

University of Wales Trinity Saint David

Motion Simulation and FEA

M6X00408

Cornelius May

1701329

26/01/2020

Introduction

Throughout this investigation, the aim will be to investigate the suitability of a component of an MCR sports 2000 race car's suspension assembly. In order to assess the suitability, the Motion Sim tool within Solidworks will be used in order to produce motion loads that can be imported into Solidworks' FEA analysis tool.

The MCR is a single seater, open top race car, used for racing on track. Considering its use, strength and mass are the main areas of importance to consider for each component, as their integrity must stand up to abuse over its lifetime without reducing the vehicle's capabilities by weighing it down (Racing Car Dynamics, 2020). In order to reduce the amount of real-world tests and to reduce wastage of prototype components, computer simulation methods are often used in industry in order to estimate a part's suitability for a role prior to it being manufactured (Blundell and Harty, 2015).

Scenario

The aim of this investigation is to look at the performance of the front left lower wishbone, on an MCR Sports 2000 race car. When designing components such as wishbones, there is always a trade-off between mechanical strength and the weight of a component (Jones, 1992). Within motorsport the emphasis usually leans towards light components, none the less the driver never wants the part to fail during a race.

In order to estimate the maximum forces that a car is likely to see on an average race track, the simulation consisted of running the front left wheel over a short step followed by a bump and then a drop off another step. The steps both had a height of 15mm, demonstrating a curb or piece of debris on the track. For the sake of consistency, this bump was defined as having the shape of a $1-\cos(\theta)$ with an amplitude of 40mm. This shape is easily repeatable and re-creatable. Furthermore, the amplitude allows for an aggressive bump, without coming into contact with any of the chassis or body.

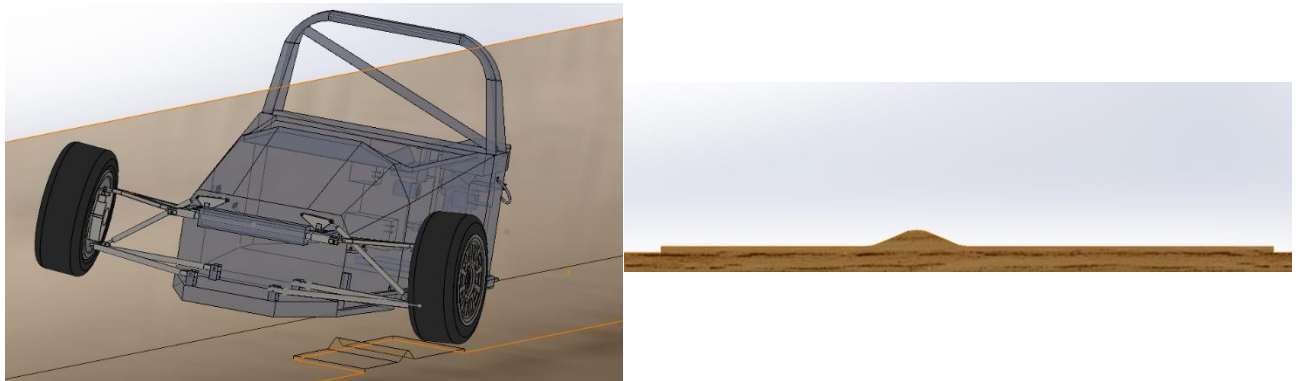


Figure 1: Motion Simulation Scenario

Finite Element Analysis

The finite element analysis (FEA) was used, in order to calculate forces and stresses at various points throughout the component. The FEA converts the infinite number of points that make up a real 3D object and converts them into a finite number of elements, for each of which the forces can be calculated (BOSCH GMBH, 2018). The creation of the points or nodes, is called meshing and can be seen in Figure 2. In the case of this analysis, the component was converted into a mesh consisting of 15,992 nodes. At each node the forces can be converted from 3D into 1D components along the lines connecting nodes to each other. This way the 1D equation for stress can be used to calculate

the stress at each node (Omnicalculator.com, 2020). Solidworks offers to possibility to investigate 1st, 2nd and 3rd Principal stresses. These are stresses at each node in the X, Y and Z direction with the 1, 2 and 3 referring to the direction in descending order of magnitude of stress (Lombard, 2013). The Von Mises Stress, which was primarily used throughout this investigation, consists of the resulting stress calculated by taking into account magnitude and direction of the 3 component stresses(Dassault Systems, 2019).

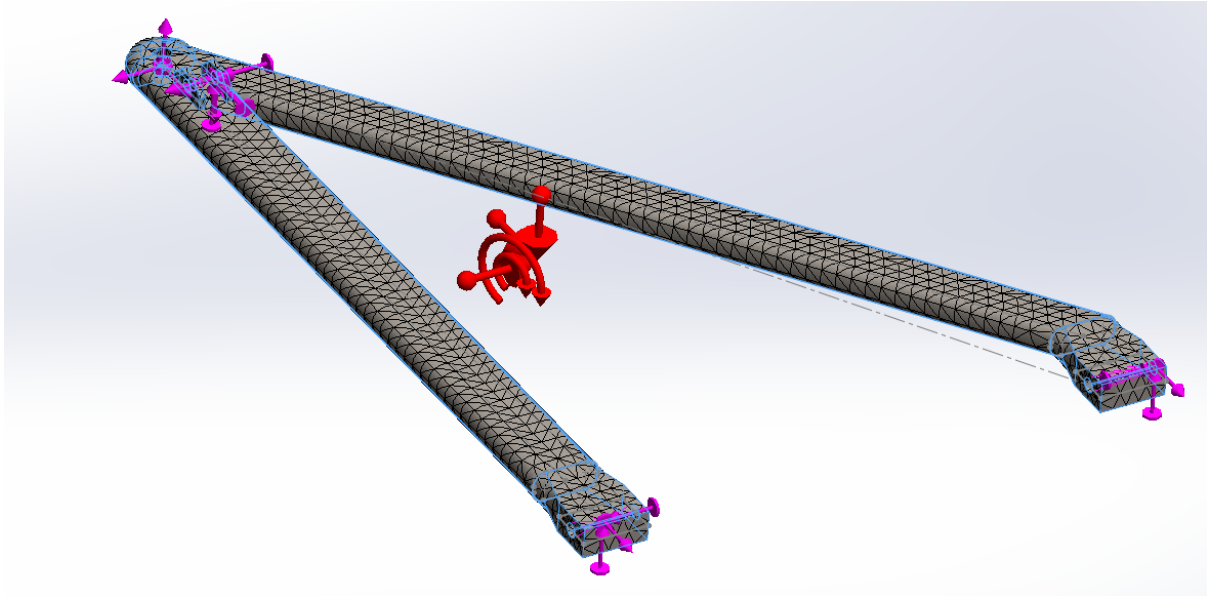


Figure 2: Lower Wishbone Mesh

Method

In order to eventually be able to complete an FEA analysis on any one component of the MCR, a number of other steps must first be undertaken.

The first step is to create a CAD model of the relevant components of the MCR. For this investigation, as well as the required joints and fixings, these components consisted of the front suspension assembly including wishbones, hubs, push-rods, tie rods, rims and tyres, the chassis itself and the sledge replacement for the rear suspension.

Before the CAD model can be created, the individual components need to be measured and weighed. This involved disassembling the relevant areas of the car in order to remove these components. At this point, their geometry can be measured using a combination of Vernier callipers and steel rules, and the parts can be weighed. This point can be the difference between accurate results and results that bare no relevance to the real world, so it is important to take care and where possible, double check measurements. Once this is done, the geometry can be re-created in CAD.

Once all the components have been constructed, it is possible to assign material properties to the components within Solidworks. Using the assigned material Solidworks will estimate the mass of the component. Comparing these to the measured masses has shown that, whilst these are a good estimate of the mass of a component, they are rarely correct. This is because fine details, such as welds, are difficult and time consuming to model and are therefore often ignored. Instead, Solidworks provides the option to override the mass of a part without removing material properties, such as maximum yield strength.

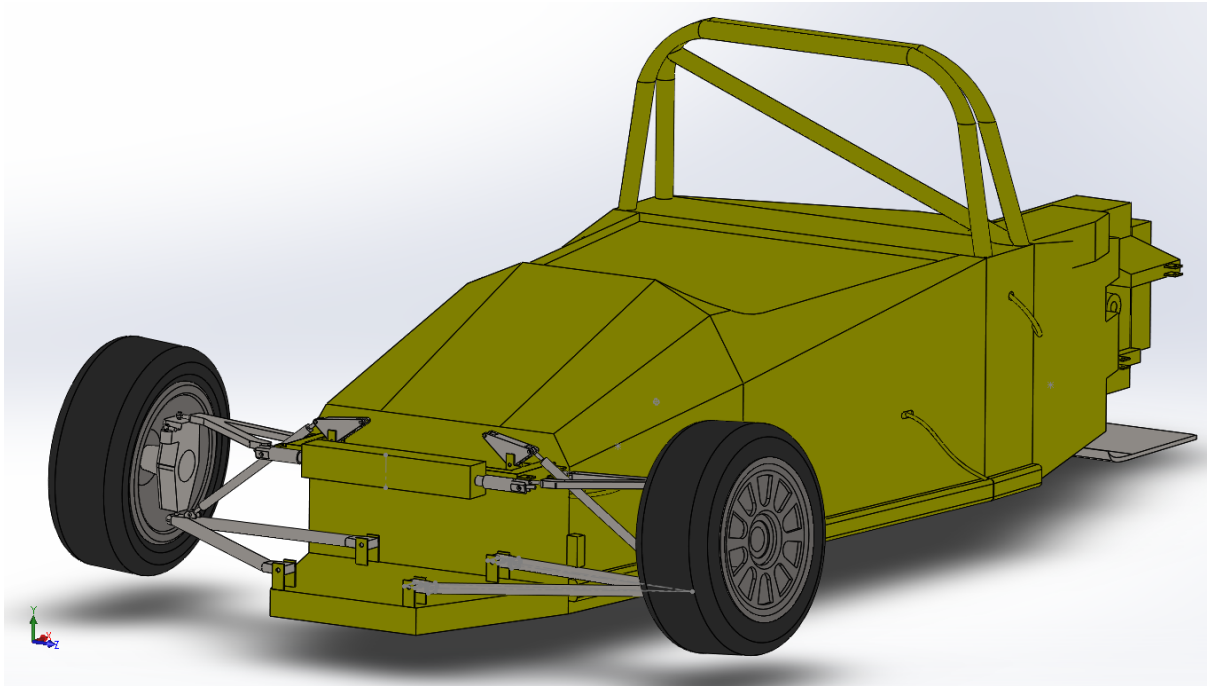


Figure 3: MCR Assembly CAD Model

Now that the individual components are ready they can be assembled into a full vehicle model. As part of the assembly process, the mates between components are included, defining the way that components interact with each other. In many places, mates can replace the need to model real fixings as they can dictate the parts behaviour as if the part was there. An example of this is the mechanical hinge mate. It is important to make sure that, whilst allowing the components to move only the way they would in real life, they don't over constrain the components motion. Whilst creating the assembly and defining the mates, the full range of mates should be used. Often a single, more complicated mate, can replace multiple more simplistic mates. Doing this will reduce the number of redundant mates later on. Furthermore, some mates can be removed completely by combining multiple components into a "rigid group". This again would reduce the number of mates being made redundant in the motion simulation. As this assembly contains components that are symmetrical left to right, the individual files were created once on the same side of the car and then mirrored.

Furthermore, equations can be used in order to define distances that are common across multiple components. This is especially useful in the case of the symmetrical components whose distances are the same left to right. They can be defined centrally, or even externally from Solidworks and then called up as required. As well as this, for the components being investigated in FEA, as part of the process of defining mates, the load bearing faces need to be defined. Later, when the FEA is carried out, these are the points at which forces are applied to the component. This process defines, within Solidworks, which loads it needs to record during the course of the Simulation.

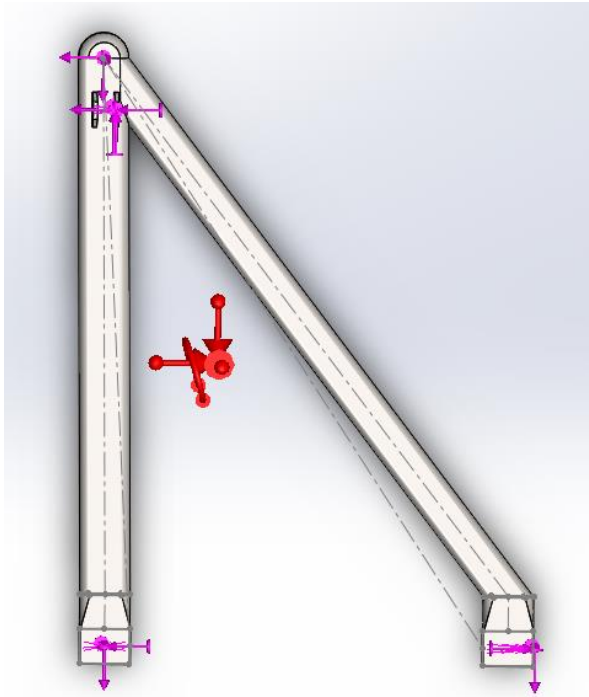


Figure 4: Lower wishbone motion loads

At this point, the assembly can be opened in Solidworks Motion Study. Here the motion of the assembly is defined, including external forces applied to the assembly and how external forces interact with the individual components. An example of the forces applied at this point, are the force to propel the vehicle, usually supplied by the engine. At this point, the spring and damper relationship between the rocker and the chassis were introduced. Finally, rigid contacts between the assembly and the external components (in this case the road) can be defined. Once the motion simulation has started to run it is a good idea to perform a visual inspection of the scenario. This simple check can often catch out any mistakes that might otherwise have been missed, possibly causing issues when looking at the resulting data.

Once the Motion Sim has been calculated, Solidworks can use the forces calculated during the simulation to investigate the forces being applied to the component under investigation. Exporting these motion loads into the individual file allows them to be used in the FEA tool. For this it is necessary to specify which part of the motion simulation is needed as often the entire simulation is too long and not required in its entirety for the FEA.

Finally, the part under investigation can be opened separately from the assembly. From here it will be obvious that there are new tabs that were not there before the motion sim was completed. These contain the motion load data retrieved from the motion sim. After Solidworks has had the opportunity to complete the FEA calculations from the imported motion loads, it is possible to view stress, strain and displacement across the surface of the component for each frame of the motion sim. This makes it possible to identify weak spots in the design of the components for evaluation and improvement.

Results

The first analysis that was carried out was completed using a model of the lower wishbone which, using measured data, represented the real-world component as closely as possible. After a few runs

were completed in order to determine the optimal section of the run to investigate the results shown below were retrieved.

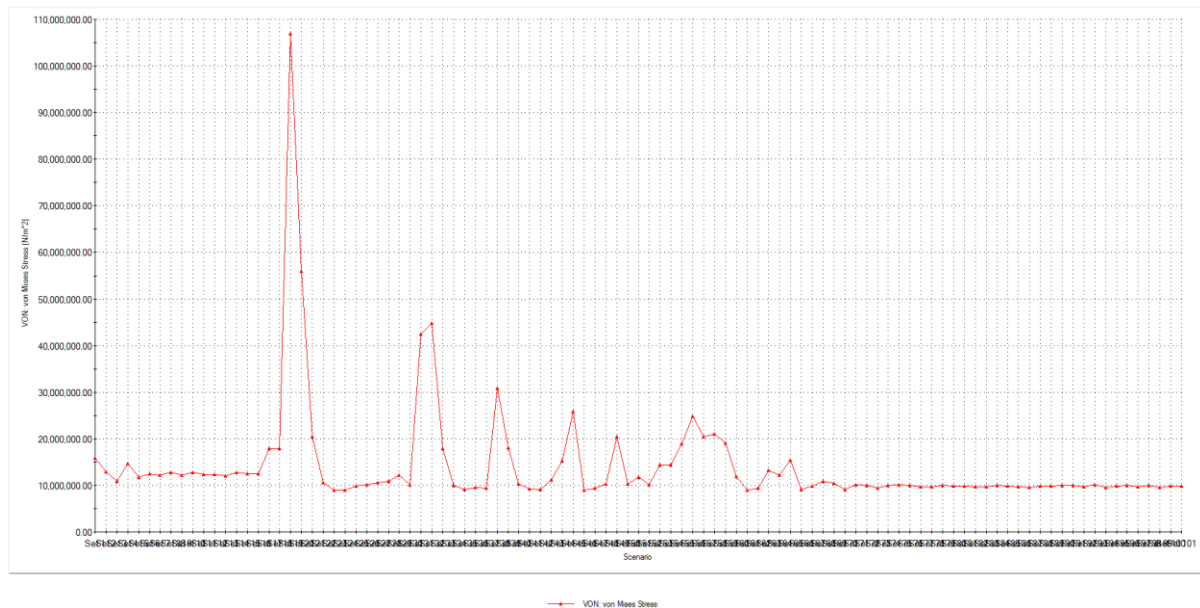


Figure 5: Original Wishbone max von Mises Stress per Scenario

As previously mentioned, using the method described above produces multiple FEA analysis per Motion Simulation run, with one “scenario” being created at each frame of the motion sim. Figure 5 shows a plot of the maximum Von Mises Stress from each scenario of the Motion Sim. Using this the maximum overall stress can be determined to occur at scenario 18.

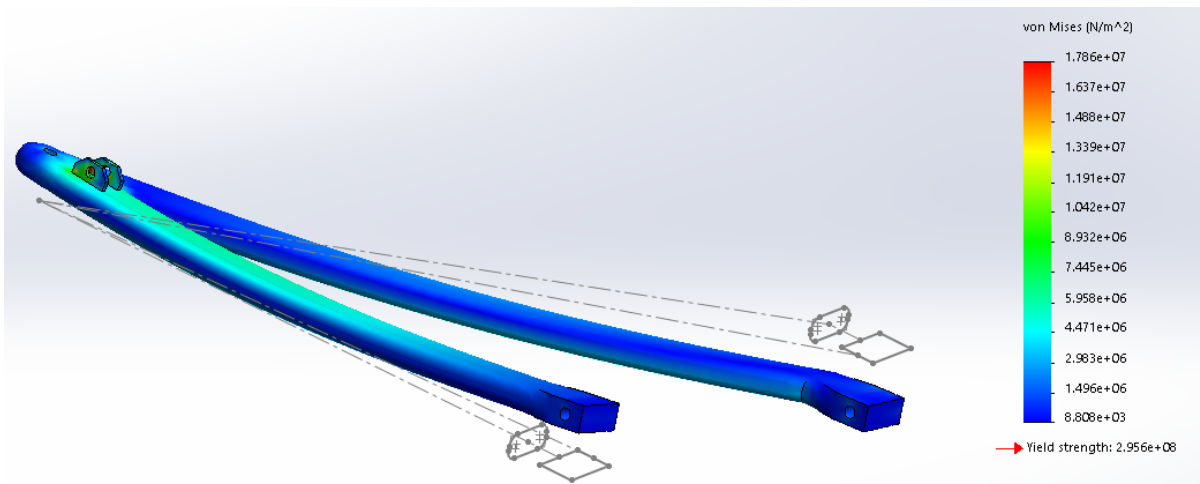


Figure 6: Original Wishbone Scenario 18 von Mises Stress

Knowing the maximum stress is important, as it shows whether the overall structure is likely to be strong enough for the scenario. If the yield strength is never exceeded, the part will not fail within the given scenario (Study.com, 2020). The right side of Figure 6 shows what the maximum stress experience was and also shows the yield strength of the material being used. Since the yield strength was not exceeded at any point, it is displayed below the scale however, if it had been exceeded, a red arrow on the scale would denote which colour is representative of the yield strength.

This becomes useful when investigating the 3D surface colour of the part, which is created as part of the FEA in order to show what stress is seen at each point on the part. If the yield strength had been

exceeded, this colour plot would make it easy to identify areas that would have failed. Since this did not happen we can assume that the part would not have failed throughout the given scenario. Instead, the colour plot makes it possible to identify weak spots that can be improved, or strong areas where a weight reduction might be possible.

For this wishbone, 3 areas for improvement were identified. The first two areas concentrated on relieving stress concentrations as identified by the FEA analysis. The analysis showed that the area where the pushrod's rose-joint connected to the wishbone caused a stress concentration as can be seen in Figure 7. Another stress concentration was identified at the point where the pushrod bracket attached to the wishbone as Figure 8 shows.

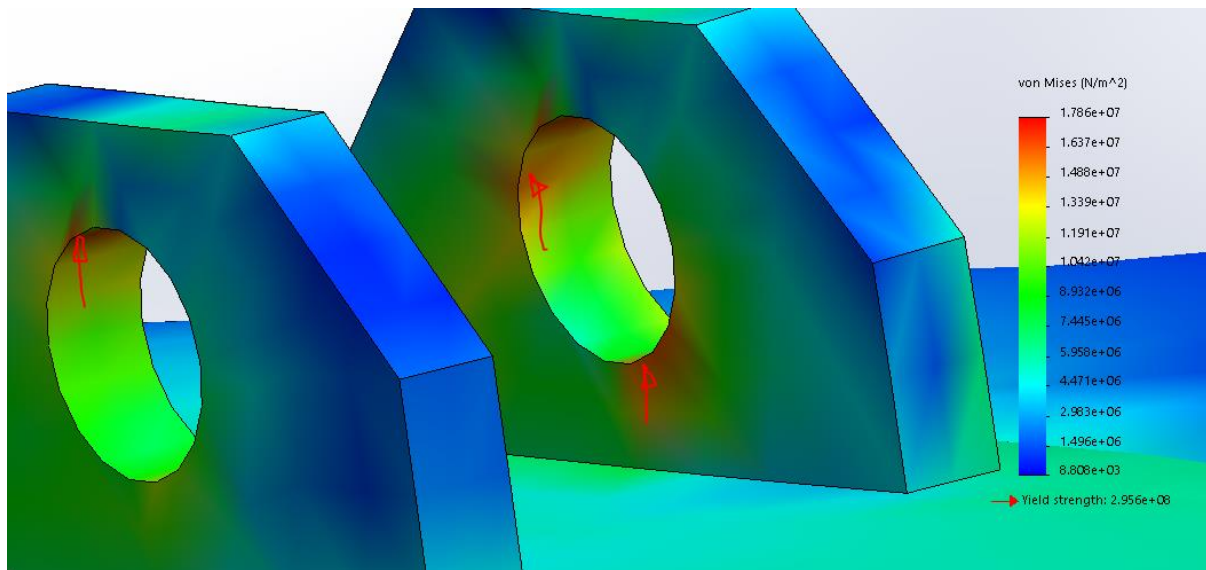


Figure 7: Push-Rod Bracket to Rose-Joint Original Wishbone Scenario 18 von Mises Stress

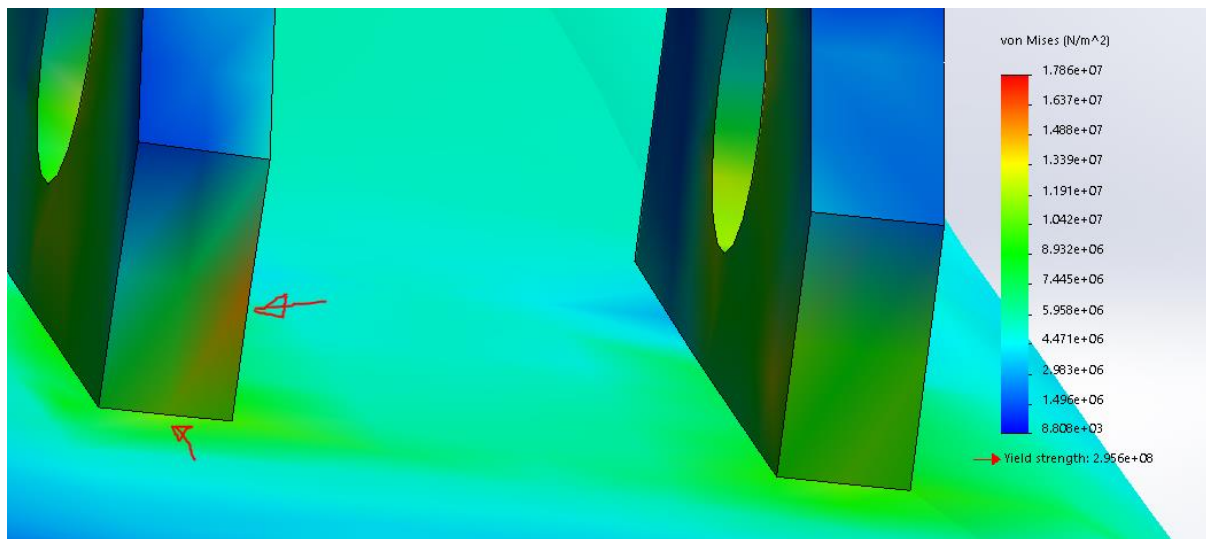


Figure 8: Push-Rod Bracket to Wishbone edge Scenario 18 von Mises Stress

In an attempt to alleviate these stress concentrations, 1mm radii were applied to all of the affected areas. The results can be seen in Figure 12. This had the opposite result of what was expected, increasing stresses throughout the wishbone to the point where the yield strength was exceeded in multiple areas. Furthermore, whilst the stress concentration around the rose-joint attachment was decreased, the stress concentration in the key area at the bottom of the bracket became worse than

it was in the previous iteration. The next attempt to reduce the stress concentration and the overall stresses throughout the component was to increase the radius of just the sides of the bracket to 2mm. This further compounded the issue, resulting in the final improvement, increasing the radius around the base of the bracket to 2 mm. This final improvement had the desired result of relieving the stress concentration as can be seen in Figure 9.

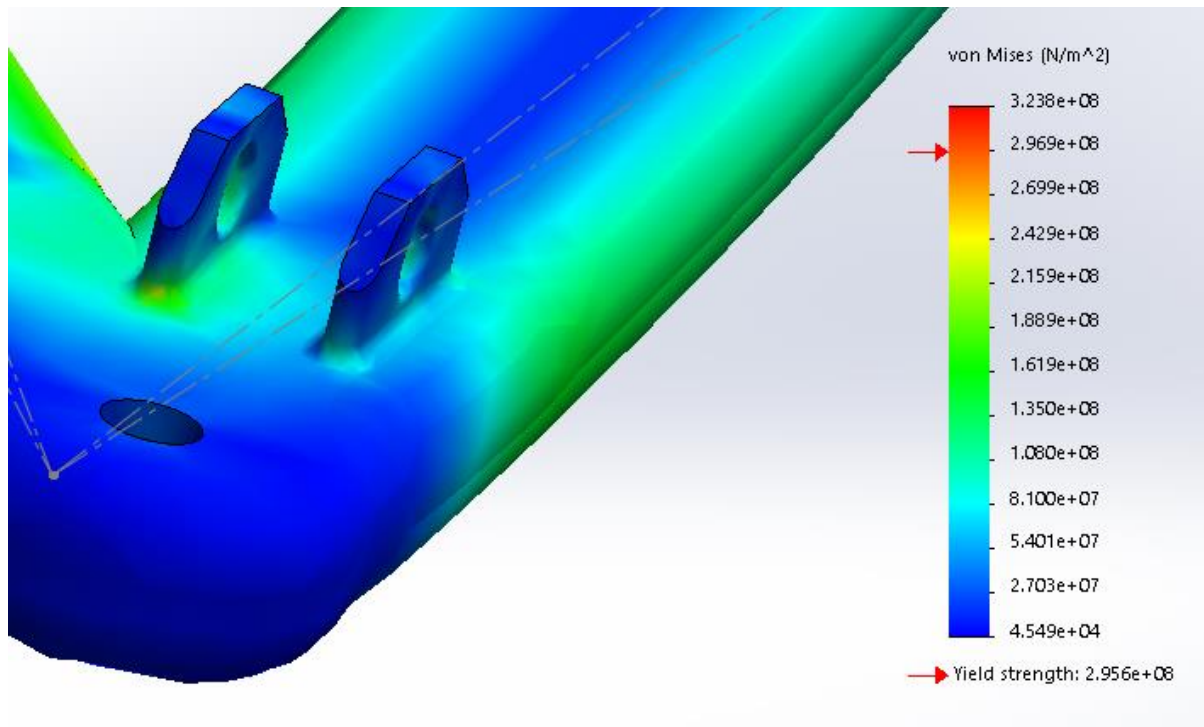


Figure 9: Push-Rod Bracket to Wishbone with 2mm radius on sides and base Scenario 41 von Mises Stress

The drawback of this improvement was that another stress concentration became apparent. Stress concentrations, exceeding the yield strength were seen on the inside of the joint between the front and rear sections of the wishbone (Figure 14). Initially a similar approach as previously was taken by introducing a 4mm radius to the affected area. Whilst reducing the stress in the affected area, the improvement was insufficient to drop the stress below the yield strength of the material. The final attempt to resolve this issue can be seen in Figure 16 where a bar was introduced, in order to brace the longitudinal motion of the wishbone. This introduced sufficient strength into the material to reduce the stiffness to below the yield strength.

The final area of improvement looked at the effect of reducing the weight of the component. Although currently the wishbones are made of a solid piece of steel alloy, it would be possible to hollow out the structure, using hollow bar for its construction. This would result in a part with the same external dimensions but a significant weight reduction.

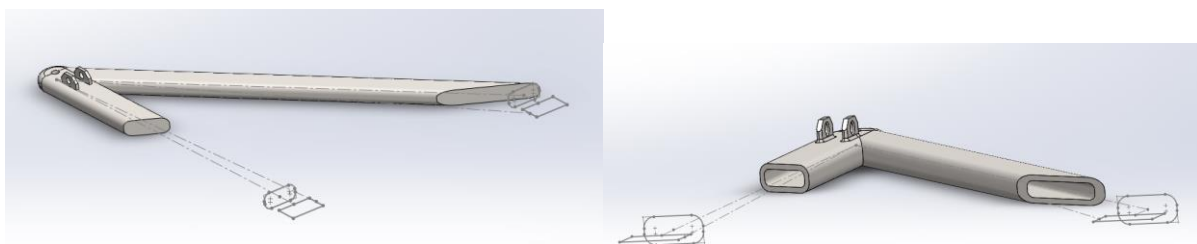


Figure 10: Cross-Section of Wishbone

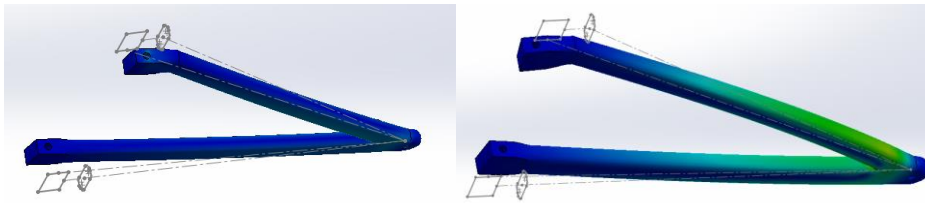


Figure 11: FEA comparison of solid and hollow Wishbones

As the analysis in Figure 11 shows, the proposed wall thickness would result in failure of the component as the strength in the walls becomes too low. This could be alleviated with a higher wall thickness.

Conclusion

As a result, it is inadvisable to replace the part currently in use with a part implementing the radius improvements outlined above. Considering that the part was not exceeding the yield strength, the introduction of these features would result in a weaker part than the original.

Furthermore, the bracing bar could be considered, as it does provide a significant increase in strength of the part. This has the downside of increasing the parts mass and since the original part does not require any increase in strength, would only serve as a drawback.

It is advisable to ensure that there is a safety margin between the maximum stress seen in the FEA and the maximum yield stress of the material. Whilst the FEA gives a good idea of where the stress concentrations lie, it is impossible to say with any certainty, if the magnitude of the stresses seen are representative of those that the part might see in real life. It would be ideal to conduct some real-world testing, to ensure that the forces coming out the motion sim are representative of the real-world scenario. Figure 1

Future testing could involve further increasing the wall thickness of the hollowed-out wishbone, until stress concentrations fall below the yield strength of the material. This would make it possible to find the lightest version of the wishbone that does not fail under load. Ideally, a non-constant wall, where the thickness is reduced and even removed completely depending on the forces moving through an area, would result in the lightest possible component.

Furthermore, before any parts are considered for real world use an investigation, to determine the accuracy of the forces and stresses displayed by the Solidworks FEA tool, must be carried out. This would ideally be carried out using the wishbone to be improved for the most accurate results.

Bibliograph

- Study.com. (2020). [online] Available at: <https://study.com/academy/lesson/what-is-yield-stress-definition-formula.html> [Accessed 20 Jan. 2020].
- Blundell, M. and Harty, D. (2015). *The multibody systems approach to vehicle dynamics*. Kidlington, Oxford, UK: Butterworth-Heinemann is an imprint of Elsevier.
- BOSCH GMBH, R. (2018). *AUTOMOTIVE HANDBOOK 10E*. [Place of publication not identified]: WILEY-BLACKWELL.
- Dassault Systems (2019). *2019 Solidworks Help*.
- Racing Car Dynamics - Your Uncomplicated Resource to Race Engineering. (2020). *Race Car Design: 6 Steps to Design like a Pro*. [online] Available at: <http://racingcardynamics.com/race-car-design-in-6-steps-2/> [Accessed 20 Jan. 2020].
- Omnicalculator.com. (2020). *Stress Calculator - Omni*. [online] Available at: <https://www.omnicalculator.com/physics/stress> [Accessed 20 Jan. 2020].
- Jones, P. (1992). *CAD/CAM*. Macmillan.
- Lombard, M. (2013). *SolidWorks 2013 bible*. Indianapolis, Ind.: Wiley.

Appendix

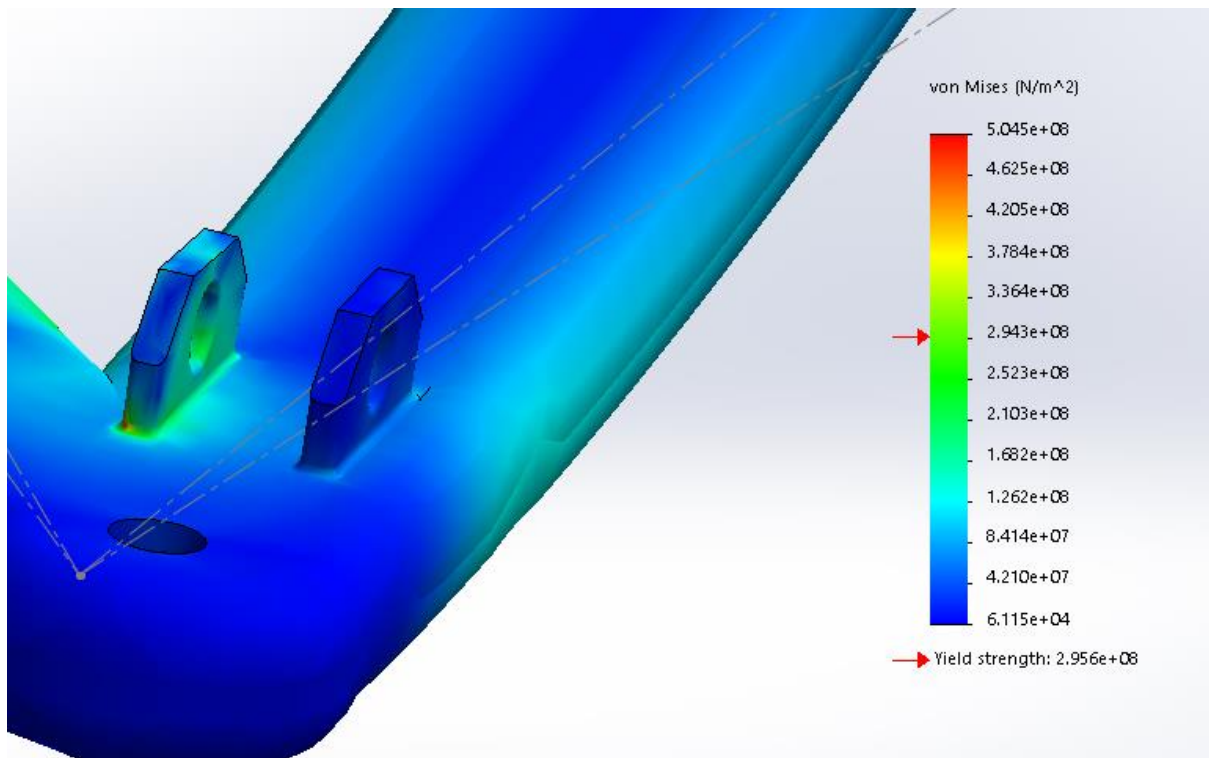


Figure 12: Push-Rod Bracket and Rose-Joint to Wishbone with 1mm radii Scenario 51 von Mises Stress

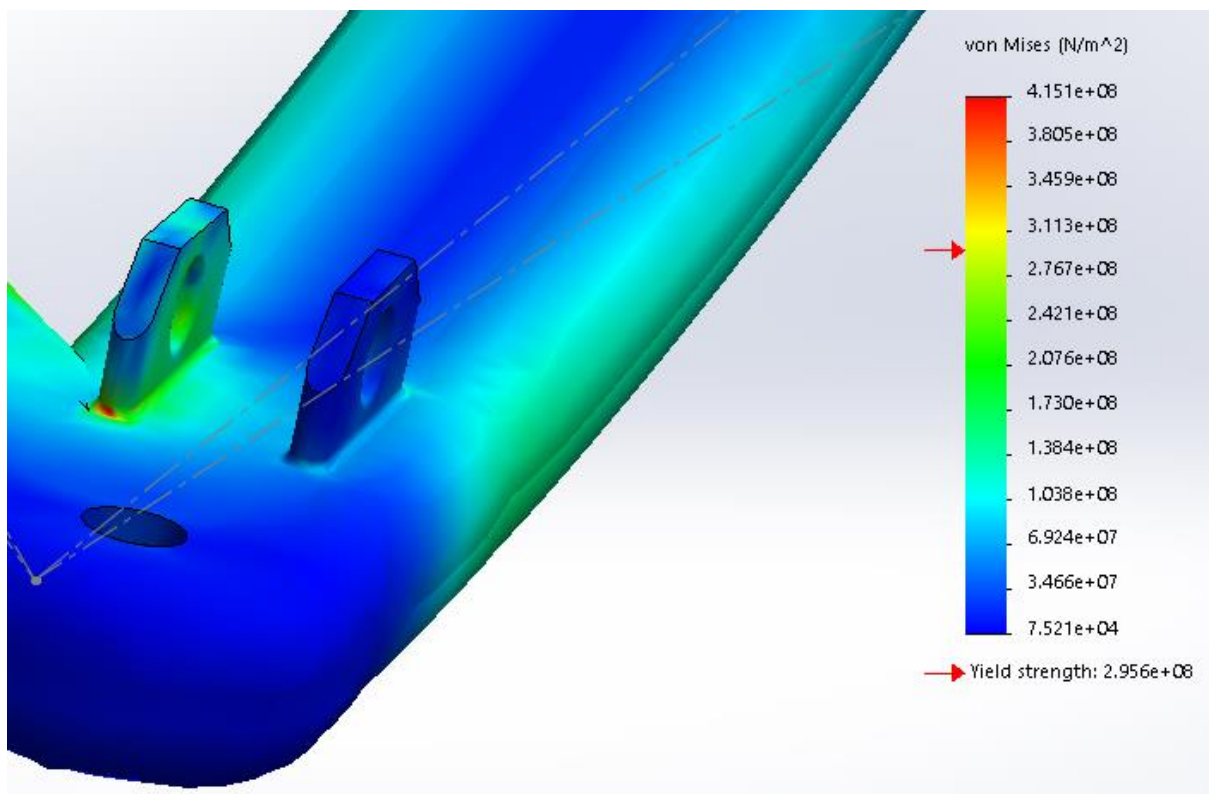


Figure 13: Push-Rod Bracket to Wishbone with 2mm radius on sides and 1mm on base Scenario 14 von Mises Stress

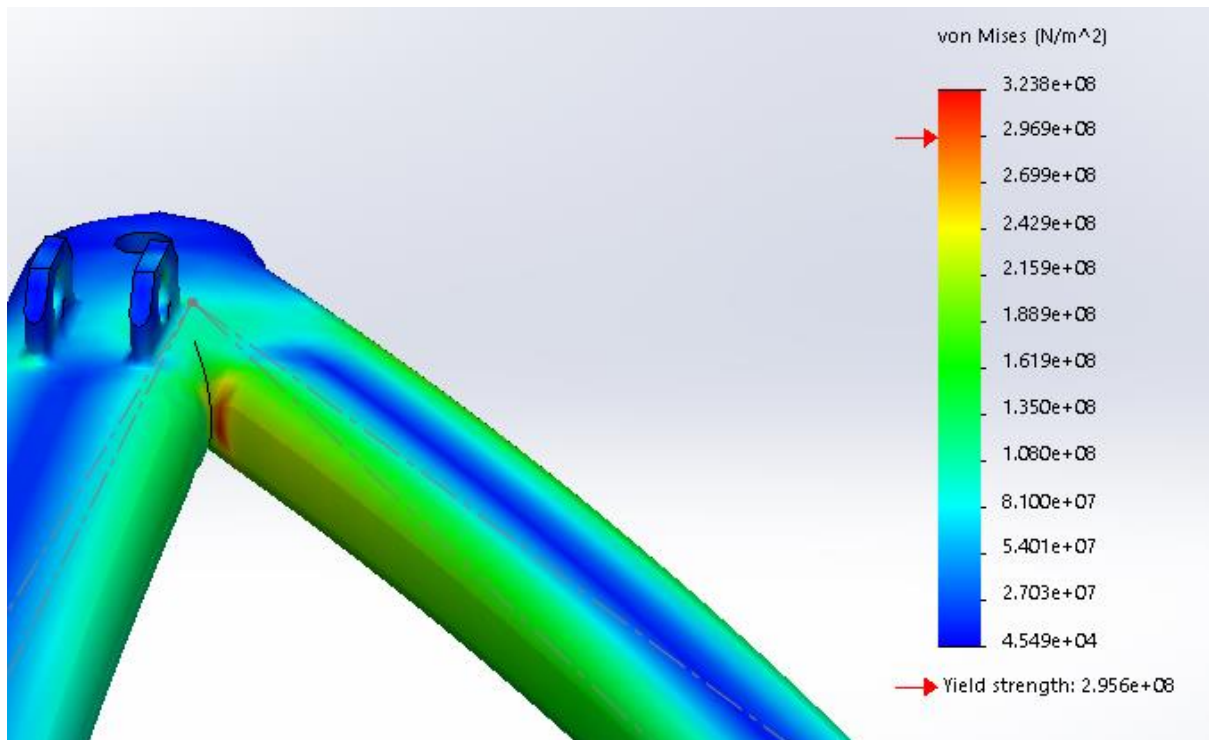


Figure 14: Push-Rod Joint Scenario 41 von Mises Stress

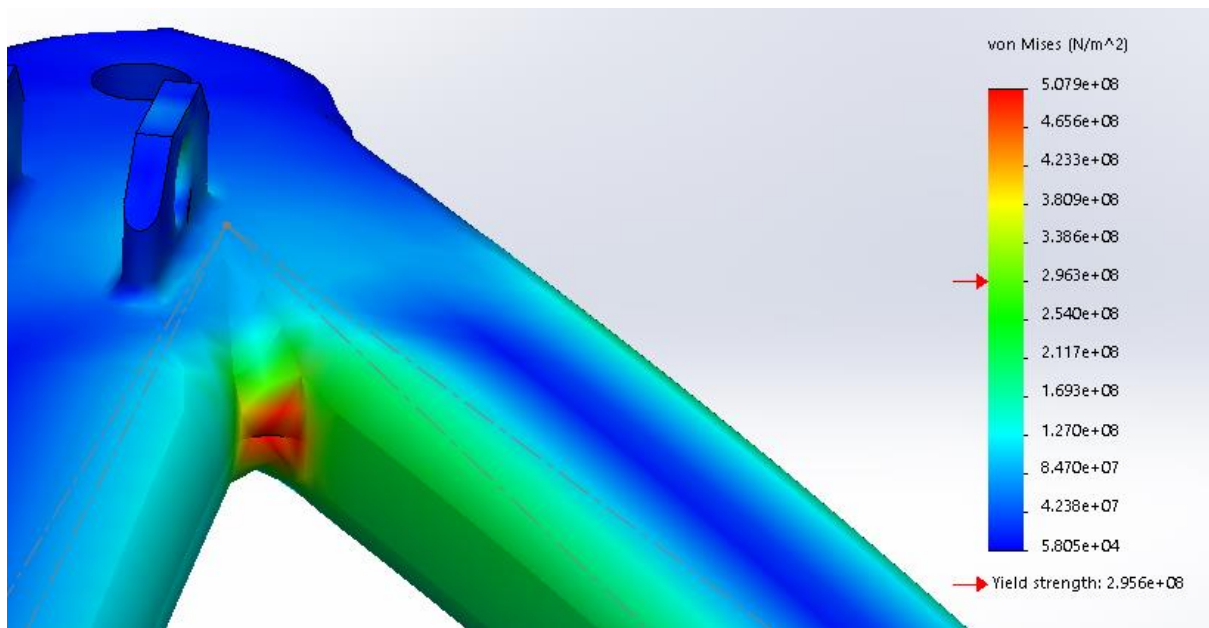


Figure 15: Push-Rod Joint with 4mm radius Scenario 28 von Mises Stress

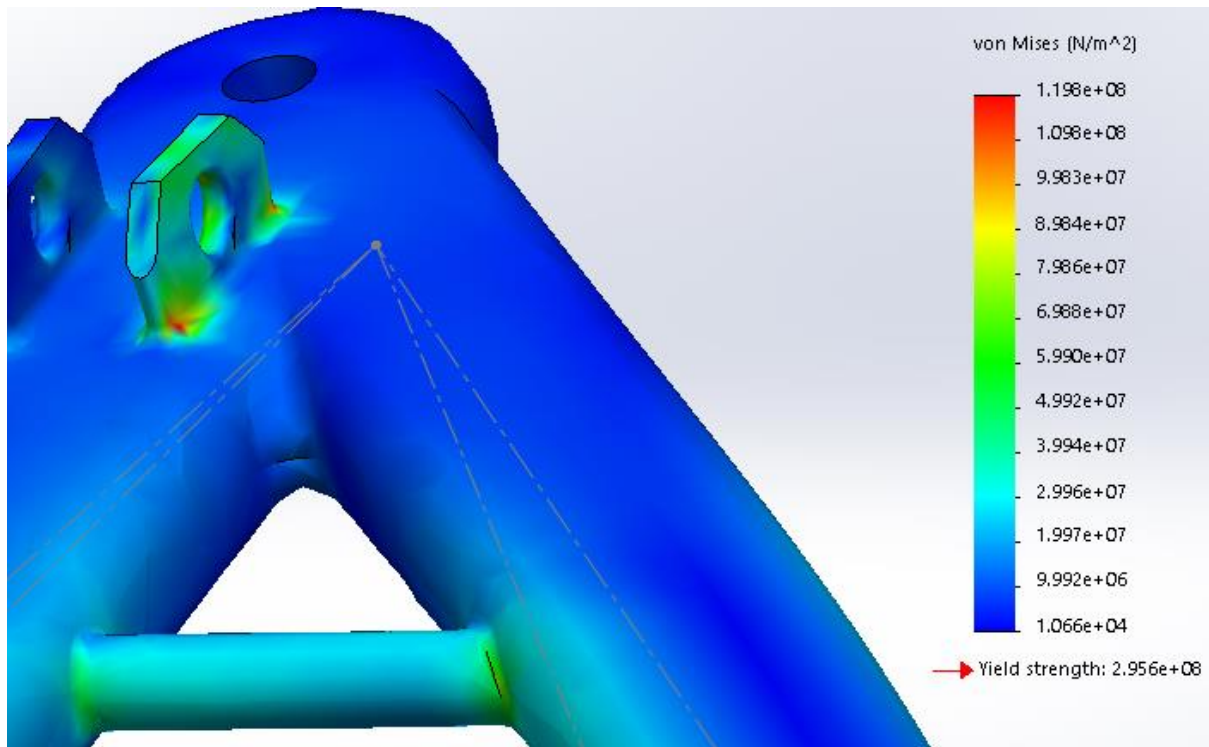


Figure 16: Push-Rod Joint Scenario 1 von Mises Stress

Report generated by Solidworks can be found in the folder with the CAD files – Leo_LowerWishbone-CM1-ALT-Frames-1300-1400-1-1.docx. This report is generated automatically by Solidworks using data recorded throughout the FEA process. It shows the highlights including stress concentrations to show where the maximum Yield Stress is exceeded.

Please find all relevant files at:

https://uowtsd-my.sharepoint.com/:f/g/personal/1701329_student_uwtsd_ac_uk/EisZZ_UG69ROuuvZRusj83ABiTc14b_wjTKqYiSn7QdoQ?e=fUdRSj