

University of Wales Trinity Saint David
School of Engineering

Advanced Computational Methods
M6X00408

Samuel Treleaven
1702453

24/01/20

Motion Simulation and FEA

Contents

1	Introduction	3
2	Modelling	3
3	Motion Simulation	5
4	Finite Element Analysis	7
4.1	Results.....	8
4.1.1	Original Design	8
4.1.2	Improved Designs	10
5	Limitations and Improvements.....	15
6	Conclusion.....	16
	Bibliography.....	17

1 Introduction

This report shows the creation of a vehicle suspension and a finite element analysis using a motion study on the pushrod rocker in computer-aided design software, CAD, which is called Solidworks. The force applied to the rocker is the loads exerted on it when the car goes over a simulated bump. The rocker will then be analysed and improvements made to optimise mass reduction as well as improving the stress value, especially if it falls below the safety factor. The aims of the project are to improve the performance of the vehicle's pushrod rocker by reducing the mass and keeping the stress at a safe level.

2 Modelling

The first part of the investigation was to create an accurate representation of the suspension of the vehicle through a CAD model. To do this the vehicle's suspension components were removed from the vehicle, measured and weighed, this allows mass to be input into the model. The image below displays components, such as a wishbone, being disassembled.

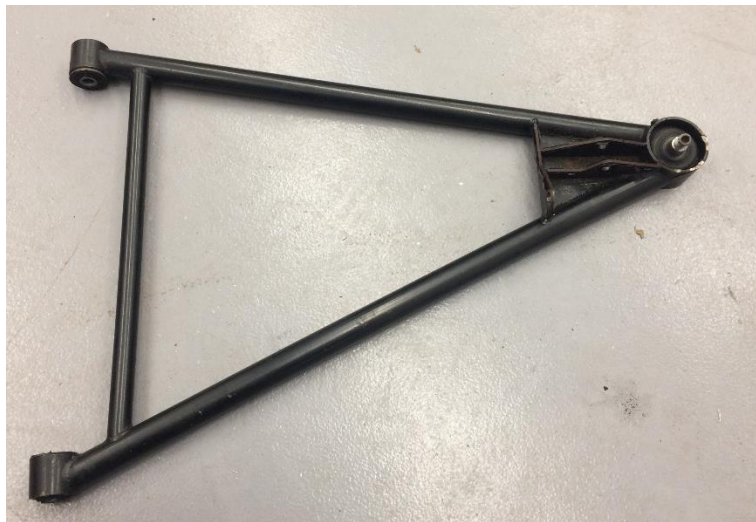


Figure 2.1 – lower wishbone

The material for each component needed to be selected with some components being approximated, such as silicon rubber for the tyre. The rocker which is being investigated can be seen in figure 2.2, this was assigned the material alloy steel.

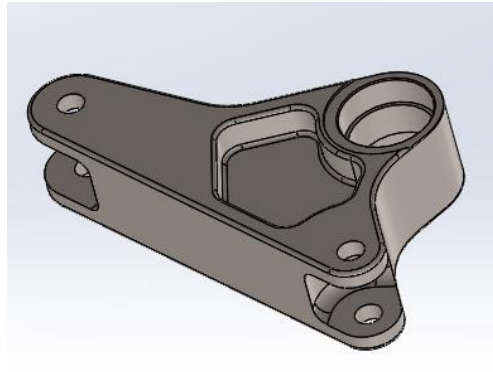


Figure 2.2 – suspension rocker

The individual components were then created and assembled in an assembly format in Solidworks.

Parts which needed to move had to be made as separate parts, such as the rose joint in the pushrod. When it was first made the rose joint was made incorporated into the entire pushrod but during the assembly, it was noticed that it didn't function. This meant they had to be remade as separate parts. Once this error had been completed the entire assembly could be made. See the following images for the assembly at various stages. The final stages of modelling were to input a road into the model to allow the tyre to rest on when gravity is input so it doesn't continually fall down.

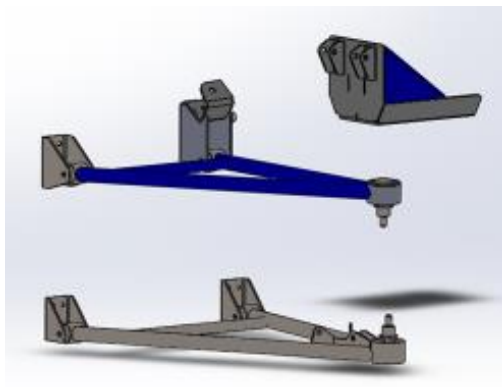


Figure 2.3 – the beginning of assembly

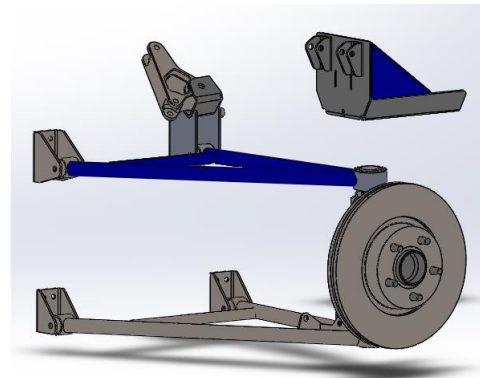


Figure 2.4 – assembly including upright and hub

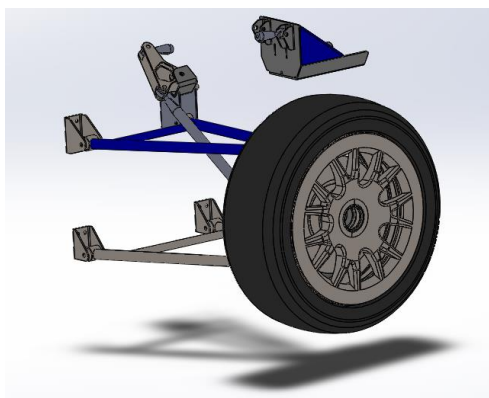


Figure 2.5 – final assembly

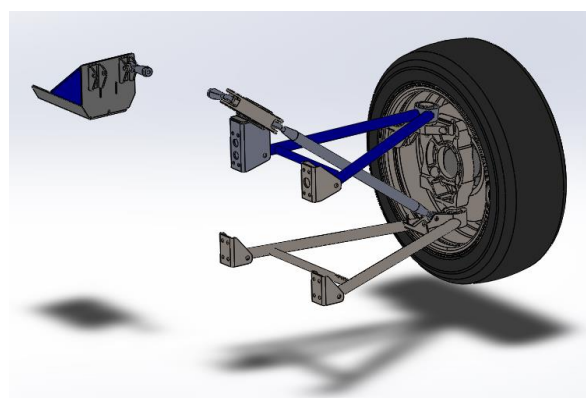


Figure 2.6 – final assembly rear view

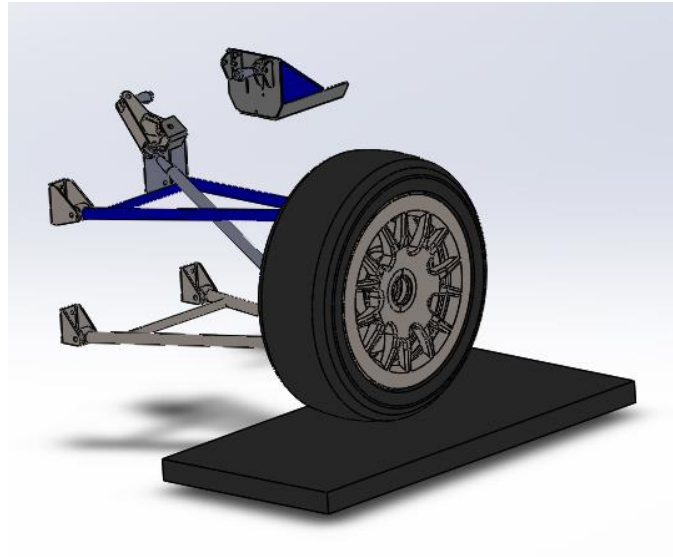


Figure 2.7 – assembly with road

3 Motion Simulation

A motion analysis was carried out to collect the loading data of the rocker when the vehicle goes over a bump. The reason a motion analysis is used to calculate the load data is that it is more convenient and quicker to allow a computer programme, Solidworks, carry out the mathematical equations rather than a person by hand. It also emits the risk of human error in miscalculating. To begin the motion analysis, the mates need to be analysed to ensure the model is functioning as expected. This could be to check that nothing is over-constrained, preventing the parts to move and function as they should, as well under constrained, where parts move about when they should be still. Once this had been analysed, bushings were added to certain mates. This was done to allow deformation in certain directions which would occur realistically. This helped to bring the number of mate redundancies down. Particular attention was paid to the geometry which effected the rocker as this would make the finite element analysis, FEA, more accurate. The number of redundancies didn't reduce to zero like desired. This was due to certain parts, such as the tyre, not being able to be mated in a way that removed the redundant mate. However, these mates didn't directly affect the rocker.

The next part of the motion study was to add the parameters which will cause motion on the model. This first parameter that was added was gravity which was applied perpendicularly into the road. The next parameter to be applied was the motor that would simulate the bump. This was applied as a distance motor of 100mm that acted up on the bottom of the upright for half a second, see *figure 3.1*. 100mm was chosen as it is the maximum height of a speedbump on UK roads (The AA, 2016). The key points were edited for the motor, see *figure 3.2*, so that it was only on whilst it displaced and turned off when the maximum distance was reached. If this wasn't considered, the motor would have held the suspension compressed and not allowed it to function correctly.

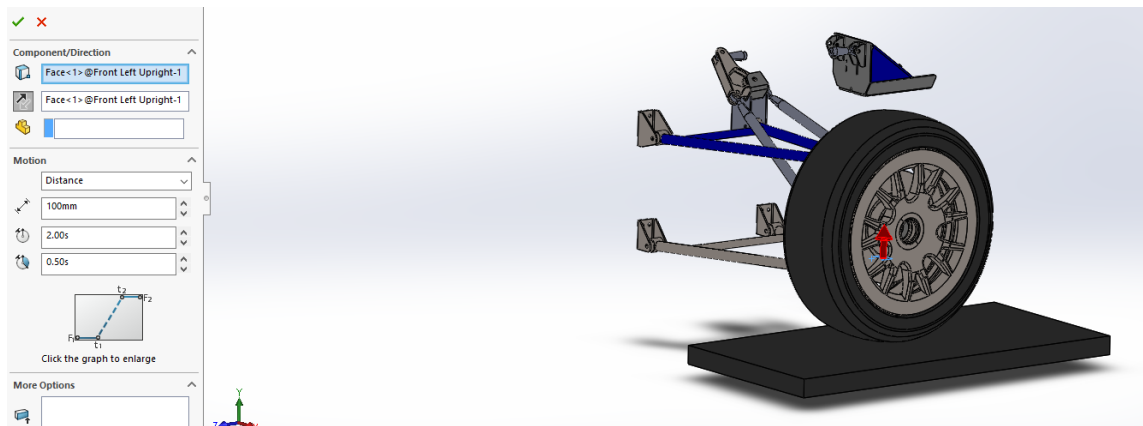


Figure 3.1 – motor input

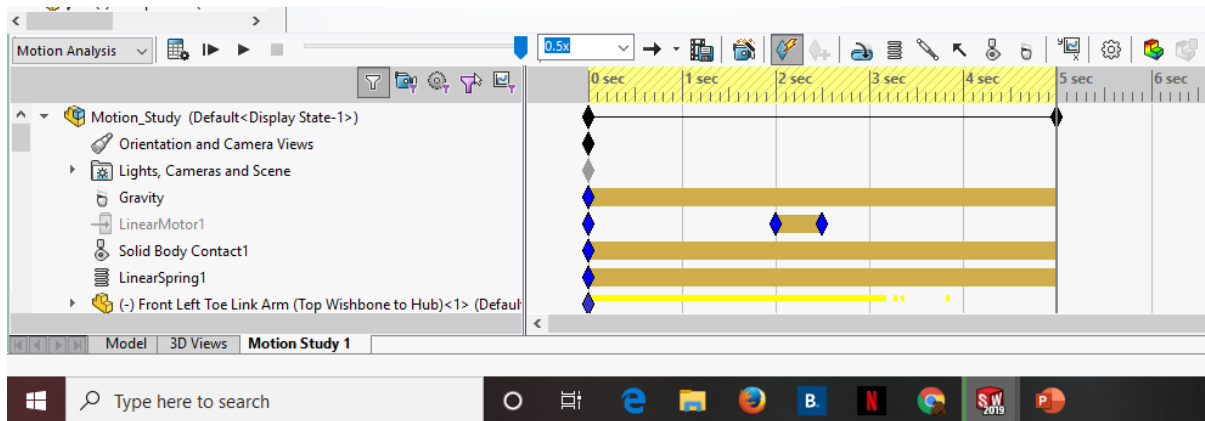


Figure 3.2 – parameters key points

A contact force was applied between the tyre and road, see figure 3.3 so that the tyre could sit on the road under gravity but be lifted off the road when the wheel went over the bump.

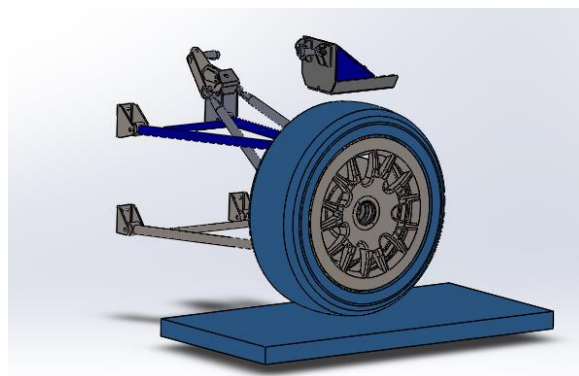


Figure 3.3 – contact force

The final parameter to set was the spring and damping. This was added using the cars spring and slow speed damper rates to give a realistic result.

The motion analysis was then carried out on the model, however, a visual error occurred which meant more investigation into the mates was required. When the motion took place the wheel turned and span around. To prevent this a steering arm was added, see figure 3.4, to remove the free motion in the wheel during the displacement. The analysis was re-run and no further issues arose.

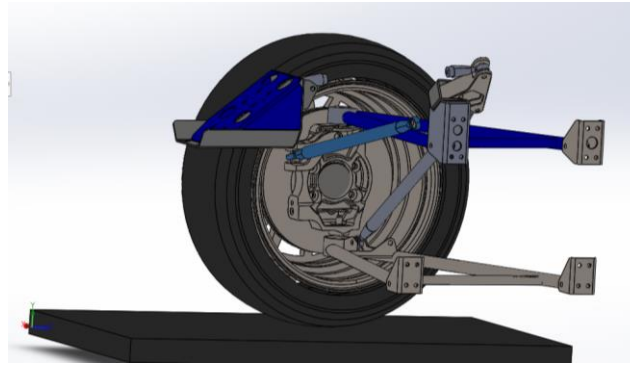


Figure 3.4 – assembly with steering arm

To be able to move on to analysing the specific component the motion loads on the rocker arm were imported into the rocker arm model for evaluation. The part could then be opened with the loads already imported ready for analysis.

4 Finite Element Analysis

Within Solidworks, the model could be analysed using the dynamic loadings which were acted upon the specific component during motion analysis. A static FEA analysis was carried out with the loading values taken from frame 75, which occurred just before peak displacement at 2.5 seconds, see *figure 4.1*.

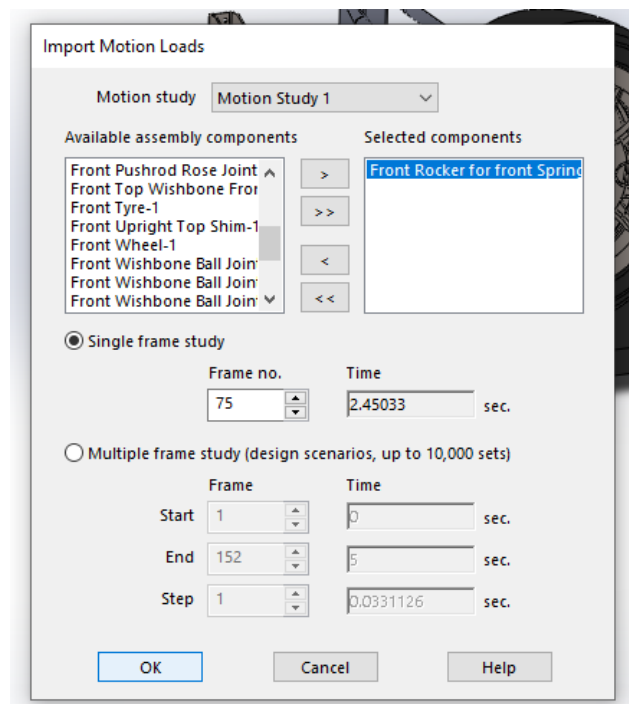


Figure 4.1 – importing motion loads

4.1 Results

Solidworks produce an array of data for a finite element analysis. It includes coloured images to stress, displacement or strain. These can be produced normally or with deformation added which exaggerates the visual image to look bent. More information about the mesh type, material specification and other constants can be found when a report is produced. The report contains the coloured images previously stated as well as some other images, such as the standard part and even the mesh, *see figure 4.2*.

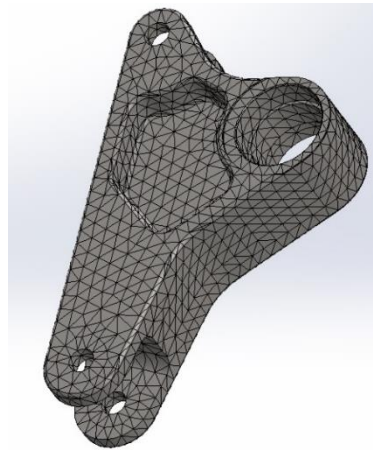


Figure 4.2 – a mesh of original design

4.1.1 Original Design

The stress from the original design displays a very small amount of stress on the component, *see figure 4.3*. This implies that the component is well made but will be easy to remove material to reduce its mass. The yield strength of the model is $6.20422 \times 10^8 \text{ N/m}^2$, whereas the max stress experienced on the rocker was $2.032 \times 10^8 \text{ N/m}^2$. From this information a static safety factor can be produced, using the equation *Safety Factor (SF)* = $\frac{\text{Yield Strength}}{\text{Max Stress}}$ (Cyprean, 2016) For the original design, a safety factor of 3.05 is calculated. This value proves that the design is safe as it falls below the value recommended of 2 (The Engineering ToolBox, 2017).

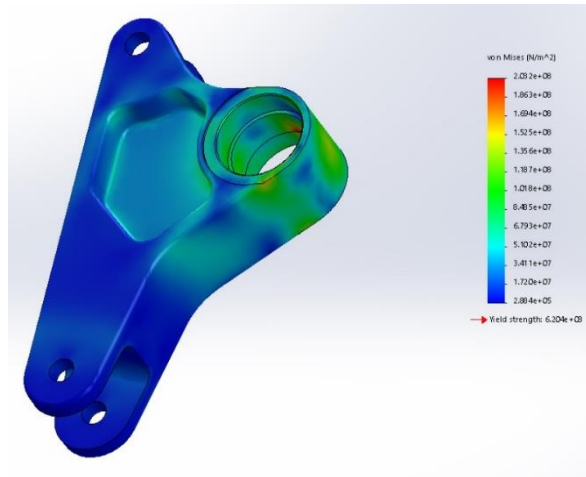


Figure 4.3 – stress (original)

The area of high concentration of stress can be seen in *figure 4.4*, This shows that the max stress is experienced on the curved edge where the rocker mounts to the wishbone bracket. This value may not be very accurate as the bolt and washers that would be mounting this haven't been modelled in the simulation.

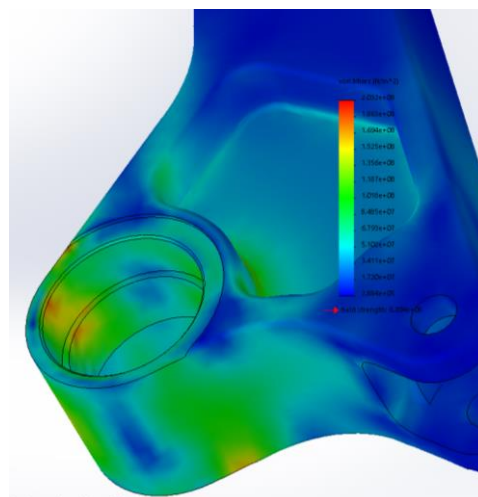


Figure 4.4 – peak stress points (original)

The maximum displacement of the graph is very small of 0.083mm, which is unnoticeable to the human eye but will still affect the performance of the component. The maximum displacement is on the top edge of the mounting are to the wishbone bracket.

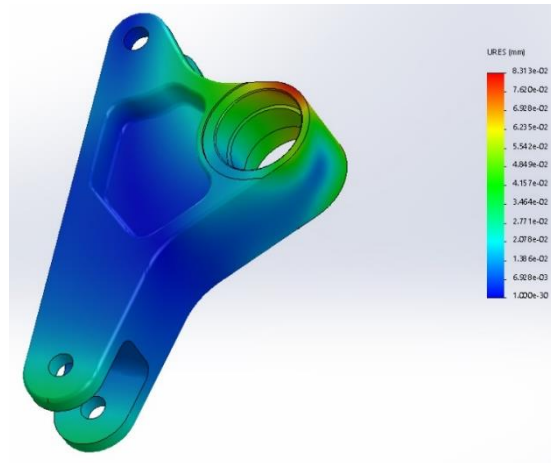


Figure 4.5 – displacement (original)

Overall the original design is well made, however, it has a mass of 1.5263 kg. which could easily be reduced for such a small part. By reducing the mass of the rocker arm this will help in reducing the unsprung mass of the vehicle.

4.1.2 Improved Designs

4.1.2.1 Design One – shape change

The first iteration of the design process was to remove more material along the main area of the rocker. This would hopefully lead to a reduction in the mass of the component without affecting the safety factor. The new design can be seen in *figure 4.6*

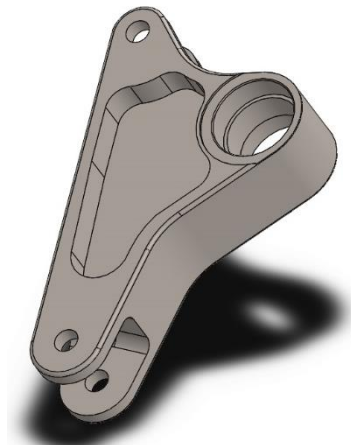


Figure 4.6 – Improved design V1

For the new design, the maximum stress increased by $0.2 \times 10^8 \text{ N/m}^2$ to $2.2213 \times 10^8 \text{ N/m}^2$ which is a 9.8% increase. This is an okay result as this means the new design keeps a safe safety factor of 2.80.

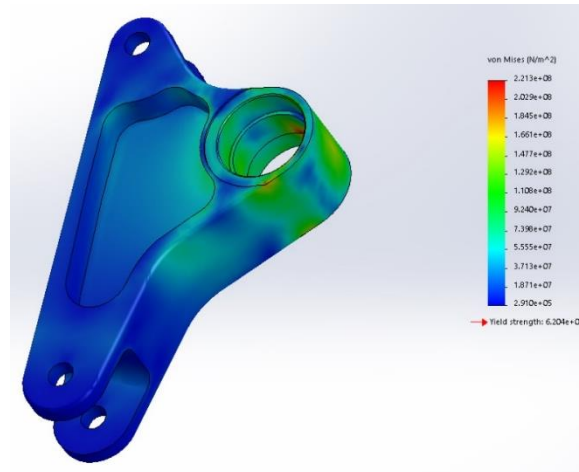


Figure 4.7 – stress (V1)

The point of high stress is located in the same place as it was in the original design. This was expected as no design effort was put into eradicating this point as it wasn't deemed necessary.

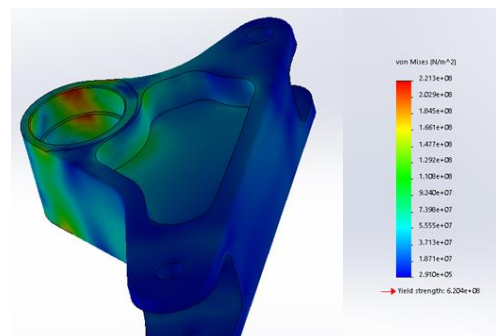


Figure 4.8 – peak stress points (V1)

As well as the stress increasing, so did the maximum displacement from 0.083mm to 0.09mm. even though this seems negligible the fact the numbers are very small anyway makes the change an 8% increase.

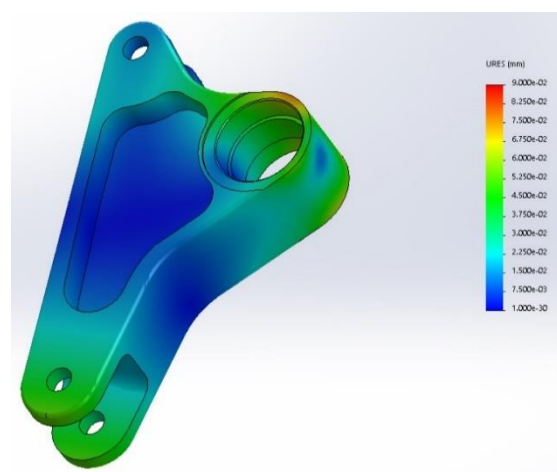


Figure 4.9 – displacement (V1)

The new design did its purpose and that was to reduce the mass of the overall component. This was done by a reduction of 16% from 1.52kg to 1.28kg. This mass saving would be a benefit but at such a small percentage change it may not be noticeable. More investigation will need to be looked into to see if the mass can be reduced any further.

4.1.2.2 *Design Two – material*

The next, and final iteration, that was made was changing the material for a lighter material. This material selected was an aluminium 2018 alloy which is much lighter than the alloy steel originally used. The final design can be seen in *figure 4.10*.

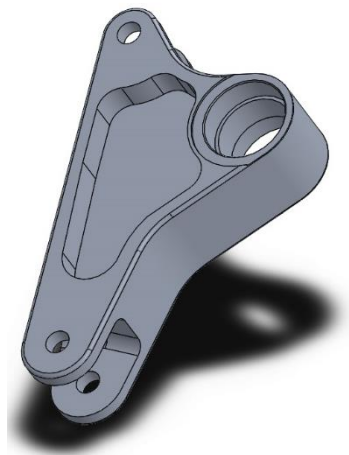


Figure 4.10 – improved design V2

The maximum stress is very similar to the stress that was calculated in iteration one, however, this material has a much lower yield strength of $3.17104 \times 10^8 \text{N/m}^2$. This means this final iteration has a safety factor of 1.4, which is quite low. This figure would not be accepted for normal automotive use, however, it could be accepted in the motorsport industry as the parts are being inspected regularly for buckling. The risk of using this component would not be worth taking since the mass reduction can not be hugely beneficial on an original 1.5kg component.

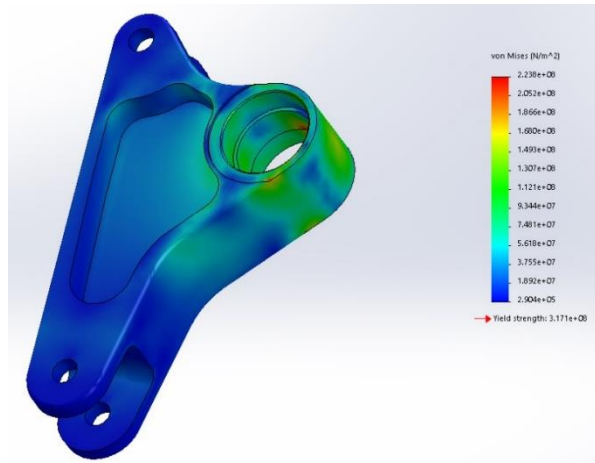


Figure 4.11 – stress (V2)

The area of high concentration of stress has reduced inside the rocker and is now being exerted right on the top of the rocker next to the hole. It is important that this area doesn't bend or break as this could cause the rocking movement to seize up which would cause there to be no suspension movement in the front suspension.

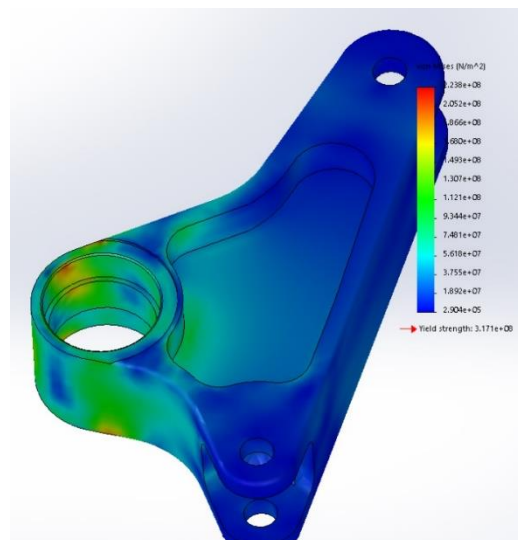


Figure 4.12 – peak stress point (V2)

The displacement has got much larger in this iteration to 0.26 from the original 0.083. This is a massive increase of 213%. This means far more deformation and flexing will occur in this rocker than the original. However, the amount is still quite small that the difference will not be noticeable by the driver.

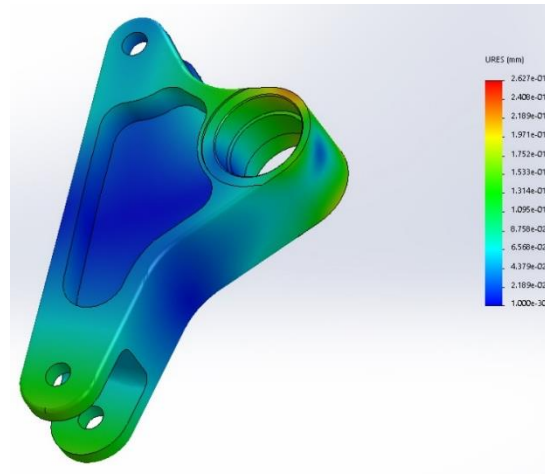


Figure 4.13 – displacement (V2)

The main aim of this iteration was to reduce the mass as much as possible and this has been done successfully by reducing the rocker to 0.47kg. This is a massive reduction of 69.4%. If this was used on both front rockers it would reduce the mass of the car by over 2kg. This amount of mass is meaningless in the automotive industry however, in the motorsport industry this value could help bring a competitors car down to the minimum allowed mass in its class.

Moving forward, if another iteration was to be completed the main focus would be to retain the current mass whilst increasing the safety factor. This could be done by adding curves to the areas of high stress or adding some more material, but more material could still be removed from the rocker such as drilling holes all the way through it.

5 Limitations and Improvements

There are a few limitations to carrying out a motion analysis and FEA in Solidworks. The first issue is that particular attention needs to be made to mate and fully define the model but this may be done in an unrealistic way to the real car. Also for motion analysis, it is advised to use minimal components to prevent too many redundancies. However, if a model could be built with no redundancies but include all the components, the motion loads in it would be more accurate. Another limitation to the Solidworks package is that it assumes damping to be linear and the same for both bump and rebound. So if the bump was hit at a high speed the user would have to decide if they wanted to use the low or high-speed damping rate of their car but this isn't accurate for a real-world car. The dampers on a car have four different damping constants, not the one that Solidworks assumes there is (Jugulkar, et al., 2016).

To improve the analysis next time, the model would need to be rebuilt with more mechanical mates to reduce the redundancies to zero. This would have been carried out if the project had less of a time constraint. Also, a full-size model that would hold the entire mass would be beneficial to do, if it can be mated correctly, so that the accuracy of the loads could be improved.

To improve the actual motion analysis the car could drive along a road and interact with a speed bump to acquire more accurate results as the loading won't just be in one axis but multiple axes. Whilst investigating the loads an analysis could be created to compare how the variable spring and damper settings would affect the motion loads, which would affect the stress experienced by the bump.

Another improvement, if the road was to move, would be to improve the model of the tyre as it is only modelled as a solid rubber material where in fact a tyre is pneumatic (Geng, et al., 2007). This means a tyre would experience variable damping under compression, as well as the tyre pressure affecting the loads as some would be absorbed by the tyre. Also, other components materials were only approximated as to get a more accurate result so it would have been beneficial to carry out tests on some components to field the correct characteristics of them, such as the yield strength and the youngs modulus of the material.

6 Conclusion

To conclude, the investigation was conducted in a way that only the trends of the stress on the component would be useful. It allowed improved designs to be made, however, if this was to be used on a real car a better model would need to be used to ensure that the max stress values were accurate so that the new design wouldn't break during use. However, if a quick analysis was needed to be done the finite element package was very quick and useful. The loads could either be imported from the motion analysis or they could be manually put in as a static investigation. This allows users to get fast results to see how a component would act in any situation but if the actual stress values were required more time would need to be invested into developing the model.

Bibliography

Cyprein, N., 2016. *Calculating Safety Factor: FEA for All*. [Online]

Available at: <https://feaforall.com/calculate-safety-factor/>

[Accessed 15 January 2020].

Geng, Z., Popov, A. & Cole, D. J., 2007. Measurement, identification and modelling of damping in pneumatic tyres. *International Journal of Mechanical Sciences*, Volume 49, pp. 1077-1094..

Jugulkar, L. M., Singh, S. & SM, S., 2016. *Analysis of suspension with variable stiffness and variable damping force for automotive applications*, Los Angeles: Sage.

Ponginan, R., 2015. *Practical Aspects of Finite Element Simulation*. 3rd ed. na: Altair University.

Schumacher, A., 2015. *Practical Aspects of Structural Optimisation*. 2nd ed. n/a: Altair University.

The AA, 2016. *Speed Bumps: Advice: Breakdown cover: The AA*. [Online]

Available at: <https://www.theaa.com/breakdown-cover/advice/speed-bumps>

[Accessed 4 January 2020].

The Engineering ToolBox, 2017. *Factors of Safety: Engineering ToolBox*. [Online]

Available at: https://www.engineeringtoolbox.com/factors-safety-fos-d_1624.html

[Accessed 15 January 2020].

https://uowtsd-my.sharepoint.com/:f/q/personal/1702453_student_uwtsd_ac_uk/Eq5NOx89hZZlumex8gHTik0BBJo6HOZliULBIjPu-jUIWg?e=UdNIgu