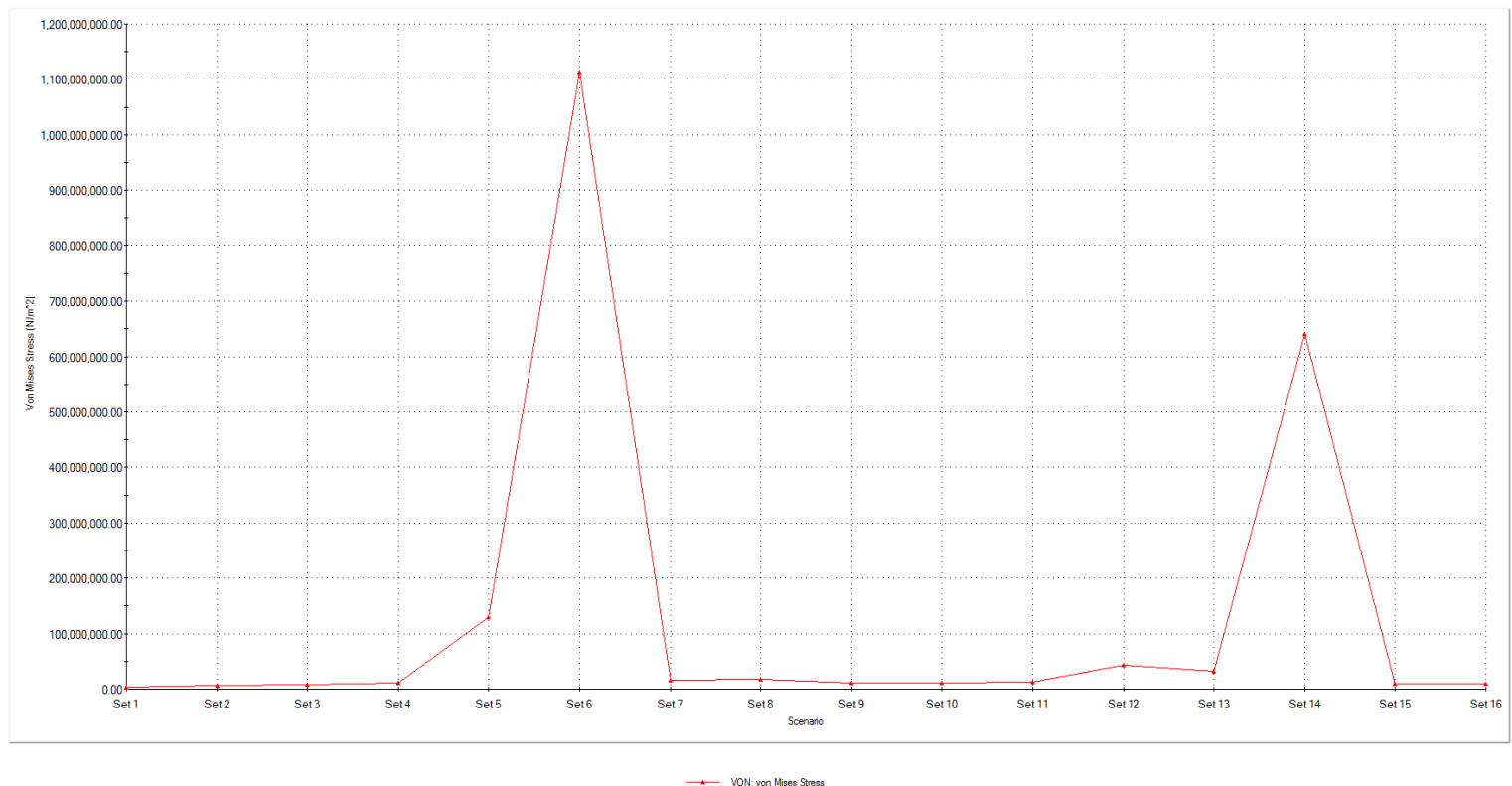


Figure 6 - Von Mises Stress for Each Scenario

5 Finite Element Analysis

For this report, the author will assess maximum Von Mises stresses and maximum displacements. Von Mises stress is a value used to analyse how a ductile material, such as metal will behave under complex loading; it gives a good indication of if the component is likely to reach yield, the point at which permanent deformation will occur, or fracture (Everett, 1994). The yield stress for the alloy steel upper wishbone is $6.204 \times 10^8 \text{ N/m}^2$.

Von Mises stress is a combination of first, second and third order principal stresses. At any point throughout the component the stress can be calculated as a scalar quantity and compared to observed yield points (Ajay Harish, 2019).

For these calculations, the author decided that the default mesh created by SolidWorks was sufficient and therefore was not altered. From each calculation, the component factor of safety (FoS) was calculated using:

$$FoS = \frac{\textit{Maximum Stress Experienced}}{\textit{Yield Stress}}$$

It is suggested that generally safety factors should be around 4, however in motorsport applications the desire to use lightweight components means a factor of safety around 2 is commonly seen. It should be remembered that this report studies a component loading more drastic than would usually be expected in average conditions, so FoS values calculated may be lower than a real-world application (McCartney, 1979).

6 Results - Optimisation of Upper Wishbone

6.1 Original Design (First Iteration)

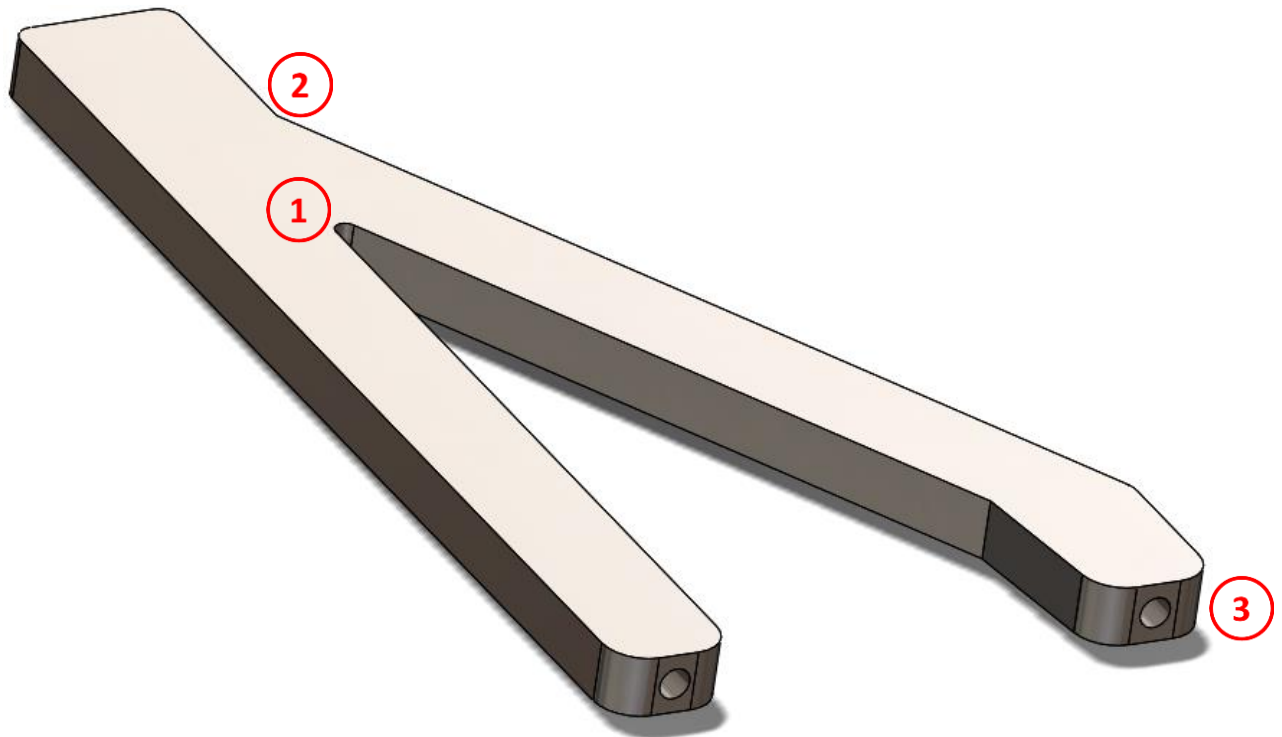


Figure 7 - Original Design CAD

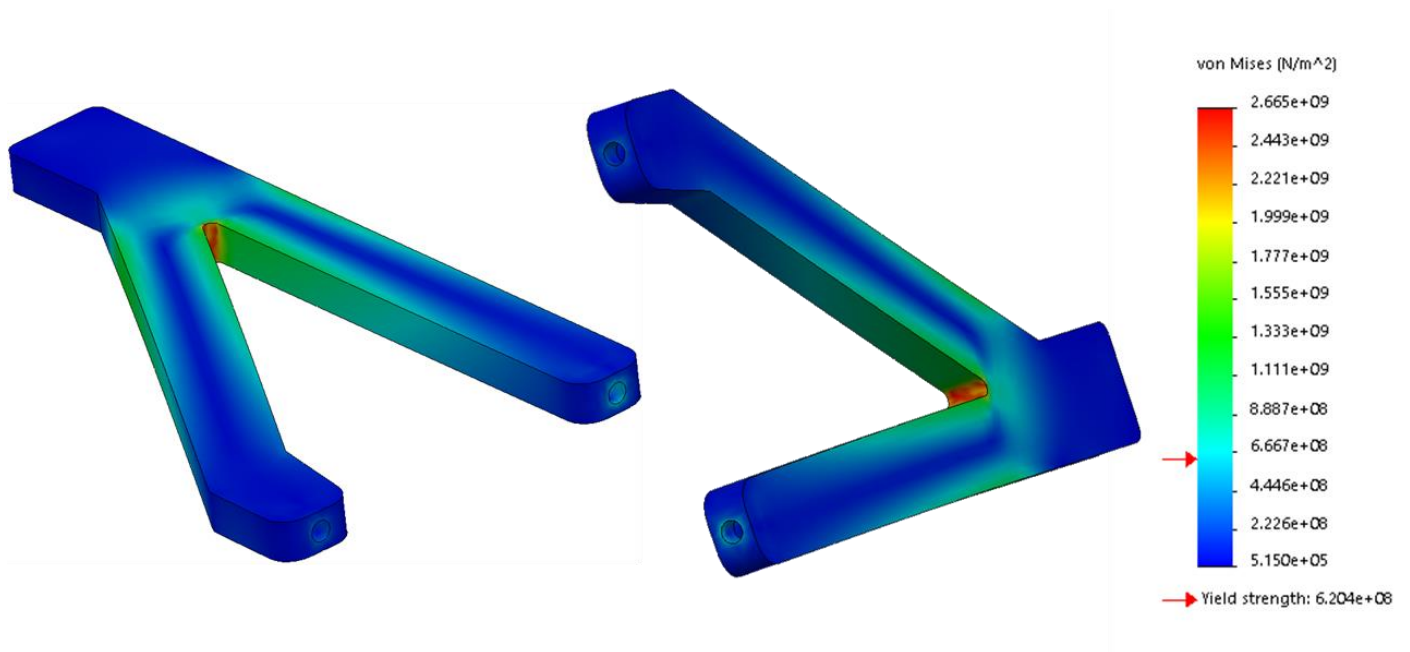


Figure 8 - Original Design Von Mises Stress

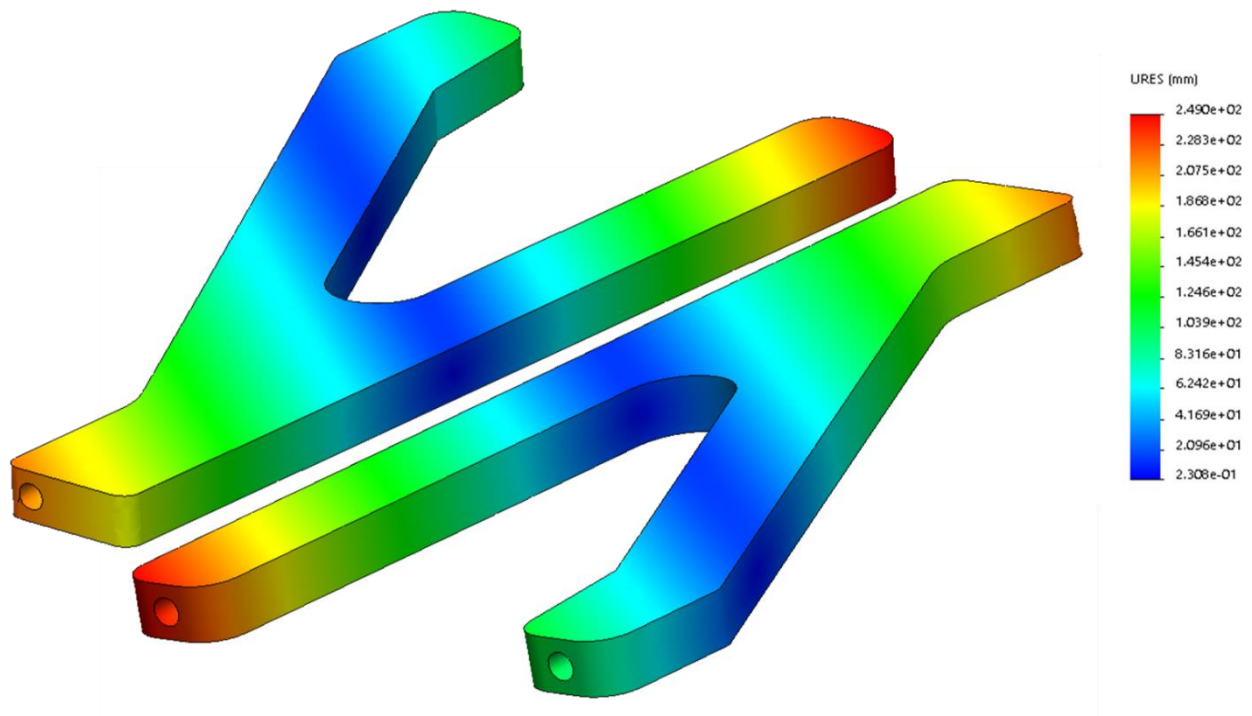


Figure 9 - Original Design Displacement

Table 2 - Original Design FEA Data

Maximum Von Mises Stress (N/m²)	2.665×10^9
Minimum Von Mises Stress (N/m²)	5.150×10^5
Maximum Deflection (mm)	2.490×10^2
Minimum Deflection (mm)	2.308×10^{-30}
Component Mass (kg)	2.154
Factor of Safety	N/A

The initial design had the following design features, as seen on figure 7:

- Inside radius (1): 3mm
- Outside radius (2): None
- Edge fillets: None
- End fillets (3): 2.5mm

It is clear that there is a significant increase in stress between the two arms. The loads applied are pulling them apart and taking the metal past yield point. This is due to the small radius in this location. A small radius fillet creates an abrupt change in geometric shape, meaning the internal stresses do not get distributed evenly within the component, this is known as a stress concentration and must be removed from the design (Sengupta, 2009).

Although not at yield point, there are also stress concentrations around the edges, a smooth fillet in later iterations will solve this.

Figure 9 shows an alarming 249mm deflection at each end of the component. This shows a lack of rigidity within the component, caused by the arms being pulled apart.

6.2 Second Design Iteration

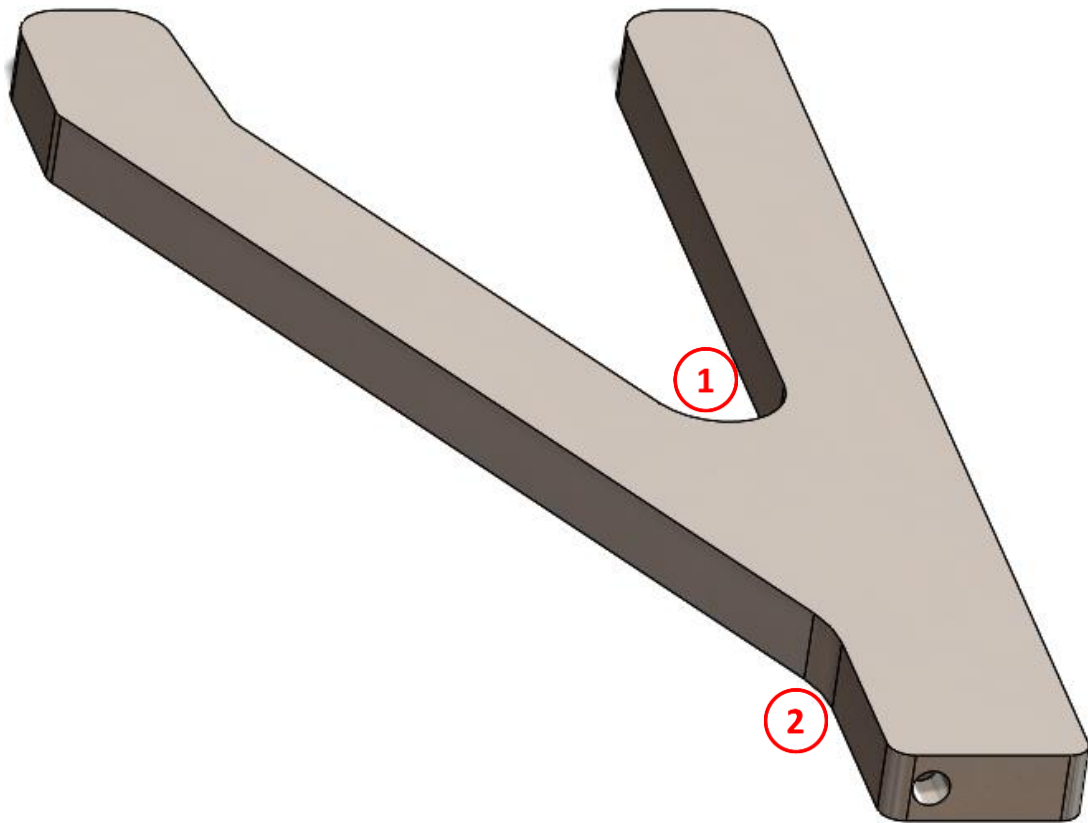


Figure 10 - Iteration 2 CAD

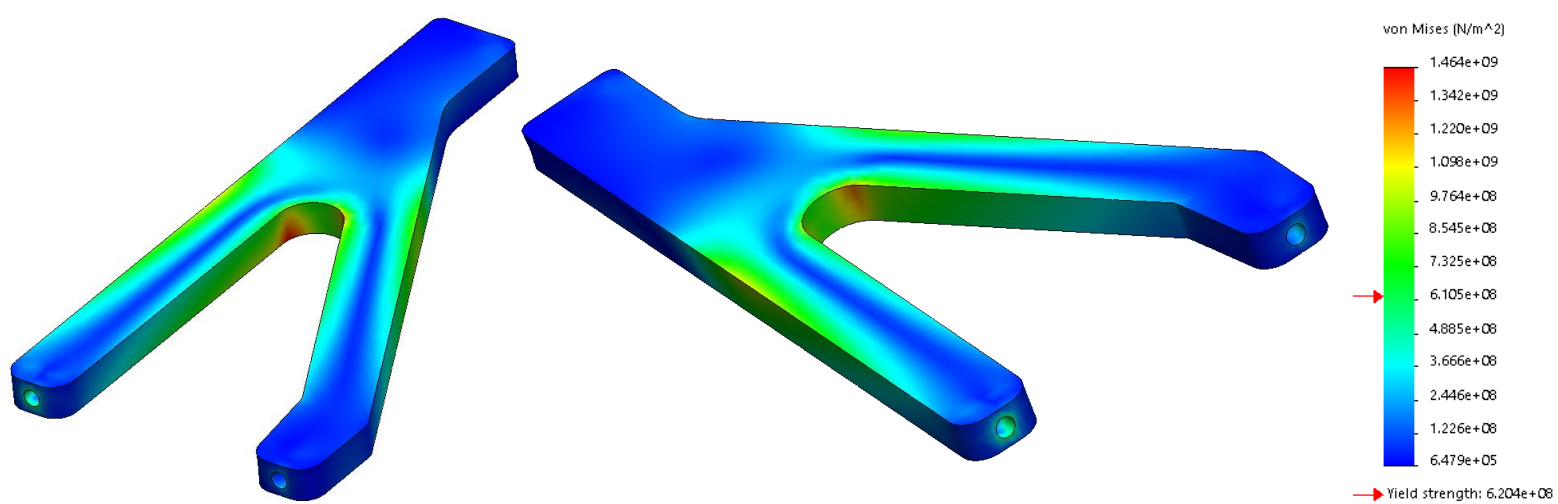


Figure 11 - Iteration 2 Von Mises Stress

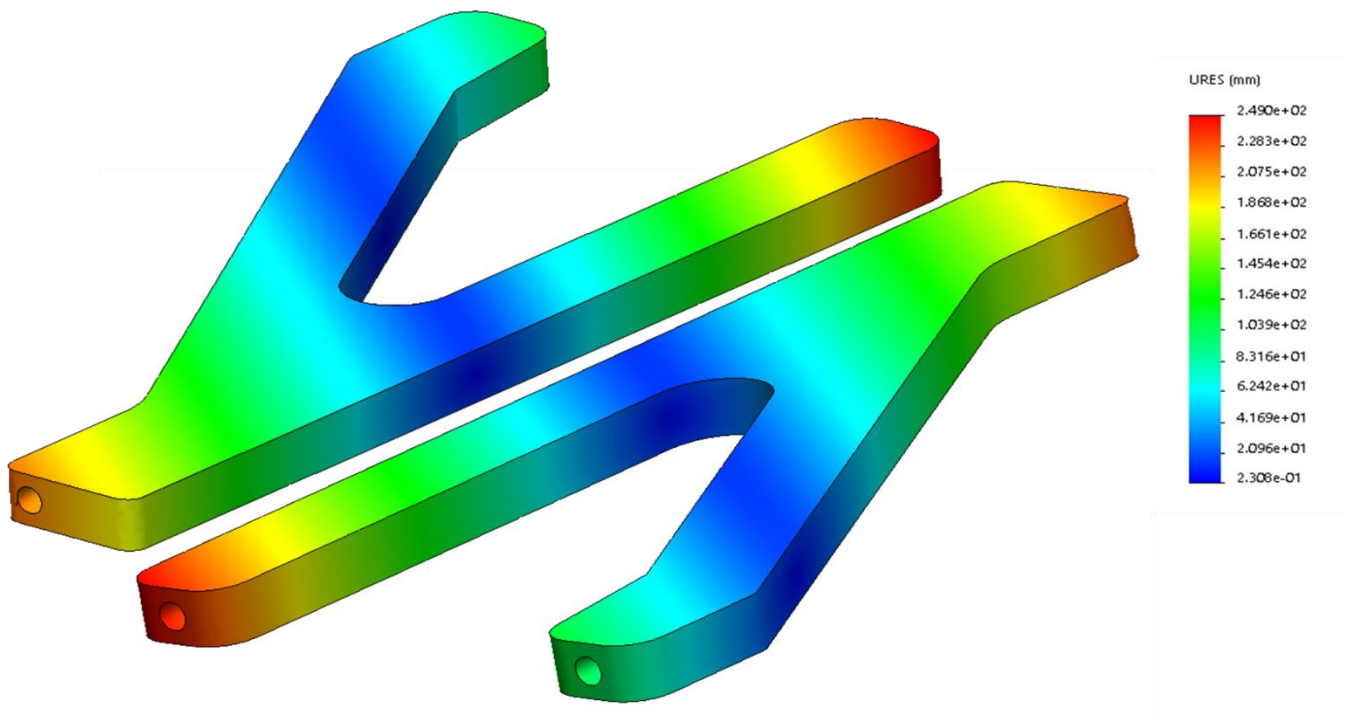


Figure 12 - Iteration 2 Displacement

Table 3 - Iteration 2 FEA Data

Maximum Von Mises Stress (N/m²)	1.464×10^9
Minimum Von Mises Stress (N/m²)	6.479×10^5
Maximum Deflection (mm)	2.490×10^2
Minimum Deflection (mm)	0.2308
Component Mass (kg)	2.221
Factor of Safety	N/A

The second iteration had the following design features, as seen on figure 10:

- Inside radius (1): 15mm
- Outside radius (2): 25mm
- Edge fillets: None
- End fillets: 2.5mm

Figure 11 shows the increased fillet in between the wishbone arms has reduced the magnitude of stress concentration by 45% as this change allows the stresses to distribute more equally, exerting a more even load on the component. However, rather than one central concentration, there are now two equal concentrations at either side of the fillet. This is confirmed by figure 12, showing exactly the same deformation under load. It is clear further strengthening is required (Zhuming, 2018).

Adding a larger fillet also increased the component's mass by 67 grams. The author wishes to reduce overall mass of the component, so this shall be addressed in later iterations.

6.3 Third Design Iteration

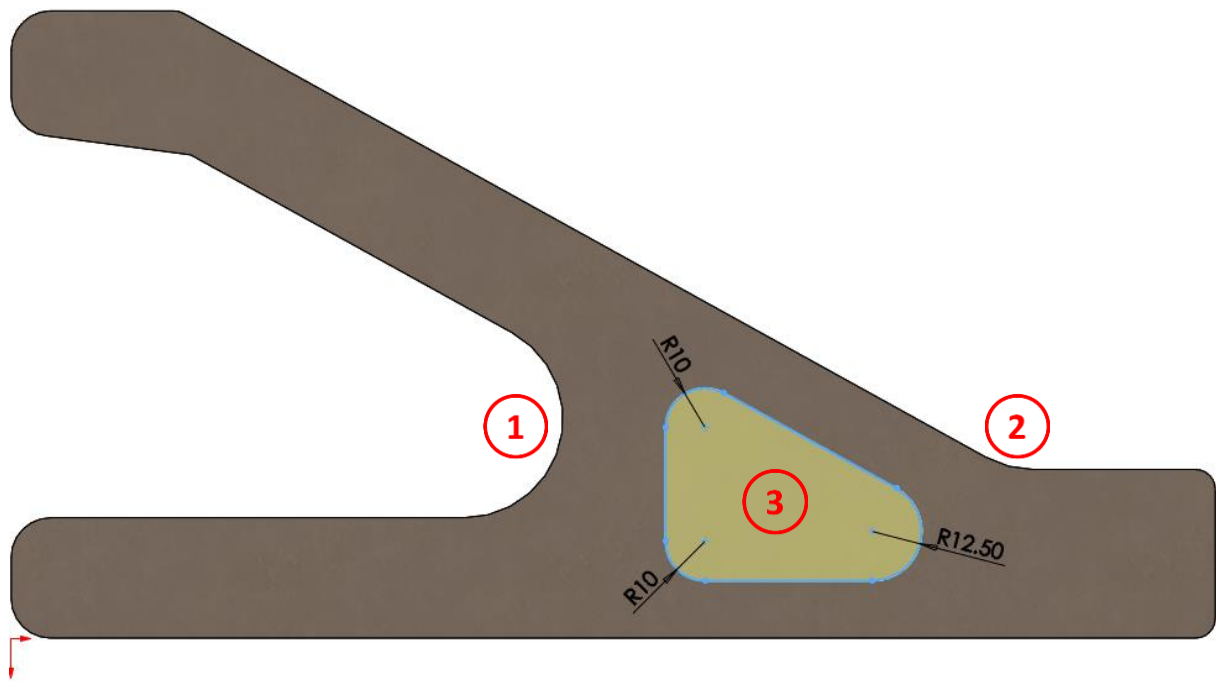


Figure 13 - Iteration 3 CAD

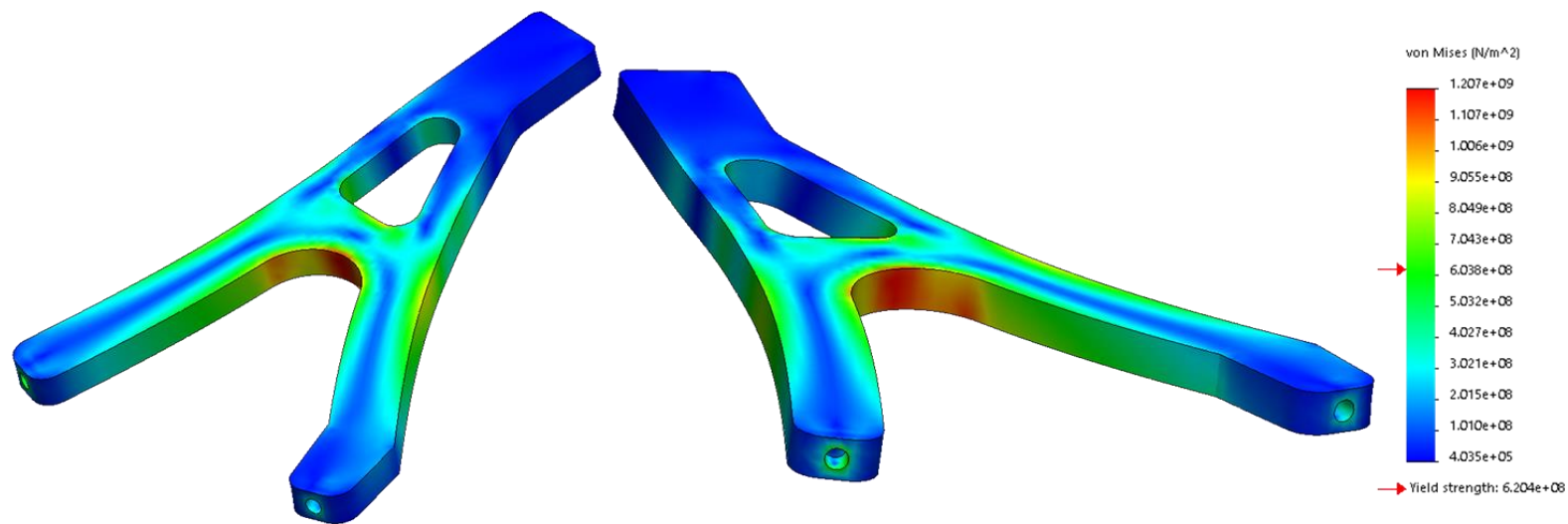


Figure 14 - Iteration 3 Von Mises Stress

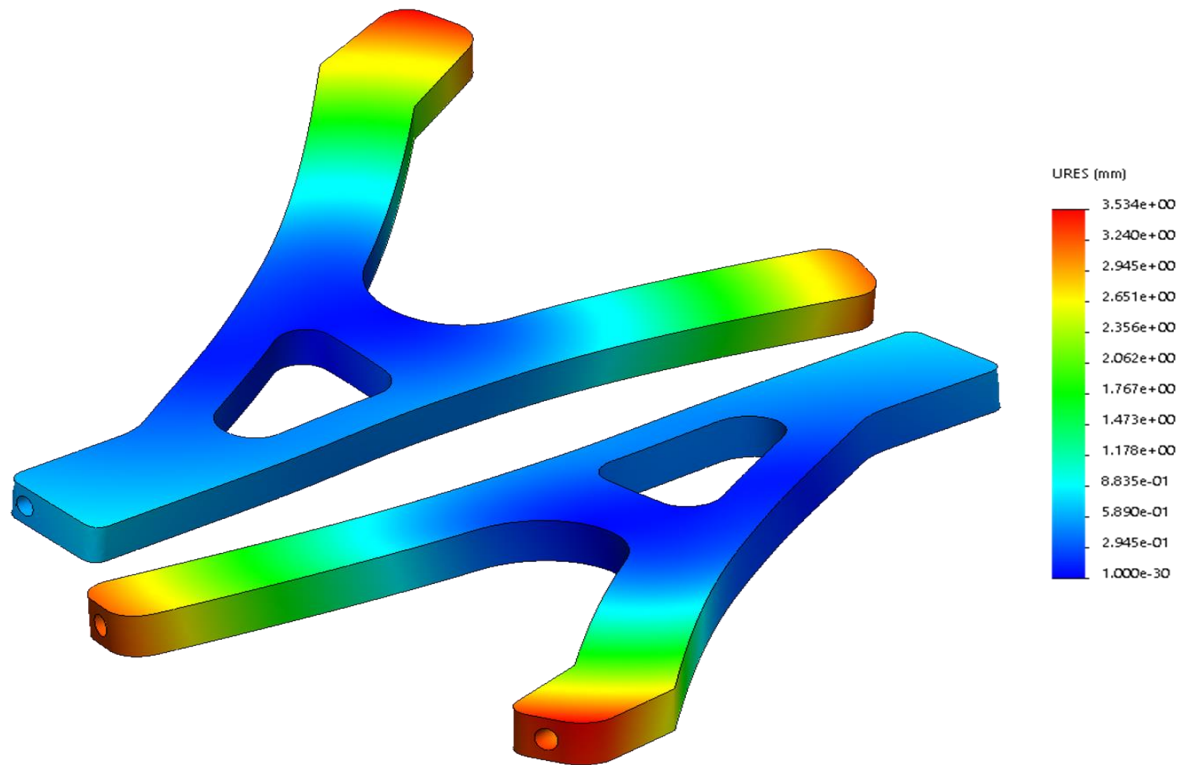


Figure 15 - Iteration 3 Displacement

Table 4 - Iteration 3 FEA Data

Maximum Von Mises Stress (N/m²)	1.207×10^9
Minimum Von Mises Stress (N/m²)	4.035×10^5
Maximum Deflection (mm)	3.534
Minimum Deflection (mm)	1.000×10^{-30}
Component Mass (kg)	2.028
Factor of Safety	N/A

The third iteration had the following design features, as seen on figure 13:

- Inside radius (1): 25mm
- Outside radius (2): 25mm
- Edge fillets: None
- End fillets : 2.5mm
- Removed section of material (4)

Figure 14 shows that a 25mm radius is too high to maintain structural rigidity. Although reducing concentrated stress, the component is weakened by the geometry of the increased fillet, thus compromising strength (J. Trevelyan, 1986). Although maximum Von Mises stress seen has decreased, as has deformation, these values are still too high.

The mass is reduced by 8% as a result of the removed material, but stress is increased around the edges, although not to the point of yield. This feature will be carried forward and improved.

6.4 Fourth Design Iteration

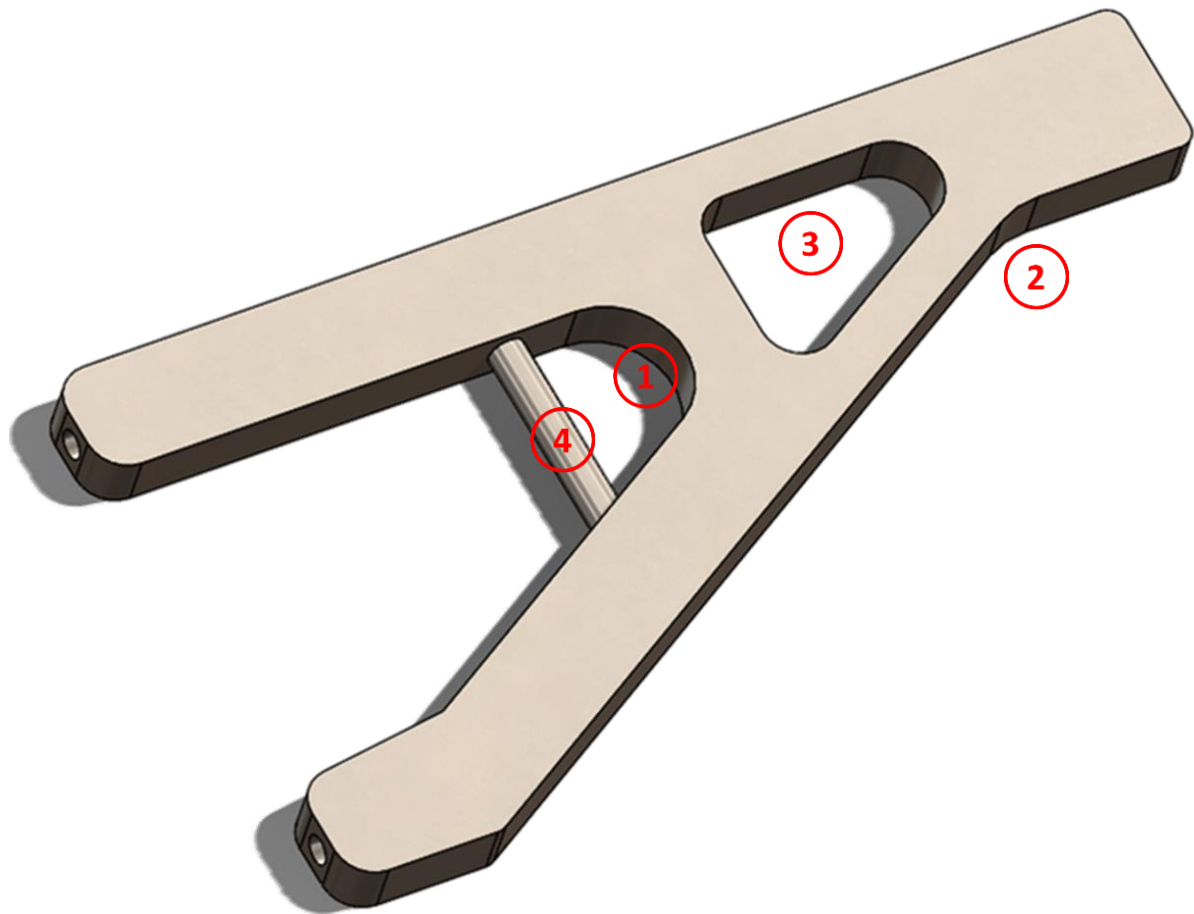


Figure 16 - Iteration 4 CAD

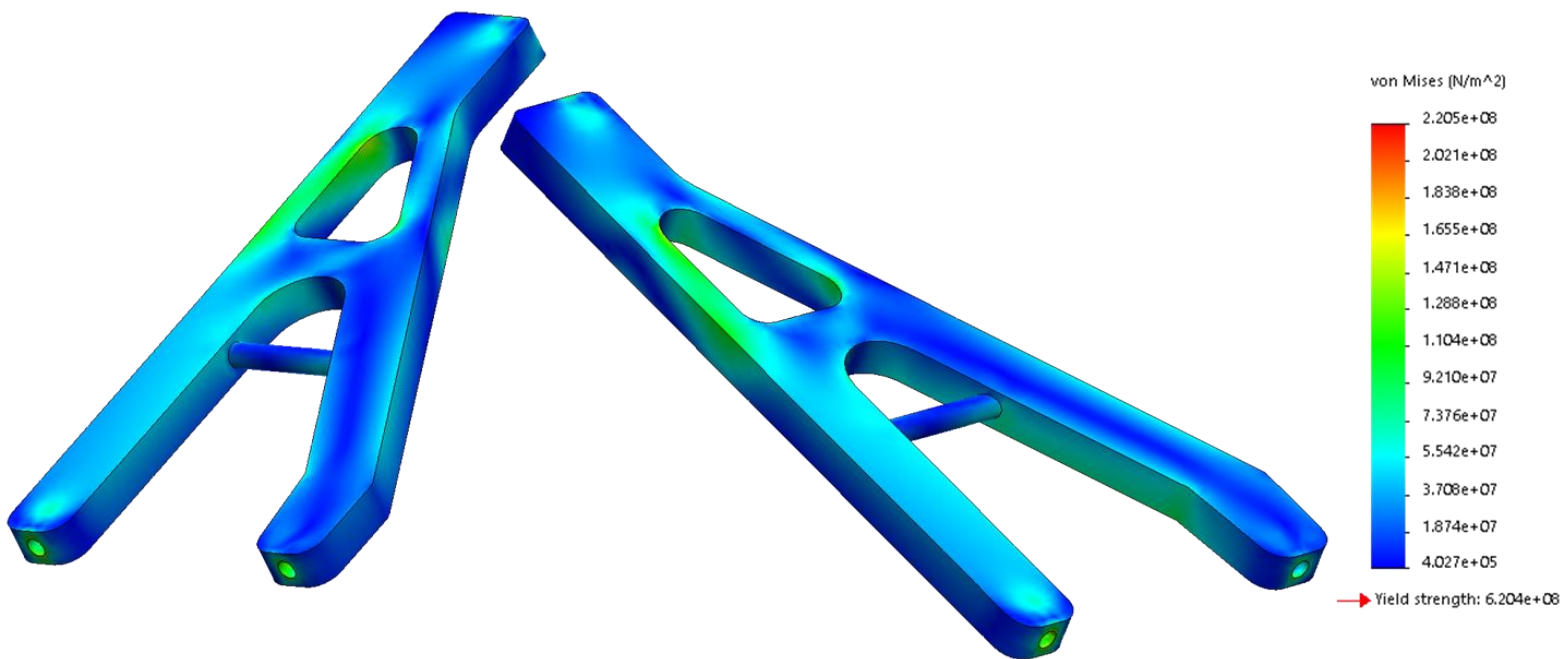


Figure 17 - Iteration 4 Von Mises Stress

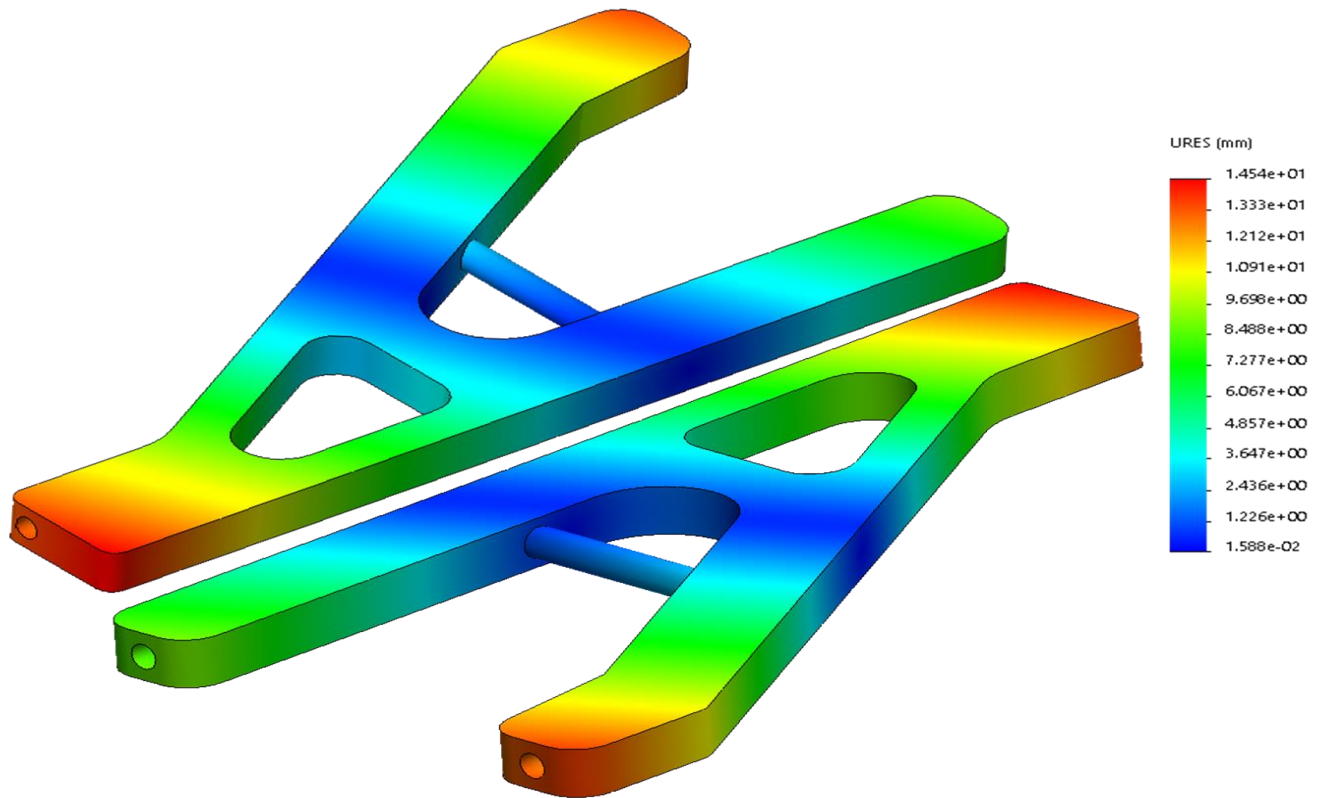


Figure 18 - Iteration 4 Displacement

Table 5 - Iteration 4 FEA Data

Maximum Von Mises Stress (N/m²)	2.205×10^8
Minimum Von Mises Stress (N/m²)	4.027×10^5
Maximum Deflection (mm)	14.540
Minimum Deflection (mm)	1.588×10^{-2}
Component Mass (kg)	2.065
Factor of Safety	2.813

The fourth iteration had the following design features, as seen on figure 16:

- Inside radius (1): 20mm
- Outside radius (2): 25mm
- Edge fillets: None
- End fillets: 2.5mm
- Removed section of material (3)
- 10mm diameter support strut (4)

This iteration has reduced the internal diameter for added strength and includes a 10mm support strut between the arms, designed to stop them spreading apart. This component does not fail and has a safety factor of 2.813, above the lower acceptable limit for a motorsport application.

Now that the component does not fail, the author shall endeavour to reduce the mass, whilst maintaining structural rigidity.

6.5 Fifth Design Iteration

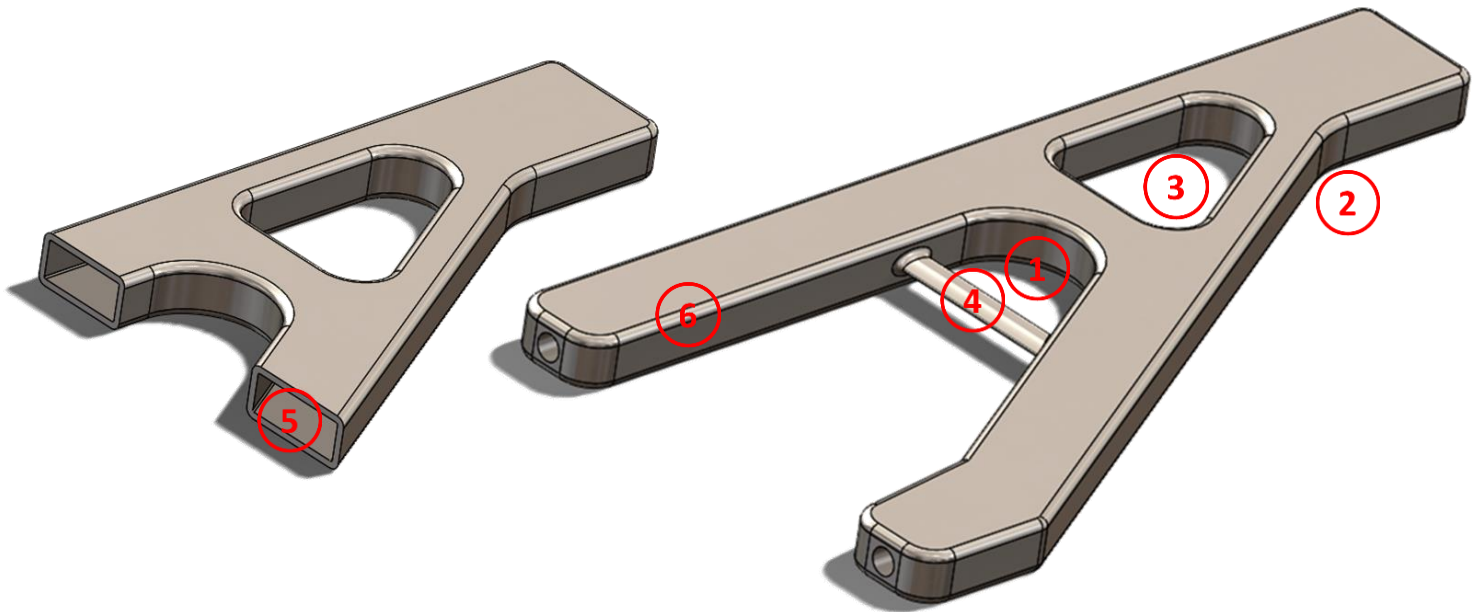


Figure 19 - Iteration 5 CAD

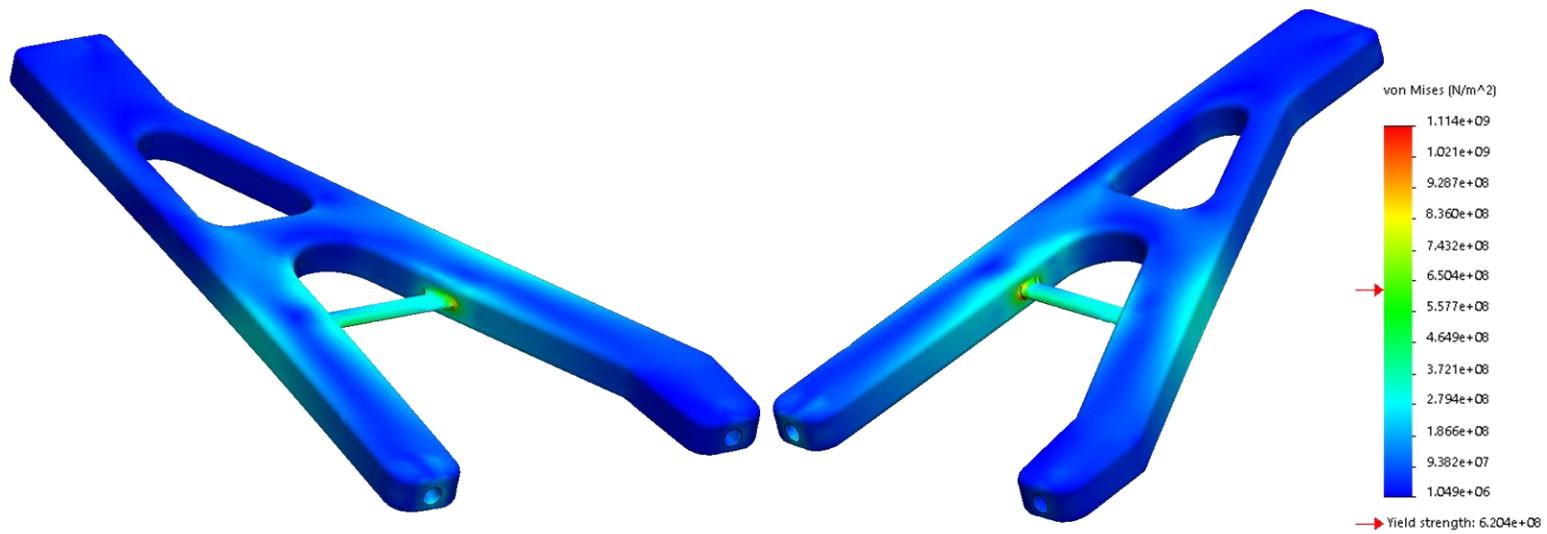


Figure 20 - Iteration 5 Von Mises Stress

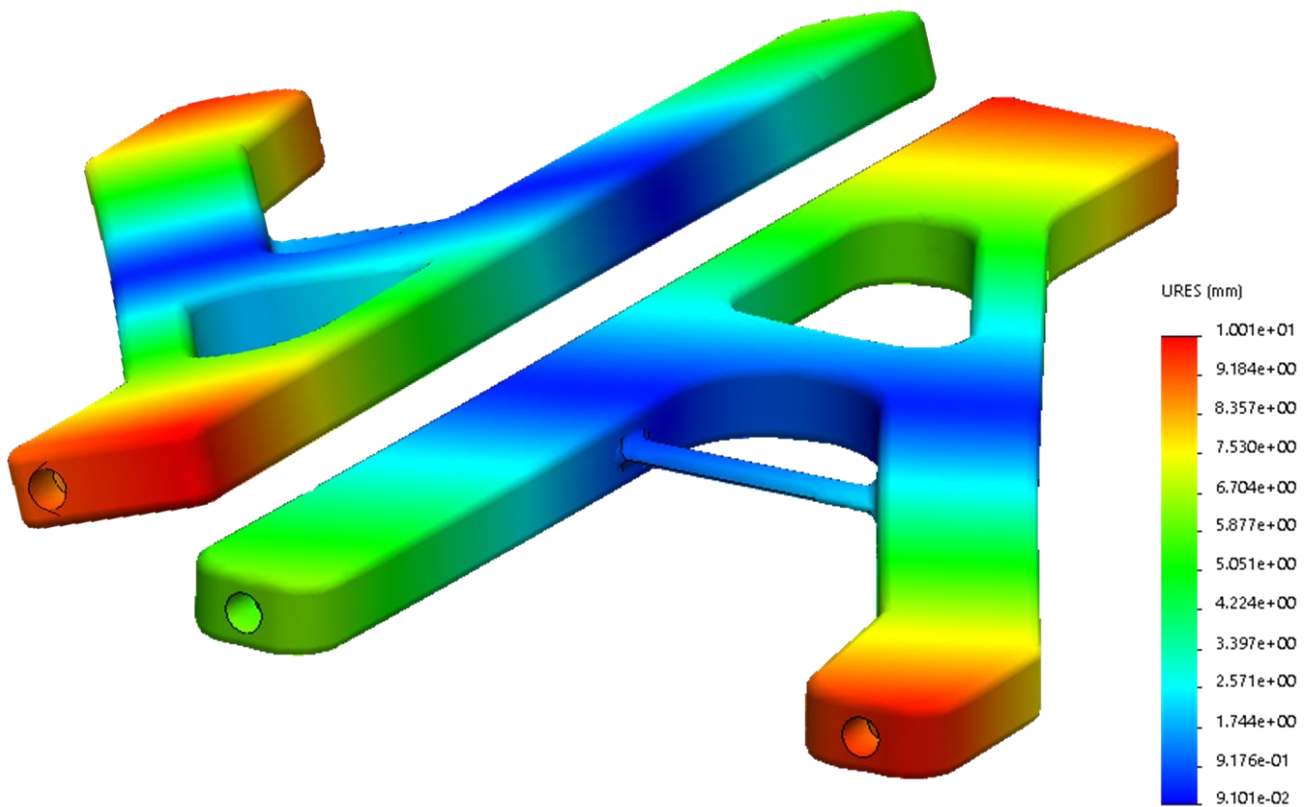


Figure 21 - Iteration 5 Displacement

Table 6 - Iteration 5 FEA Data

Maximum Von Mises Stress (N/m²)	1.114×10^9
Minimum Von Mises Stress (N/m²)	1.049×10^6
Maximum Deflection (mm)	10.01
Minimum Deflection (mm)	9.101×10^{-2}
Component Mass (kg)	0.742
Factor of Safety	N/A

The fifth iteration had the following design features, as seen on figure 19:

- Inside radius (1): 20mm
- Outside radius (2): 25mm
- Removed section of material (3)
- Varying radius support strut (4)
- 2mm shell (5)
- Edge fillets (6): 2.5mm

This iteration was produced to explore the minimum safe values for the shell thickness, and support strut radius.

The support strut was drawn as an ellipse, a shape with one curve that resembles the size of various circle diameters. The author analysed the stress concentration point and saw it was until around the section resembling a 6.5mm radius, this information can be used in the next iteration.

The shell was set to 2mm, this was decided to be the minimum thickness worth testing, however figure 20 shows this does not effect structural rigidity and thus can be carried on to further iterations (Ohara, 1984). Moreover this feature reduced component mass by 64%, proving it is a useful feature to have in the design.

Material yielding here has resulted in an increased deformation, this will be reduced in the next iteration.

6.6 Final Design (Sixth Iteration)

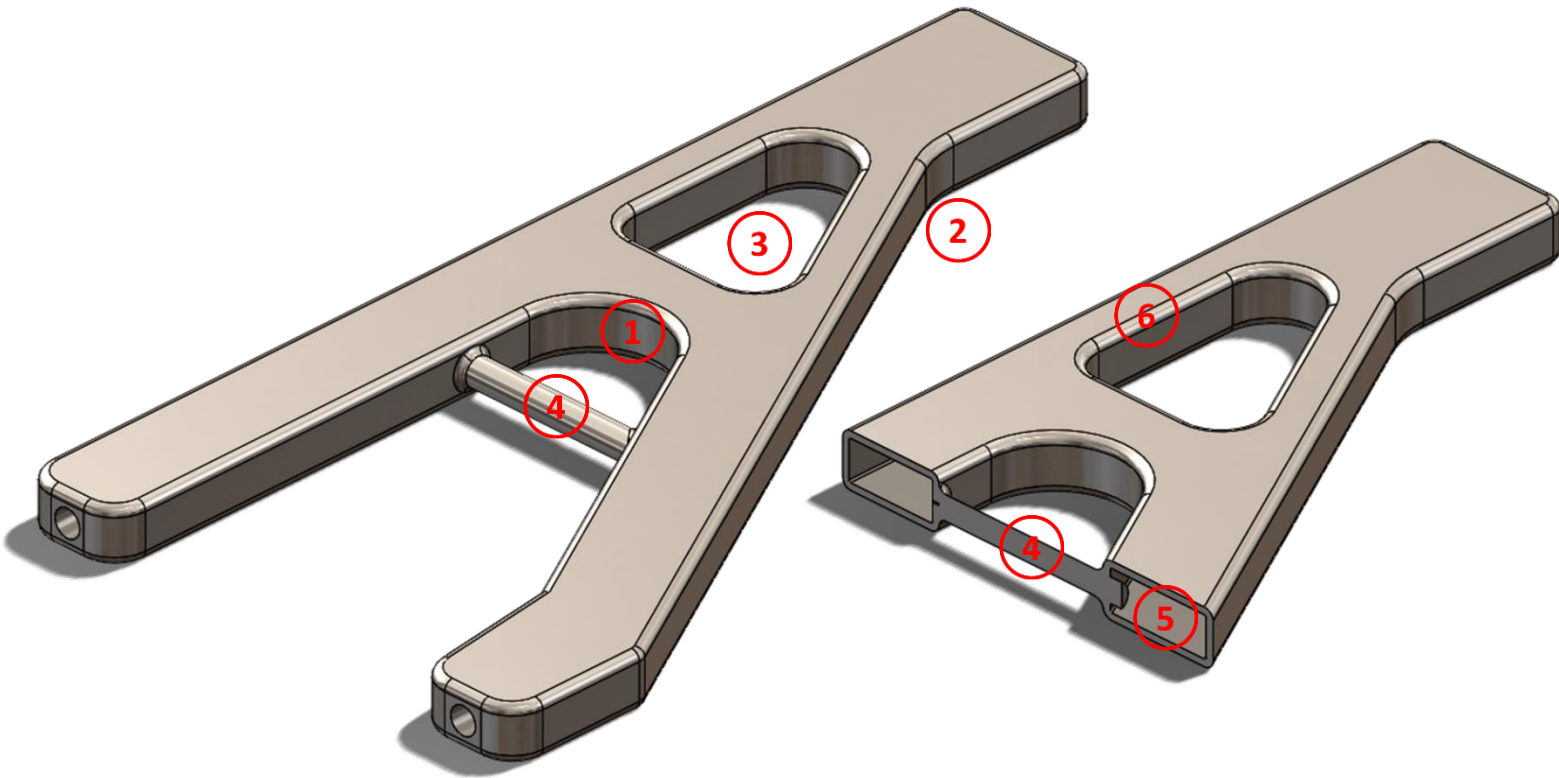


Figure 22 - Final Design CAD

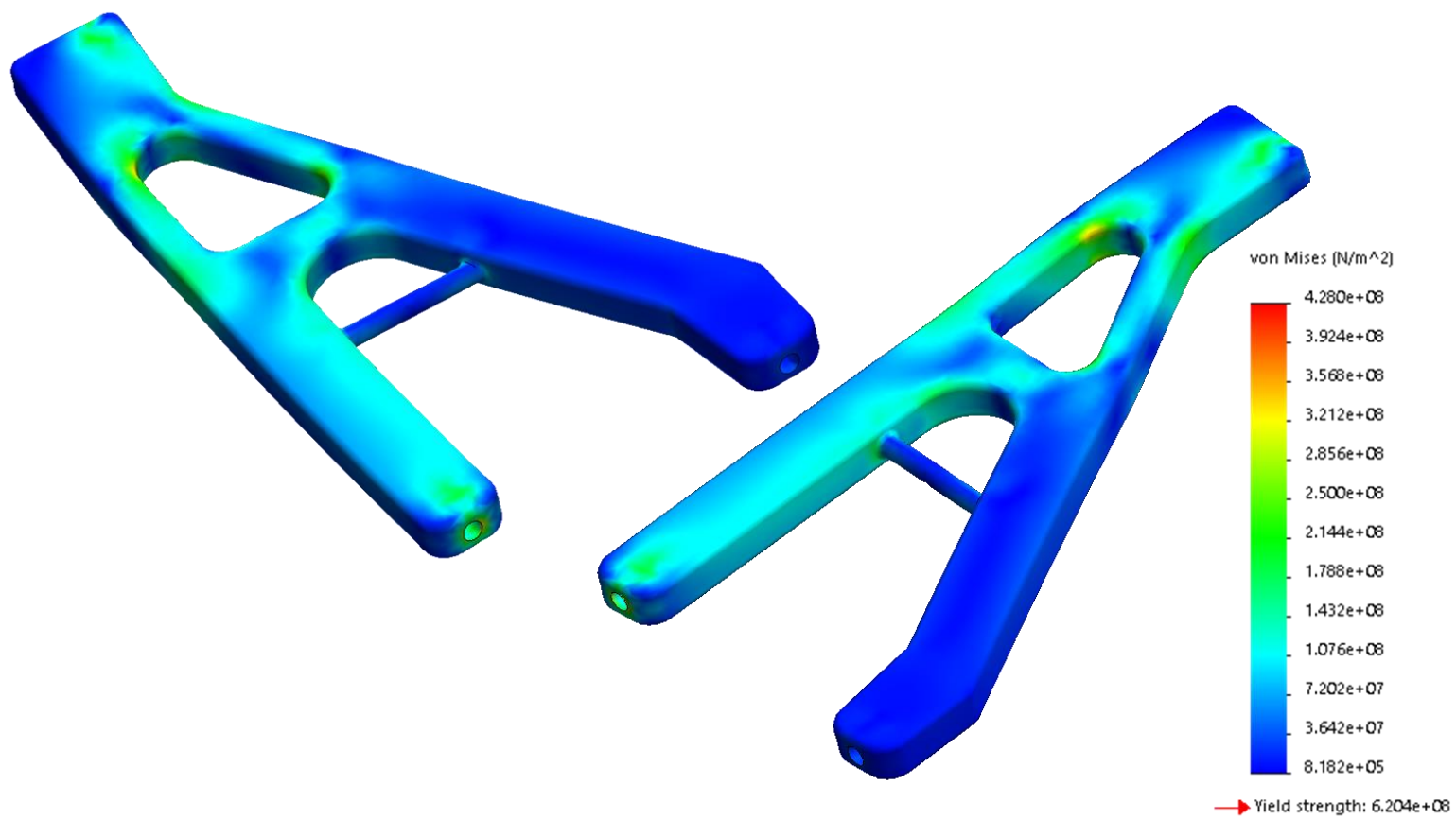


Figure 23 - Final Design Von Mises Stress

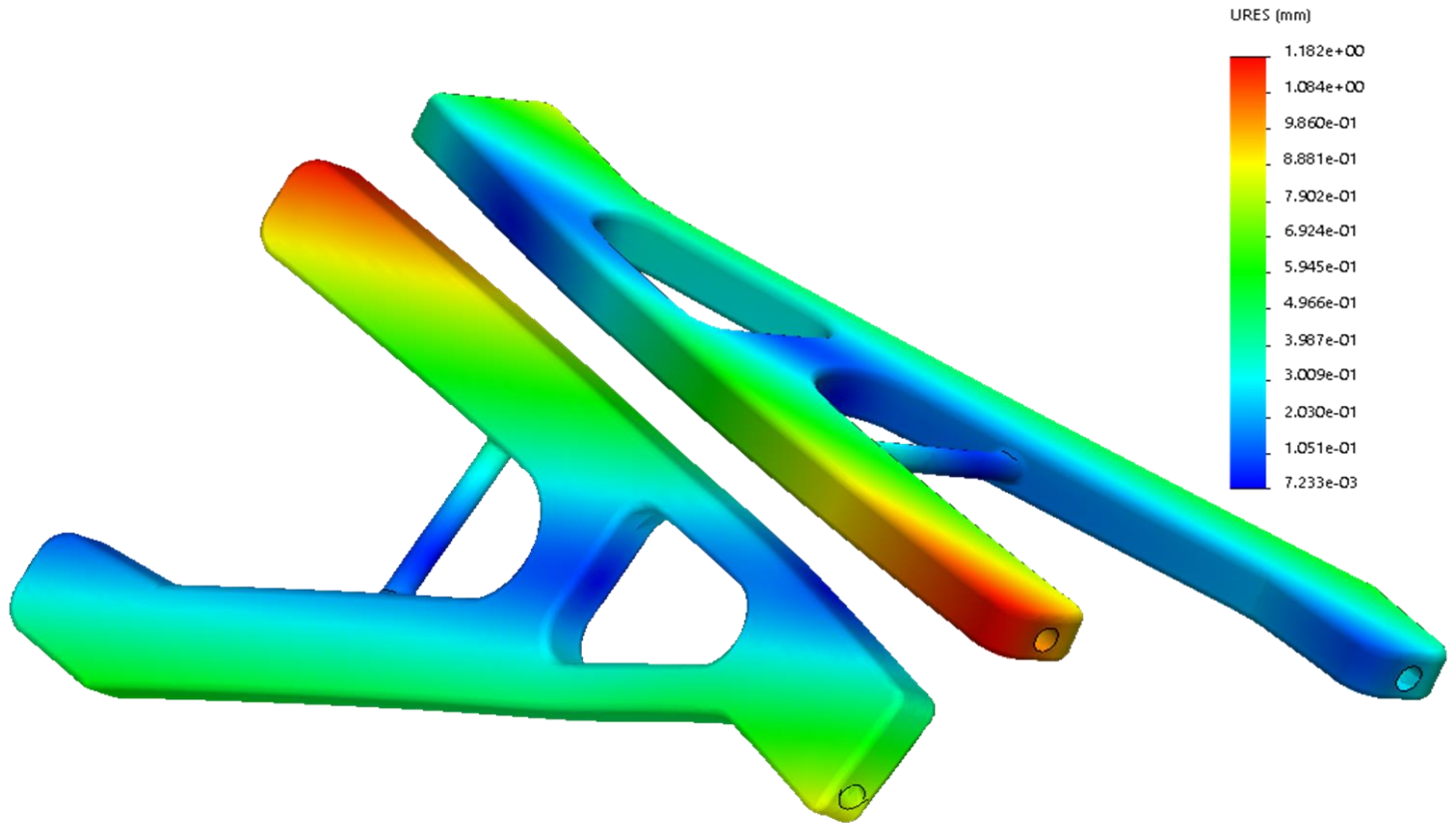


Figure 24 - Final Design Displacement

Table 7 - Final Design FEA Data

Maximum Von Mises Stress (N/m²)	4.280×10^8
Minimum Von Mises Stress (N/m²)	8.182×10^5
Maximum Deflection (mm)	1.182
Minimum Deflection (mm)	7.233×10^{-3}
Component Mass (kg)	0.752
Factor of Safety	1.012

The final design had the following design features, as seen on figure 22:

- Inside radius (1): 20mm
- Outside radius (2): 25mm
- Removed section of material (3)
- 8mm solid support strut (4)
- 2mm shell (5)
- Edge fillets (6): 2.5mm

The final design has no stress concentrations above the yield point of the material. There are very small stress concentrations around the bolt holes holding the rose joints, but this is where all the force of the system is first placed, so this is expected. Even in these extreme areas there is a safety factor of over 1, but the safety factor across most of the component is over 2.5. It must be remembered this wishbone has been simulated within an extreme scenario, in normal use this wishbone will be even stronger, with a normal life cycle.

Even with 10g added mass due to the solid support bar, the mass of this iteration is still 65% lighter than the original component, with maximum stresses seen reduced by 83.94% and maximum deflection reduced by over 99%. Even though deflection is seen, it is very small. Clearly, this component has been improved by an iterative design optimisation process.

7 Analysis and Evaluation

7.1 Results

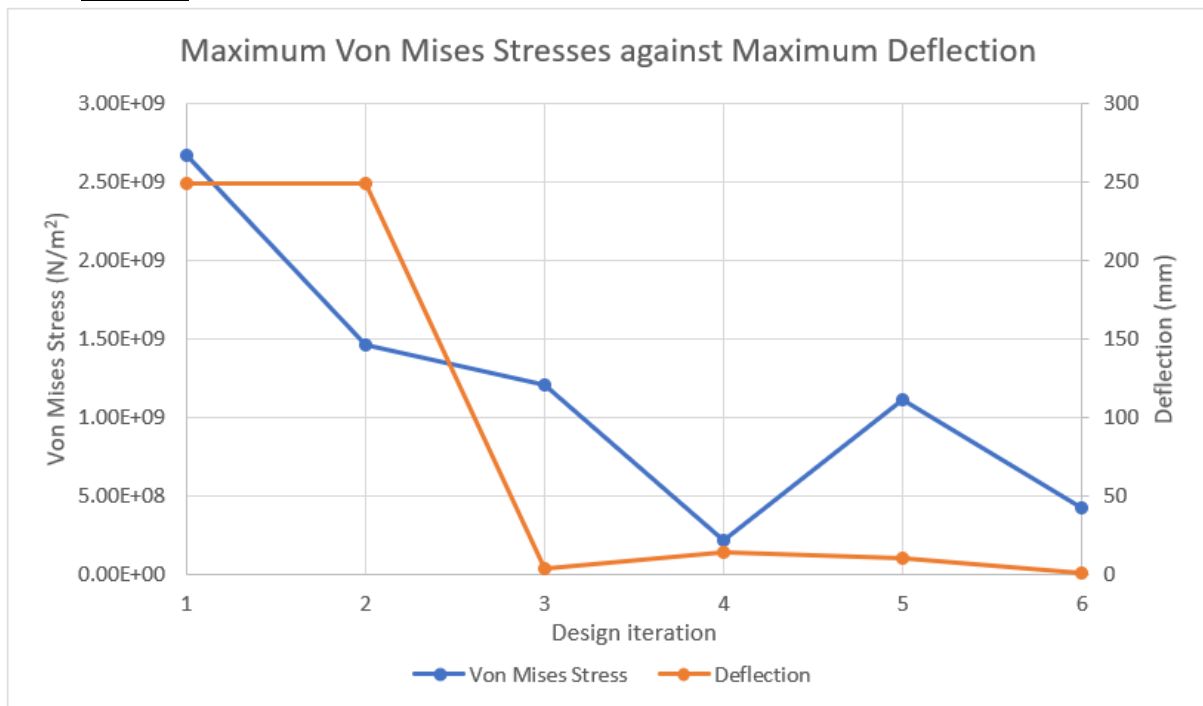


Figure 25 - Von Mises Stress Against Deflection

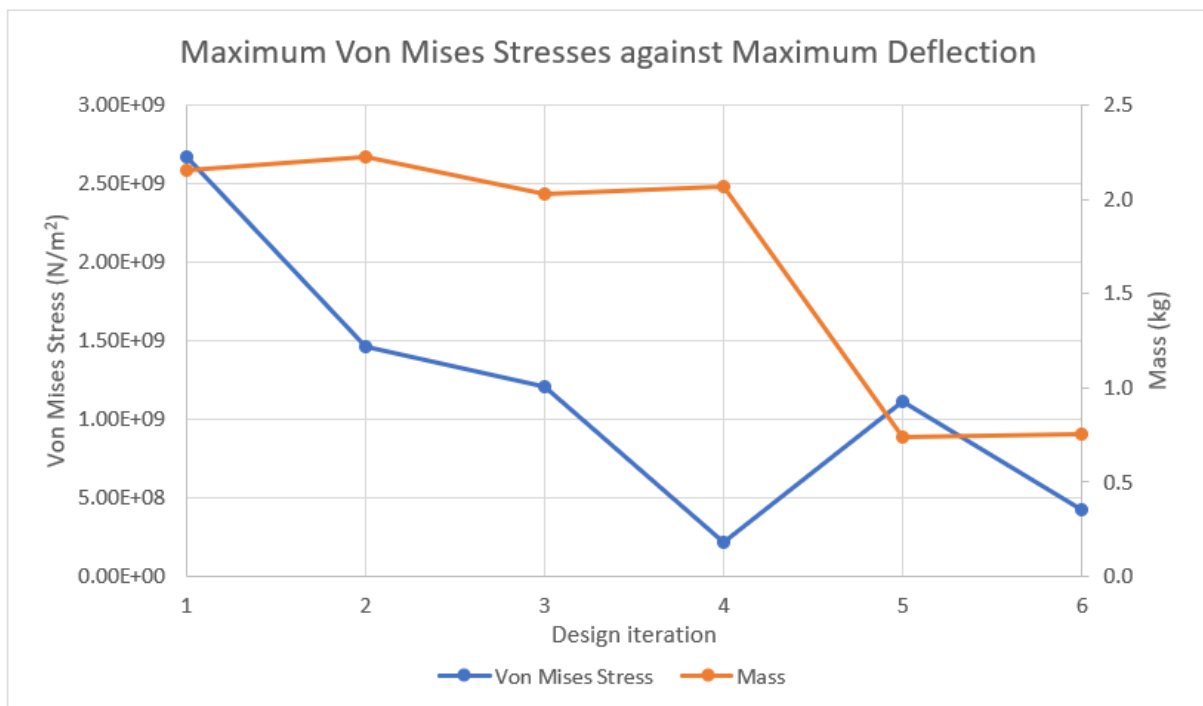


Figure 26 - Von Mises Stress Against Mass

Figures 25 and 26 summarise the results shown in section six. It is clear that iteration 4 is the strongest, with by far the smallest Von Mises stress experienced; however, this design is also far too heavy, moreover it deforms more than other iterations.

The author stopped the iterative design process at iteration six because, as shown above, the design showed the second smallest stress experienced, was only 10kg heavier than the lightest mass, and had the smallest deflection. This is a good compromise of all measured factors.

7.2 Simulation Improvements

There are a few things that the author would change if this analysis were to be carried out again that could improve accuracy of results:

- Although it was decided the amount of redundancies within the motion simulation did not affect results, more mechanical mates could be added to reduce the number further, thus improving the running of the motion simulation and improving FEA results even more.
- The simulation was run at 25 frames per second as this was adequate for extracting the data required. For a more detailed analysis, the framerate could be increased to allow more scenarios to be analysed when importing motion loads.
- The author used the default mesh set by SolidWorks. This could have been investigated further, introducing a finer mesh into geometrically complex areas, or areas prone to stress concentrations to understand the loadings further.
- The author did not add loadings onto the pushrod to simulate the mass of the vehicle. The data acquired was within an acceptable tolerance to what was expected, so the tests were not re-run. However for extra accuracy this should have been considered.
- A topology study could have been carried out to further understand how the design of the component could change whilst maintaining strength and rigidity.
- Different materials could have been assessed to understand if the material chosen was the best for the application. This was not explored as the author wished to understand how best to change the original wishbone through re-designing alone.
- Frictional losses within touching components such as joints could be included in the simulation for further accuracy. Due to the nature of the assembly, this was decided to be negligible.

7.3 Conclusion

This report shows that SolidWorks motion simulation and Finite Element Analysis packages can produce valid results with relative accuracy when analysing the loading of automotive components. Forces calculated were within expected regions, matching well to MoTeC data, and the amount of stress experienced compared to mass and deformation correlated well.

FEA analysis has proven to be a fantastic engineering tool, significantly saving cost and time when analysing the design of an upper wishbone.

The author, through this analysis, has produced a significantly improved wishbone design with reduced stress, mass and deformation.

8 Referenced Work

Ajay Harish (2019) *What Is the Meaning of the Von Mises Stress and the Yield Criterion?*, March 14. Available at: <https://www.simscale.com/blog/2017/04/von-mises-stress/>.

Bodein, Y., Rose, B. and Caillaud, E. (2013) 'A Roadmap for Parametric CAD Efficiency in the Automotvie Industry', *Computer-Aided Design*, 45(10), pp. 1198–1214.

Bodein, Y., Rose, B. and Caillaud, E. (2014) *Explicit reference modeling methodology in parametric CAD system*, *Computers in Industry*. doi: 10.1016/j.compind.2013.08.004.

Everett, A. (1994) *Materials*. Harlow: Longman Group Ltd.

Hart, M. *et al.* (2018) 'UWTSD Aerodynamic Group Project CAD File'. Swansea.

Hart, M. (2019) 'Dissertation Research - Max Hart'.

J. Trevelyan (1986) *Finite element analysis and applications*, *Engineering Analysis*. doi: 10.1016/0264-682x(86)90048-1.

Korunovic, N. *et al.* (2014) *FEM based parametric design study of tire profile using dedicated CAD model and translation code*. Niš. Available at: https://www.researchgate.net/publication/269222790_FEM_based_parametric_design_study_of_tire_profile_using_dedicated_CAD_model_and_translation_code.

Levesley, M. C. *et al.* (2003) *Dynamic Simulation of Vehicle Suspension Systems for Durability Analysis*, *Materials Science Forum*. Leeds. doi: 10.4028/www.scientific.net/msf.440-441.103.

Lombard, M. (2013) *SolidWorks Bible*. 1st edn. Indianapolis: John Wiley & Sons.

McCartney, L. N. (1979) *Can Safety Factors be Reduced Safely When Designing Against Fatigue?*, *Fatigue & Fracture of Engineering Materials and Structures*. doi: 10.1111/j.1460-2695.1979.tb01096.x.

Morris, A. J. (2008) *A Practical Guide to Finite Element Modelling*. Chichester: John Wiley & Sons.

O.Z Racing (2018) *O.Z Racing Alloy Wheels*. Available at: <https://www.ozracing.com/uk/alloy-wheels>.

Ohara, M. (1984) 'Cad/Cam in the Automotive Industry.', *Bulletin of the Japan Society of Precision Engineering*, 18(2). doi: 10.1299/jsmemag.86.776_759.

Rieg, F., Hackenschmidt, R. and Alber-Laukant, B. (2014) *Finite Element Analysis for Engineers*. Cincinnati: Hanser Publications

Sengupta, A. K. (2009) *Stress Concentration Caused by Sudden Change in Form*, Jurnal Pesona PAUD.

Tovey, M. (2002) 'Concept design CAD for the automotive industry', *Journal of Engineering Design*, 13(1), pp. 5–18. doi: 10.1080/09544820110090287.

Zhuming, B. (2018) *Finite Element Analysis Applications: A Systematic and Practical Approach*. London: Elsevier.