



Prifysgol Cymru
Y Drindod Dewi Sant
University of Wales
Trinity Saint David

BEng Motorsport Engineering

<i>Assessment Title</i>	Application of Computer Aided Engineering Processes
<i>Module Number</i>	ACHA6010
<i>Module Name</i>	Structural Design & Fluid Analysis
<i>Student Number</i>	2306046
<i>Student Name</i>	Roland Magyar
<i>Date of Submission</i>	17.12.23

School of Engineering
University of Wales Trinity Saint David
2023-2024

Contents Page

Table of Contents

Contents Page.....	1
Table of Figures	2
Introduction.....	3
Task 1.....	4
Task 3.....	8
Conclusion	11
References.....	12

Table of Figures

Figure 1: Beam Dimension	4
Figure 2: SolidWorks Model	5
Figure 3: 0.020mm Mesh.....	5
Figure 4: 0.005mm Mesh.....	6
Figure 5: 0.045mm Mesh.....	6
Figure 6: Case Study Result.....	7
Figure 7: Bimetallic Strip.....	7
Figure 8: Complete Assembly.....	8
Figure 9: Thermal Conditions.....	9
Figure 10: Aluminium Deformation	9
Figure 11: Aluminium Stress	9
Figure 12: Steel Deformation.....	10
Figure 13: Steel Stress	10

Introduction

Throughout this assignment the process of validation will be carried out to show the accuracy of the Ansys software, this will be done by comparing case studies through simple simulations. To achieve this a comparison will be made between two different materials, aluminium (6061T6) and low carbon steel, which will be used on a piston to simulate how they would be produced in assembly. The goal is to produce a realistic simulation that provides accurate calculation results for deformation and stress.

Task 1

The software used for the structural analysis was Ansys. Firstly, to check the accuracy of the Ansys software you must do the process of validation, to do the validation a simple simulation was chosen. This simulation was a cantilever beam study, the dimension of the beam can be seen in Figure 1 with the created beam in Figure 2 which was created on SolidWorks before it was imported into the Ansys. Once in Ansys the beam had a constant 2.750 kilo pascals applied over the whole top of the beam evenly when doing the simulation. The mesh chosen was 0.020m this is because this is the same size as the one from the case study with the validation the same mesh size was chosen to see if it gets the same numbers as on the case study. Alongside comparing it to the case study the max deflection was also calculated by hand, the hand calculation was calculated with the equation:

$$u_{z, \max} = \frac{qL^4}{8EI_y} = \frac{3pL^4}{2Ew^2t}$$

Modulus of Elasticity	E	210000.000 MPa
Length	L	1.000 m
Width	w	0.050 m
Thickness	t	0.005 m
Uniform Pressure	p	2.750 kPa

Answer: 1.5714×10^{-3}

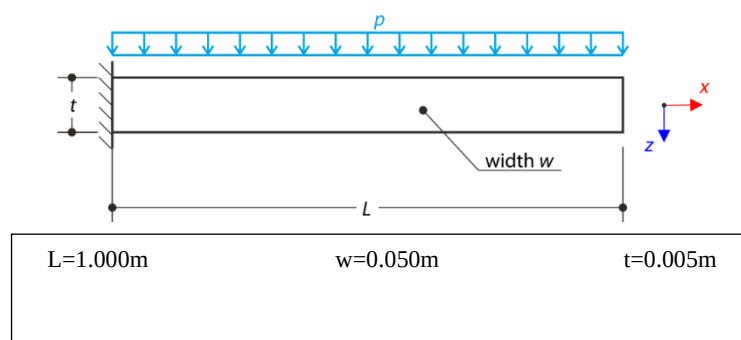


Figure 1: Beam Dimension

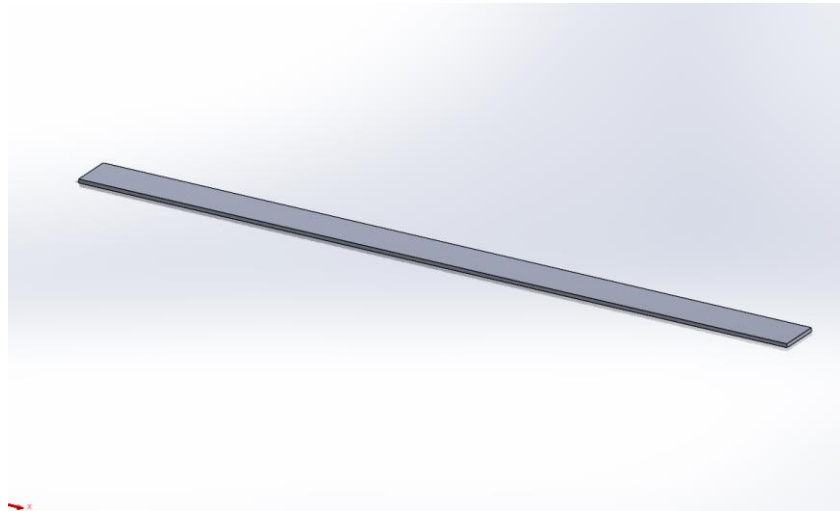


Figure 2: SolidWorks Model

After the hand calculations are completed, they can be compared to the results of the Ansys results in Figure 3. The results show the results from minimum deflection to maximum deflection, the maximum deflection is 0.007 over the hand calculation as well as for the case study which shows Ansys accuracy. From there different mesh sizes were used to see how accurate it still is for this, the mesh sizing was put to 0.005m and 0.045m the results can be seen in Figure 4 and Figure 5. When looking at the deflection of the beam the numbers don't change, and that the maximum deflection is the same throughout the three-mesh sizing's. The case study used can be found from (Dlubal Software GmbH, 2001-2023).

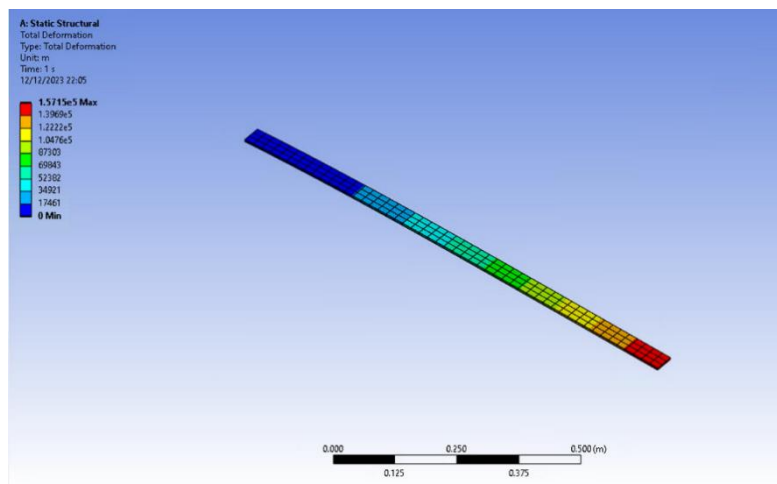


Figure 3: 0.020mm Mesh

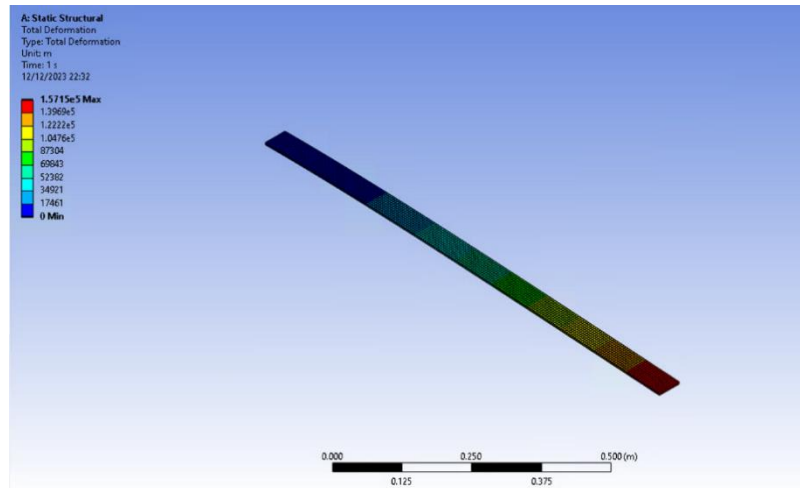


Figure 4: 0.005mm Mesh

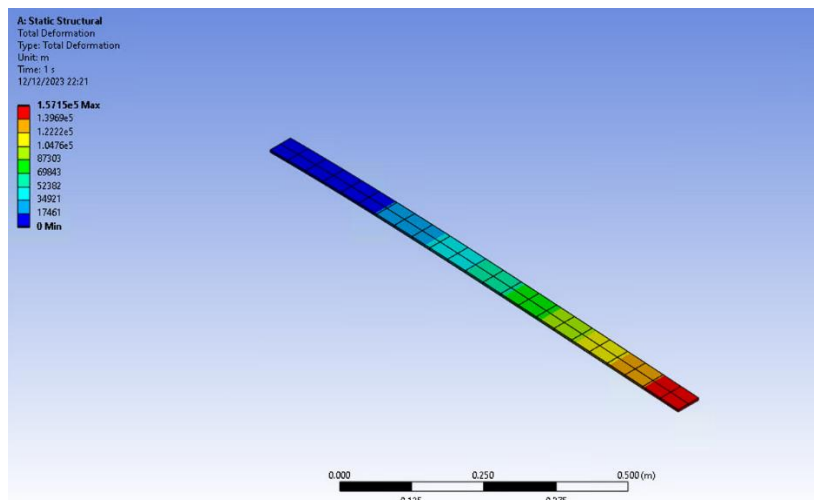


Figure 5: 0.045mm Mesh

A bimetallic strip was done to test the accuracy of the thermal simulations and to see how heat effects the material the case study that it will be compared to was found on (Dlubal Software GmbH, 2001-2023). This case study showed the dimensions of the model but also the materials and all the other conditions. In Figure 6 it shows the maximum deflection from the case study whilst comparing it to Figure 7, the results can be seen for the simulation with its maximum deflection. The results aren't exactly the same there is a slight difference between the two with the difference being 0.0723 this difference being very minimal. The difference could be due to the mesh sizing but also the calculations the software is doing could be creating different compound errors and therefore causing a deviation in the final answer.

Analytical Solution
$U_{z,max}$ [mm]
-7.049

Figure 6: Case Study Result

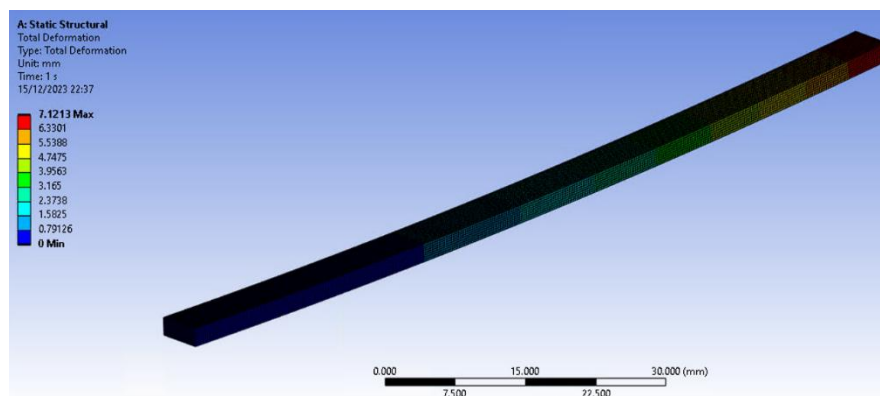


Figure 7: Bimetallic Strip

Task 2

Computer Aided Engineering (CAE) is used to help simulate different situations to see how it would affect a component, these simulations can help with the design of a component but also with the improvement of them. This can be done by simply adding changes to a component and running the simulation again, this means that the component doesn't need to be created every-time which means its also quicker but also a more cost effect way. This can be seen in the previous task of the use for validation and how it can be used to show testing of different beams. But it can also be seen through this task how different simulations can be used to test the boundaries of different components, with how changes can be done to improve the results afterwards. To show a example of how CAE can be useful a kinematic simulation with thermal influence was fed into a static structural simulation to see how it would affect a piston, the assemble created and simulated on can be seen in Figure 8. The results being looked at was the total deformation of the piston and the stress (Von Mises), first set of results is with aluminium 6061 T6. The first step was to do Rigid Dynamics simulation this was done to find the motion load being applied to the piston for this, one bit of data was selected and this being the highest amount of force. As for a structural analysis the worse case of scenario is what the component is tested for. From there the motion load is loaded into the Static Structural this is so that the data is in there and doesn't cause any issues later on. After the Steady State Thermal followed the temperature conditions which the piston was simulated under can be seen in Figure 9, these conditions where set to create a more realistic condition over the piston. When the thermal simulation was done that solution was fed into the Static Structural, this was all done to give a realistic simulation and give a idea of what could happen to the piston. When running in the engine at 5000 RPM with 1.5 Bar of pressure being applied to the top of the piston to help simulate combustion. The final results of the total deformation of the aluminium can be seen in Figure 10 with the stress being in Figure 11, for the total deformation of the aluminium piston is was 0.99024 millimetres and the maximum stress on the piston being 354.55 Megapascals.

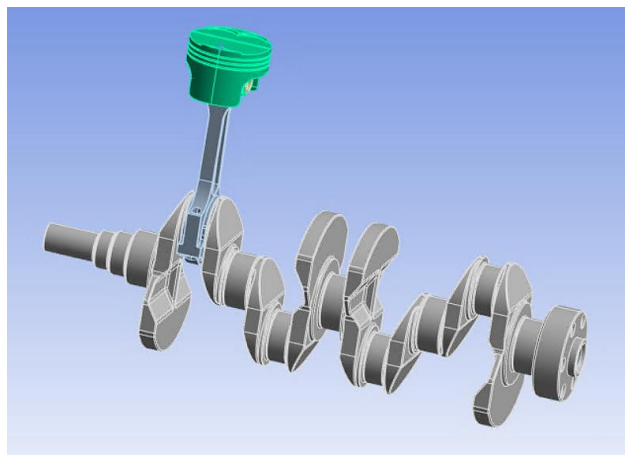


Figure 8: Complete Assembly

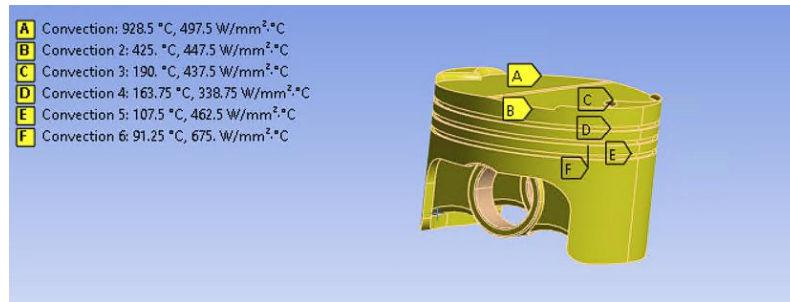


Figure 9: Thermal Conditions

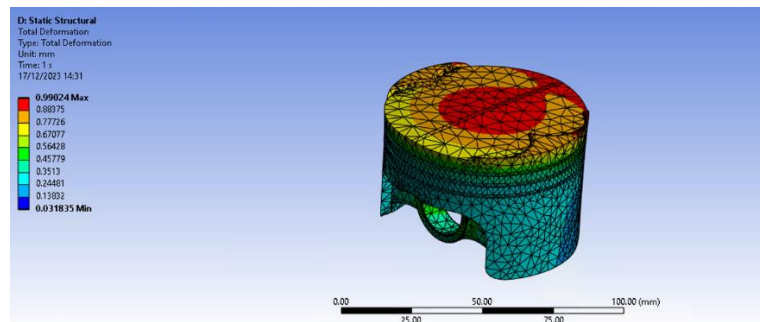


Figure 10: Aluminium Deformation

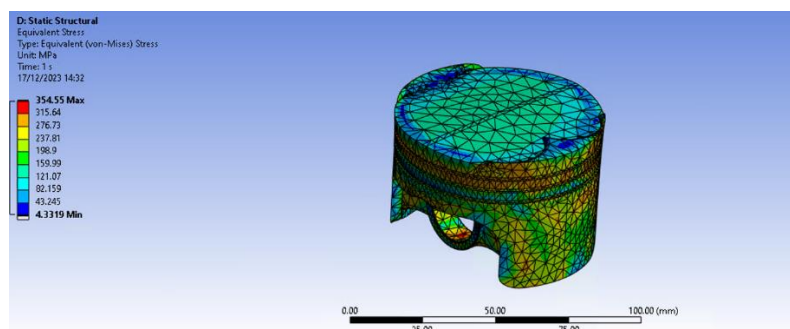


Figure 11: Aluminium Stress

To give a comparison the simulation was done again with the same boundary conditions and settings but with a different material. The different material was a 4340 low alloy steel hardened and tempered at 315 Degrees Celsius this material was chosen to see the differences between the two pistons. The results for the simulations can be seen in Figure 12 and in Figure 13, the maximum deflection is 0.40693 millimetres and a maximum of stress of 1763 Megapascals. So, for the difference between the deformation is that the Steel had deformed less by 0.58331 millimetres compared to the aluminium. But the stress being more on the Steel by 1408.45 Megapascals so overall the steel piston deformed less but had more stress going through it. With this example it can be seen how CAE is useful as the material could be changed easily without the need of creating a new component to see how the materials would fare when being used for a piston.

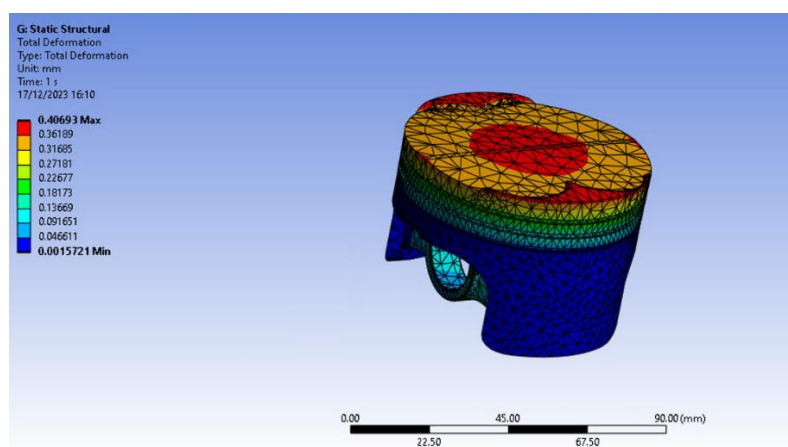


Figure 12: Steel Deformation

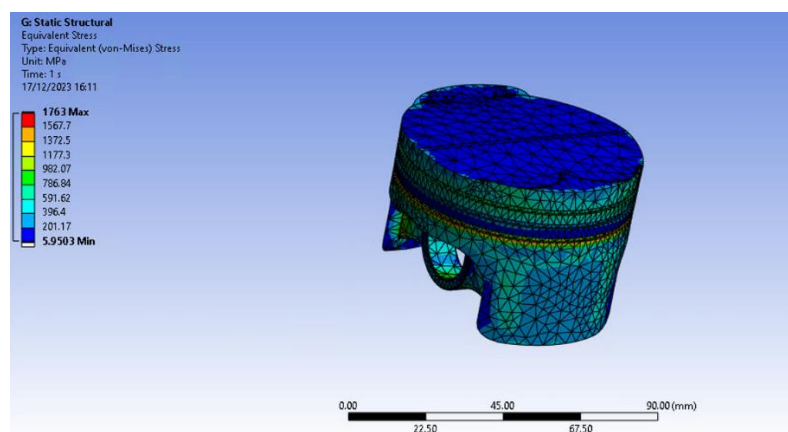


Figure 13: Steel Stress

Conclusion

Through the assignment it can be seen how validation can be used to help show the accuracy of a software. Before using it to test out different components on there own as well as on assemblies. For the second task Ansys was then used to show how CAE can be useful, but also used by showing different types of simulation on a assembly to get a comparison on two different materials used for a piston.

References

Dlubal Software GmbH, 2001-2023. *Dlubal*. [Online]
Available at: <https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/009016>

[Accessed 11 12 2023].

Dlubal Software GmbH, 2001-2023. *Dlubal*. [Online]
Available at: <https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/000078>

[Accessed 15 12 2023].

Doko, Andoko and Saputro, Nanang, 2018. Strength analysis of connecting rods with pistons using finite element method. *International Mechanical and Industrial Engineering Conference*, 01 01, 204(MATEC Web of Conferences), pp. 1-6.