

Josh Martin - CAE

by Joshua Martin

Submission date: 18-Mar-2022 11:09PM (UTC+0000)

Submission ID: 173912369

File name: 12205_Joshua_Martin_Josh_Martin_-_CAE_411906_1219770504.docx (18.41M)

Word count: 4327

Character count: 20767

2022

Computer Aided Engineering

LIMITATIONS OF CAE – TANK SUSPENSION SYSTEM – 3294 WORDS

JOSH MARTIN | 1810346

Contents

1	Introduction	3
1.1	Types of Computer Aided Engineering Systems	3
1.2	FEA	3
1.3	Meshing.....	3
2	Case Study.....	3
2.1	Overview	3
2.2	Components.....	3
2.3	Motion Study.....	4
2.4	Understanding the Initial Model	4
3	Model Validation.....	5
3.1	Hand Calculations.....	5
3.2	FEA	5
4	Model Redesign	8
4.1	Model 1	8
4.2	Model 2	9
4.3	Model 3	9
4.4	Model 4	10
4.5	Model 5	11
4.6	Model 6	11
4.7	Model 7	12
4.8	Model 8	13
5	Conclusion.....	14
5.1	Model Improvements	14
5.2	CAE limitations	14
5.3	Solidworks limitations	14
6	References	15
7	Appendix A – Drawn Components	16
8	Appendix B – Initial Model	16
9	Appendix C – Mesh Comparison	17
10	Appendix D – Model 2.....	18
11	Appendix E – Model 5	19
12	Appendix F – Model 6	20
13	Appendix G – Model 7.....	21

Figure 1 Suspension Assembly	3
Figure 2 Initial Model	4
Figure 3 Reaction Force Graph.....	4
Figure 4 Cantilever Beam - 2000N - 340mm Max Displacement.....	6
Figure 5 Cantilever Beam - 2000N - 340mm - Von Mises.....	6
Figure 6 Cantilever Beam - 2000N - 170mm Max Displacement.....	6
Figure 7 Cantilever Beam - 2000N - 170mm - Von Mises.....	6
Figure 8 Cantilever Beam - 2000N - 85mm Max Displacement	6
Figure 9 Cantilever Beam - 2000N - 85mm - Von Mises	6
Figure 10 Cantilever Beam - Failed 5mm mesh	6
Figure 11 Cantilever Beam – 500kN - 340mm mesh - Max Displacement	7
Figure 12 Cantilever Beam – 500kN - 340mm mesh - Von Mises.....	7
Figure 13 Figure 8 Cantilever Beam – 500kN - 170mm mesh - Max Displacement.....	7
Figure 14 Cantilever Beam – 500kN - 170mm mesh - Von Mises.....	7
Figure 15 Figure 8 Cantilever Beam – 500kN - 85mm mesh - Max Displacement.....	7
Figure 16 Cantilever Beam – 500kN - 85mm mesh - Von Mises.....	7
Figure 17 Model Redesign 1.....	8
Figure 18 Model Redesign internals	9
Figure 19 Model Redesign 2 - Strengthened	
Figure 20 Model Redesign 3.....	9
Figure 21 Model redesign 3 - Factor of Safety.....	10
Figure 22 Model Redesign 4.....	10
Figure 23 Model Redesign 5.....	11
Figure 24 Model Redesign 6.....	11
Figure 25 Model Redesign 7.....	12
Figure 26 Model Redesign 7 - Large Displacement Issue.....	12
Figure 27 Model Redesign 7 Factor of Safety - Axle	13
Figure 28 Model Redesign 7 Factor of Safety - Bracket	13
Figure 29 Model Redesign 7 Factor of Safety - Main Body.....	13
Figure 30 Model Redesign 8.....	13
Figure 31 Model Redesign 8 - Bracket	13

1 Introduction

1.1 Types of Computer Aided Engineering Systems

The computer aided technology used today can be split into different categories, most heard of as CAD/CAM, these are Computer Aided Design and Computer Aided Manufacturing respectively. 'In its simplest form, it is a computerized communication and design function within and between design engineering and manufacturing engineering' [1]. That quote encompassed the understanding of CAD/CAM in 1987, however a more modern development is the introduction of CAE, Computer Aided Engineering. Computer Aided Engineering is built on the foundation of CAD/CAM but refers to the use of the software to simulate loads and constraints of a product or structure and analysing the effects of the different conditions [2]. All three disciplines work together, first the part must be designed using CAD, then CAE to analyse the conditions the part will be subjected to through its life, then to manufacture the part, CAM can be used, completing a fully computerized manufacturing cycle.

1.2 FEA

CAD is limited to kinematics; modelling of dynamic systems usually must go through a dedicated finite element analysis package [3]. 'Finite Element Analysis is a mathematical technique used to obtain approximate solutions to complex problems that cannot be solved using basic theories' [4]. Whilst this is true, it should also be noted that Finite Element Analysis should always be validated, this can be done using basic mathematical theories. Regardless of how accurate the CAD models are, the FEA is reliant on the physical properties of materials and experimental data loaded into the software [3].

1.3 Meshing

Using Finite Element Analysis, the part to be analysed needs to be divided into a mesh of finite sized elements of simple shape [5]. Generally, as the number of elements increases, a more accurate result of the analysis can be achieved however element shape can have an impact on accuracy as well. Triangular elements are a standard for automatic meshing in the authors experience, however elements that can have curved sides and can model a quadratic variation in displacement or linear variation in stress are generally preferred [5]. Elements are called finite to emphasize they are not infinitesimally small but small in comparison to the overall model [6]

2 Case Study

2.1 Overview

A Solidworks assembly has been created using individual parts based on a real-world military vehicle suspension system. The assembly will be subjected to a motion simulation and Finite Element Analysis (FEA) will be used on the Radius Arm. The radius Arm will then be redesigned to meet performance targets to stay under the materials maximum Yield Strength with a minimum factor of safety of 1.5, and a maximum mass of 110kg.

2.2 Components

Each component model that was created from the engineering drawings can be found in Appendix A, all dimensions are correct to the given drawings.

- Test Plate – Provides a fixed point for the assembly to work around during the motion study.
- Test surface – Acting as the 'real world' floor, it allows for the simulation of suspension movement using an oscillating linear motor.
- Radius Arm – The part to be structurally analysed, being acted on by the opposing forces of the oscillating linear motor and the damper and spring assembly.
- Wheels/Tyres – Provides the connection between the floor and the radius arm, there's a direct interaction between the wheels and the radius arm, a likely point of high stress.
- Wheel Spacer – no impact on the analysis, just representative of the real-world setup to space the wheels apart.

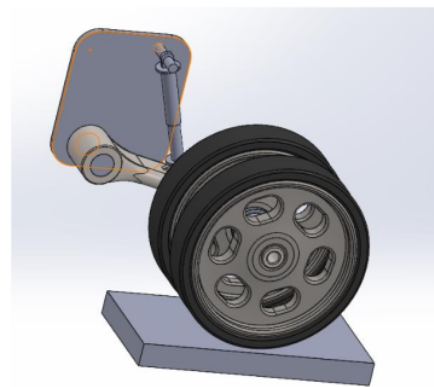


Figure 1 Suspension Assembly

- Upper/Lower Damper – Representative part of the real-world suspension, dampers will have a load rating as well as providing a surface for the spring to be simulated onto without having to physically design a spring. Without damping, a vibration once excited would persist indefinitely [7]

2.3 Motion Study

To perform the FEA analysis, you must create a motion study using the motion analysis tool, to access this you must turn on the motion simulation add on from the tool bar. The disadvantage of this is that Solidworks does not tell you to do this, and you may create a basic motion study but not be able to perform the Finite Element Analysis. To someone untrained in Solidworks this could be a frustration as it is not easy to find immediately.

2.4 Understanding the Initial Model

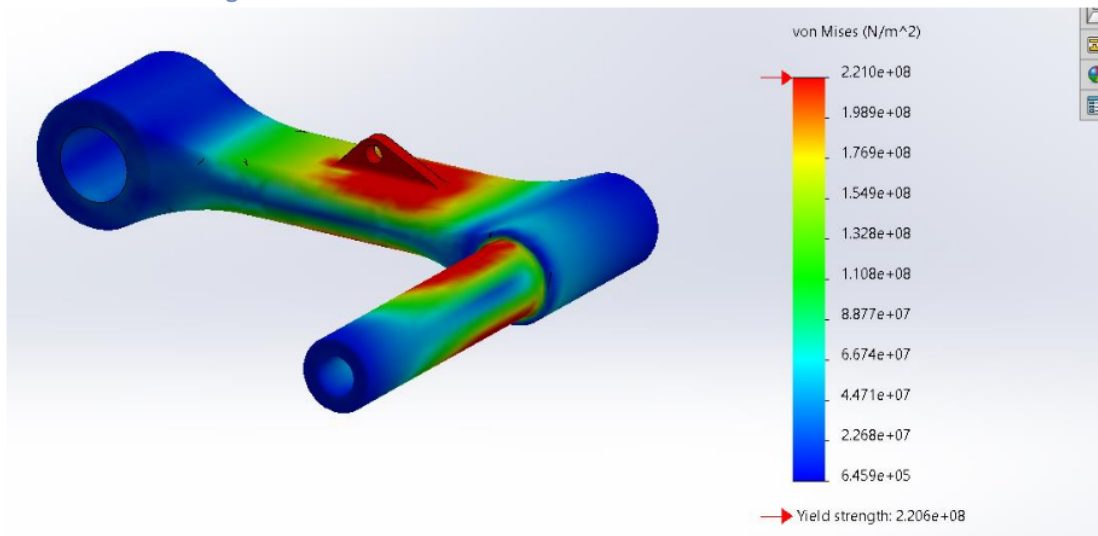


Figure 2 Initial Model

The initial Radius Arm fails on Yield Strength, factor of Safety, and has a mass of 118kg. The model showed significant weakness in two areas when put through the analysis, the first being the bracket that attaches to the lower damper. The scale shows this to be above the maximum yield strength of the material and the same goes for the 'axle' section of that connects to the wheels. The first thing to notice is the areas in dark blue are the furthest away from the yield strength, meaning material could potentially be removed from these areas without a significant change to the performance of the part. The axle section is fully hollow so an initial point of improvement would be to add material in that internal area. Some strengthening work needs to be done along the main body and particularly on the bracket. The bracket size is limited by the dimensions of the lower damper arm.

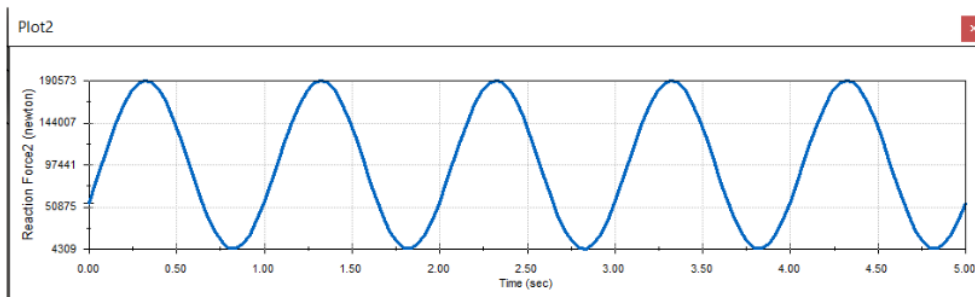


Figure 3 Reaction Force Graph

Using the motion study feature in Solidworks, a graph for reaction force between the damper and the radius arm shows where the peak stresses will be, the first wave peaks at 0.31s, this allows for easier analysis going forwards, instead of a multi frame analysis, a single frame study can be conducted freeing up some processing power.

3 Model Validation

An FEA model result should not be taken as a correct answer without validating it first. This can be done in several ways, comparison with expected results is a simple method to ensure the model has been calculated correctly [4]. A situation this would be useful for, is having an FEA model and a physical model that has been subjected to the same loads that have been simulated, for example a tensile test of a dog bone. You could simulate the test and the results should be representative of the real world results. However, in this case, a real-world simulation is not possible. Another option is comparison against hand calculations using simple mathematical theory. A cantilever beam model to work out maximum displacement can be done relatively simply in an FEA package and using hand calculations.

3.1 Hand Calculations

First, when conducting a calculation by hand, it's best practice to use SI units, therefore the figures in bold are the conversions of the given units into SI units that will be used in the calculation. The equations used below are appropriate for the type of Cantilever Beam example being used [8], with a single Force being applied to the end of a fixed beam. The material is Plain Carbon Steel, and its material properties are taken directly from Solidworks as this is representative of the military suspension system to be modelled later.

Force = **2000N**

Base = 1000mm = **1m**

Height = 1000mm = **1m**

Length = 5000mm = **5m**

Young's Modulus = 210GPa = **$210 \times 10^9 \text{ N/m}^2$**

$$I = \frac{bh^3}{12} \quad I = \frac{1 \times 1^3}{12} = 0.083m^2$$

$$\delta B = \frac{FL^3}{3EI} \quad \delta B = \frac{2000 \times 5^3}{3 \times (210 \times 10^9) \times 0.083} = 4.76 \times 10^{-6} = 0.0048mm$$

Force = 500kN = **500,000N**

Base = 1000mm = **1m**

Height = 1000mm = **1m**

Length = 5000mm = **5m**

Young's Modulus = 210GPa = **$210 \times 10^9 \text{ N/m}^2$**

$$I = \frac{bh^3}{12} \quad I = \frac{1 \times 1^3}{12} = 0.083m^2$$

$$\delta B = \frac{FL^3}{3EI} \quad \delta B = \frac{500000 \times 5^3}{3 \times (210 \times 10^9) \times 0.083} = 0.001195 = 1.20mm$$

The result of the hand calculation shows that the beam subjected to 2000N of force should have a maximum displacement of 0.0048mm. As the result is such a small number that couldn't realistically be measured easily, a second validation was done with a force of 500kN. The result of this is a maximum displacement of 1.2mm. Identical scenarios should be built in Solidworks and subjected to FEA Analysis, if the results are comparable then the model can be considered valid.

3.2 FEA

Figures 4-9 show the cantilever beam created using Solidworks, made from Plain Carbon Steel, with a force of 2000N applied in a downward direction against the end face of the beam and a fixed geometry at the other end of the beam. Each beam has been modelled using a different mesh size to see how it impacts the results and an automatic yield scale.

The variety of mesh sizes used had a negligible effect on the results for maximum displacement, the mesh's were selected using the slider bar in Solidworks, a coarse mesh at 340mm, an average mesh was 170mm in diameter, with the fine mesh coming down to 85mm. One finer mesh at 5mm was attempted as this gave a good mesh pattern on

the Radius Arm shown in Appendix B. However as seen in Figure 10, the software took over 2 hours trying to create this mesh and achieved no progress, so it was unfeasible to continue the attempt.

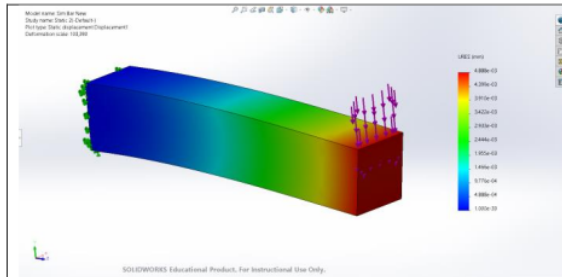


Figure 4 Cantilever Beam - 2000N - 340mm Max Displacement

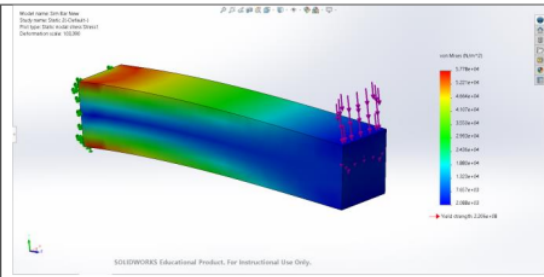


Figure 5 Cantilever Beam - 2000N - 340mm - Von Mises

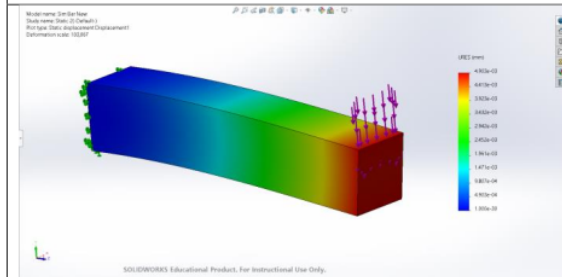


Figure 6 Cantilever Beam - 2000N - 170mm Max Displacement

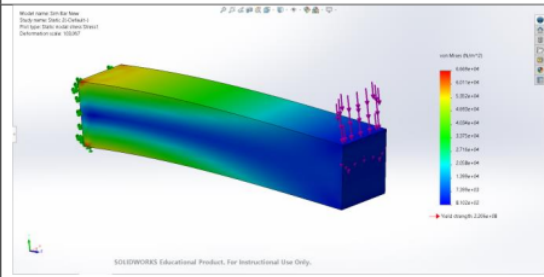


Figure 7 Cantilever Beam - 2000N - 170mm - Von Mises

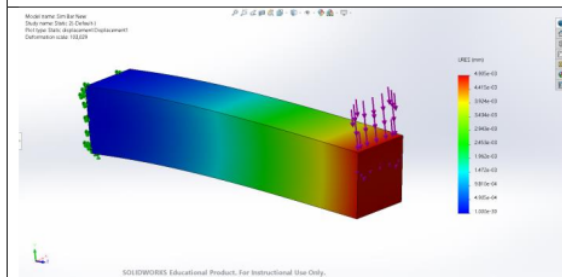


Figure 8 Cantilever Beam - 2000N - 85mm Max Displacement

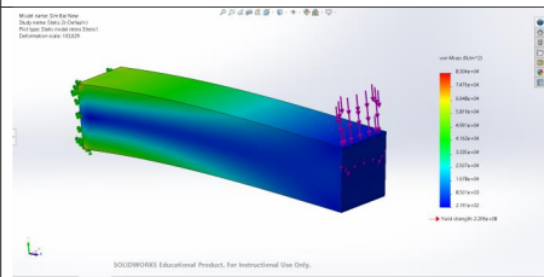


Figure 9 Cantilever Beam - 2000N - 85mm - Von Mises

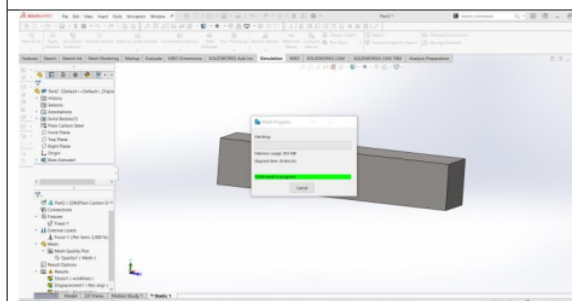


Figure 10 Cantilever Beam - Failed 5mm mesh

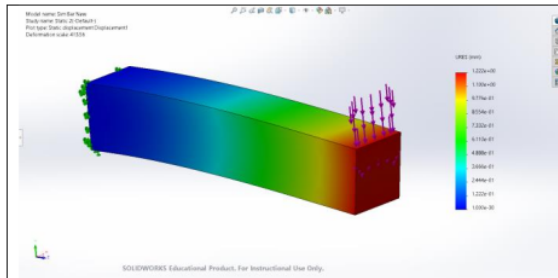


Figure 11 Cantilever Beam – 500kN - 340mm mesh - Max Displacement

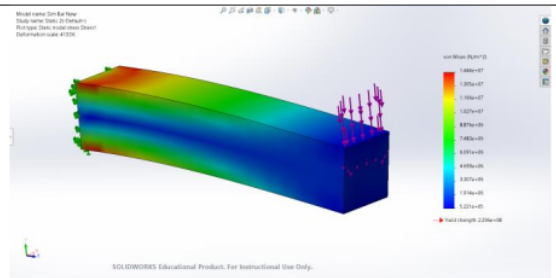


Figure 12 Cantilever Beam – 500kN - 340mm mesh - Von Mises

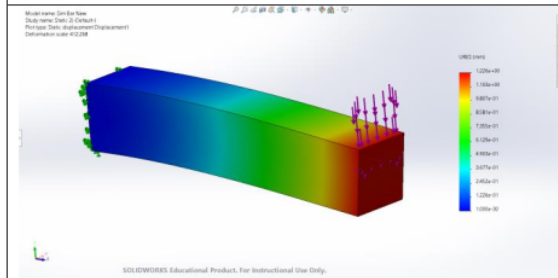


Figure 13 Figure 8 Cantilever Beam – 500kN - 170mm mesh - Max Displacement

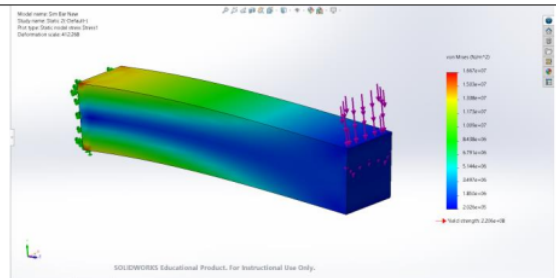


Figure 14 Cantilever Beam – 500kN - 170mm mesh - Von Mises

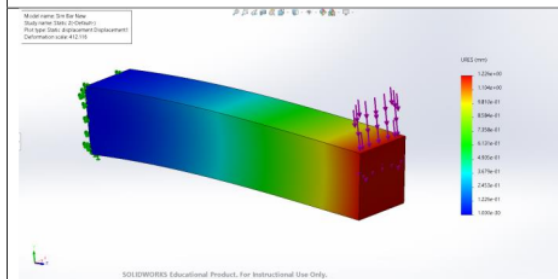


Figure 15 Figure 8 Cantilever Beam – 500kN - 85mm mesh - Max Displacement

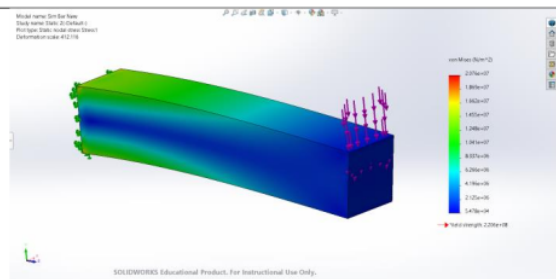


Figure 16 Cantilever Beam – 500kN - 85mm mesh - Von Mises

Mesh	Maximum Displacement @2000N	Maximum Displacement @500kN
Automatic	0.0049mm	1.226mm
170mm	0.0049mm	1.226mm
85mm	0.0049mm	1.226mm
340mm	0.0049mm	1.222mm

The results from this study show that between the 2000N cantilever beam and the respective had calculated beam scenario, there is a difference of 0.0001mm for maximum displacement, this gives confidence that the model is correct as it is such a small, almost immeasurable number. To give further confidence we can look at the separate beam problem at 500kN, the difference between the FEA software and hand calculations for this one is 0.02 to 0.06mm, although this seems like a much larger difference, in the scale of things with the fact that the force is so much higher, the author accepts this as a reasonable comparison and deems the model validated.

Whilst the Von Mises simulations won't be used for model validation, it is useful to show how different meshing affects the problem. The colour scales change dramatically between a fine and coarse mesh, this is because the smaller elements allow a higher degree of accuracy on the problem, showing more accurate stress values across the bar, while the coarse mesh calculates that the highest stress within its element is on the corner, and it applies this value across the whole element, giving a wider spread red colour.

One limitation with CAE is that you are always limited by the hardware you are using, the 5m³ cantilever beam split into 5mm elements proved too much for the computer to handle in a reasonable timescale. This could be overcome

with better hardware however this adds in a cost issue, buying the most powerful computer possible may not be suitable for all users of this software.

4 Model Redesign

Additional screenshots of model redesigns can be found in the appendices.

4.1 Model 1

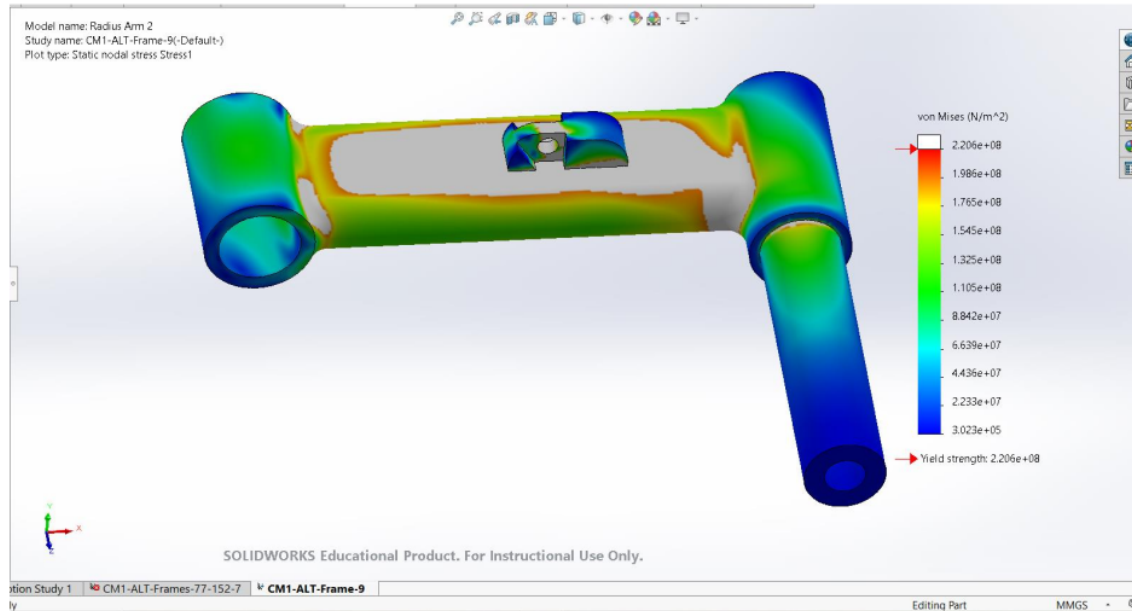


Figure 17 Model Redesign 1

The first redesign has essentially aimed to add more material around the bracket to try and stop any twisting, material has also been added into the centre of the axle. To get below the weight target of 110kg material has been taken off areas that were within their stress limits, the cylinder on the left has been thinned to the same diameter as the right-hand cylinder, and the 300mm fillet has been reduced. The thinning of the cylinder has kept within the stress limits however removing the fillet on the material has significantly increased the amount of stress across the main body and is not a feasible option.

The scale has been changed with a maximum value equivalent to the max yield strength of Plain Carbon Steel, with any value exceeding this to be shown in white, this gives a clear image of the areas that need modification.

The factor of safety will not be checked on every model, only models that show a reasonable Von Mises distribution will be checked for factor of safety as it is unnecessary to check each one if it is already showing as a significant failure.

4.2 Model 2

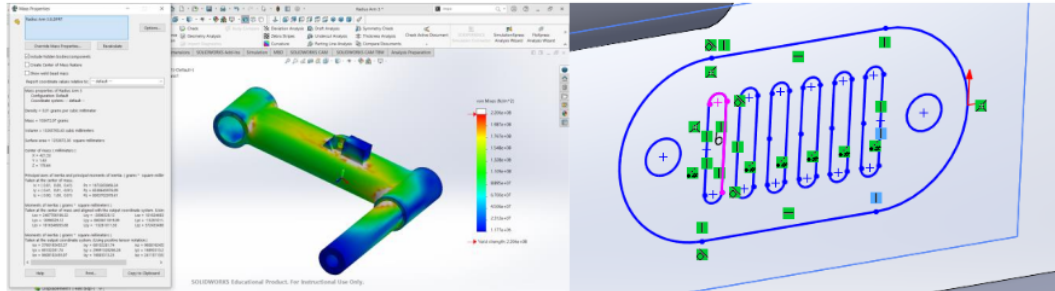


Figure 18 Model Redesign

Figure 19 Model Redesign 2 - Strengthened internals

In Model 1, the centre main structure was the main point of failure, in this model, the main structure has been thickened with an internal slot pattern cut from the material to save weight and add strength, this has significantly improved the strength of the material however the joints between the main body, the bracket and the cylinders is still failing.

4.3 Model 3

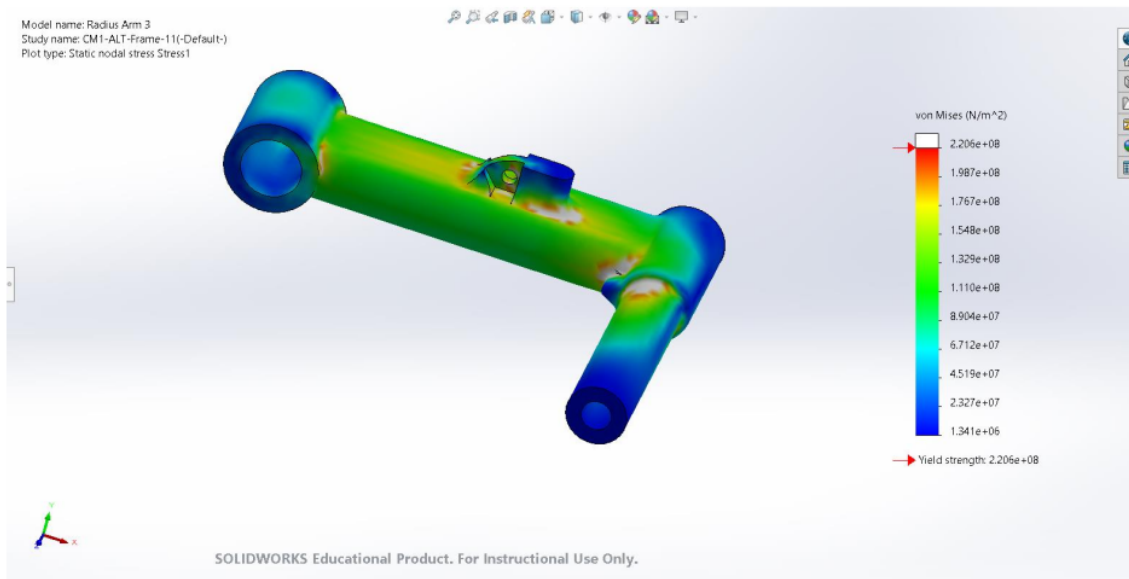


Figure 20 Model Redesign 3

A small fillet was placed around the bracket support to try and ease the stress seen at the sharp angle as it will also add some material to take the stresses, along with a supporting piece joining the main body to the axle area. Unfortunately, this hasn't remedied the issues and is visibly very similar to the previous model.



Figure 21 Model redesign 3 - Factor of Safety

Although the stresses don't look too bad, the factor of safety on this model is also far below acceptable, the white area shows any point of the material that has a factor of safety below 1.5 and it can be seen that a significant portion of the radius arm is failing.

4.4 Model 4

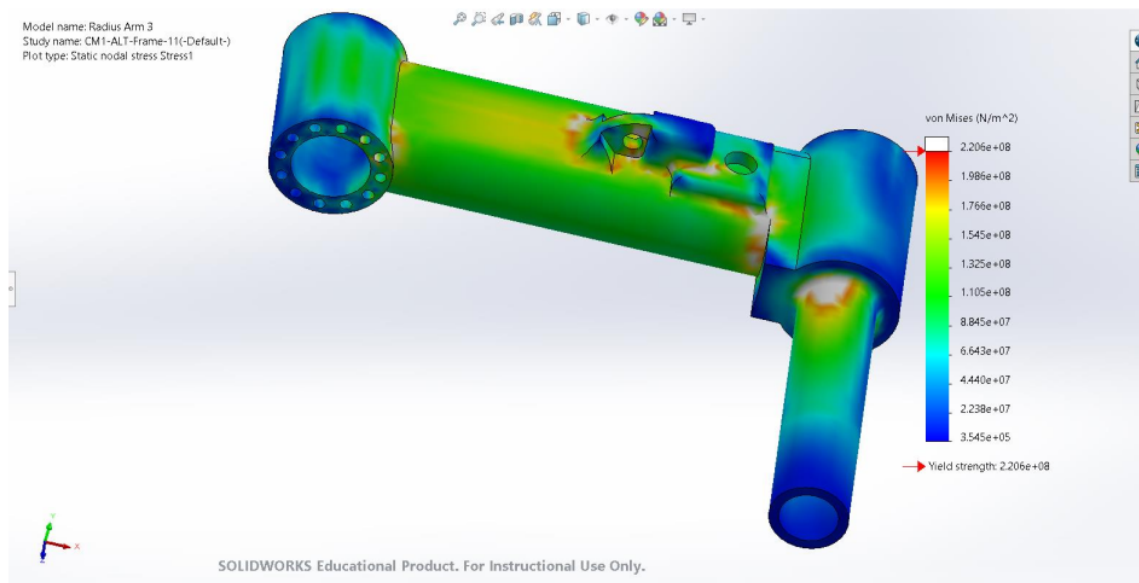


Figure 22 Model Redesign 4

The next iteration built on model 3, trying to significantly reinforce the bracket, and the area by the axle, all that has happened is the stress is localised to the joints of the new structural supports. Removing material from the internal dimension of the axle and smaller cuts through the cylinder material have got the weight down to 105kg but this is the only target that has been met.

4.5 Model 5

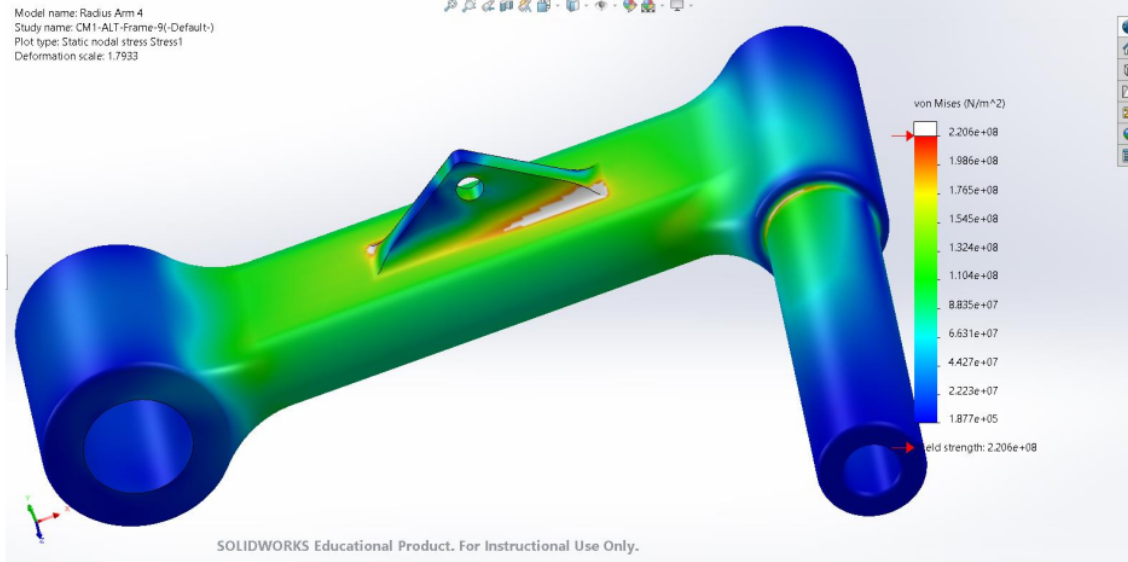


Figure 23 Model Redesign 5

This iteration moved away from previous versions and back to the triangular type of bracket along with internal reinforcement in the shape of 'I Beams'. This significantly improved the stresses seen in the main area, however there is still high stress along the edges of the bracket, the next step would be to reinforce these edges. Another issue was found in the assembly, the triangle bracket met the lower damper as it oscillated, this was only discovered using the reaction force graph and noticing a bump in the curve. Solidworks did not pick up on this and allowed the model to run even though in real life this would be a catastrophic failure.

4.6 Model 6

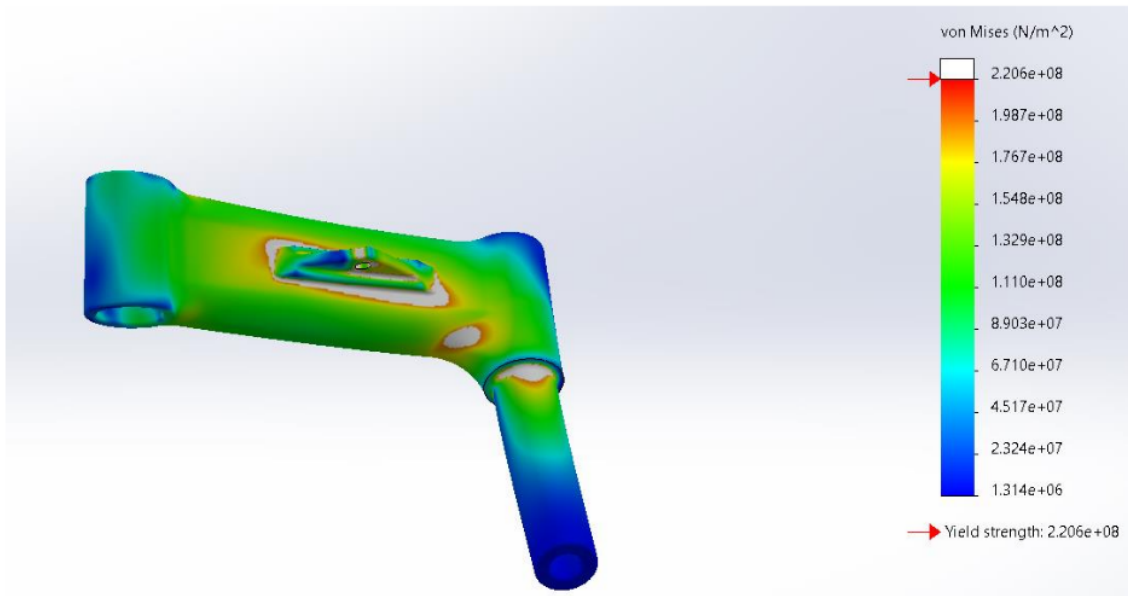


Figure 24 Model Redesign 6

Reinforcing the edges in this way has not helped the strength of the material and it has worsened it, along with making the cylinders thinner to save on weight, the radius arm is in a worse state than model 5.

4.7 Model 7

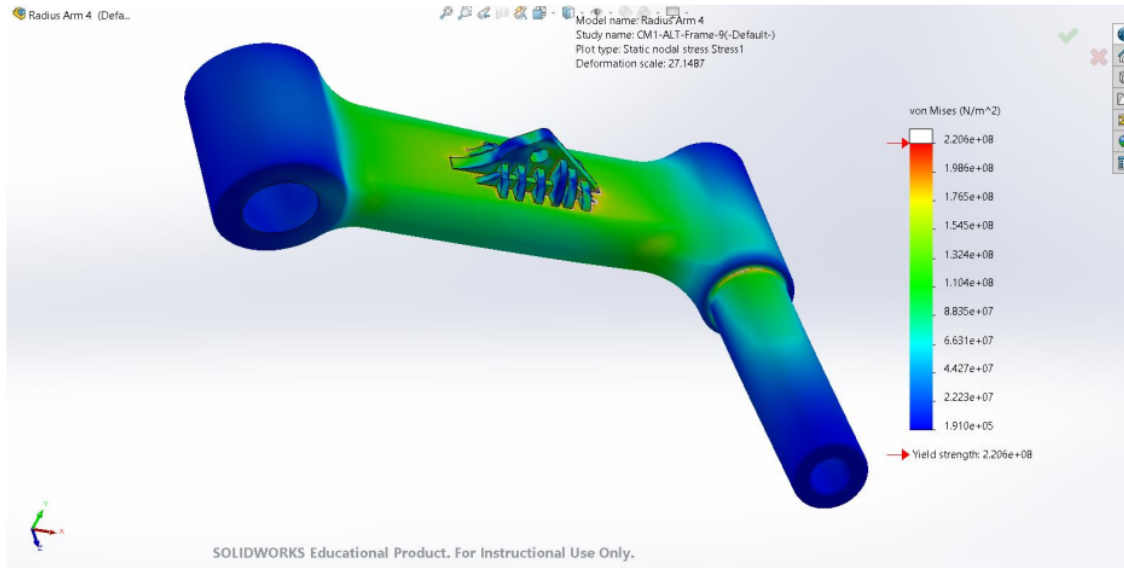


Figure 25 Model Redesign 7

Reverting to thicker cylinders, using the same internal geometry that seems to have strengthened it while saving weight, a different type of reinforcement has been used for the bracket, this seems to have significantly improved the amount of stress we are seeing in the component. The geometry of the bracket has also been modified to allow the damper to move without interference. The only areas of failure for stress are a very small area on the joint of the axle, along with the edges of the reinforcement structure for the bracket.

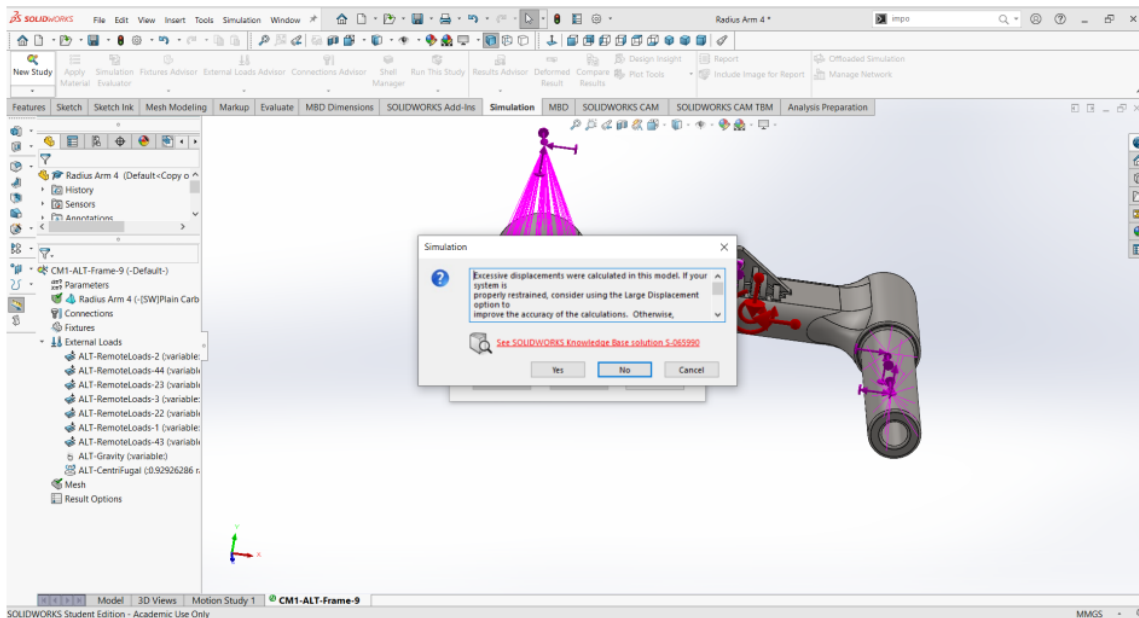


Figure 26 Model Redesign 7 - Large Displacement Issue

One limitation within Solidworks is that these messages do not give a good explanation of 'why'. It would be interesting to know why between the previous model and this one, we are now seeing excessive displacements. It states there would be improved accuracy using a large displacement option however when trying this, the model failed to calculate.

An improvement would be to highlight the point in the model that it thinks has the large displacement to learn more from geometry that works and geometry that does not work.

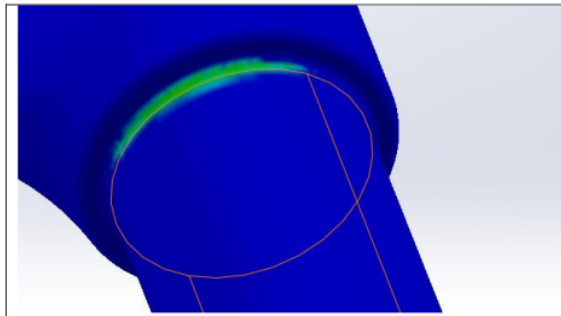


Figure 27 Model Redesign 7 Factor of Safety - Axle

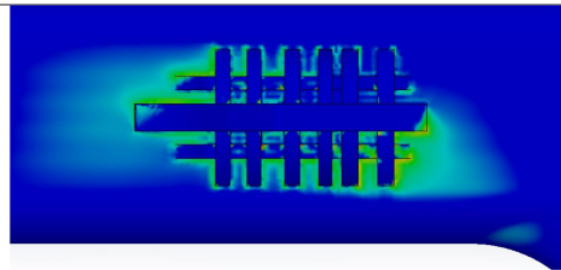


Figure 28 Model Redesign 7 Factor of Safety - Bracket

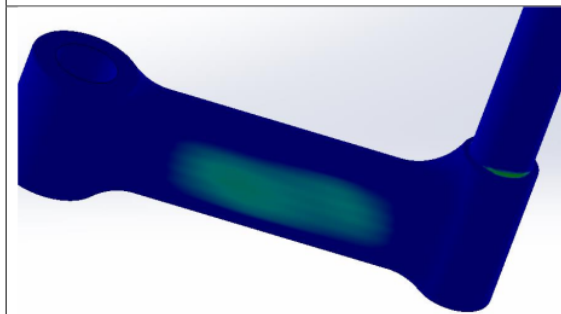


Figure 29 Model Redesign 7 Factor of Safety - Main Body

Figures 27 - 29 show the factor of safety for this model, a factor of safety value was set at 1.5 to give a clear picture of failures, anything that isn't the solid dark blue is under the factor of safety requirement, these mirror where the failures of the material due to stress are. The area in figure 27 is minimal in both FOS failures and stress failures, this is a difficult area to reinforce as it is restricted in geometry by the mate with the wheel rim. To strengthen this area, you need to modify the thickness of the axle, in turn modifying the diameter of the wheel 'hub'. Another option would be to change the material however for the purpose of this study, everything is attempted with Plain Carbon Steel to determine if it is possible to meet the targets.

4.8 Model 8

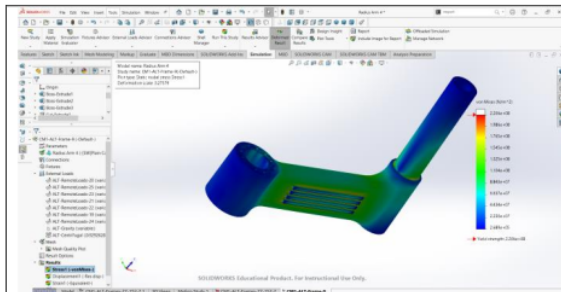


Figure 30 Model Redesign 8

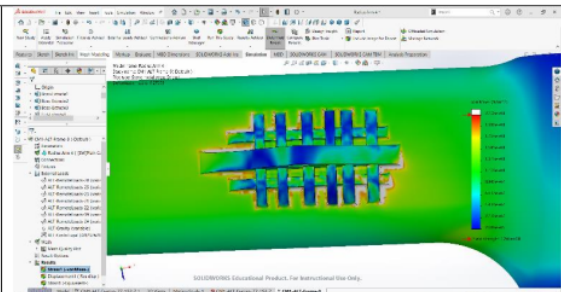


Figure 31 Model Redesign 8 - Bracket

Material has been removed once again from the left-hand cylinder to save weight, and in place it was put onto the lower section of the main structure to try and improve where the factor of safety failed, a fillet was also added to the bracket section to try and ease the stress, but it had no noticeable effect on either. Unfortunately, due to a crash of the software, a factor of safety wasn't captured, however, it was almost identical to the previous model and as such, model 7 will be the model to go conclude the redesign attempts. Crashes and corruption are significant downfalls of CAE that is hoped to be avoided but is possible at any moment [9].

5 Conclusion

5.1 Model Improvements

In conclusion, it has not been possible to redesign the Radius Arm to meet the performance targets, however it is extremely close. A real-world recommendation for the Radius Arm would primarily be to choose a different material, there are lighter and stronger materials available, Titanium would be a good option however that would come down to the cost restrictions for the real-world platform. The suspension system here isn't clear whether it is an active or semi active suspension unit for a military vehicle, semi active uses control algorithms and an active damper that consumes virtually no power and it generally improves ride quality [10], this would be an option if the redesign could've been extended to the rest of the assembly and it may alter the stresses seen on the Radius Arm. Another option would be to simply change the width of the mate with the lower damper, this is a very thin section of metal that could be strengthened by being thicker.

5.2 CAE limitations

The first limitation with any CAE software is that it has a high associated cost, not only the purchase price but the hidden costs of training and the supporting hardware/maintenance [9]. This is exaggerated if the CAE software being used isn't capable of complex FEA some software needs to marry up with other software for certain features [9], then you'll need an additional license for an FEA software such as ABAQUS. A commercial ABAQUS License costs roughly \$20,000 [11]. This also comes down to the skill of the user, a standard design engineer should be capable of simple FEA tasks but industries like Formula 1 would have dedicated FEA specialists for complex scenarios [6]. CAE can also be affected by idealisation errors, these are physical errors with the CAD model that cause the analysis side of things to be incorrect, although the FEA itself might be accurate [6], the model lets it down.

5.3 Solidworks limitations

A significant limitation is the hardware needed to run CAE software, there are memory issues, failed simulations or simulations that at least take too long to be processed on a laptop. Along with the hardware needed, the author was unable to use an external mouse towards the closing stages of analysis which made it virtually impossible to manoeuvre the model as desired.

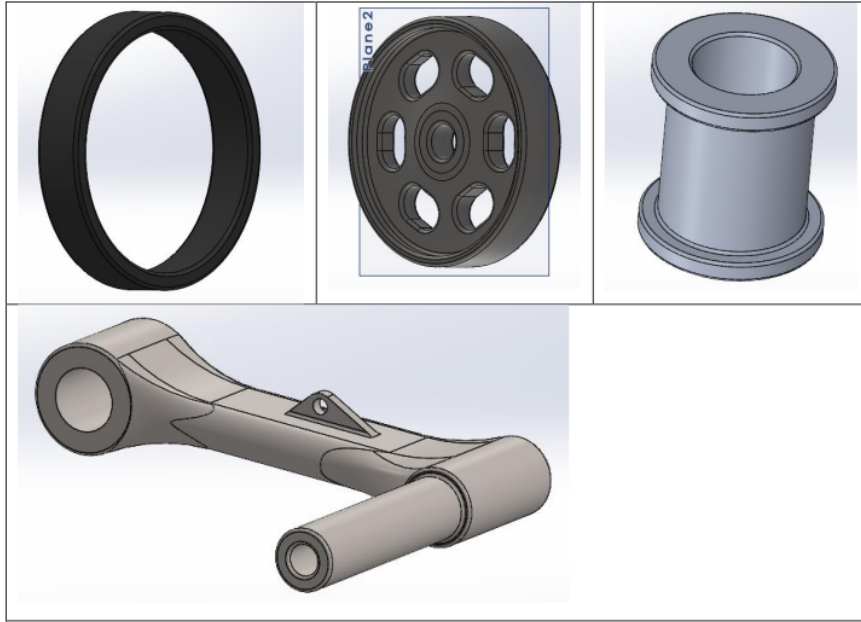
The motion study also came with its own challenges, when assemblies had been rebuilt or slightly modified, they would sometimes fail in ways that shouldn't be possible, despite having mates in place that restrict the movement of the model, the components would move to random locations and the movement would not be as it's supposed to. There seems to be no explanation for this and the only way to remedy it is to rebuild the assembly from scratch. This made it very difficult to input each new radius arm to the assembly as they would often crash in one way or another.

Certain parts of the components are reliant on each other, the bracket was reliant on the main body section being present, for each modification of the main body, the bracket would be deleted, as the bracket is one of the more difficult parts to model, it is extremely time consuming to re-design it each time you need to structurally change the centre piece. It may be possible to isolate the bracket to a new plane, but when this was attempted, the change in thickness of the centre section made it unsuitable to use the bracket anyway.

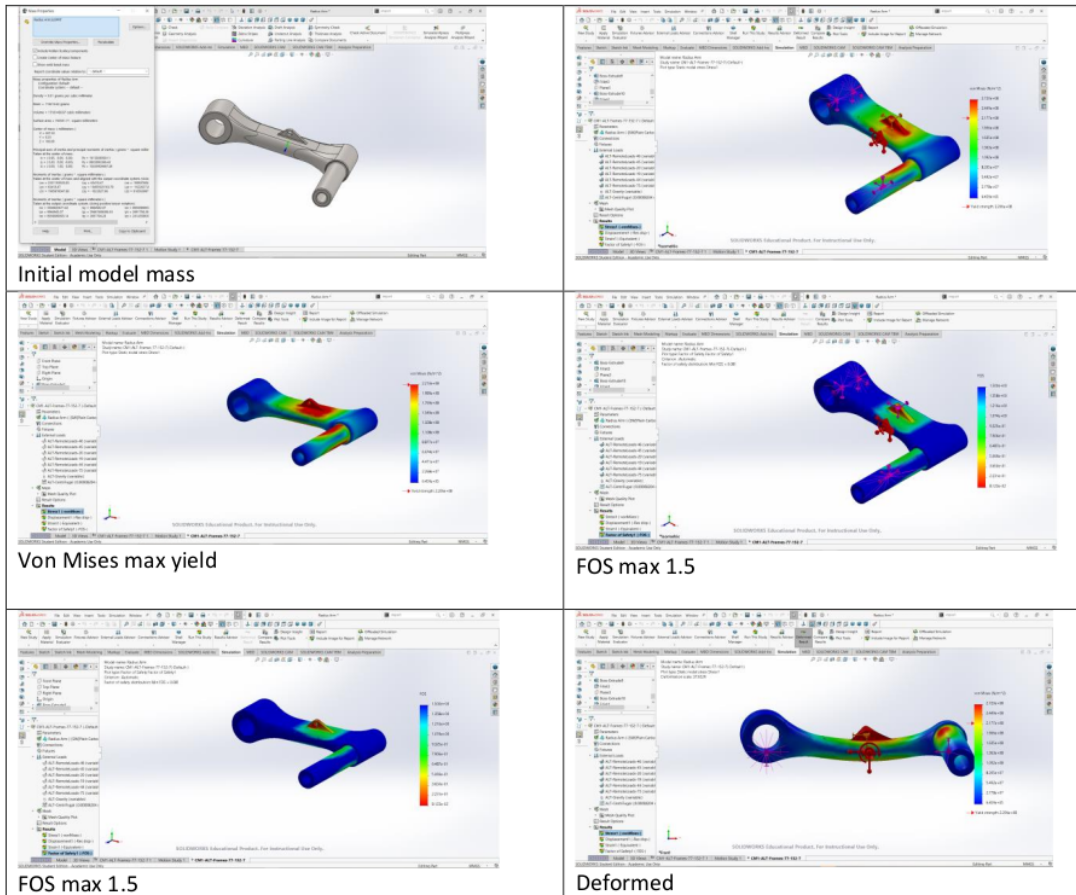
6 References

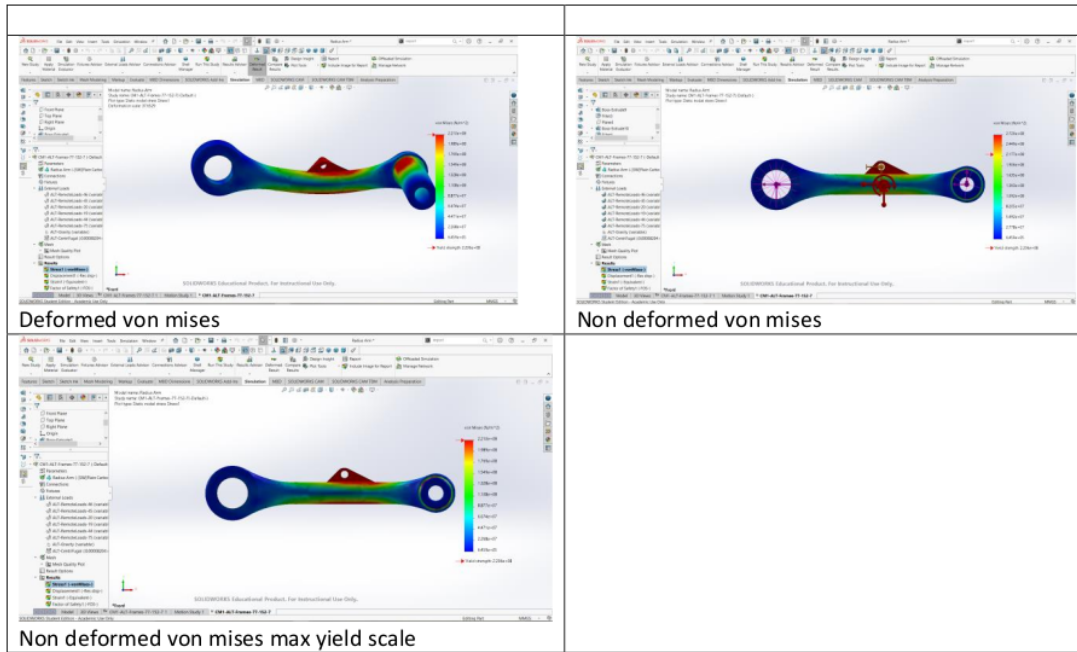
- [1] D. Bowman and A. Bowman, Understanding CAD/CAM, Indianapolis: Howard W. Sams & Co., 1987.
- [2] Michigan State University, "What is the Difference between CAD, CAE and CAM?," April 2021. [Online]. Available: <https://online.egr.msu.edu/articles/cad-vs-cae-vs-cam-what-is-the-difference/>. [Accessed March 2022].
- [3] K. Lee, Principles of CAD/CAM/CAE Systems, Seoul: Addison Wesley, 1999.
- [4] B. MacDonald, Practical Stress Analysis with Finite Elements, Dublin: Glasnevin Publishing, 2013.
- [5] R. Crawford and E. Al, Mechanics of Engineering Materials, Harlow: Longman Group Limited, 1996.
- [6] P. Kurowski, Engineering Analysis with Solidworks Simulation 2013, Mission: SDC Publications, 2013.
- [7] D. Bastow, Car Suspension and Handling, Warrendale: Professional Engineering Publishing, 1993.
- [8] A. Noble, "Validation of Computational Models," UWTSD, Swansea, 2022.
- [9] P. Jones, CAD/CAM Features, Applications & Management, Basingstoke: The Macmillan Press, 1993.
- [10] L. Miller, "The Design and Development of a Semiactive Suspension for a Military Tank," *SAE 1984 transactions*, vol. 97, pp. 347-355, 1988.
- [11] K. Donne, Interviewee, *OME Physics*. [Interview]. October 2018.

7 Appendix A – Drawn Components

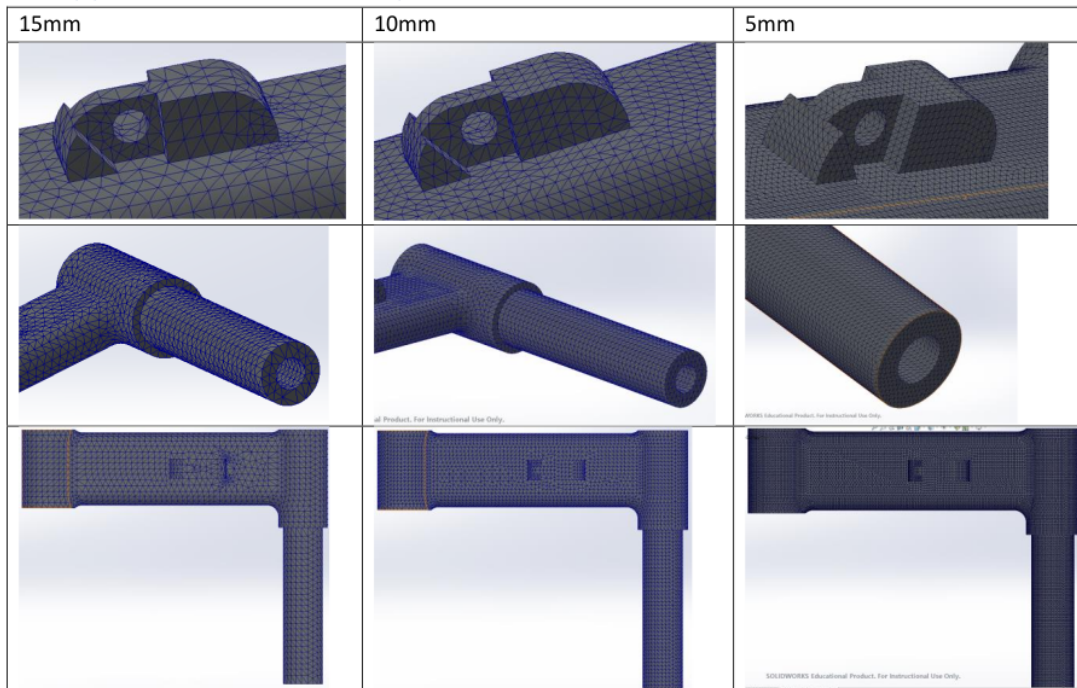


8 Appendix B – Initial Model

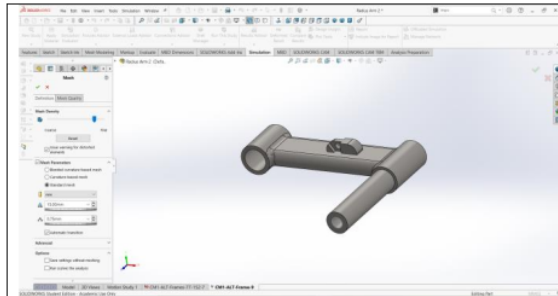




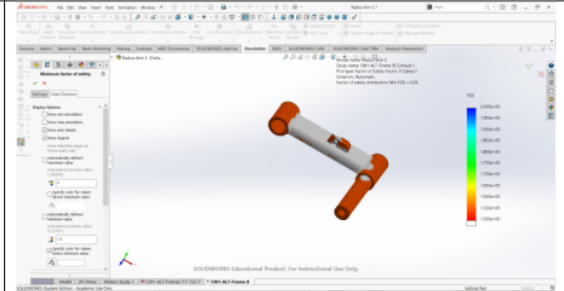
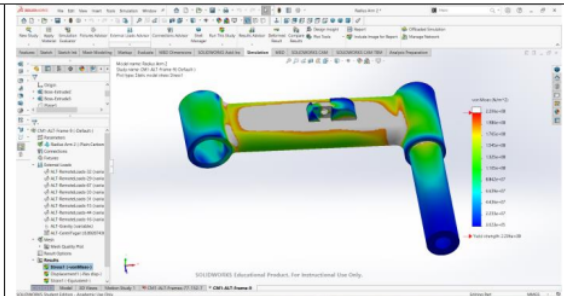
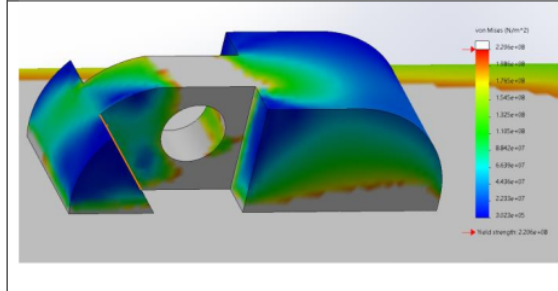
9 Appendix C – Mesh Comparison



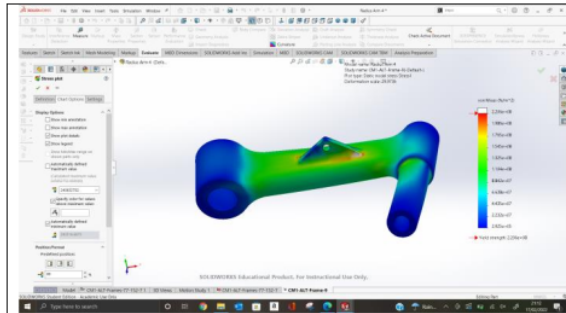
10 Appendix D – Model 2



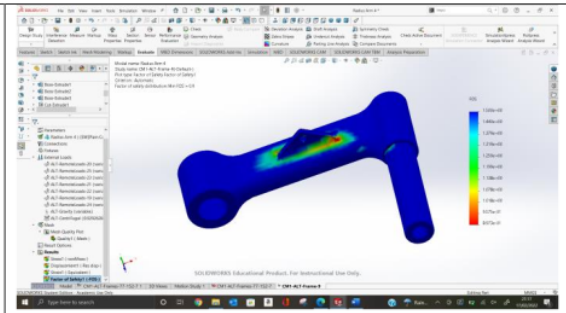
Model 2



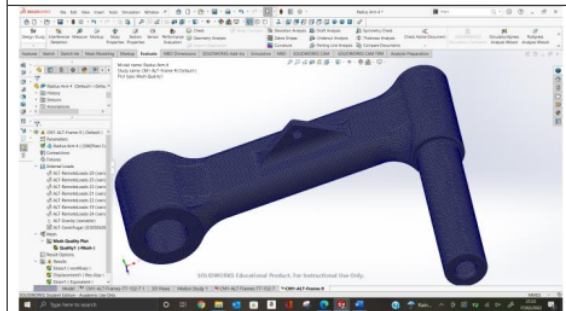
11 Appendix E – Model 5



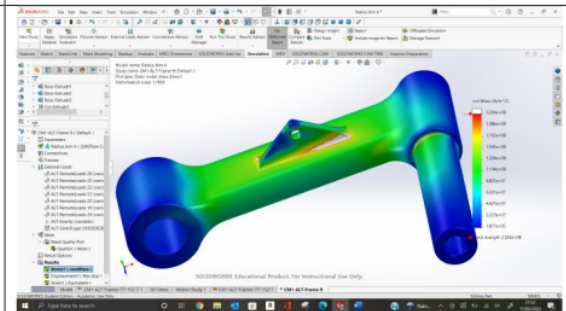
Von Mises auto mesh



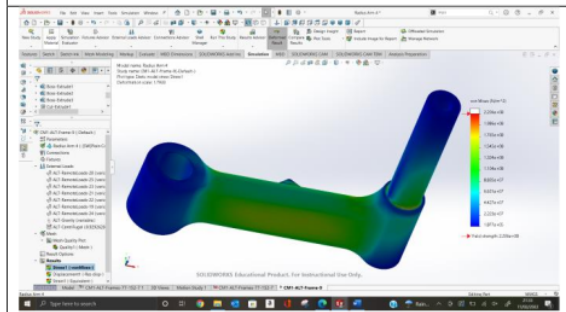
FOS max 1.5 auto mesh



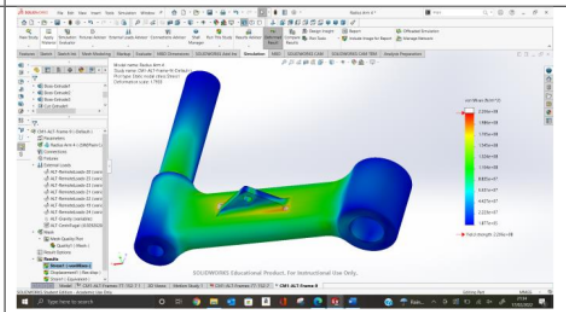
5mm mesh



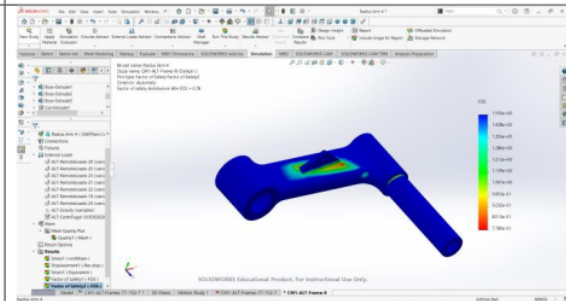
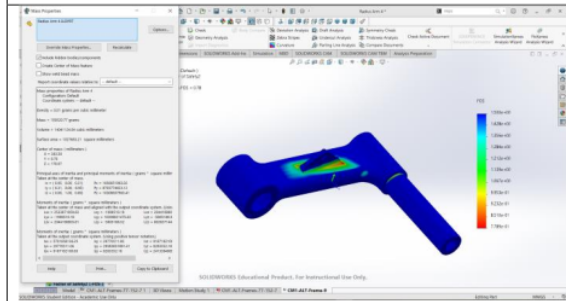
Von Mises 5mm mesh



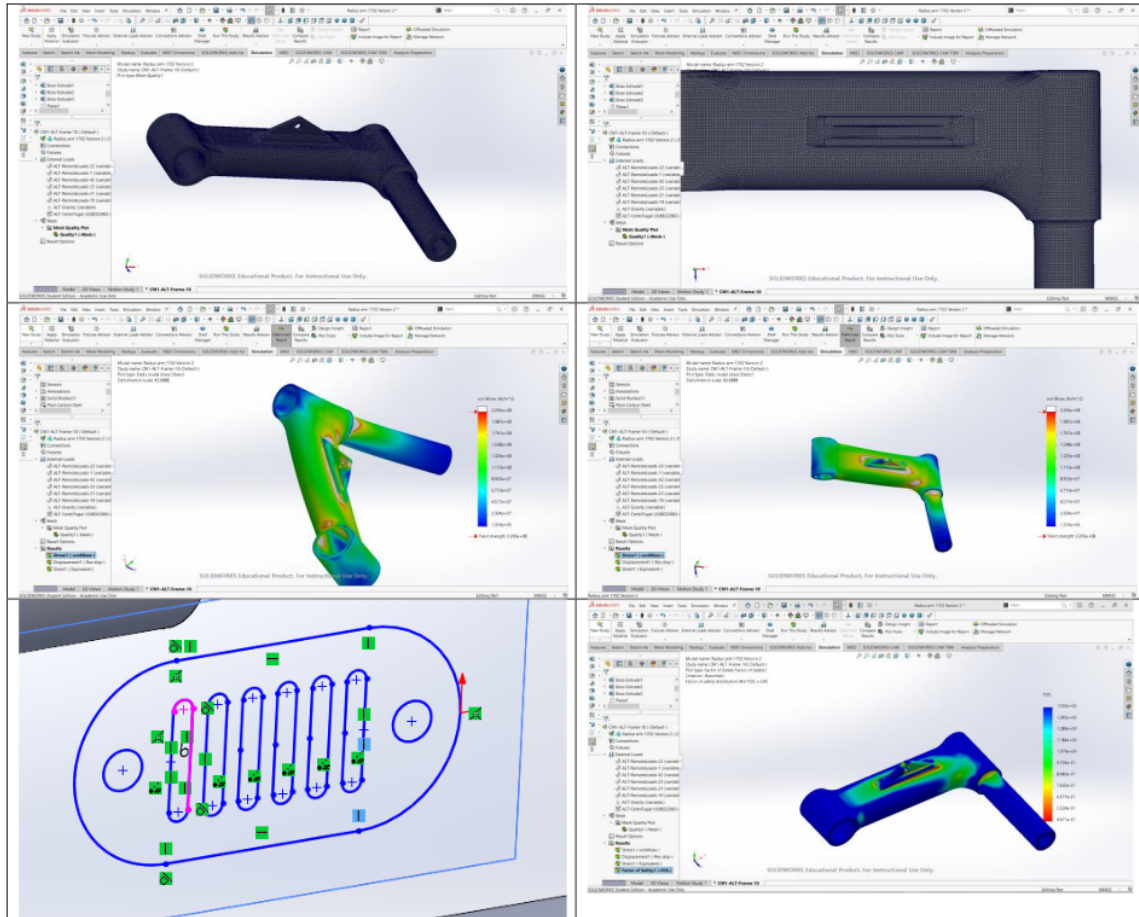
Von Mises 5mm



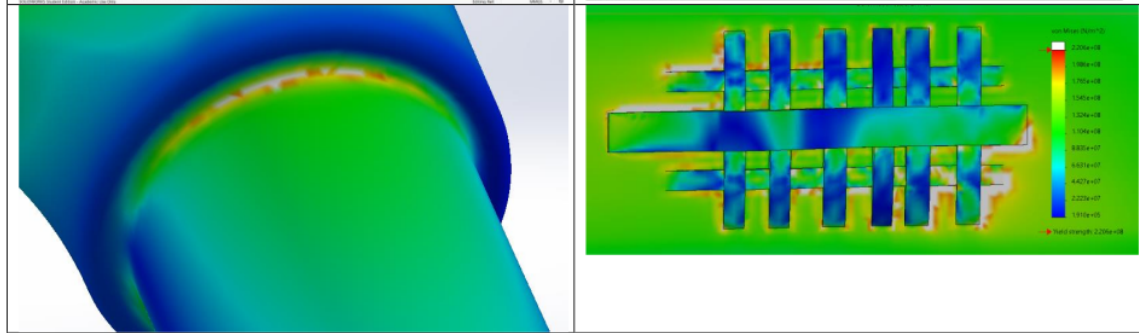
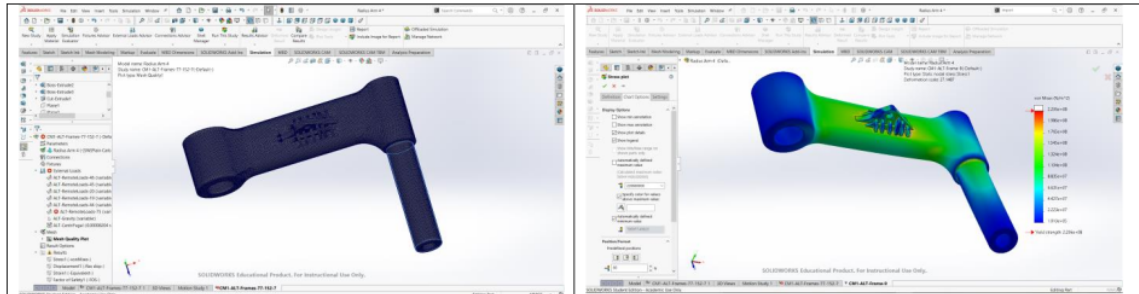
Von Mises 5mm



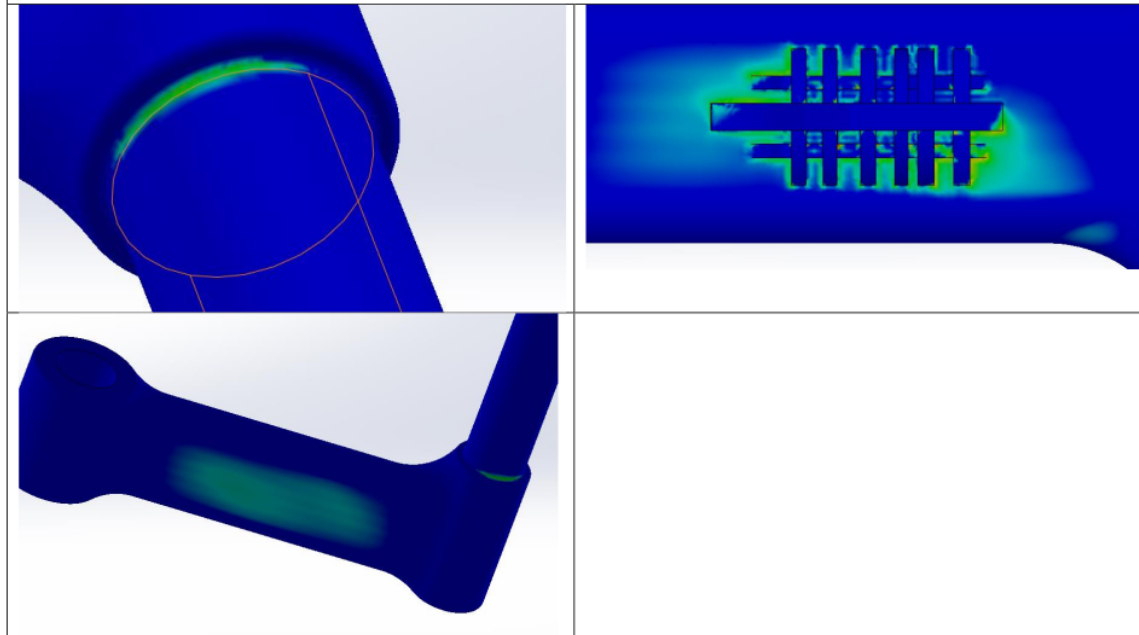
12 Appendix F – Model 6



13 Appendix G – Model 7



FOS



FINAL GRADE

71 / 100

GENERAL COMMENTS

Instructor

Your use of literature throughout the report is good, you have consulted a good range of sources and embedded them well through the document in support of your work. There could have been more sources used in some areas to create a critical and comparative discussion.

You show a good fundamental understanding of the different sub-divisions of CAE and their industrial applications as well as noting some of the limitations of implementing this type of approach for engineering problems. Your validation works are good to see, although it would have been nice to have seen more information given on the mesh study that was conducted, for example more information about mesh independence could have been shown through a more varied study of mesh sizing's.

You have shown some information to the reader about the study being conducted here, with the assembly shown and seemingly accurate as well as clear demonstration of the boundary conditions to be applied. You could have gone further and supplemented this with a screenshot of the simulation setup that showed the boundary conditions applied to the assembly to make it even more clear to the reader.

Your iterative design approach was good to see, you have presented the results and changes made clearly. Good that you have considered aspects such as the colour bar scales needing to be consistent etc. Each design is explained to a reasonable level, and it is clear that you have made effort to analyse the previous design to inform the following iteration. Some of the presentation of results was a bit inconsistent unfortunately, especially with the final iterations

where the screen shots were either lacking the colour bars, or the screenshots could have been more focused on the main viewer window to make the information clearer to the reader.

Your summary of the work is decent, with some valid points raised in your conclusion. Your sections noting the limitations of the software have some good information contained in them, but could have been a bit more critical in their analysis of the problem investigated. Overall though, a good effort here.

PAGE 1

PAGE 2

PAGE 3

PAGE 4

PAGE 5

PAGE 6

PAGE 7

PAGE 8

PAGE 9

PAGE 10

PAGE 11

PAGE 12

PAGE 13

PAGE 14

PAGE 15

PAGE 16

PAGE 17
