

COMPUTER AIDED ENGINEERING

Structural Dynamics (ACHA6010)



Prifysgol Cymru

Y Drindod Dewi Sant

University of Wales

Trinity Saint David

Nathanael Potts

12/2022

Abstract

Within this report, the use of computer aided engineering will be discussed and critically analysed while demonstrating how different software's can inform and help develop improved designs. Validation cases are discussed and investigated to prove the accuracy of the methods being used within Ansys and how this can give engineers confidence moving forward with each design knowing that the results being taken can be trusted.

Contents

Abstract.....	1
Introduction	2
1. Literature Review	2
2. Methodology and Validation.....	3
2.1. – Structural Validation Case	3
2.1.1. – Analytical Solution.....	3
2.1.2. – Ansys Simulation	4
2.2. – Thermal Validation Case.....	5
2.2.1. – Analytical solution.....	5
2.2.2. – Ansys Simulation	5
3. Front Wing Mounts Analysis	6
3.1. Front Wing Mounts CAD assembly	7
3.1.1. Aerodynamic Analysis of Front Wing	7
3.1.2. CAD Assembly of Front Wing	8
3.2. FEA Analysis.....	8
3.2.1. Engineering data	8
3.2.2. Geometry Setup	9
3.2.3. Mesh	9
3.2.4. Boundary Conditions.....	10
3.2.5. Results.....	10
4. Wing Mounts Development.....	12
4.1. CAD Model Changes.....	12
4.2. FEA Test	13
5. Conclusion.....	15
Bibliography	15
Appendix	16
1. Mount Component Drawings.....	16
2. New component technical drawings.....	18

Introduction

This report will focus on the use of Computer Aided Engineering and how it benefits the engineering industry. This will be demonstrated by analysing a SYSTEM which was designed in Solidworks CAD package and then structural and dynamically tested using Finite Element Analysis on Ansys. This system was then developed and tested again to show how quick it can be to design and develop a system using Computer Aided Engineering.

Two validation cases were conducted to investigate the accuracy of Finite Element Analysis using Ansys. Within the validation cases, mesh sizing was investigated also to find how finer or coarser meshes effect the simulation results. The validation cases simulated on Ansys were compared to an analytical solution to find the percentage error between both cases. The aim is to get as close as possible to the analytical solution so that the software and methods can be validated.

The second part of this report will focus on the design and development of a front wing mount for a hill climb car. The current wing mounts used on the hill climb car are old and made up of two parts bolted together so the team have asked for new wing mounts which are simplified and lighter than the old ones. Using the existing mounts as a baseline, new components will be developed and tested within FEA to investigate the comparison between the old and newly developed mounts.

1. Literature Review

Computer Aided Engineering (CAE) is the use of computer software to design a component or assembly from prototype designing to simulation testing (PTC, 2022). CAE is widely used within the engineering industry to assist the product development by reducing the costs and time as well as the product durability. Siemens gives a brief overview of a few of the many applications of CAE (Siemens, 2016):

- Finite Element Analysis to analyse the stresses and dynamics of a component
- Computational Fluid Dynamics focusing on Thermal and Fluid Analysis
- Multibody Dynamics simulating the kinematic and dynamic of mechanisms
- Boundary Element Method looking at the Acoustics – can also be done through FEA
- Mechanical Event Simulation
- Control System Analysis

CAE is a powerful tool that enables engineers to gain knowledge in the early stage of development so they can make design changes reducing the cost of development (Bi, 2018). This is because the performance of a component or assembly can be simulated so that component/assembly can be further developed based off of the simulation results. When running simulations of a component/assembly, CAE offers visualization of how the component/assembly will react under a given condition (Urban, 2019). This allows the engineers to critically analyse the simulation results to understand how the component/assembly can be developed to withstand conditions but to also improve the overall performance.

The focus within this report will be mainly the use of Finite Element Analysis. (English, 2019) describes Finite Element Analysis (FEA) as the “simulation of a physical phenomenon using a numerical mathematic technique” used widely in the engineering industry. FEA is a very useful tool for developing prototype designs before physically manufacturing and testing them. FEA applications are now normally integrated withing CAD programs, such as Solidworks, to helping engineers to go from detailed designs and structurally, thermally analyse them quicker (Dunkley, 2021). To validate an FEA solution practically, the solutions results would be compared to the component physically

loaded with the same values and boundary conditions and tested in the real world in a controlled environment (Srinivas, 2020). In this report however, the focus will be on validating the results through an analytical solution.

2. Methodology and Validation

Validating results is a critical part of the process as this will tell you whether your results are representable. If the results aren't validated, then an engineer won't know whether the component/assembly design will work the way they hope it would. Validating results can be done either by an analytical solution or through a practical validation (McCaslin, 2021).

2.1. – Structural Validation Case

To know that the results from the simulations on the system designed will be reliable, a validation case was conducted to understand and compare the percentage error from Ansys. To do this, an analytical method was used using a cantilever beam. All the numbers including length, young's modulus and other parameters were kept consistent throughout the case and within Ansys.

2.1.1. – Analytical Solution

The analytical method chosen to be used is on a cantilever beam and adding a point load to one end. The screenshot below represents what type of cantilever beam the analytical solution will be based off.

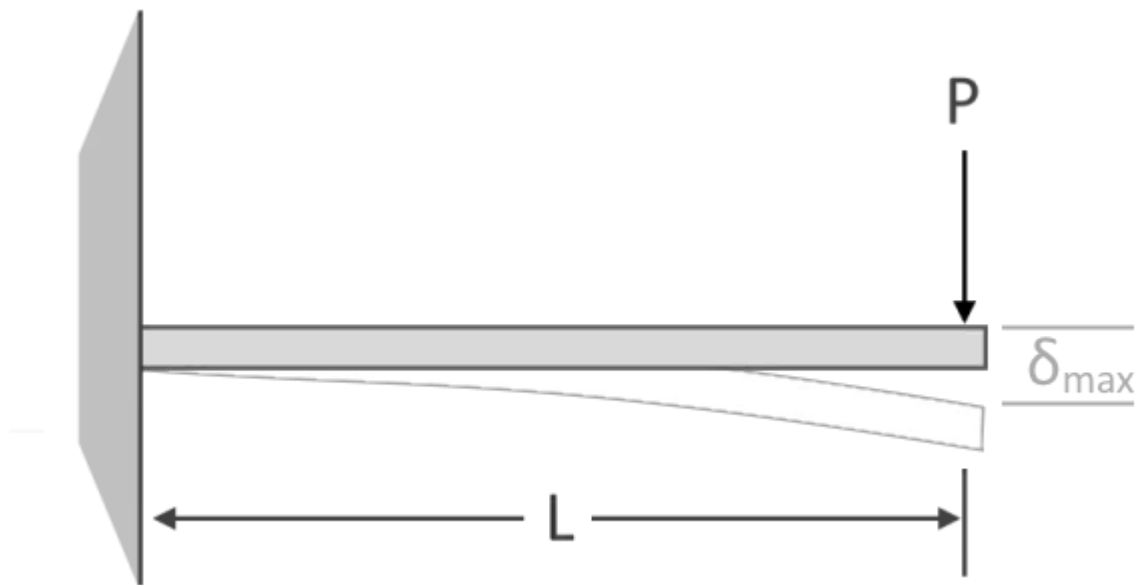


Figure 1 - Screenshot of cantilever beam (Alambra, 2022)

The first step of the analytical solution is to work out the cross section of the cantilever beam. These dimensions are kept consistent throughout Ansys to make sure the results gathered are consistent. The first formula is for the cross sectional area of the cantilever beam:

$$I = \frac{wh^3}{12} \quad 1.67 * 10^3 mm^4 = \frac{20 * 10^3}{12}$$

Equation 1 - Cross sectional area of Cantilever Beam

The next step is to work out the displacement of the beam which we can then compare to the results taken in Ansys. The young's modulus for Stainless steel 304 data was taken from (World Material,

n.d.) and then used in the formula above to generate an analytical solution that can be used when validating the results from Ansys. The formula for this cantilever beam is (Alambra, 2022):

$$Deflection = \frac{FL^3}{3EI} \quad 131.3mm = \frac{1000 \cdot 500^3}{3 \cdot (190 \cdot 10^3) \cdot (1.67 \cdot 10^3)}$$

Equation 2 - Cantilever Beam Formula

The deflection calculated from a steel cantilever beam was 131.3mm which can then be compared to the FEA results taken within Ansys.

2.1.2. – Ansys Simulation

The cantilever beam was designed within Solidworks to the same dimensions specified in the analytical solution in equation 1 and 2. The cantilever part was then imported into Ansys to setup the simulation to compare the results.

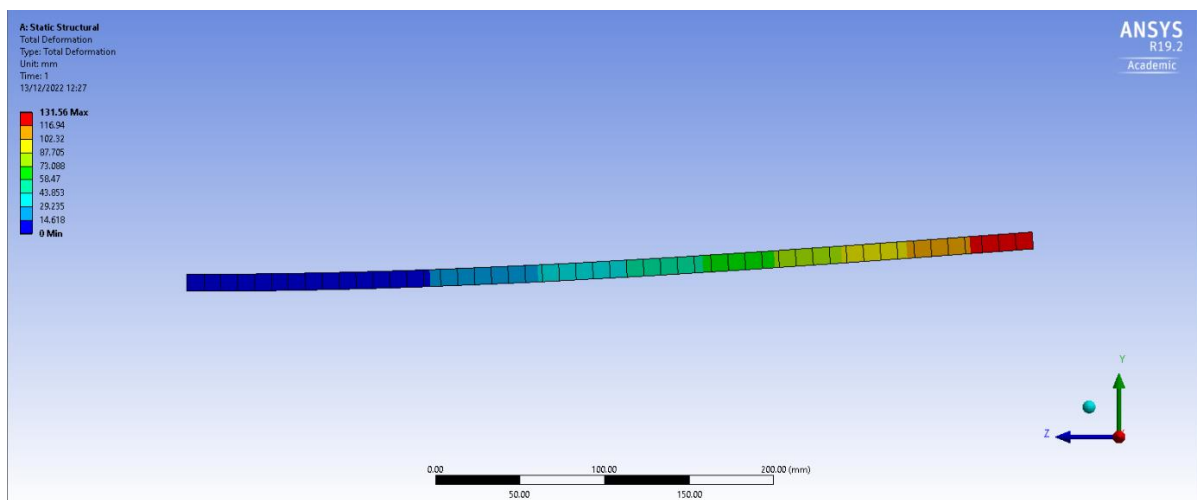


Figure 2 - Ansys Simulation showing the displacement of the cantilever beam

In figure 2 above, we can see the cantilever beam put under a load of 1000N to find how much displacement would occur. With each simulation, the mesh size was slightly adjusted to see how the results would be effected.

The table below demonstrates how the mesh sizing effects the displacement and the percentage error. The analytical solution displacement was 131.3mm which means that the results taken from Ansys are very representable being 0.2/0.3mm out. This demonstrates the power and accuracy of the Ansys software.

Table 1 - Results from cantilever beam displacement simulation in Ansys

Ansys		
Mesh size	Displacement	Percentage error
15	131.5	100.1523229
10	131.6	100.2284844
5	131.6	100.2284844
1	131.6	100.2284844

2.2. – Thermal Validation Case

The second validation case will be looking at a thermal study and validating a case against an analytical solution. For this we're going to use the Bimetallic strip which consists of 2 metals bonded together and heated up to see what happens. Like the previous validation case, all the parameters will be kept the same throughout the analytical solution and simulation to maintain consistency.

2.2.1. – Analytical solution

To find the analytical solution for the bimetallic strip, we need to use the following formulas. The formula will include both materials properties. The first formula finds the coefficient K1 which is dimensionless and can be used for the calculation of the equivalent stiffness:

$$K_1 = 4 + 6 \frac{t_c}{t_i} + 4 \left(\frac{t_c}{t_i} \right)^2 + \frac{E_c}{E_i} \left(\frac{t_c}{t_i} \right)^3 + \frac{E_i t_i}{E_c t_c}$$

$$16.003 = 4 + 6 \frac{1}{1} + 4 \left(\frac{1}{1} \right)^2 + \frac{(130 \cdot 10^3)}{(137 \cdot 10^3)} \left(\frac{1}{1} \right)^3 + \frac{(137 \cdot 10^3) \cdot 1}{(130 \cdot 10^3) \cdot 1}$$

Equation 3 – formula to find the coefficient used for the next calculation

$$\text{Total Deformation} = \frac{6(\alpha_i - \alpha_c)(t_i + t_c)T_c L^2}{t_s^2 K_1 2}$$

$$-7.049\text{mm} = \frac{6 \cdot ((1.2 \cdot 10^{-6}) - (2 \cdot 10^{-5})) \cdot (1 + 1) \cdot 100 \cdot 100^2}{1^2 \cdot 16.003 \cdot 2}$$

Equation 4 - Total deformation equation

2.2.2. – Ansys Simulation

From the results shown in the graph below, we can see that the coarser the mesh, the more the displacement varies. In-between a mesh size of 1mm and 0.5mm the results level out, giving us a bit of convergence but then drop off at 0.1mm. This could be because the mesh size is now too fine. The total deformation difference between the analytical solution and Ansys simulation is around 0.08mm. This again shows that the Ansys simulation is very accurate and can be trusted in further calculations.

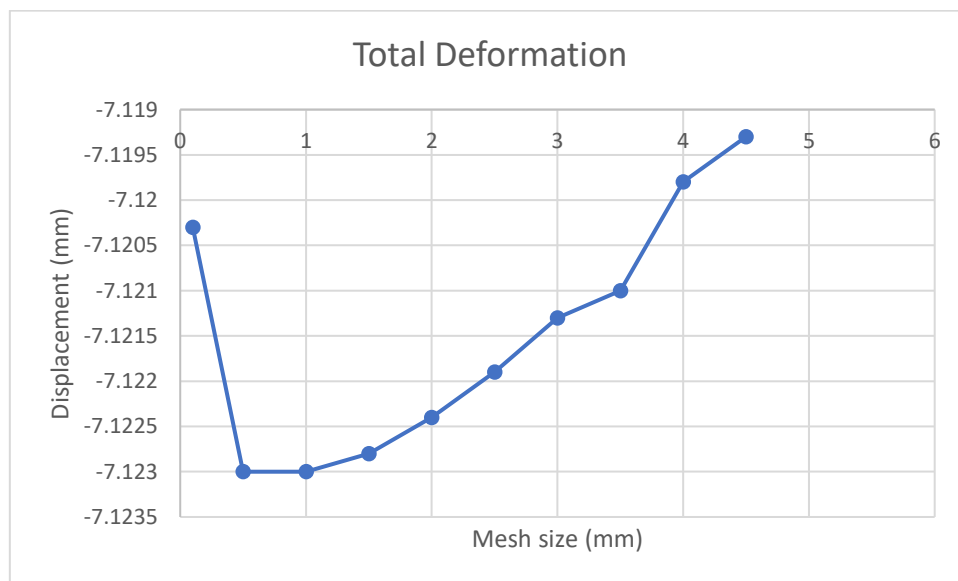


Figure 3 - Total deformation vs mesh sizing for thermal study

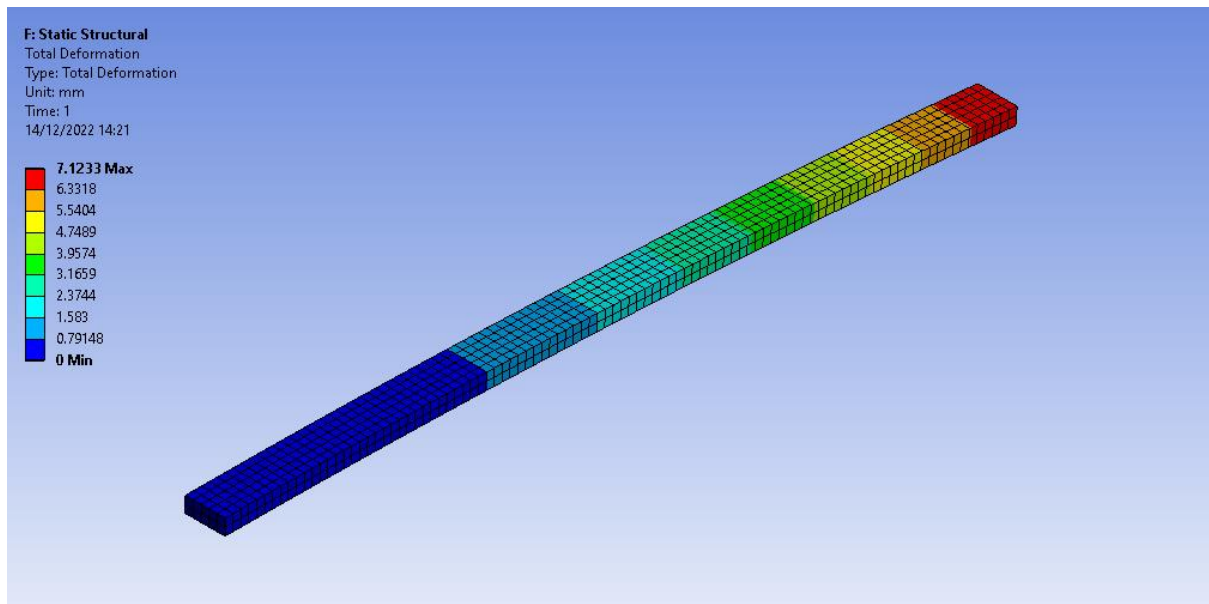


Figure 4 - Ansys simulation of total deformation under a thermal load

3. Front Wing Mounts Analysis

The Force TA hill climb car races in the British hill climb series. As the car requires large amounts of downforce at high speed, the mounts that hold the front wing to the car need to be able to withstand the forces generated by the powerful wing. In the below figure is the Force TA car with the front wing mounts circled in red:



Figure 5 - Force TA Hill Climb Car on track (Jones, 2022) – Wing mounts circled in red

The front wing mounts that are currently fitted to the car are made up of 2 separate components bolted together on each side (can be seen in figure 6). The team have recently asked for the wing mounts to be developed to maintain the strength, reduce the weight and also simplify them.



Figure 6 - Front wing mounts of the Force TA (Jones, 2022)

3.1. Front Wing Mounts CAD assembly

3.1.1. Aerodynamic Analysis of Front Wing

The wing mounts connect the wing to the car meaning the wing mounts must be able to cater for the force generated by the front wing. To be able to accurately redesign the wing mounts and to allocate the correct forces, an aerodynamic study was conducted in Ansys fluent, which a CFD package, to learn how much downforce is generated solely by the front wing. The picture below shows the pressure contours on the front wing of the car.

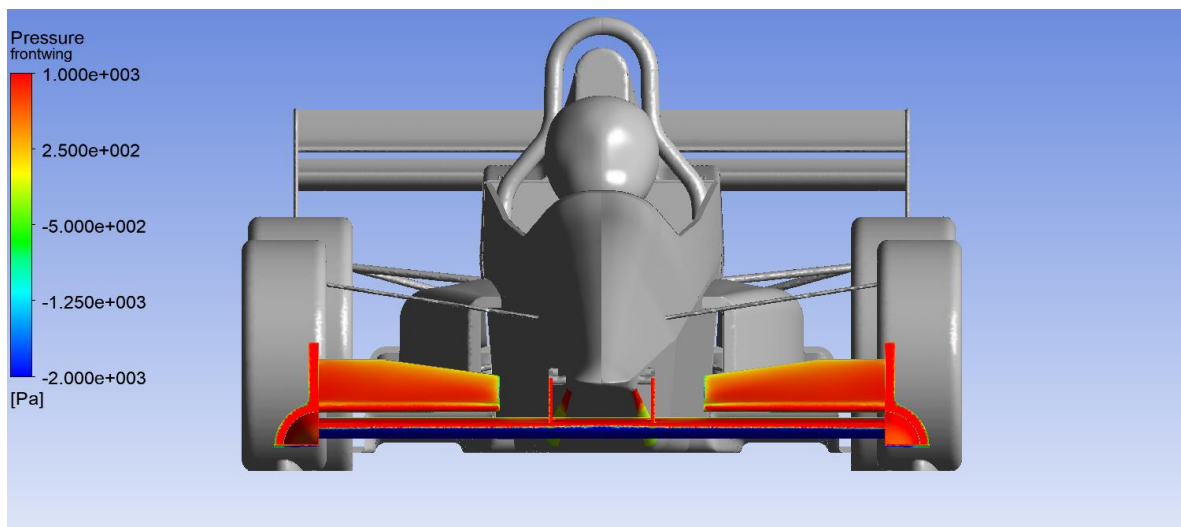


Figure 7 - Pressure Contours of Force TA front Wing

The velocity of the study set was 58m/s which is about 130mph. The cars top speed taken from data collected at a previous event was 125mph so simulating the car at around 130mph allowed for a higher margin of error (Jones, 2022). The total amount of downforce generated by the front wing was 2284N and the total drag force was 252N. These numbers can now be used when simulating the forces working on the wing mounts.

3.1.2. CAD Assembly of Front Wing

Using dimensions provided by (Jones, 2022), see appendix section 1, the components were drawn up in Solidworks accurately to represent what's on the car. The assembly included the front wing, each wing mount component, spacers and a mock-up chassis to represent the car chassis. This can be seen in figure 8 and 9.

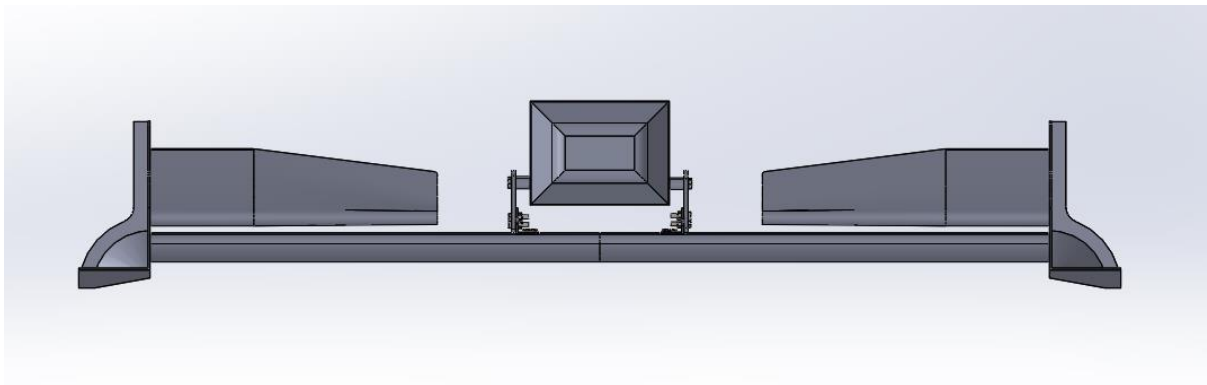


Figure 8 - Front wing assembly in Solidworks (front view)

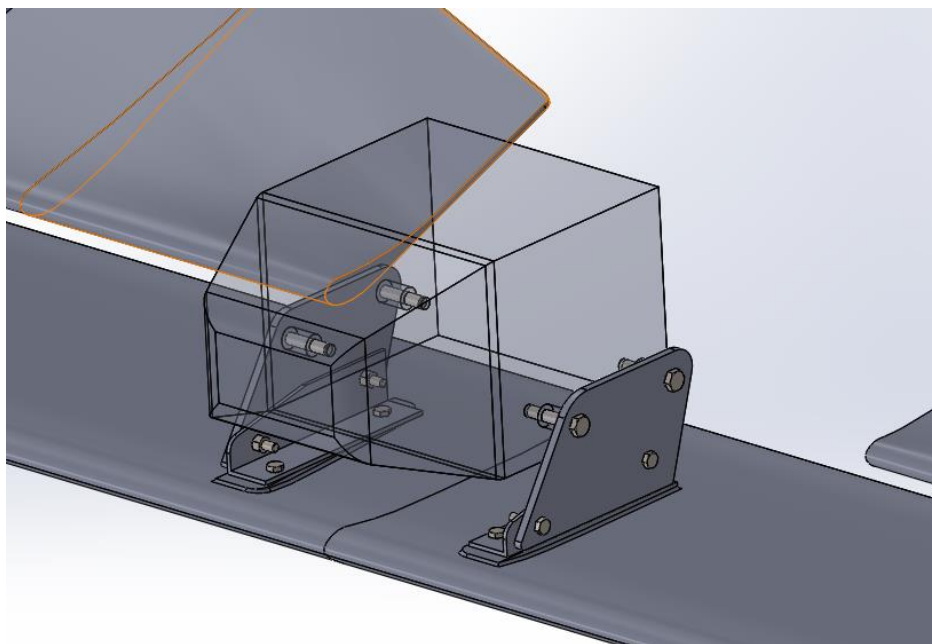


Figure 9 - Detailed view of wing mounts in Solidworks (Isometric View)

3.2. FEA Analysis

3.2.1. Engineering data

Before starting to setup the simulation, the material needs to be defined within engineering data. Using data found (Material Properties, 2022). The main piece of information needed is the youngs modulus of aluminium. In the below screenshot we can see the engineering data filled in for aluminium.

Outline of Schematic B2, C2, D2: Engineering Data				
	A	B	C	D
1	Contents of Engineering Data			Source
2	Material			Description
3	aluminium			C:\Users\2003504\OneDrive - University of Wales Trinity

Properties of Outline Row 3: aluminium				
	A	B	C	D
1	Property	Value	Unit	
2	Material Field Variables	Table		
3	Isotropic Elasticity			
4	Derive from	Young's Modulus and Poisson...		
5	Young's Modulus	70000	MPa	
6	Poisson's Ratio	0.3		
7	Bulk Modulus	5.8333E+10	Pa	
8	Shear Modulus	2.6923E+10	Pa	

Figure 10 - Engineering data for aluminium

3.2.2. Geometry Setup

The first step in the process of gathering some FEA data is to setup the assembly within the geometry. To do this, the assembly was imported into the geometry section as a step file, and then in the model, made sure all the appropriate connections and mates were in the correct places. The material was also specified for the wing mount components using the data inputted into engineering data tab.

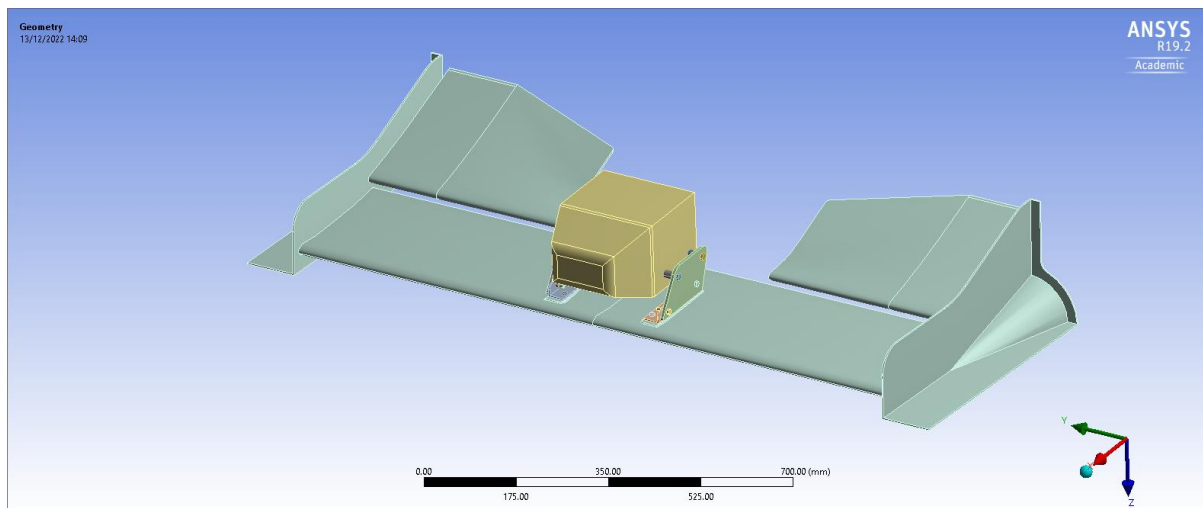


Figure 11 - Front Wing Geometry in Ansys

3.2.3. Mesh

The next step of the process was to mesh the assembly. The mesh size, based off the results taken from the cantilever beam validation earlier in the report, was set to max global element size of 5mm. This is to make sure that the results are accurate. If the mesh was to be finer, for example lower than 1mm, the mesh generating time would take longer as well as the solution time taking a bit longer. This is because there a lot more mesh elements to create and simulate.

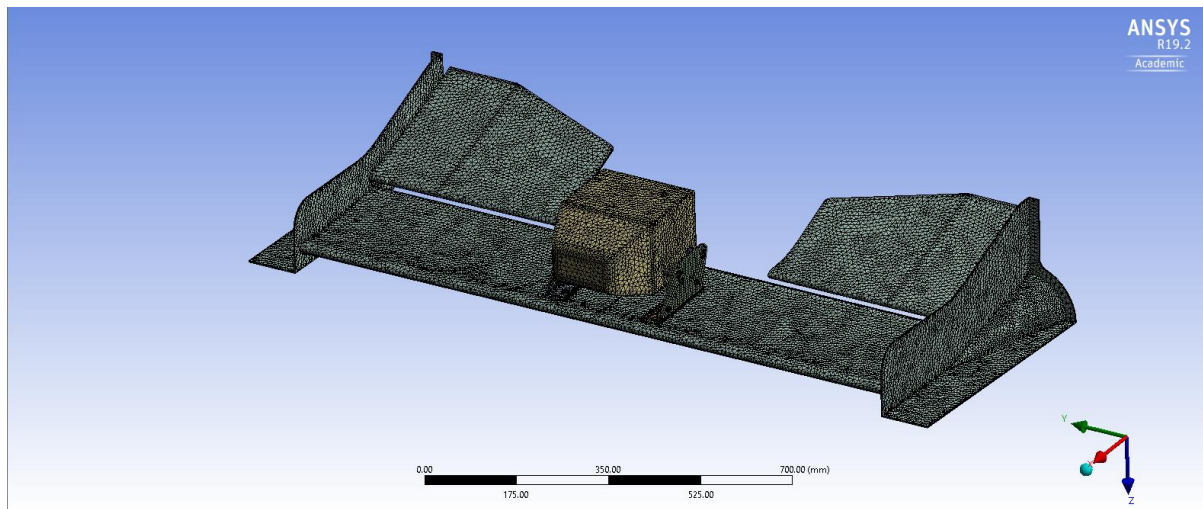


Figure 12 - Front wing assembly with a 5mm fine mesh

3.2.4. Boundary Conditions

The next step before running the simulation is to setup the boundary conditions. The first condition set was to make a fixed connection to constrain the model. This constraint was added to the mock-up chassis as the wing hangs from the chassis. The next boundary conditions were to add a force pulling down on the wing, acting as the downforce, and the force pulling back on the wing, acting as the drag, which was the force taken from the CFD results. The downforce was set at 2300N which was rounded up from the CFD results to allow for tuning changes, and drag which was set at 260N pulling back on the wing which was also rounded up. This can be seen in the below picture.

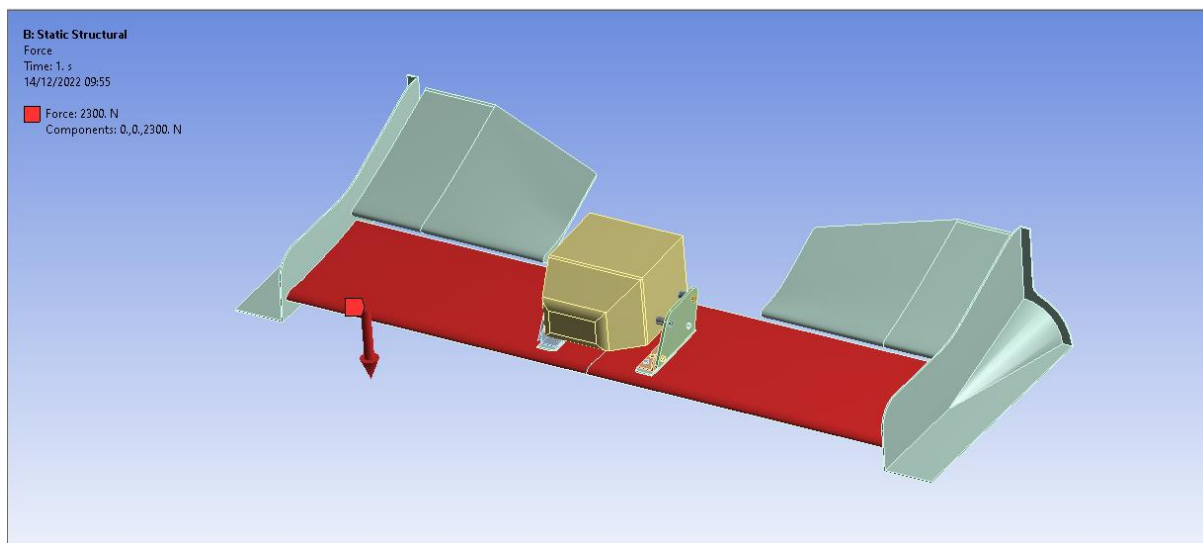


Figure 13 - Force applied to wing pulling down on wing mounts

3.2.5. Results

Once everything was setup, the simulation was set to solve to get the results. Once solved, the total deformation was presented to see how the components reacted under the forces loaded up on them. This is demonstrated in figure 14 and 15 where we can see the deformation of the components. The legend on the left shows the minimum and maximum deformation and the colours of the in between so that we can compare how much deformation is happening in each area and the components. From this, we can see that the maximum deformation of the component is just under 0.1mm which is happening on the front bolts. The main components are getting around 0.02mm to

0.05mm of deformation. This isn't a huge amount but with fatigue life of the components, they will eventually start to feel the effect.

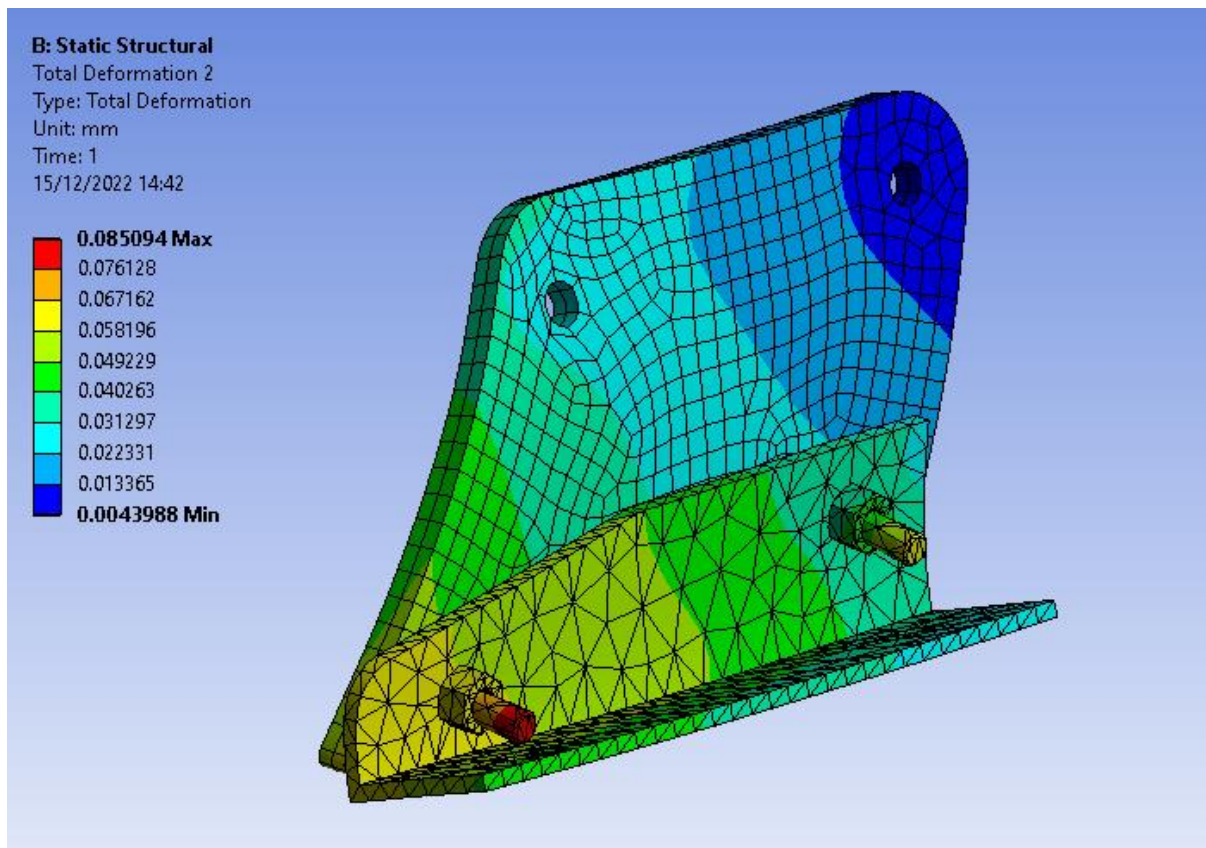


Figure 14 - Total deformation of wing mounts when load applied

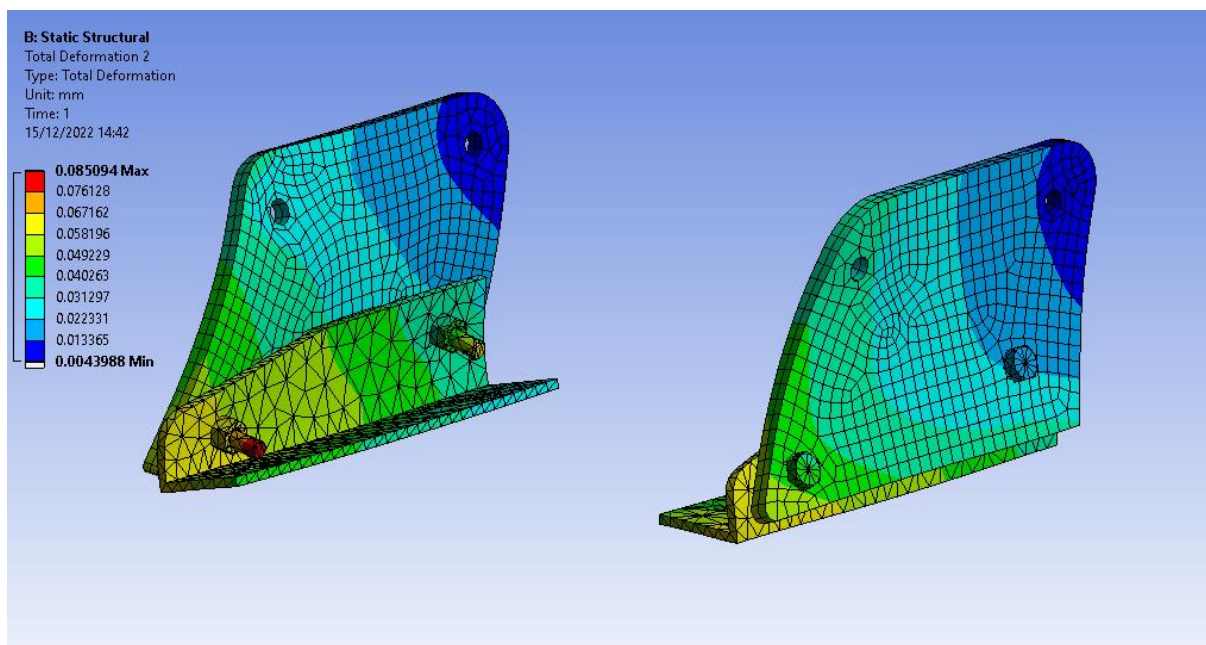


Figure 15 - Total deformation of both wing mounts when load applied

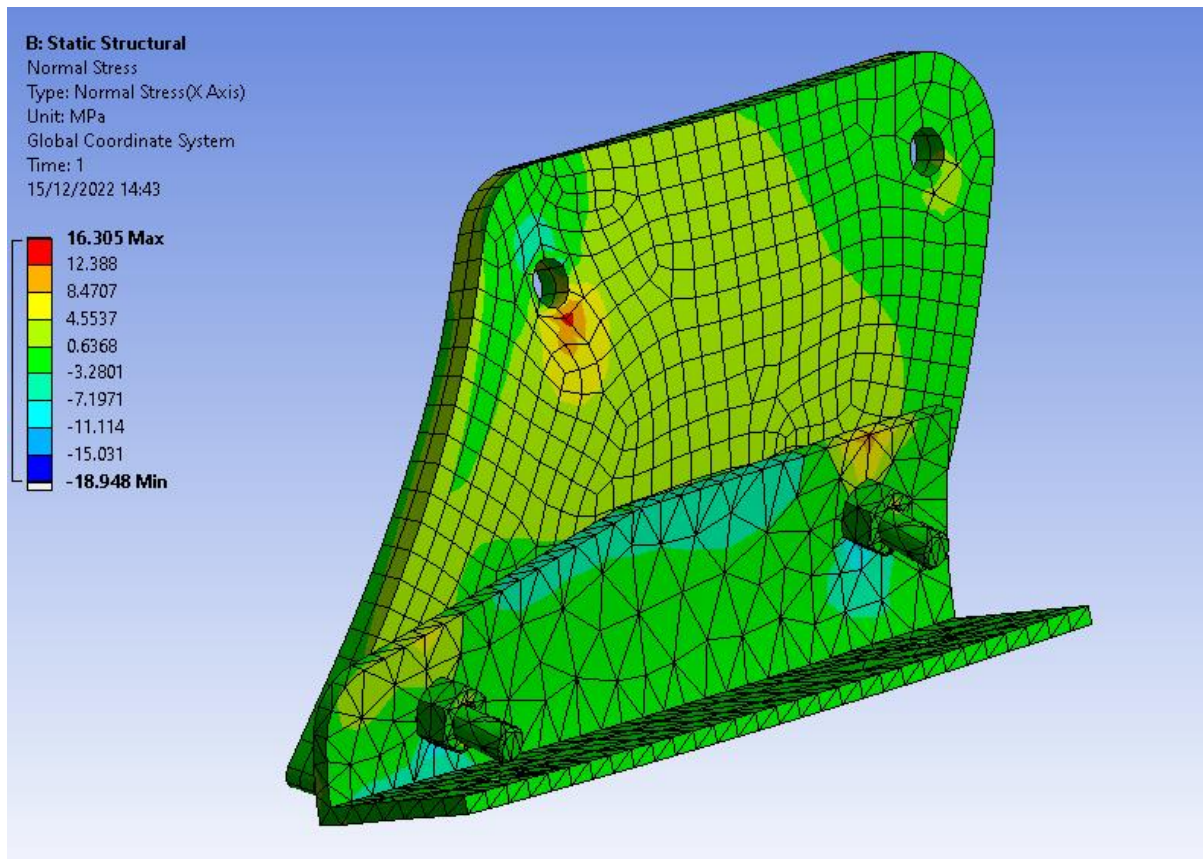


Figure 16 - Stress contours on wing mounts when load applied

In figure 16, we can see the stress contours of the components. The main areas of stress concentration are around the bolt holes. The max stress around the bolt holes is around 16.3MPa. Stress concentration is when the stress builds up in one area around a hole or an edge and the stress is higher than the average stress seen around the body of a component (Murakami, 2019).

4. Wing Mounts Development

4.1. CAD Model Changes

The main development change is that the front wing mounts are now 1 separate part on each side instead of 2 parts bolted together on each side. The first reason for doing this was to simplify the component/assembly to reduce the amount of connections. Eliminating the bolting together of the two parts in the previous version to reduce the stress concentration around where the bolts joining the two components together. The shape of the component is very similar to the previous components combined but will hopefully maintain/reduce the amount of deformation that would happen under load as the aim is to maintain the current strength of the components. The screenshot below was taken from Solidworks displaying the new wing mounts. (See appendix 2 for technical drawing)

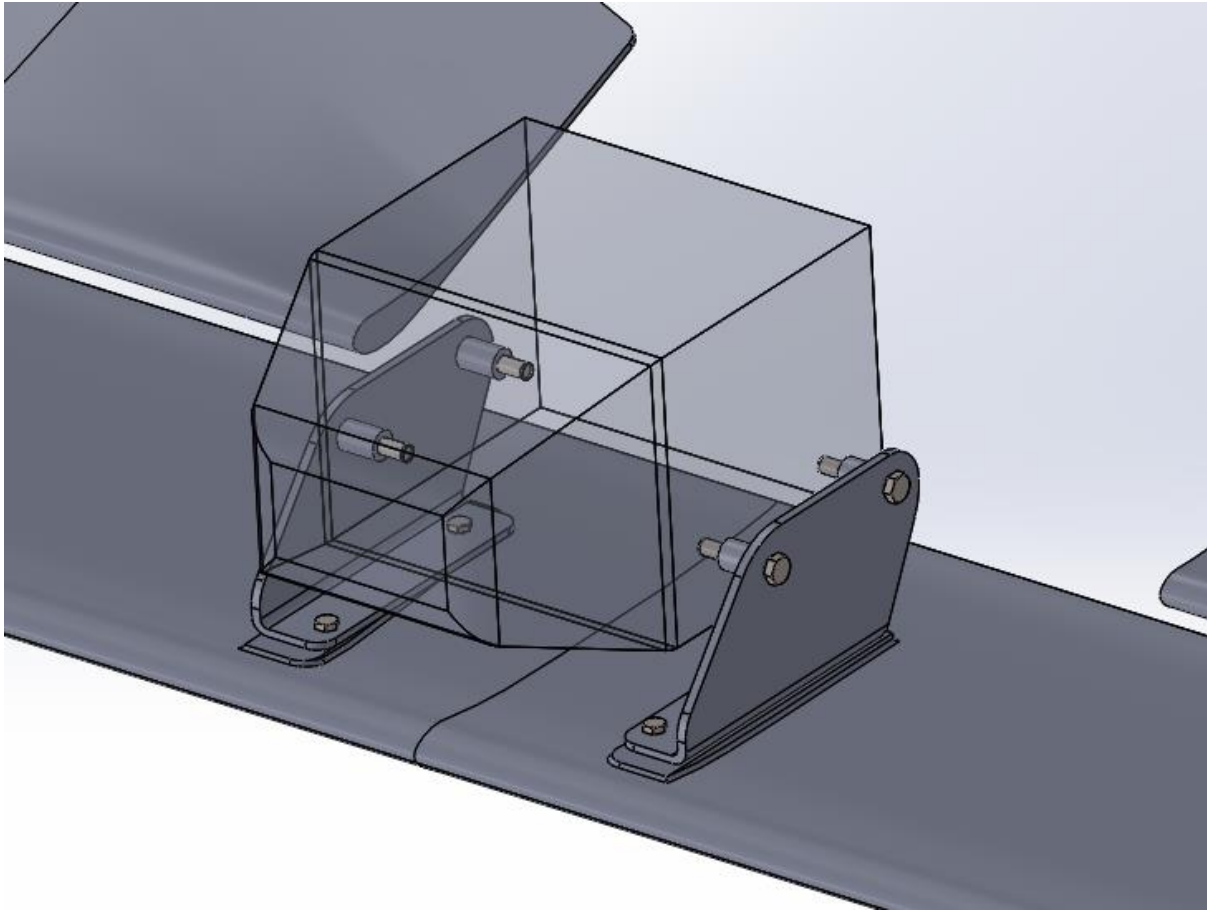


Figure 17 - Developed wing mounts in solidworks

4.2. FEA Test

To analyse the strength of the new components, the same method; materials, mesh and boundary conditions, were used within FEA to consistently investigate the comparison in strength and deformation. We can clearly see in figure 18, the maximum deformation shown on the legend has reduced to a max of 0.027mm but in the main, we're seeing around 0.003mm to 0.02mm of deformation across the component. In figure 19 we can see that the stress concentration areas are similar around the bolt holes but the maximum stress has also decreased. Overall we can see an improvement in component performance while also reducing the amount of parts used to simplify the assembly.

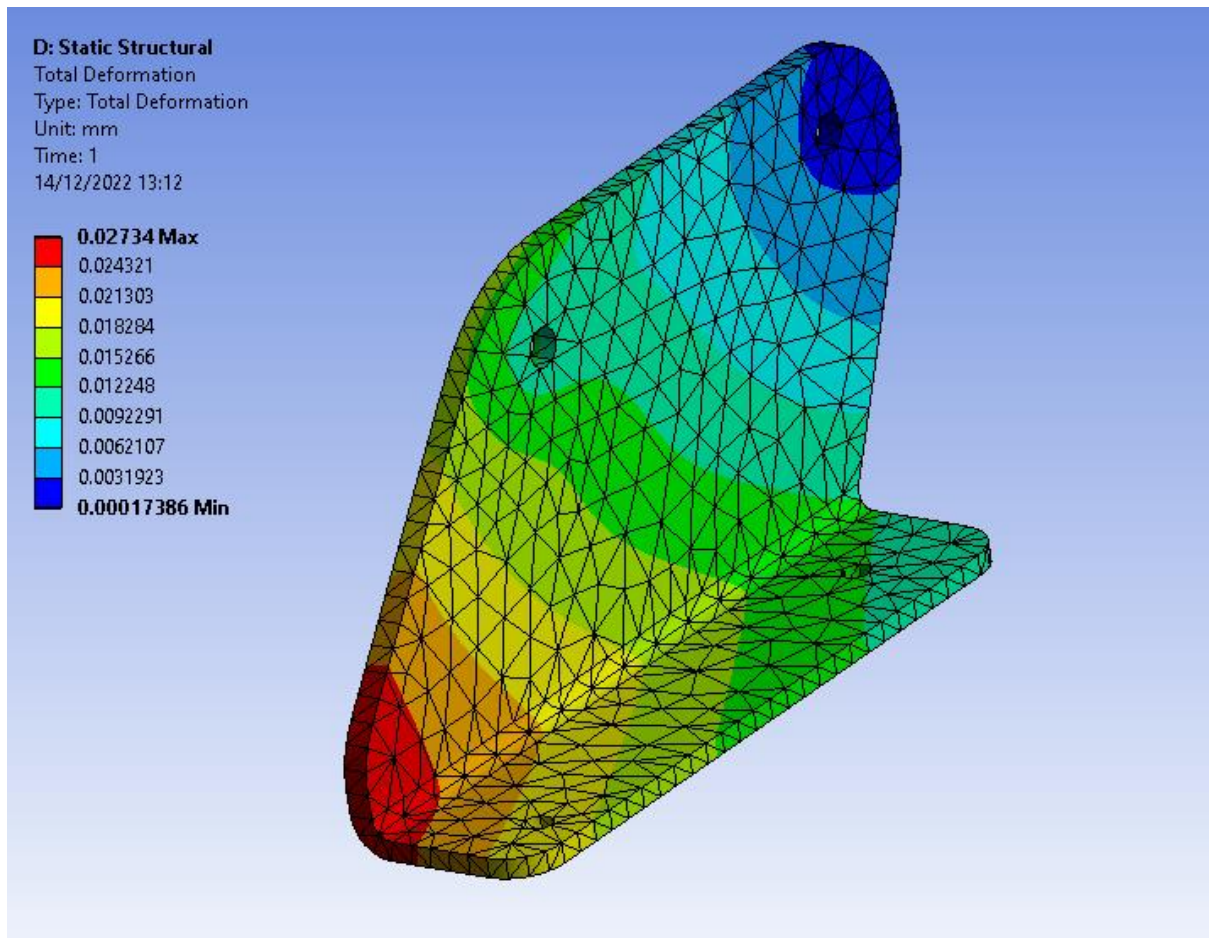


Figure 18 - Total Deformation of the new wing mount

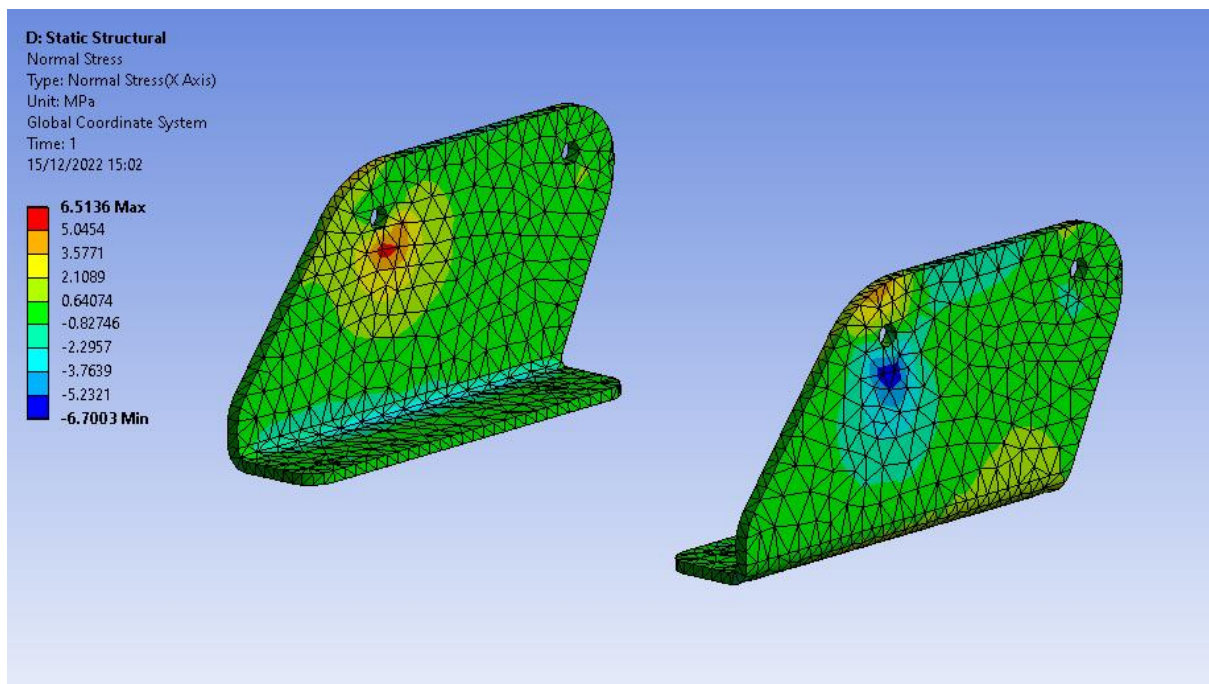


Figure 19 - Total stress of the new wing mounts

5. Conclusion

In conclusion, computer aided engineering is a powerful tool that can be utilised in a variety of industries to improve and develop the methods and components designed by the engineers without having to physically test a component or assembly. This not only speeds the design process up, it also allows the engineers to go through and test more design iterations computationally before having to manufacture the final design for physical testing. This then can allow the engineers to validate their final findings with their computational results to validate the component as well as their method so moving forward the engineers have confidence in the software's accuracy as well as the methods they are using.

The validation cases conducted in this report proved that you don't have to manufacture a component to validate your results. A simple analytical solution can be conducted to compare against a computational simulation to find the accuracy in the methods being used. The mesh sizing can also be investigated to help improve the accuracy but also indicate for potentially more complex components the accuracy they might have.

The wing mounts development was a good example of using computer aided engineering as they were designed and simulated in a few different steps. Finding the forces within computational fluid dynamics, another form of CAE, acting on the wing that the mounts will be holding up was important to accurately simulate the loads the mounts would be under. The results taken from CFD, were then used within FEA to find the deformation and stresses the components were under so that the new mounts could be developed using the visible results presented by Ansys. The final wing mount design proved to increase the overall strength of the component while reducing the amount of deformation happening throughout the part. The final design can now be manufactured to go on the hill climb car and hold the wing up with minor performance gains.

Bibliography

- Alambra, N. S. (2022, 06 17). *Beam Deflection Calculator*. Retrieved from OMNI Calculator: <https://www.omnicalculator.com/construction/beam-deflection>
- Bi, Z. (2018). Overview of Finite Element Analysis. *Finite Element Analysis Applications*.
- Dunkley, J. (2021, August 20). *How Finite Element Analysis (FEA) can support the New Product & Medical Device Development Process*. Retrieved from Houghton Design: <https://haughtondesign.co.uk/how-finite-element-analysis-fea-can-support-the-new-product-medical-device-development-process/>
- English, T. (2019, November 07). *What Is Finite Element Analysis and How Does It Work?* Retrieved from Interesting Engineering: <https://interestingengineering.com/science/what-is-finite-element-analysis-and-how-does-it-work>
- Jones, C. (2022). *Force TA Car Data*.
- Material Properties. (2022). *Aluminium – Properties – Price – Applications – Production*. Retrieved from Material Properties: <https://material-properties.org/aluminium-properties-applications-price-production/>
- McCaslin, S. (2021). Validation and Verification in Finite Element Analysis (FEA). *Control Automation*.
- Murakami, Y. (2019). Stress concentration. *Material Fatigue*, 13-27.

PTC. (2022). *Computer Aided Engineering (CAE)*. Retrieved from PTC:

<https://www.ptc.com/en/technologies/cad/computer-aided-engineering>

Siemens. (2016). *Computer-Aided Engineering (CAE)*. Retrieved from Siemens:

<https://www.plm.automation.siemens.com/global/en/our-story/glossary/computer-aided-engineering-cae/13112>

Srinivas, K. (2020). *Verifications and Validations in Finite Element Analysis (FEA)*. Advances.

Urban, M. (2019). What is the role of Computer-Aided Engineering in Manufacturing Industry?
Medium.

World Material. (n.d.). *304 Stainless Steel Properties: SS304 Composition, Density, Yield Strength, Thermal Conductivity, Hardness, Modulus Of Elasticity*. Retrieved from The World Material:
<https://www.theworldmaterial.com/304-stainless-steel/>

Appendix

1. Mount Component Drawings

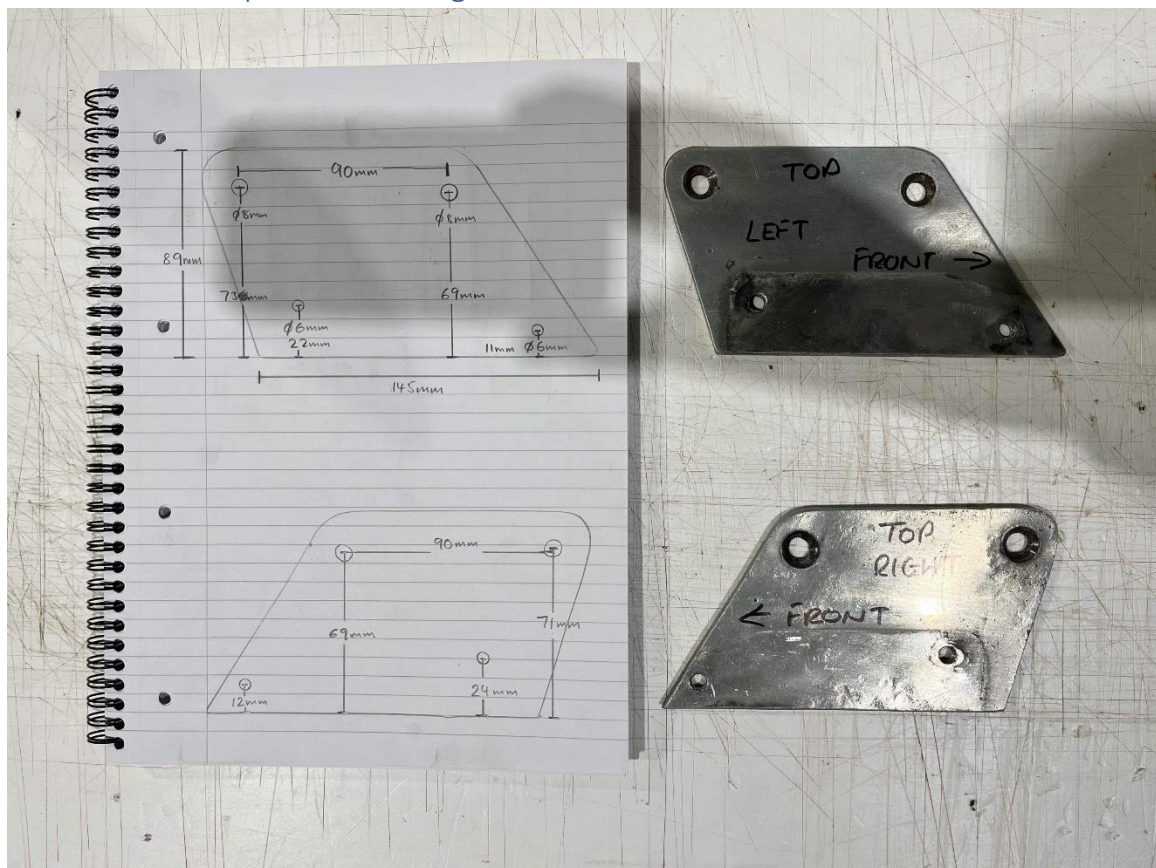


Figure 20 - Drawings for baseline wing mounts (top parts)

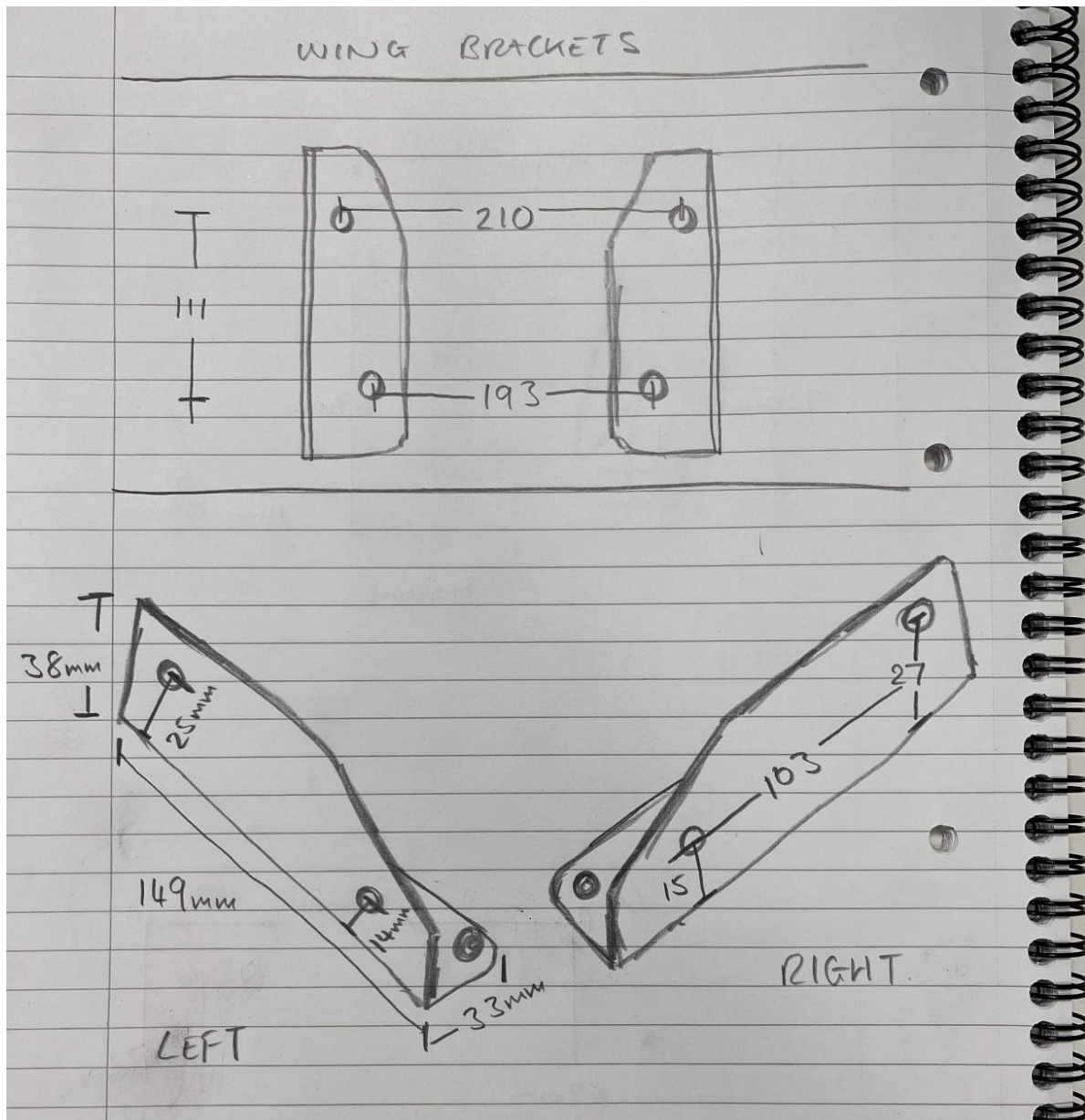


Figure 21 - Drawings of baseline wing mounts (bottom parts)

[illegible]

18