

University of Wales Trinity Saint David

Wales Institute of Science and Art



Prifysgol Cymru

Y Drindod Dewi Sant

University of Wales

Trinity Saint David

Structural and Fluid Analysis

ACHA6010

Oliver Wilson 2203893

December 2025

Contents

1	Introduction	1
2	Aims.....	2
3	Methodology.....	3
4	Validation	4
4.1	Static Structural	6
4.2	Bimetallic Thermal.....	9
5	Engine Assembly	11
6	Optimisation	18
7	Limitation	21
8	Conclusion.....	22
9	References	23

Figure 1- cantilever beam (© Dlubal Software 2015, n.d.).....	4
Equation 1-Ux max (© Dlubal Software 2015, n.d.)	4
Equation 2-Uz max (© Dlubal Software 2015, n.d.)	4
Figure 2- Bimetalic Beam (© Dlubal Software 2018, n.d.).....	4
Equation 3- coefficient K equation (© Dlubal Software 2018, n.d.).....	5
Equation 4-Uz max (© Dlubal Software 2018, n.d.)	5
Figure 3-Deformation of cantilever beam , total deformation-.....	6
Figure 4-Tetrahedral mesh on beam	6
Figure 5-Automatic meshing on beam.....	6
Figure 6-Uz Max Deformation Automatic V tetrahedral	7
Figure 7-Uz Max Deformation % difference Automatic V tetrahedral	7
Figure 8-Uz Von Mises Automatic V tetrahedral	8
Figure 9- total deformation (4mm).....	9
Figure 10-Bimetalic mesh (4mm).....	9
Figure 11-Cell size v % change in accuracy	9
Figure 12- Stress V cell size	10
Figure 13-Deflection v cell size	10
Figure 14- piston assembly	11
Figure 15- workflow	11
Figure 16- piston head with motion loads	12
Figure 17-Piston head with thermal loads.....	12
Figure 18-Piston head displacement	13
Figure 19- stress on piston.....	13
Figure 20-Piston Strain.....	14
Table 1-Material 1 (6061-T6)	14
Table 2-Material 2 (Mg-12%SiC(p)).....	14
Table 3-Material 3 (A332.0,cast,T6).....	15
Figure 21-Max deformation on Materials.....	15

Figure 22- Max strain on Materials.....	16
Figure 23-Average Stress on Materials	16
Figure 24-Yield strength v temperature (Ansys, 2024)	17
Figure 25-Plate max strain	18
Figure 26- Plate total Deformation	18
Figure 27- Optimised plate	19
Figure 28-Deformation on piston underside	19
Figure 29- Piston optimised underside	20
Figure 30-Piston optimised underside	20

1 Introduction

The use of CAD software has been crucial for rapid development of parts especially in the automotive industry, along with these developments CFD and FEA where produced. This software allows for computational heavy equations to be calculated, allowing for simulated results to be processed before the actual part is made, saving time and money.

In this paper the use of FEA in Ansys will be used to validate against basic geometry such as cantilever and bimetallic thermal test, to prove out the limitations of FEA and how it effects the results and accuracy.

The report will investigate a multi component assembly of an engine and piston assembly, it will show how transient forces and heat energy affect the assembly and will investigate how different materials characteristics effect the overall results of the investigation.

2 Aims

To validate FEA simulations against hand calculations using a range of different mesh sizes and cell type

To investigate how different materials behave when heated and the effect of different thermal coefficients has upon the results

To investigate how temperature effects the stiffness, yield strength and overall structural performance of the material e.g. piston head

To simulate a rotating engine simulation that attempts to accurately replicate the temperatures and forces that would be seen in real life

To critically analyses the results and the capabilities of FEA, assessing the limitations of the software and where it struggles

To attempt to iteratively improve the piston head using built in tools to improve the piston head overall such as a weight saving or improved fatigue life

3 Methodology

Using FEA to compute loads on a piece material requires some information such as load points fixed points, the type of joint it is and how much load is being applied and in what direction. To be able to validate the model and mesh being used will help understanding in how the software computes and how accurate it is.

The FEA will be used to initially validate the results comparing them to hand calculations on a cantilever beam and a bimetallic material to prove out the mesh and potential solvers.

Then to use the assembly of an engine with a single piston to begin to calculate the forces related to the motion of the engine and piston and to also calculate the thermal loads experienced to attempt to replicate as close as possible the condition that might be seen in a real engine. Then to experiment with other materials for the piston head and see how they compare to each other in relation to the base material.

Then to test out built in optimisation software to see how FEA is able to help optimise geometries based on the forces present and how it could potentially effect performance.

4 Validation

Validating simulation software is a crucial part of the process, by validating the method that is used and understanding its limitation will give good confidence in the accuracy of results before moving into production and testing.

A simple beam was used to validate the results of the FEA

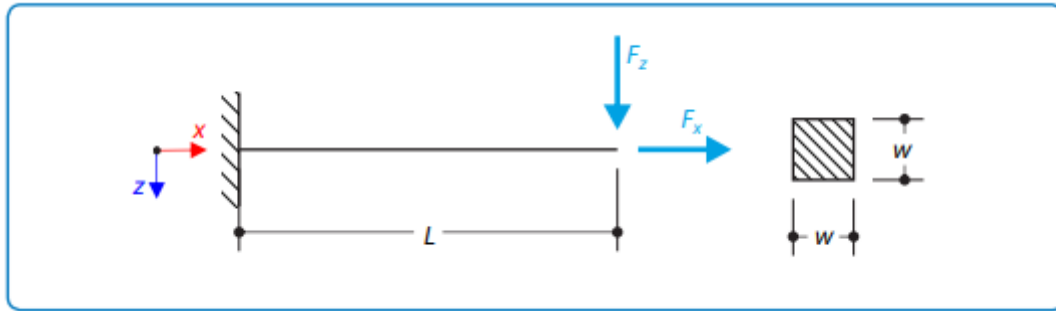


Figure 1- cantilever beam (© Dlubal Software 2015, n.d.)

The forces applied need to be calculated individually to get the correct results. There are 2 equations used in the cantilever, one to work out F_z and one to work out F_x . Using the same parameters for the steel beam a result can be found and compared to the hand calculation such as equation (1,2)

Equation 1- $U_{x\max}$ (© Dlubal Software 2015, n.d.)

$$U_{x\max} = \frac{F_x L}{EA} = 0.0012 \text{ mm}$$

Equation 2- $U_{z\max}$ (© Dlubal Software 2015, n.d.)

$$U_{z\max} = \frac{F_z L^3}{3EI_y} = 3557.478 \text{ mm}$$

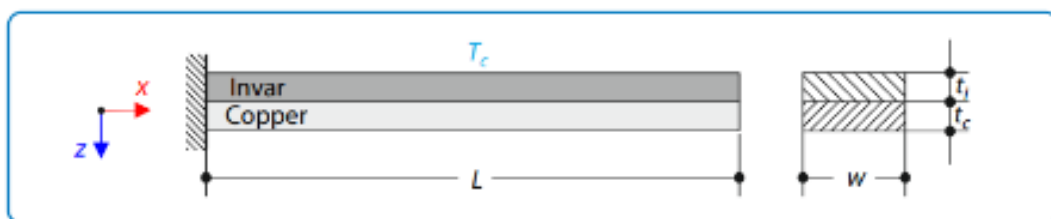


Figure 2- Bimetallic Beam (© Dlubal Software 2018, n.d.)

For the thermal validation a similar approach is used but with a different set up with 2 materials being placed on top of each other. The hand calculation used required what the thermal coefficient is to then calculate the deflection as shown in equation (3,4).

Equation 3- coefficient K equation (© Dlubal Software 2018, n.d.)

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

Equation 4- U_z max (© Dlubal Software 2018, n.d.)

$$u_{z,\max} = \frac{6(\alpha_i - \alpha_c)(t_i + t_c)T_c L^2}{t_s^2 K_1} \frac{1}{2} \approx -7.049 \text{ mm}.$$

For both methods a linear solver is being used to calculate the results.

Once validated using the same parameters, the assembly can be tested. The only part that requires a mesh is the piston head as this is the component being tested. The piston is a more complex shape.

4.1 Static Structural

To validate the software a simple cantilever beam is used, by comparing the model to hand calculation.

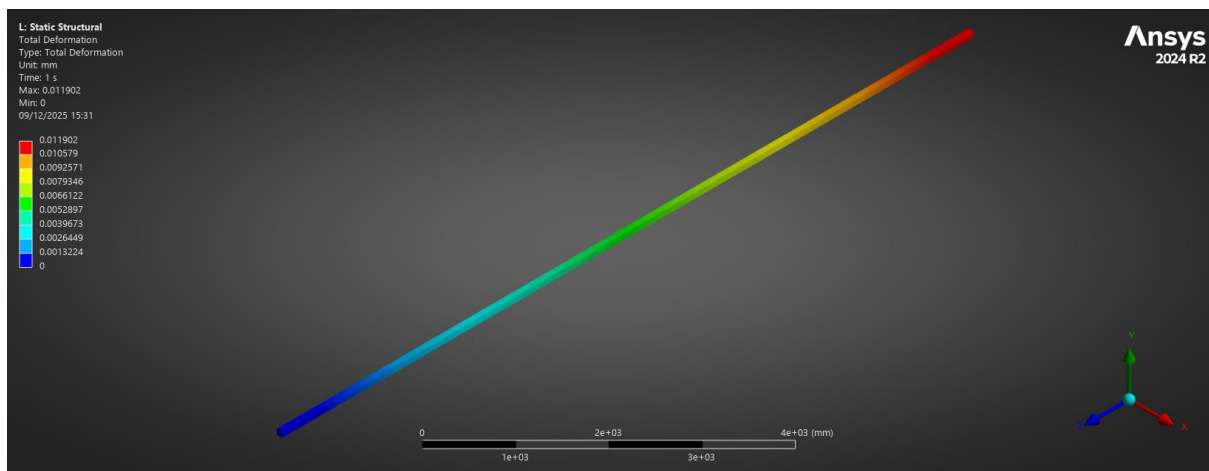


Figure 3-Deformation of cantilever beam , total deformation-

Ux max deformation stays the same at 0.0119 mm deflection

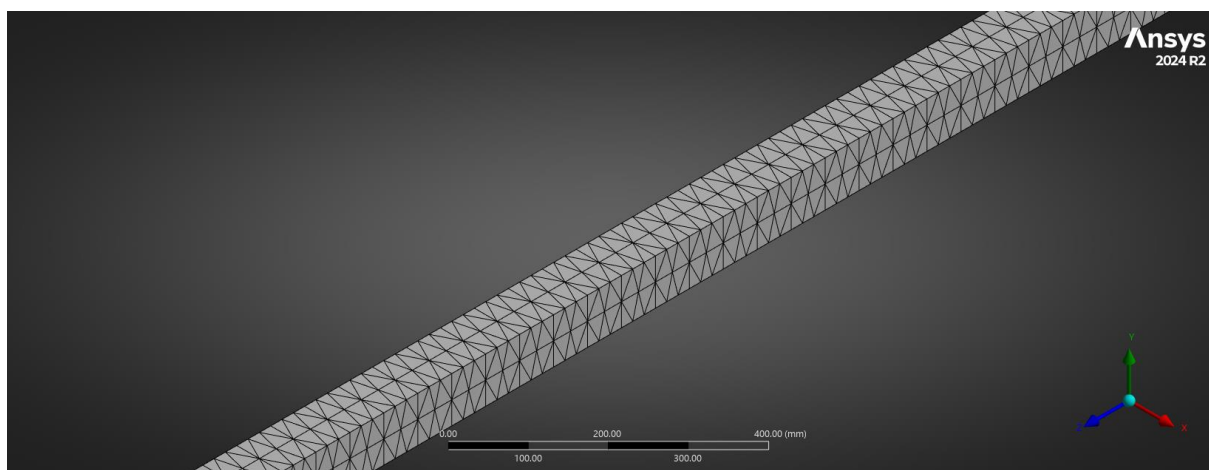


Figure 4-Tetrahedral mesh on beam

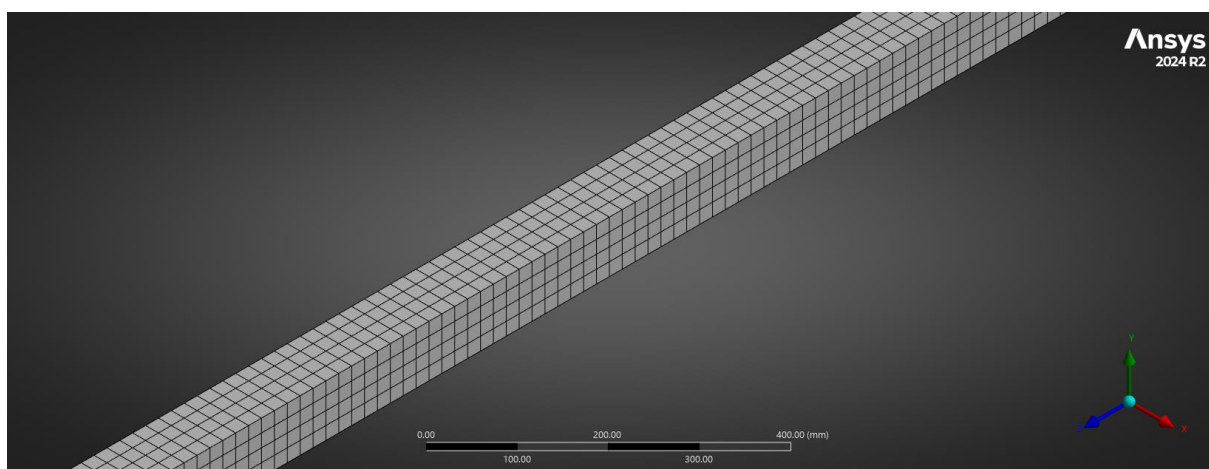


Figure 5-Automatic meshing on beam

In figure (4) shows the different mesh types used to compare the results, these are plotted in figure(5)

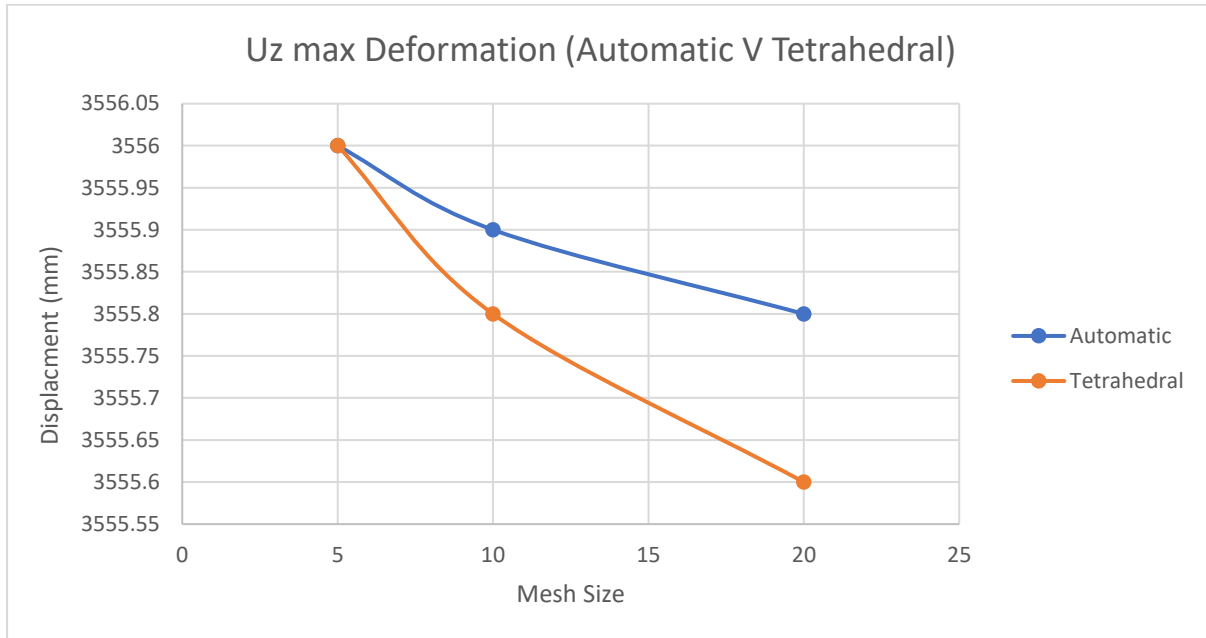


Figure 6-Uz Max Deformation Automatic V tetrahedral

Using different methods to resolve the solution show they deviate at a coarser mesh but end up converging the finer the mesh becomes. The hand calculation show Uz max is 3557.478mm. the error at the smallest mesh are within 0.05% validating this mesh showing appropriate mesh setting.

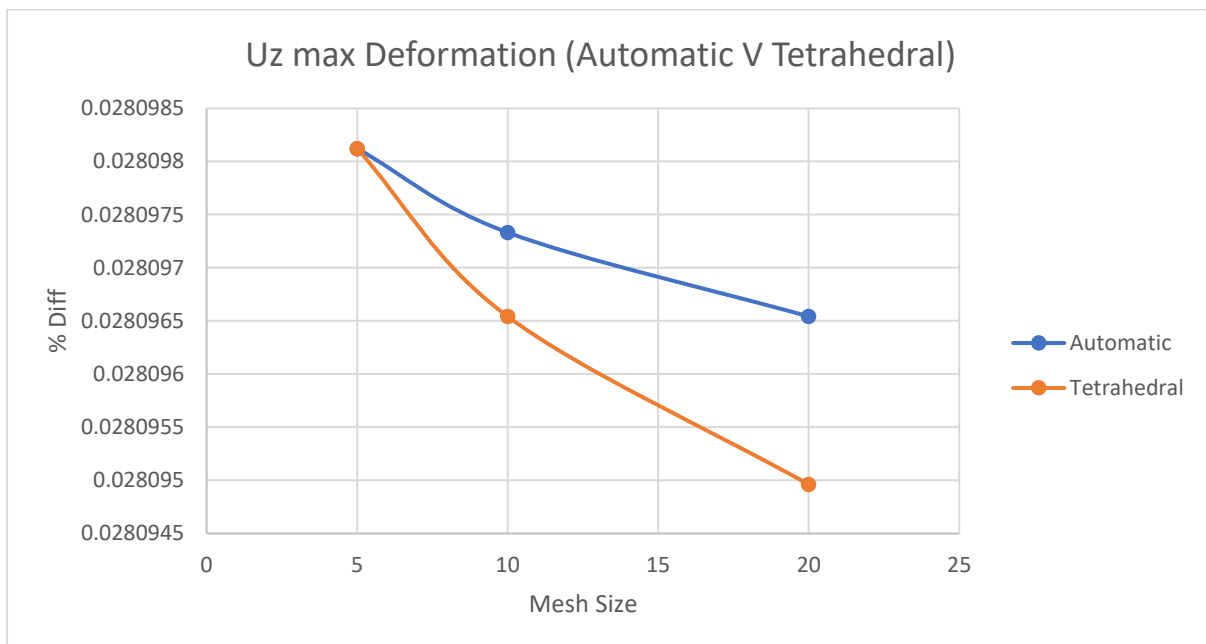


Figure 7-Uz Max Deformation % difference Automatic V tetrahedral

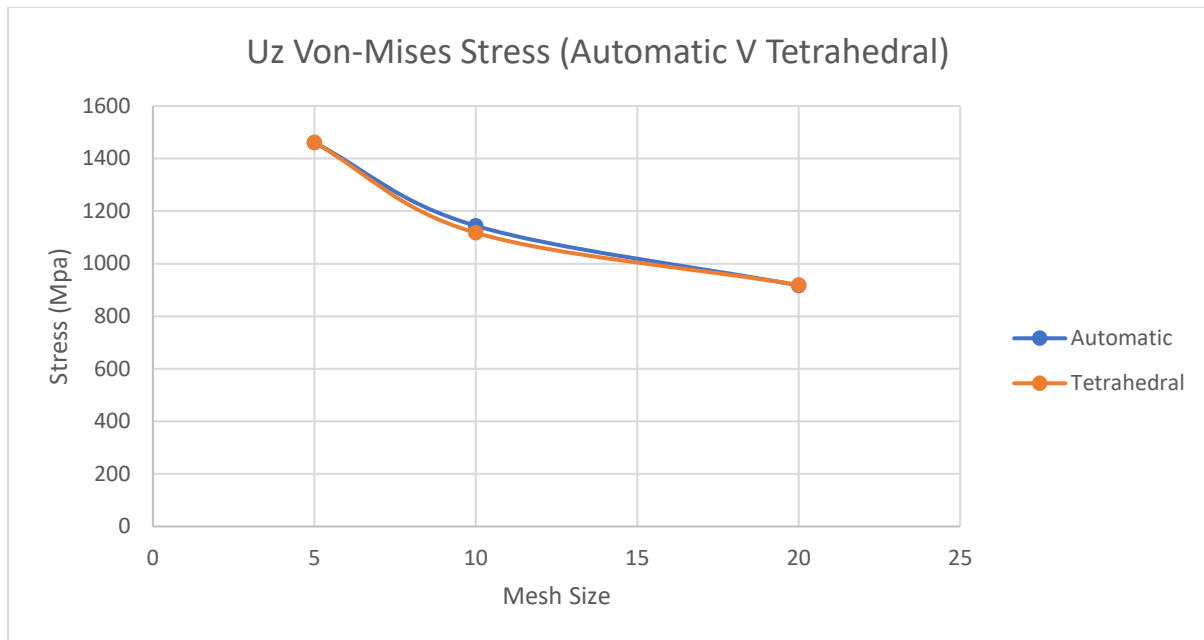


Figure 8-Uz Von Mises Automatic V tetrahedral

The graph is showing a trend upwards as the cells get smaller, the Von-Mises stress changes far more than the displacement graph. The coarse mesh shows it struggles to calculate strain on the object so as the mesh is refined the better its ability to calculate. This agrees with the findings of (Zienkiewicz, Taylor 1997) where tetrahedral elements exhibit poor stress qualities especially at a coarse mesh. Mesh refinement improves the stress prediction as its derived from displacement gradients which require a higher resolution from nodes to be able to calculate with good accuracy. Displacement suffers less due to it being predictable on simple objects.

4.2 Bimetallic Thermal

To validate the thermal component of the task a bimetallic metal was used with a copper base material and a plate of invar bonded to the top. The goal is to see how the effect of heat on different materials effect displacement of the sample and how cell size also effects it.

Starting with a coarse mesh, the test sample has been split up into square sections

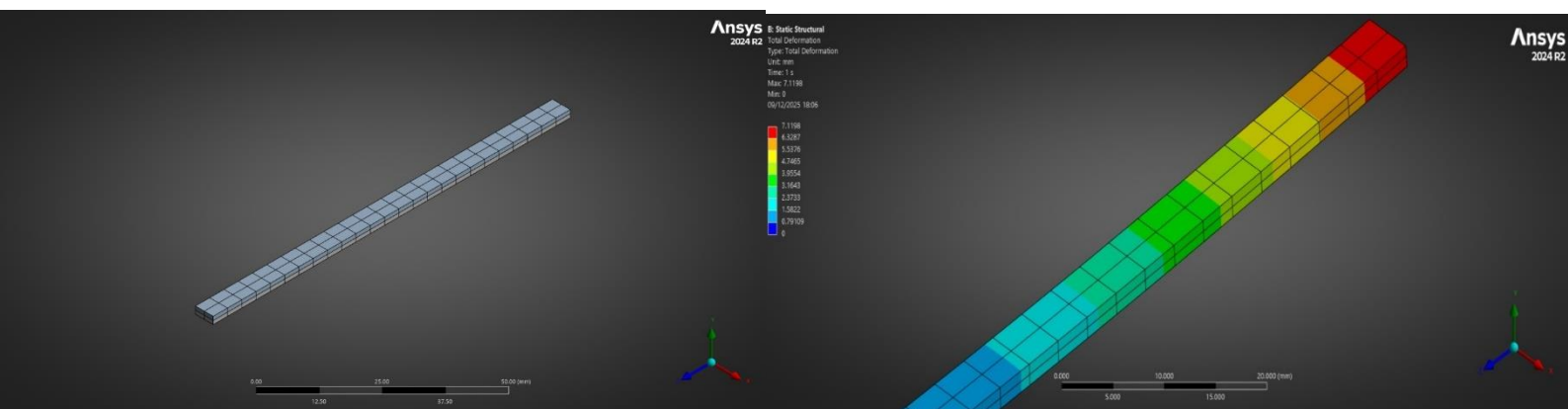


Figure 10-Bimetallic mesh (4mm)

Figure 9- total deformation (4mm)

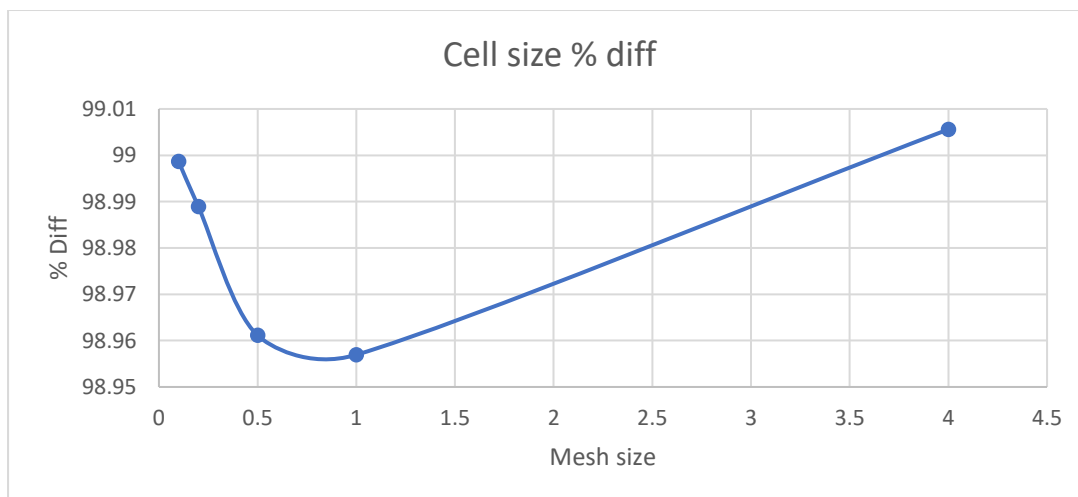


Figure 11-Cell size v % change in accuracy

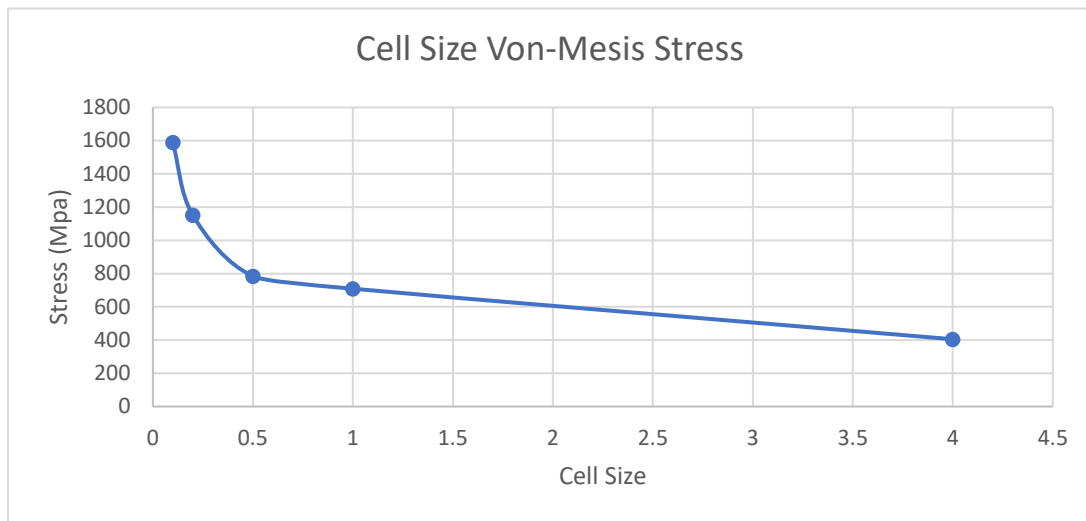


Figure 12- Stress V cell size

And the resulting difference in results, interestingly in figure(12) shows a more accurate deformation result than the finer mesh but when looking at the stress value, it shows a large discrepancy to the finer mesh. With a higher mesh count there are more cells for the simulation to solve meaning the stress calculations are more accurate as seen in the previous simple cantilever beam.

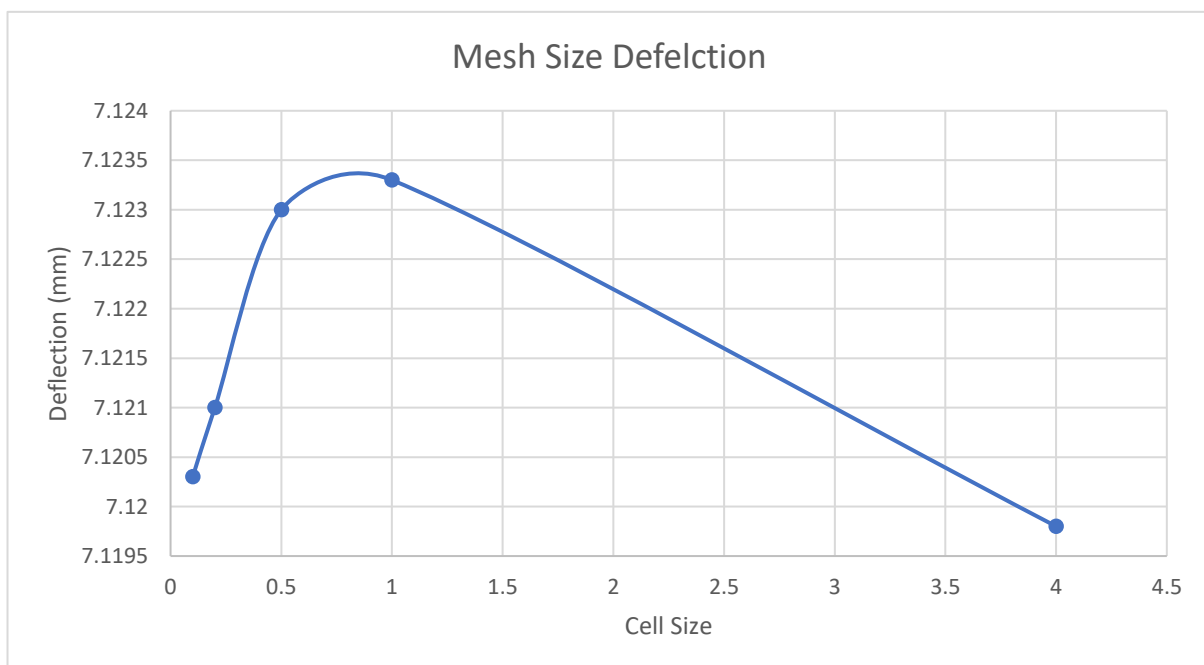


Figure 13-Deflection v cell size

5 Engine Assembly

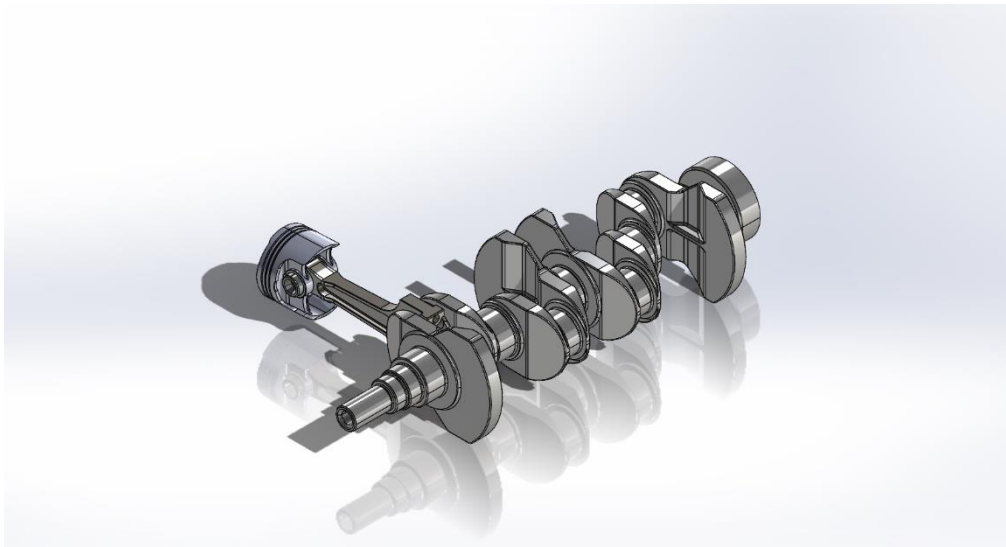


Figure 14- piston assembly

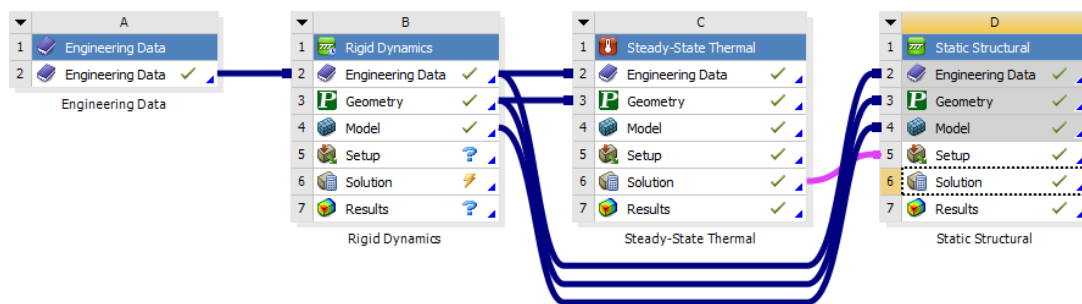


Figure 15- workflow

To complete the task a Piston and crank assembly is used to calculate an approximation of a internal combustion engine. Using a rigid dynamic set up the assembly was constrained around the crank as a revolute joint allowing rotation, which was specified at 4000 rpm. This is where peak loading occurs on the ford sigma data that was provided in previous assignments. The rest of the assembly is constrained appropriately allowing the piston to move in the Z axis in this case simulating the full rotation cycle of a piston.

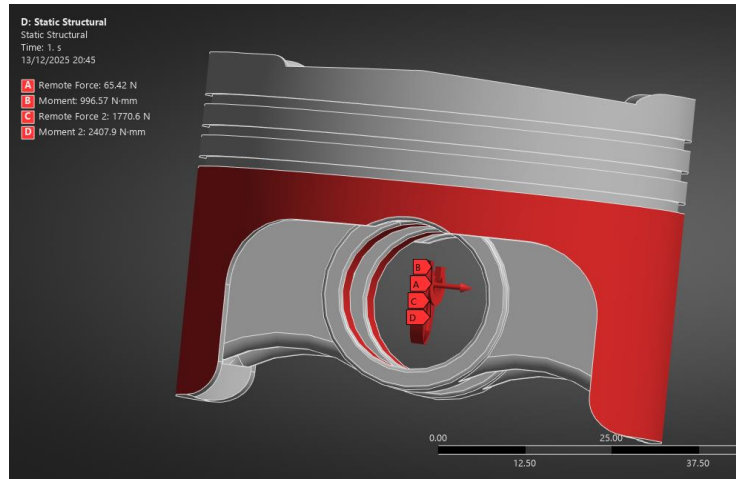


Figure 16- piston head with motion loads

The forces of the gudgeon pin were imported from the rigid dynamic study. This is used later for the static structural component on the piston. The highest load is used to simulate the most extreme conditions the piston will encounter during an engine cycle. This is common in FEA as it is where the highest fatigue will occur and the most likely point of failure will be.

The assembly is then put through a static load setup connected with the rigid dynamic. The assembly is suppressed apart from the piston; the head is changed from rigid to flexible to allow for deformation and the peak motion loads from the crank to be applied to the gudgeon pin.

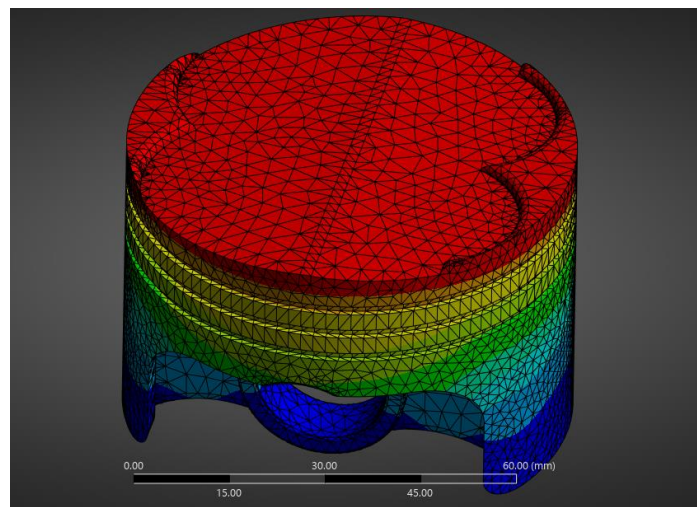


Figure 17-Piston head with thermal loads

A thermal study is conducted and linked to the entire system, coefficients of heat transfer and average piston temps were provided and applied using a convection tool on the surfaces, allowing different sections to be accurately characterised for the heat it experiences within combustion.

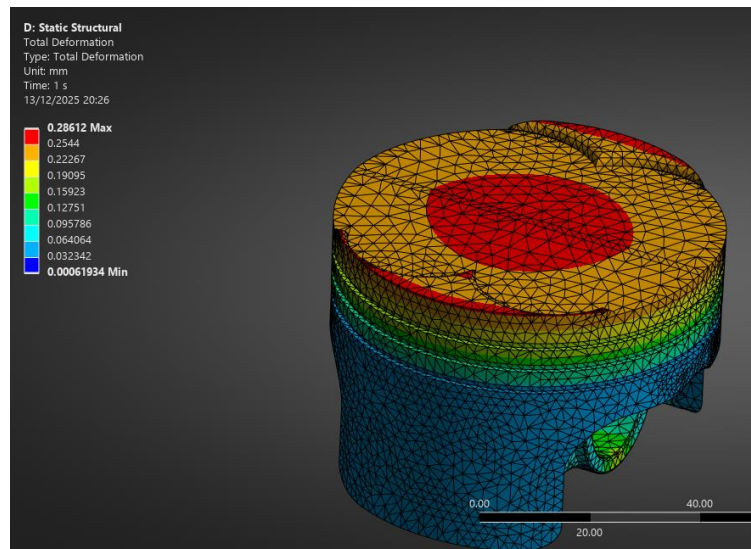


Figure 18-Piston head displacement

With all the loads associated to the engine now simulated with the combustion pressure applied to the crown, the results with deformation, stress and strain distribution can be plotted on the piston head giving an insight into how the piston is responding to the thermal loads and mechanical loading. This is an advantage of FEA, being able to visualise the stresses and deformation on components allowing the user to make informed decisions on where potential optimisation points are and where the component may be likely to fail.

Figure () shows the deformation of the piston head, maximum deformation is experienced at the crown. This is where the highest temperatures and pressure are experienced. The deformation decreases as it moves down the skirt as there is reduced load transfer with the load being transferred to stiffer more constrained regions. At the gudgeon pin there is a slight increase in deformation due to the constraints and the forces acting around them.

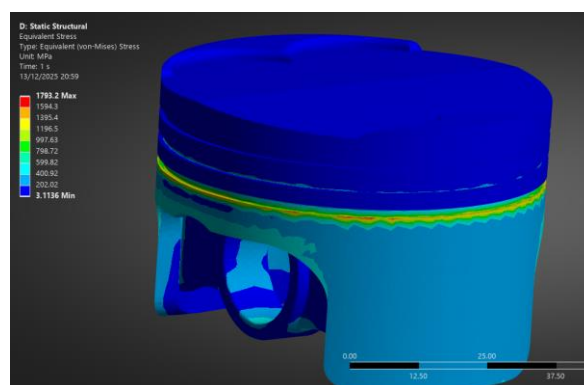


Figure 19- stress on piston

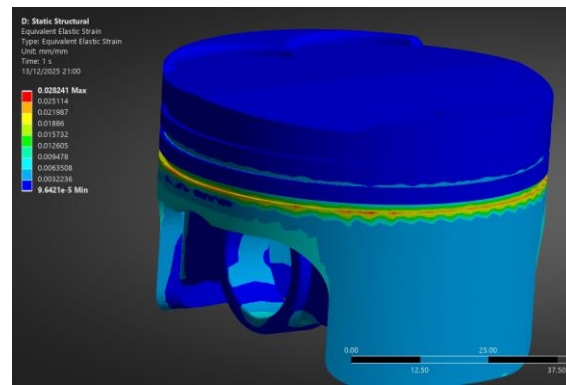


Figure 20-Piston Strain

The Von-Mises stress plot shows a stress concentration around the oil rings, very high point on the lower edge of the piston ring, showing in figure (19) at 1793.2 max Mpa which is unrealistic for an engine of this type and is well beyond the failure point of the material.

This skews the results as the high point is significant, the rest of the piston loses detail across the pressure contours. Resulting in a loss of detail across the rest of the piston making it hard to locate actual pressure points that are occurring.

To fix would potentially need to remodel the lower piston ring to make it easier for the stress to be calculated using a fillet to reduce the sharpness of the corner or a refinement in meshing could potentially help resolve the issue using the sizing tool to localise the sizing. The stress as shown in previous validation testing is highly sensitive to meshing quality. Highlighting a limitation of the software on more complex shapes with some area needing refinement to be able to simulate accurately. (W. Wunderlich, 2013)

Table 1-Material 1 (6061-T6)

6061 t6	min	max	avg	unit
total deformation	3.06E-04	0.16962	6.61E-02	mm
Equivalent Elastic Strain	4.32E-05	1.85E-02	2.60E-03	mm/mm
Equivilant Stress	7.0136	3708.8	480.94	Mpa
Temp	163.94	300.49	246.09	C

Table 2-Material 2 (Mg-12%SiC(p))

MG-12%SiC(p)	min	max	avg	unit
total deformation	6.19E-04	2.86E-01	0.10662	mm
Equivalent Elastic Strain	9.64E-05	2.82E-02	4.01E-03	mm/mm
Equivilant Stress	3.11E+00	1.79E+03	2.35E+02	Mpa
Temp	132.49	345.33	246.32	C

Table 3-Material 3 (A332.0,cast,T6)

Aluminum, A332.0, cast, T6	min	max	avg	unit
total deformation	1.43E-03	3.13E-01	0.11909	mm
Equivalent Elastic Strain	9.10E-05	3.44E-02	4.53E-03	mm/mm
Equivalent Stress	3.2505	2190.5	268.49	Mpa
Temp	145.23	325.38	244.7	C

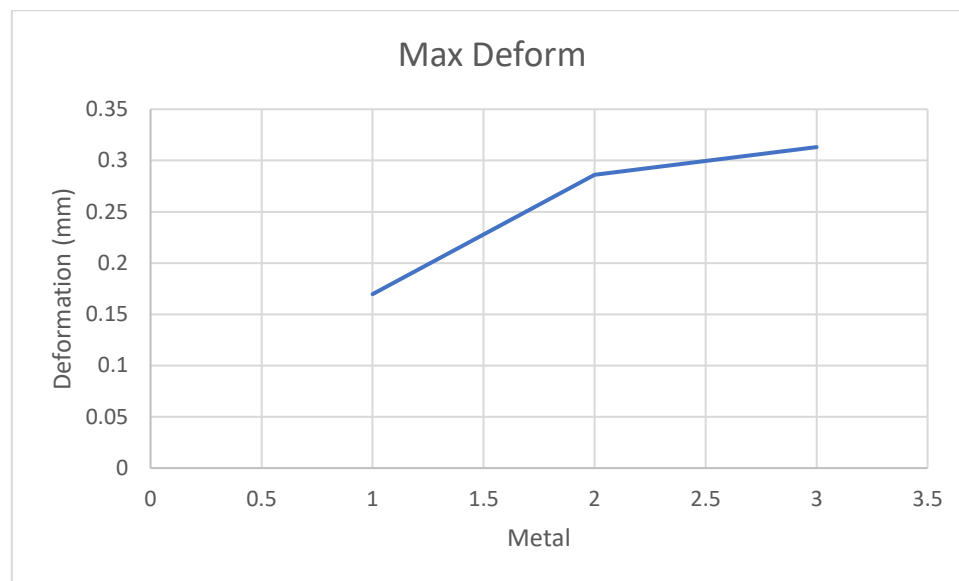
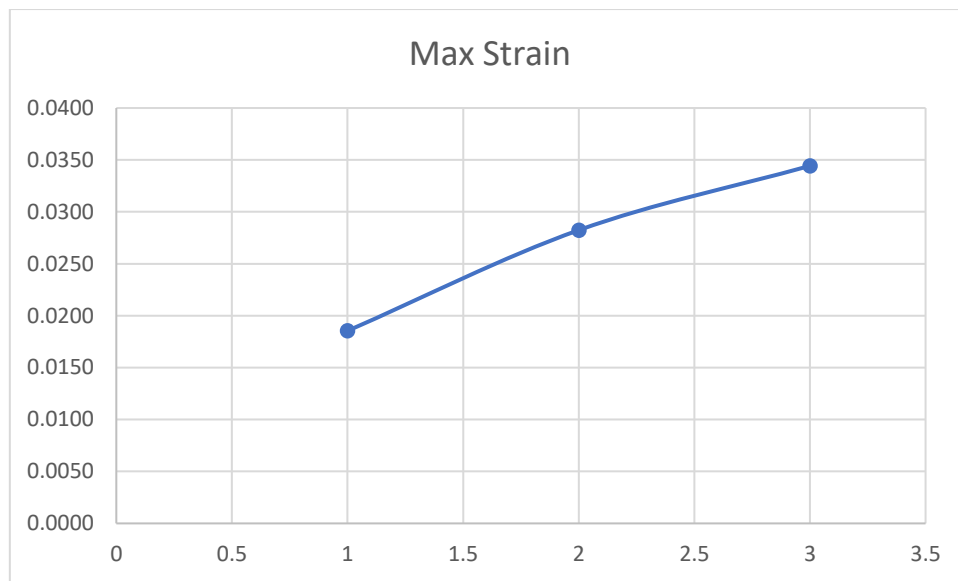
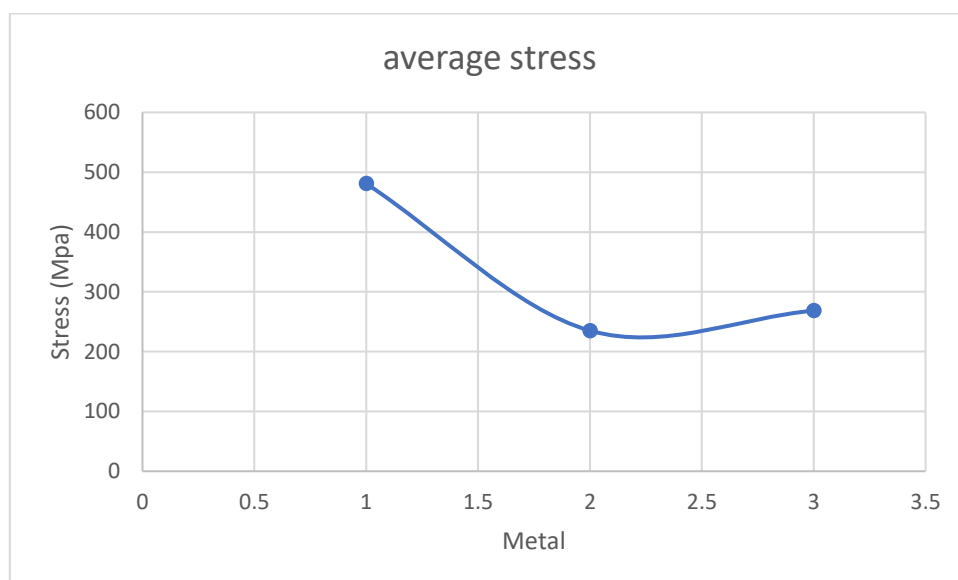


Figure 21-Max deformation on Materials

The 6061 T6 shows the least amount of deformation in comparison to the other 2 materials at 0.17mm but has a higher stress concentration. Material 2 (Mg-12%SiC(p)) shows a higher rate of deformation but figure () shows that the overall stress is lower making it safer structurally. The Alu336 has the largest deformation which potentially be less optimal for sealing the crown and rings where critical such as a piston.

*Figure 22- Max strain on Materials**Figure 23-Average Stress on Materials*

As the max stress on the pistons are excessively high due to the high stress concentration around the oil ring lip, using an average stress will give an indication on what stresses are being applied but again, these results are influenced by the high stress concentration. Figure (23) shows the average stresses, as the von-Mises stresses are always positive the extreme maxs will have a heavier influence on the accuracy of the results but is still able to compare between material to material despite this inaccuracy.

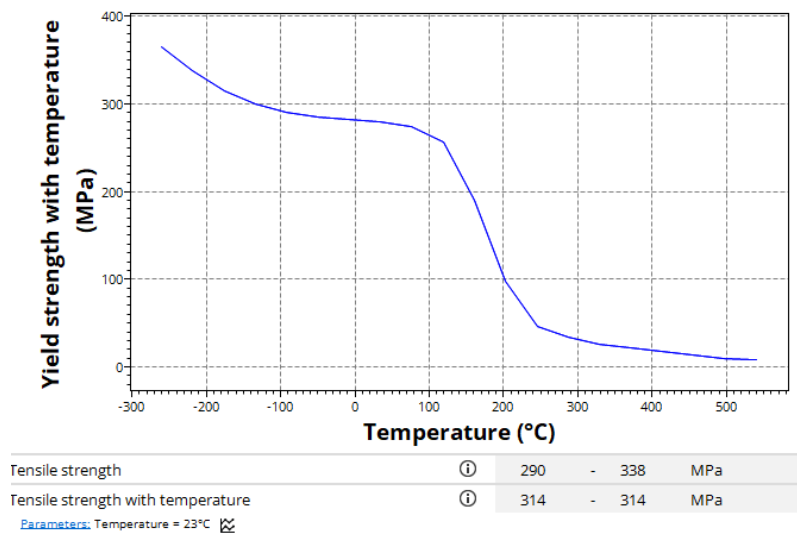


Figure 24-Yield strength v temperature (Ansys, 2024)

Figure (24) shows the temperature vs yield strength for the 6061 t6 aluminium, suggesting that material should fail under these circumstances, highlighting the importance of running simulations under the right thermal and mechanical loads despite these results being inaccurate. By removing the stress concentration a more accurate conclusion can be drawn, the Mg-12%SiC(p) has a higher Yield strength and a higher tensile strength but no data on how its effected at high temperatures.

6 Optimisation

Ansys has a option that is built into the software that allows to reduce the material content on a object, informed using pressure data

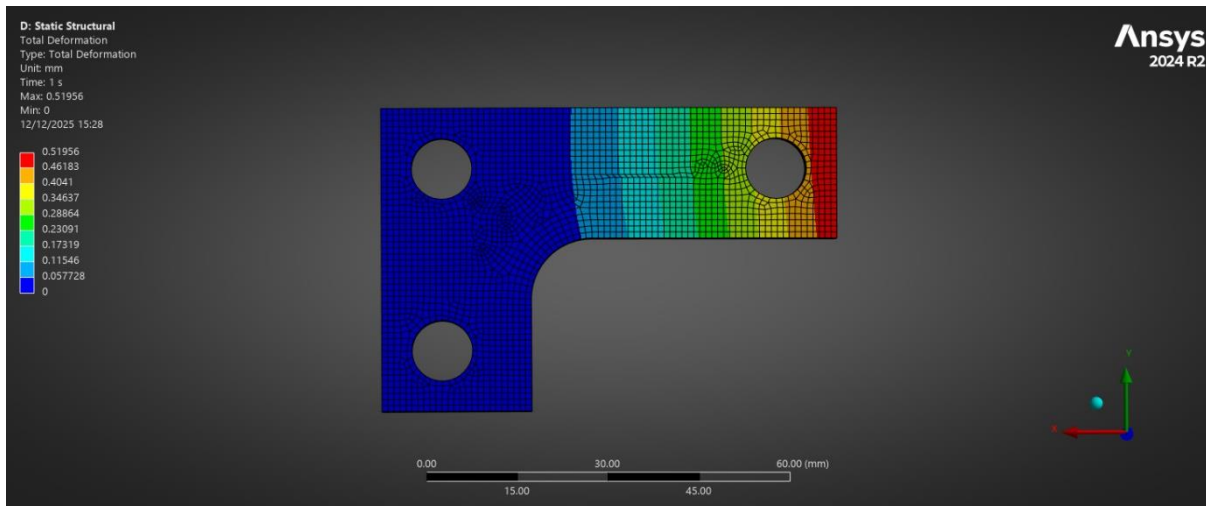


Figure 26- Plate total Deformation

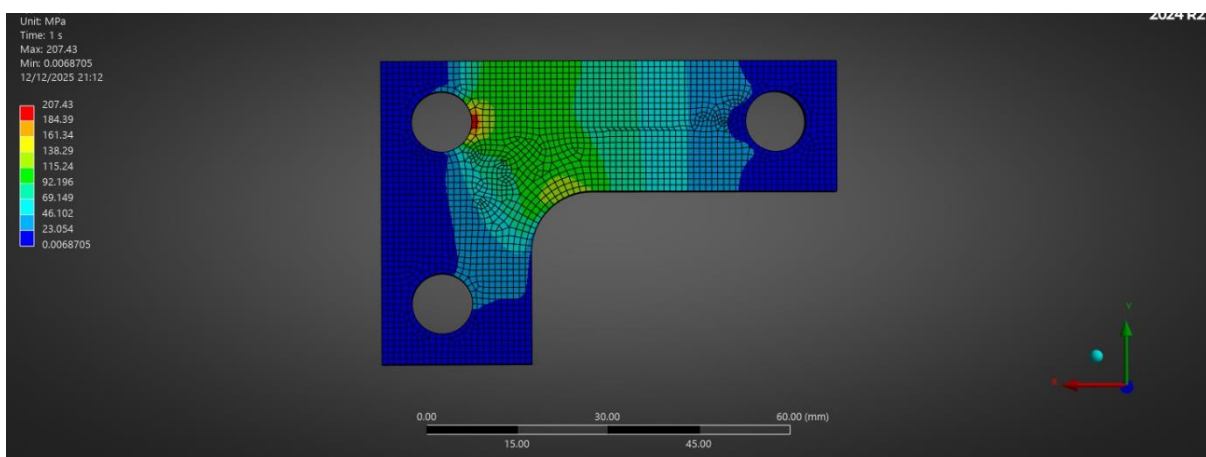


Figure 25-Plate max strain

In figure (26,27) there is a thin plate of metal that is fixed at 2 points on the left with the free end having a force of 100n put through it. There is a max deformation of 0.52mm. The software has been told to reduce the mass by 40%. Using the stress figure, it removes the material to create a more optimal design. There are different solvers that can be used that will calculate in different methods. The optimised structure focuses on load carrying sections meaning excess material is removed

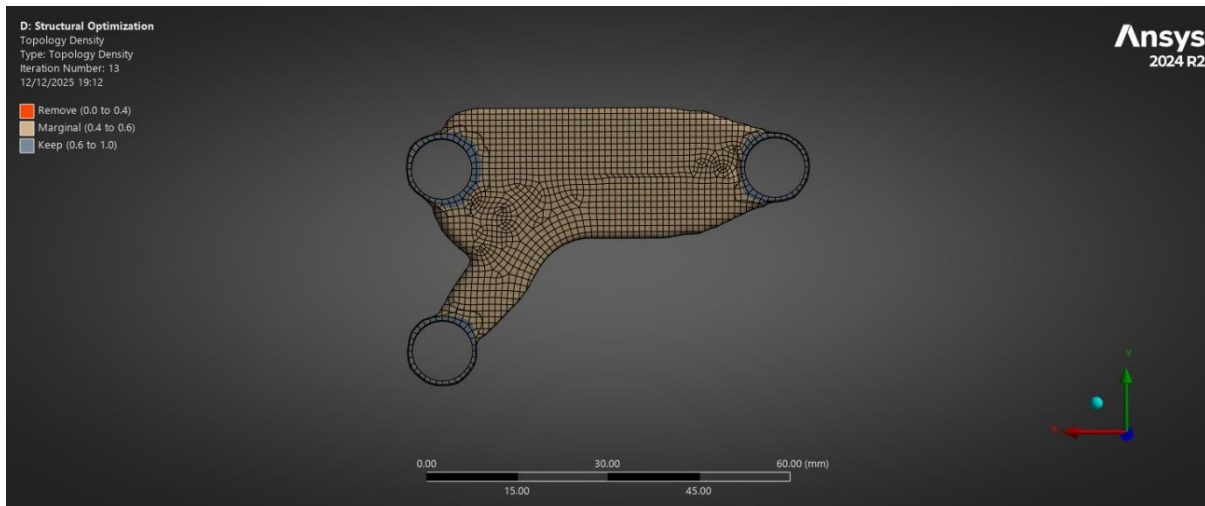


Figure 27- Optimised plate

Figure (27) shows the optimised structure from that run, the material has had a significant loss in the areas where little stress is experienced. The area where the stress is experienced remain relatively thick with this solver leaving it due to the stress applied. The part will now likely experience more force throughout as the area is reduced but its within an acceptable range. This should be determined by a safety factor.

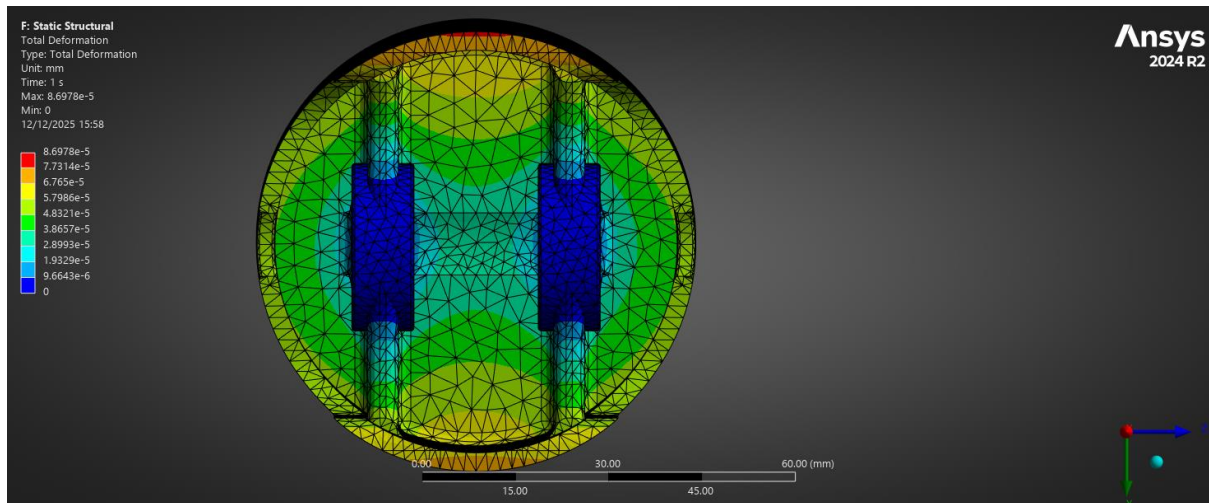


Figure 28-Deformation on piston underside

The piston is next to be optimised, using the same method of removing mass from the piston

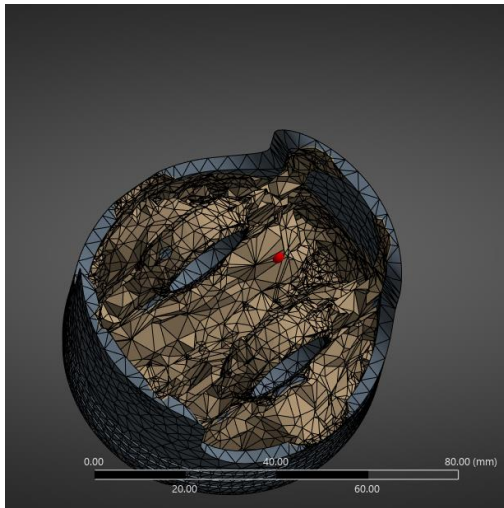


Figure 30-Piston optimised underside

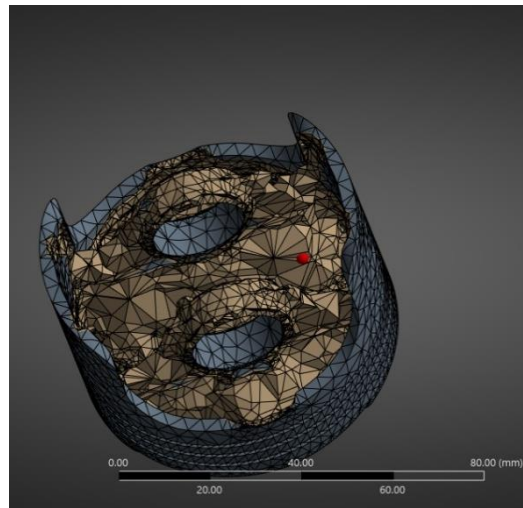


Figure 29- Piston optimised underside

Figure (29,30) shows the results of reducing the mass by 40%. The underside of the piston is the only area it was applied to due to the skirt, piston head and gudgeon pins being preserved so the shape and function remain as a piston. The optimisation tool created a tree like structure with it removing any non-load bearing material. The way the piston has been set up is the pressure is being exerted in 1 direction on crown of 6.7Mpa which is the peak pressure of the sigma engine used in a previous assignment. Meaning that not all forces are being accounted for when removing material potentially making it weaker in certain areas that will need supporting during a cycle. Having a factor of safety when optimising is a crucial part of the design process.

Starting mass 0.74364 kg

Final mass 0.50539 kg

7 Limitation

FEA tools such as Ansys give powerful insight into the component's behaviour. Several limitations affect the reliability and real-world application of simulation results. Limitations arise from geometrics simplification. Mesh related uncertainty, material and thermal assumptions, boundary condition idealisation and computational constraints.

Finite element analysis is highly dependent on mesh quality. Most noticeably the stress and strain results as discussed in the validation cases, the behaviour is consistent with finite element theory that stress and strain are a derivative of displacement making it more sensitive to mesh sizing. (Douglas H. Norrie, 1973)

Boundary conditions that are used can limit the accuracy of simulation uncertainty in boundary conditions or incorrect inputs can lead to results that differ significantly to real operating behaviour. Errors in applied force magnitude and direction, unsuitable thermos conditions or incorrect definition of support location can significantly influence the predicted stress and deformation. (Ansys, 2025)

Material selection and behaviour will also affect the result. Simulated material is consistent where the piston will in real life experience creep, plastic deformation to an extent and perfect operating conditions. Material property data obtained by Granta was used to improve the accuracy of the model, but actual material will have a degree of error during production. This includes the surface finish and heat treatment if applicable. (Ansys, 2024)

Thermal analysis was conducted using a steady state condition using average heat transfer to calculate the temperature on the piston. Although the forces were captured, a transient thermal solver would provide a more accurate representation of temperature variation over the full piston cycle. The heat was averaged from one piston to another, a degree of error will be introduced. The information for the heat transfer is assumed to be the same for the other 2 materials, as data for these specific materials and locations on a piston head is very difficult to find. As identical thermal parameters were applied to all the materials because of limited data an additional source of uncertainty is introduced.

The loads calculated are on a static structural node, which helps simplify the results and give a good idea on maximum forces applied to the piston at a single point in time and emphasis on the max loads experienced. This will not capture the inertial forces fully also the reversals of loads isn't captured as all motion loads are calculated at the same time.

Friction in this simulation is not modelled, the thermal numbers themselves will be accounting for the majority of the combined heating from the combustion and movement of the piston, but the actual implication of friction and thermal expansion of materials are not modelled fully.

8 Conclusion

In conclusion Finite Element Analysis is a proven effective tool for evaluating structural behaviour of components that are subjected to complex thermal and mechanical loading. The study shows FEA successfully applied to a piston head that replicates the forces seen at peak loading, allowing analyses of the deformation and strain that occurs and some conclusions being drawn from the findings.

The results from the validation show the sensitivity of stress depending on cell sizes and the effect on deformation. It also shows the importance of correct boundary condition being applied in relation to the accuracy of the results. The results of the piston head showed some limitations to the software where the sharp corner contributed to a miss calculation if the max stress throwing off the overall result. Showing that the validation on the simple objects isn't necessarily true of that of a complex shape.

The material study showed the 6061-t6 aluminium had the least amount of deformation but a higher stress concentration and a big reduction in yield strength at high temperatures. The Mg-12% showed improved stress distribution with increased deformation, indicating a better structural component.

The optimisation of parts is a useful accessory to FEA using the structurally loaded parts of the material to reduce excess material-based strain and stress. the result of the reduction is a messy tree like structure that will be hard to manufacture but can be used to inform future designs with reduced material based on the information.

Overall, the report demonstrates that FEA is a powerful design and assessment tool when validated appropriately, realistic assumptions and engineering judgment. While the results should not be treated as an exact representation of real-world performance it can provide a strong basis for identifying critical flaws and guide material selection and capable of refining the component further.

9 References

© Dlubal Software 2015, n.d. *Verification Example: 0042 – Bending Cantilever with Axial Force*. [Online]

Available at: <file:///C:/Users/2203893/Downloads/Cantilever%20with%20Axial%20Force.pdf>
[Accessed 14 12 2025].

© Dlubal Software 2018, n.d. *Verification Example: 0078 – Bimetallic Strip*. [Online]

Available at: <file:///C:/Users/2203893/Downloads/Bimetallic%20Strip.pdf>
[Accessed 14 12 2025].

Ansys, 2024. *Granta, level3 materials*. [Online]

[Accessed 12 12 2025].

Ansys, 2025. *ANSYS Inc. (2023). ANSYS Mechanical User's Guide..* [Online]

Available at:

https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/corp/v251/en/wb_sim/ds_Mesh.html

[Accessed 14 12 2025].

Douglas H. Norrie, G. d. V., 1973. *The Finite Element Method: Fundamentals and Applications*.

London: Harcourt Brace Jovanovich.

W. Wunderlich, . S. . B., 2013. *Nonlinear Finite Elements for Continua and Structures.*, s.l.: Springer Berlin Heidelberg.