



University of Wales Trinity Saint David

Wales Institute of Science & Art

Application of CAE Processes

Structural & Fluid Analysis

ACHA6010

Dylan Brierley

2304450

12/2025

Contents

1	Introduction	3
2	Technical Literature Review.....	4
2.1	CAE in Engine Component Design	4
2.2	Validation Practice in Published Studies	5
3	Validation Testing	6
3.1	Uniform Cantilever Mesh Study	6
3.2	Axial Cantilever Mesh Study	8
3.2.1	Mesh Quality - Skewness	10
3.3	Bimetallic Strip Thermo-Mechanical Mesh Study	11
3.4	Topology Methods Mesh Study.....	13
4	Kinematic Simulation and Structural Analysis	15
4.1	Application of Material and Boundary Conditions	15
4.2	Kinematic Analysis - Motion Outputs & Loading Conditions.....	16
4.3	Piston Steady State Thermal Study	17
4.3.1	Convection Loads	17
4.3.2	Piston Temperature Profile	17
4.4	Piston Static Structural Analysis	18
4.4.1	Face Sizing	18
4.4.2	Boundary & Load Conditions.....	18
4.4.3	Equivalent Stress (Von Mises)	19
4.4.4	Total Deformation	19
5	Optimisation and Iterative Improvement.....	20
5.1	Material Iteration 1 – 2618-T6 Aluminium.....	20
5.2	Material Iteration 2 – 42CrMo4 Steel.....	21
5.3	Material Comparison & Optimisation Choice.....	23
5.4	Design Mass Reduction – ANSYS Topology	26
5.5	Final Design Choice.....	28
6	Critical Conclusions of Results and Limitations.....	29
	References	30
	Appendices.....	32
	Appendix A – Axial Cantilever Total Deformation Fz Graph	32
	Appendix B – Piston Thermal Study Data	32

1 Introduction

This purpose of this report is to apply various computer aided engineering (CAE) processes within ANSYS Mechanical finite element analysis (FEA) software to evaluate common design practice and direct limitations. This includes testing the software accuracy against analytical solutions (validation), using an engine assembly to perform a structure analysis, and then optimising mass and structural strength through iteration and a structural optimisation solver (topology). Limitations are detailed throughout and concluded at the end, with explanations and further accuracy refinement work.

2 Technical Literature Review

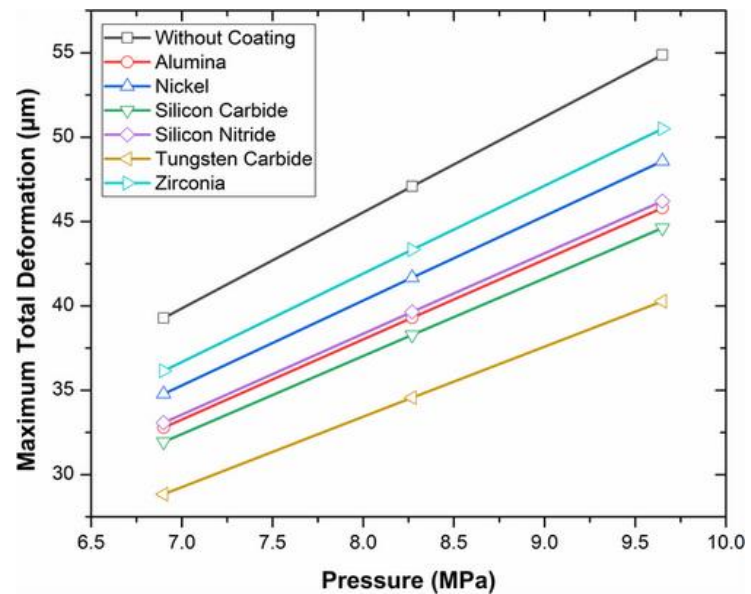
2.1 CAE in Engine Component Design

CAE within engine component design is very important to lots of scenarios are tested and to reach component limits.

The graph to the right shows some interesting research carried out in FEA to evaluate the effect on deformation with different cylinder coating materials.

The reason that FEA is a useful tool for this purpose is that the test was able to include a wide range of materials without having to manufacture or source parts to test practically.

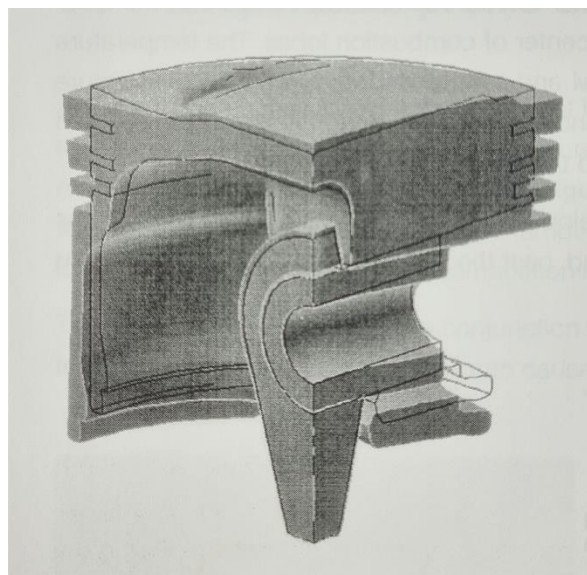
A material test like this may provide a couple of good materials to then put through to a practical test, as it is important to realise the few limitations of FEA.



(**Graph 1:** Cylinder Material Deformation (Khan, H. et al. 2025))

The study above is relevant to this report as I also aim to test multiple materials against factors like deformation to theoretically optimize an engine component.

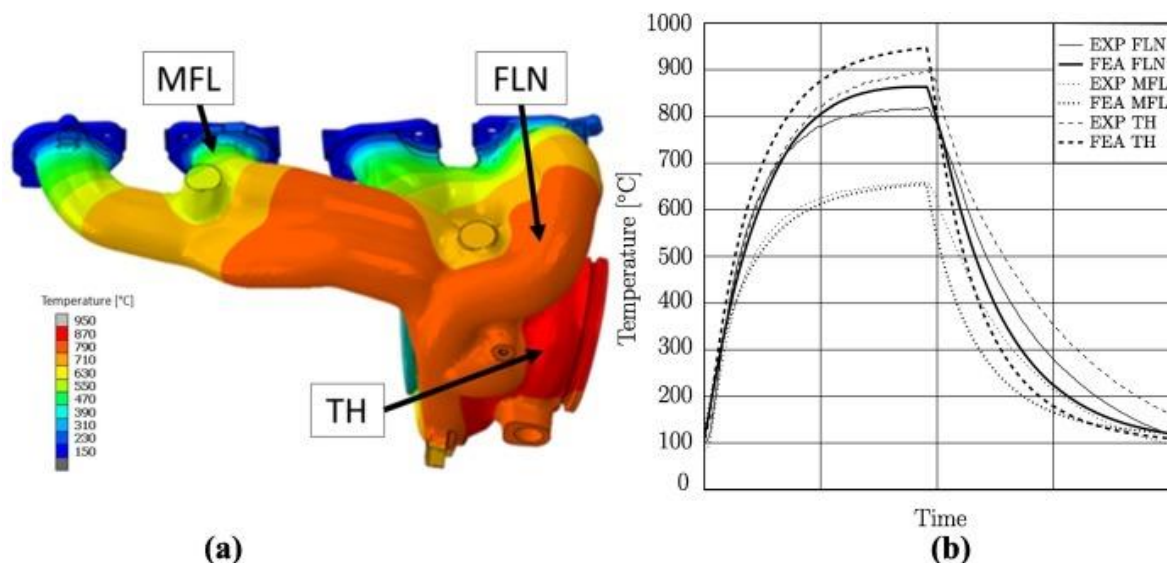
I also intend to include thermal conditions, and the figure below by engine manufacturer MAHLE shows what deformation should be seen within thermal piston testing.



(**Figure 1:** Piston Thermal Deformation (MAHLE GmbH, 2016))

2.2 Validation Practice in Published Studies

Validation within CAE is very important to ensure values are within the threshold that would be acceptable comparing to practical or analytical approaches.



(a)
(b)
(Figure 2: Exhaust Manifold FEA Comparison (Lorenzini, M. et al. 2023))

Figure 2 above is from a published study comparing a practical exhaust manifold thermal shock test against the FEA counterpart. The study quotes a 2-3% range of accuracy between the tests, which they consider to be an accurate FEA replication of the test. I will be aiming for a similar range to compare simple analytical solutions to Ansys solvers.

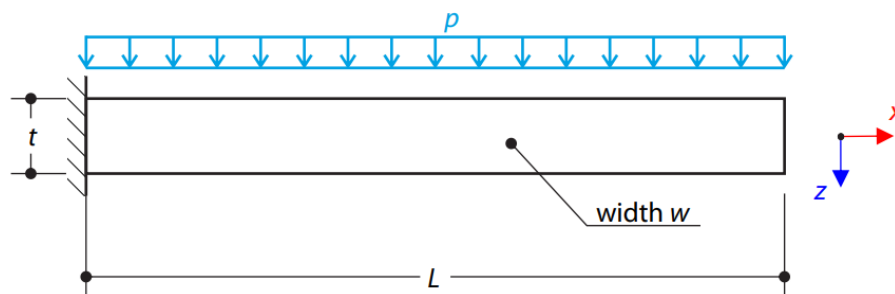
3 Validation Testing

3.1 Uniform Cantilever Mesh Study

To begin the mesh validation study, I have used a published source to copy the same load and boundary conditions to compare to their simple analytical result. Below are the properties of the material and a diagram of the conditions used for this test.

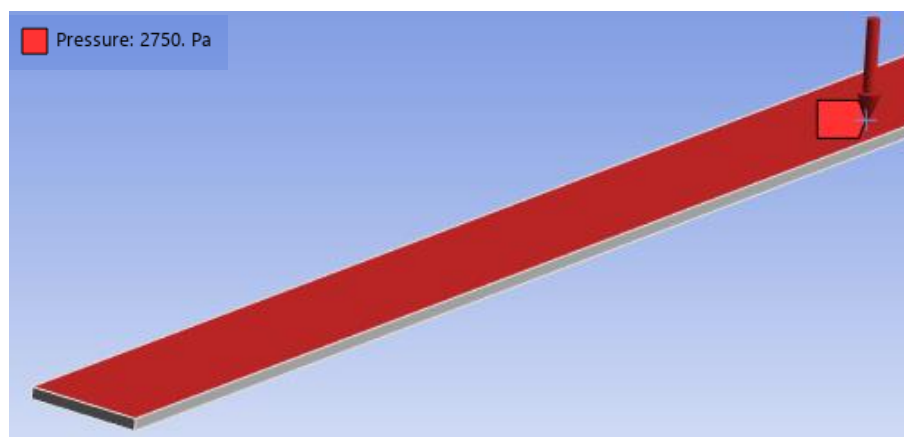
Material	Elastic-Plastic	Modulus of Elasticity	E	210000.000	MPa
		Poisson's Ratio	ν	0.000	
		Shear Modulus	G	105000.000	MPa
Geometry	Plate	Length	L	1.000	m
		Width	w	0.050	m
		Thickness	t	0.005	m
Load	Plate	Pressure	p	2750.000	Pa

(Table 1: Cantilever Uniform Load Properties (Dlupal, 2025))



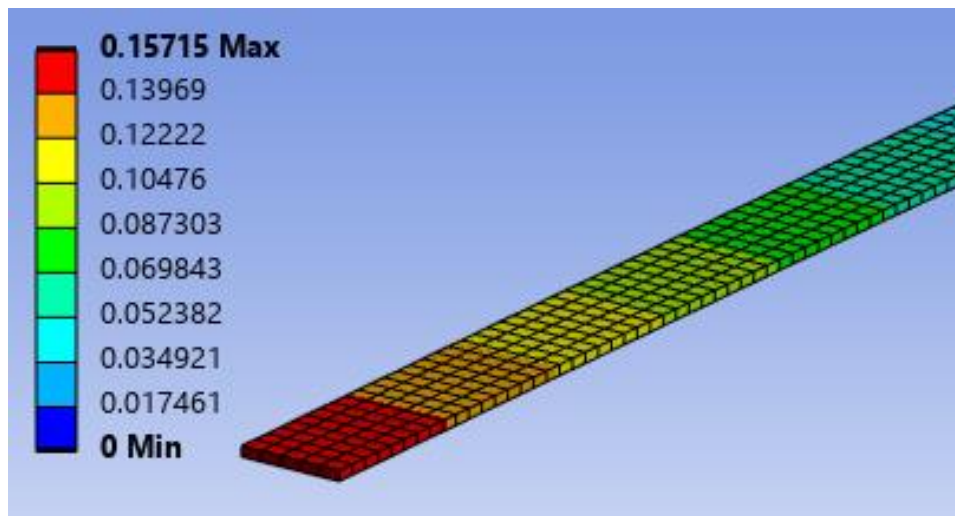
(Figure 3: Cantilever Uniform Load Conditions (Dlupal, 2025))

Shown in figure 4 below is the beam with a uniform pressure load and one end fixed.



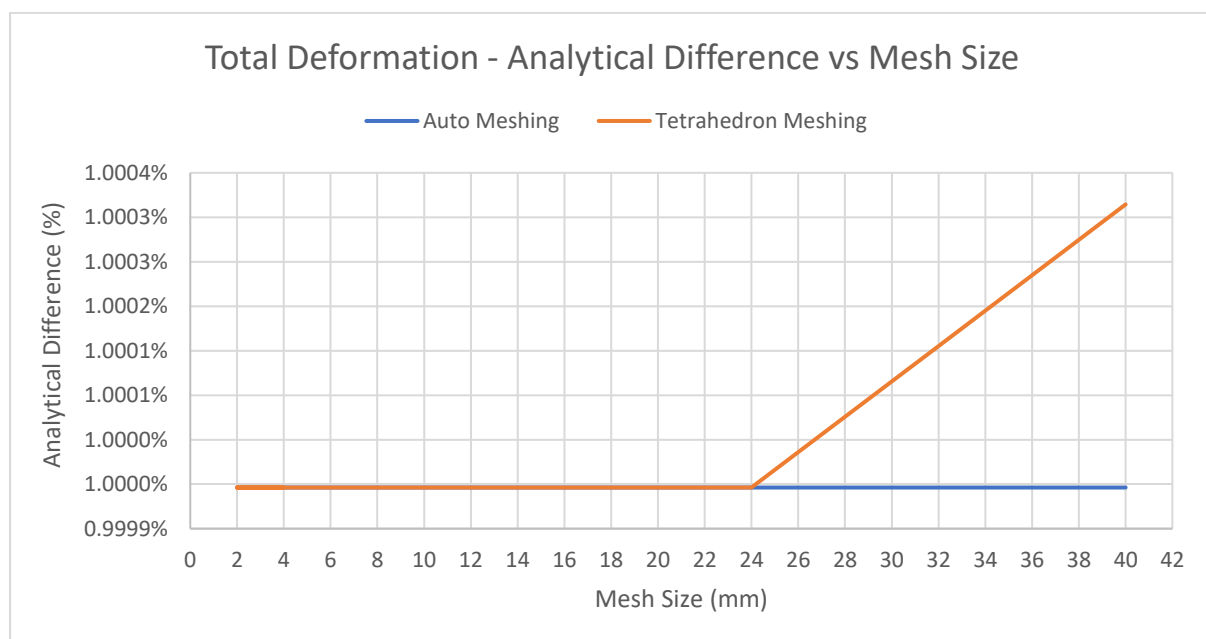
(Figure 4: Uniform Pressure Load)

I was able to test auto meshing, which chose a hex dominant based mesh approach due to the square shape, against tetrahedron meshing, at various mesh sizes. (GridPro, 2025)



(Figure 5: Total Deformation - Metres)

As shown in graph 2 below, there was no difference in deformation over mesh sizes with auto meshing, and very small differences with bigger sizes for the tetrahedron mesh.



(Graph 2: Analytical Result Comparison – Cantilever Deformation)

Compared to the analytical solution provided, the runs were around the 1% accuracy mark, which is a close enough result as the solver may have applied the maths slightly differently, including potential rounding errors within both calculations. There was very little percentage difference in the tetrahedron mesh, despite changing the deformation, as it was only a fractional decrease.

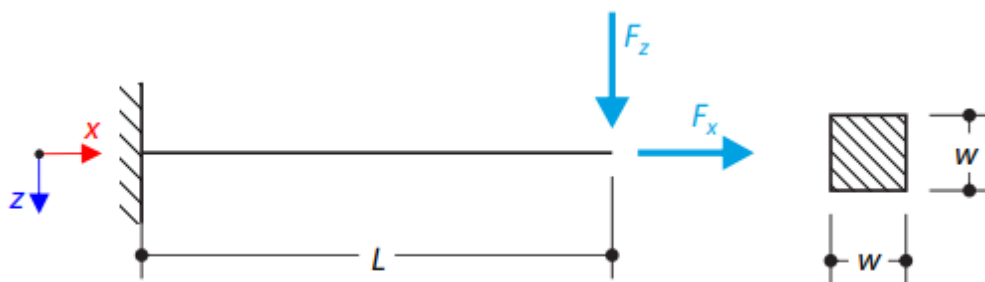
The reason for the lack of change could be due to the part being rather thin and easy to bend, this could mean there was a lack of detail needed to model the overall deformation. I expect the slight difference seen in the tetrahedron mesh would be down to a poorly fitting mesh shape for a rectangular beam, hence why auto meshing chose a square based mesh. (GridPro, 2025)

3.2 Axial Cantilever Mesh Study

Next, I applied an axial load to a different sized steel cantilever beam to evaluate meshes under a different set of conditions.

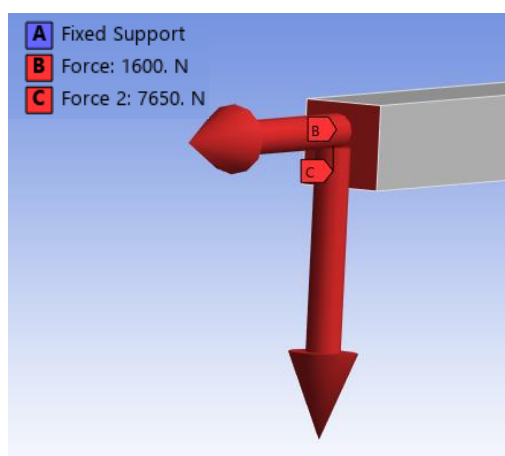
Material	Steel	Modulus of Elasticity	E	210000.000	MPa
		Poisson's Ratio	ν	0.300	—
Geometry	Cantilever	Length	L	10.000	m
		Cross-section width	w	0.080	m
Load		Transversal Force	F_z	7.650	kN
		Axial Force	F_x	1.600	kN

(Table 2: Cantilever Axial Load Properties (Dlubal, 2025))

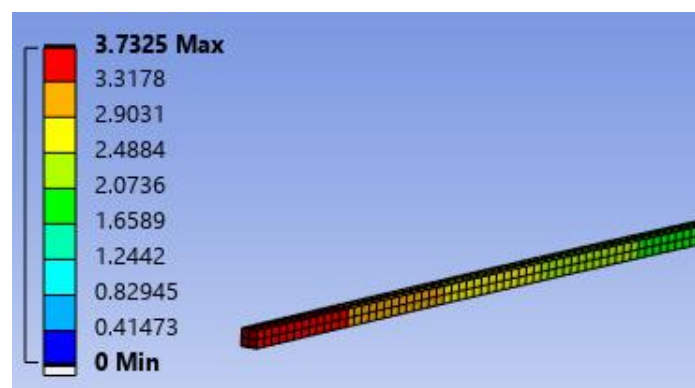


(Figure 6: Cantilever Axial Load Conditions (Dlubal, 2025))

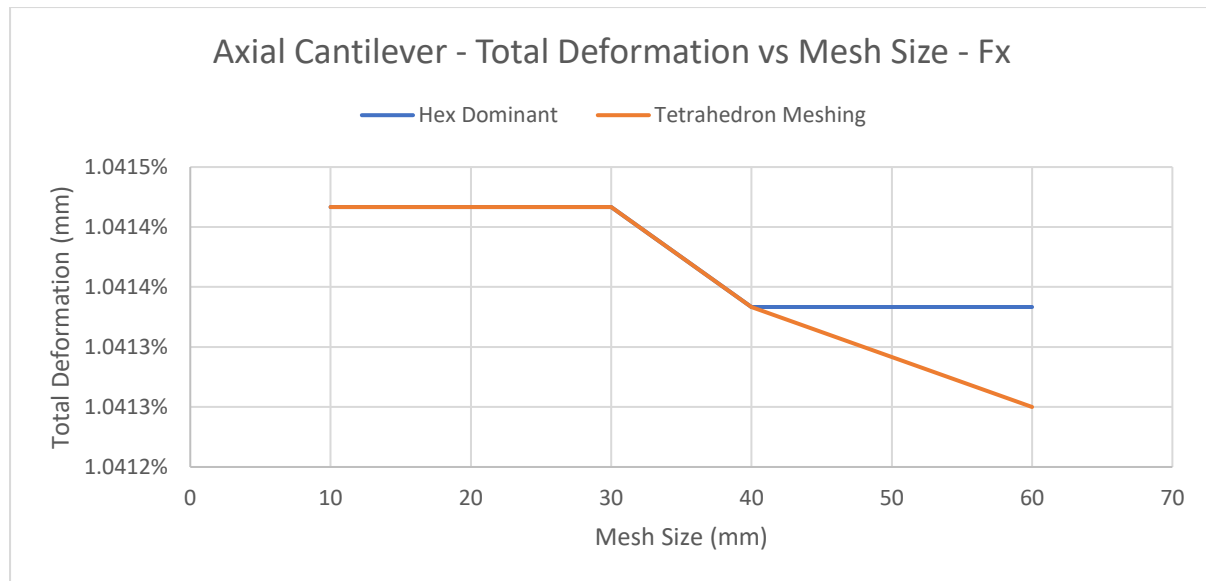
I applied the load to the free side as shown in the figure below, I was then able to split the Z and X directed deflection to match the analytical study data.



(Figure 7: Axial Loads)

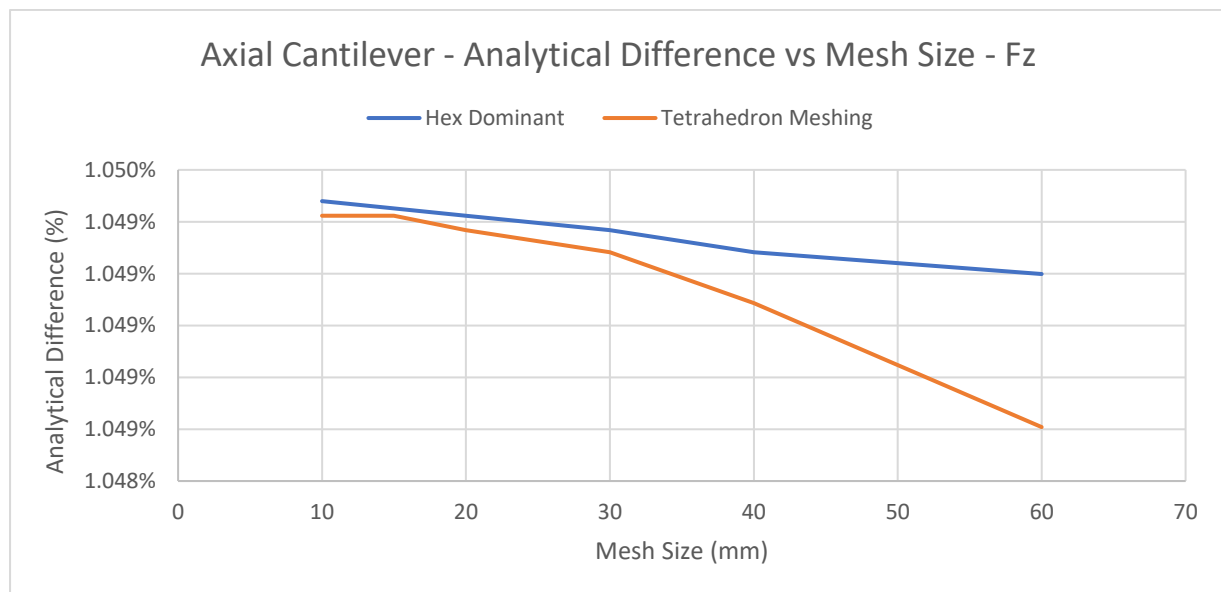


(Figure 8: Total Deformation - Metres)



(Graph 3: Analytical Result Comparison – Fx Deformation)

Graph 3 above shows a similar result to the last uniform test with minimal change and with hex dominant changing less than tetrahedron meshing. The trend suggests that bigger meshing has greater accuracy, but this could just be a coincidence of the maths application, as the change is so minor.



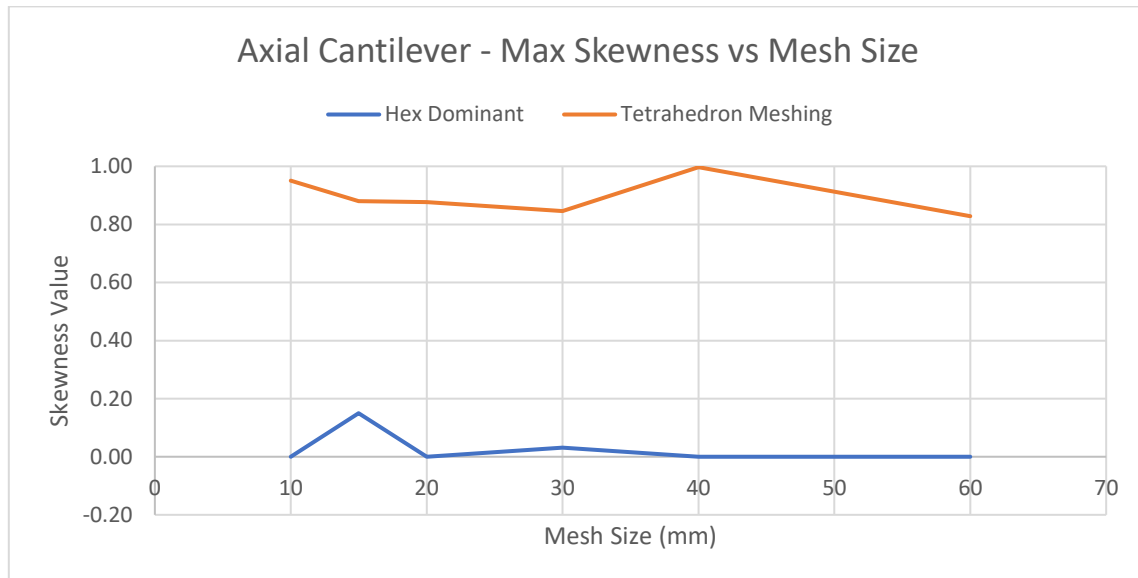
(Graph 4: Analytical Result Comparison – Fz Deformation)

Deformation in the Z direction provided slightly more of a distinct trend within the results, with the percentage change being bigger by a factor of 10. The difference is still very marginal, being around 3mm difference in deflection between the smallest and largest tetrahedron mesh (reference Appendix A).

While this mesh testing doesn't directly link to the assembly considered later in the report, it has helped to understand the mesh considerations within the software. Once convergence has been achieved, further mesh refinement does not significantly improve accuracy and may increase numerical errors due to ill-conditioning and round-off effects (Zienkiewicz, O. C., Taylor, R. L., & Zhu, J. Z. 2014).

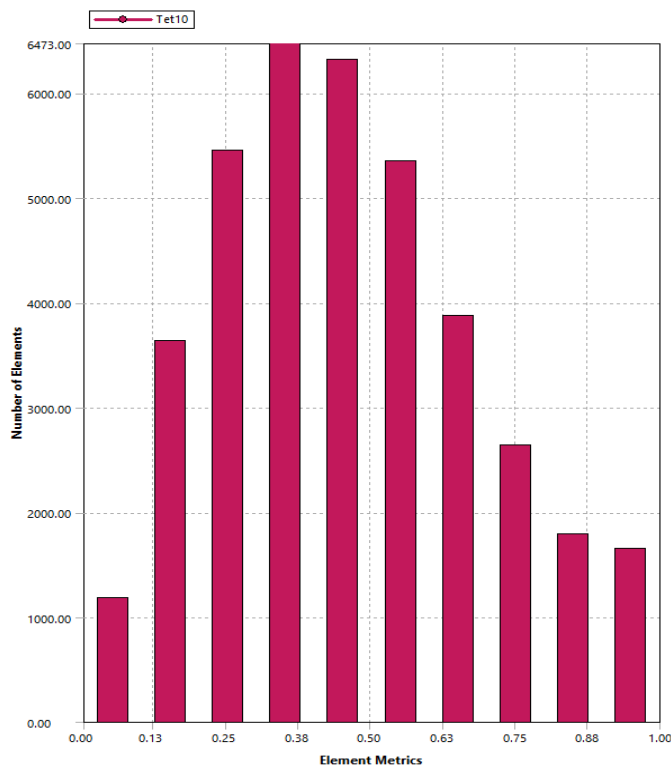
3.2.1 Mesh Quality - Skewness

As mentioned within FEA research and the ANSYS website, skewness is one of the primary tools used to measure mesh quality (Ansys, 2024). I decided to utilise this tool to check how the mesh on the axial cantilever performs.



(Graph 5: Skewness vs Mesh Size – Axial Cantilever)

The graph shown above confirms that the tetrahedron mesh is a poorly fitting mesh for this shape, as the shapes have been skewed heavily to fit the shape. Hex dominant is a lot higher quality, being near 0 on most mesh sizes apart from 15mm, which appeared to be a slight anomaly, potentially due to the odd mesh sizing not matching the even geometry.



(Graph 6: Mesh Elements Skewness Value – Piston)

This may suggest that tetrahedron meshing would be better applied to less square, more complex shapes where it wouldn't have to skew as much to fit.

However, as shown by graph 6 on the left with a 2mm tetrahedron mesh applied to a piston, there are a lot of skewed mesh elements even on the complex piston shape.

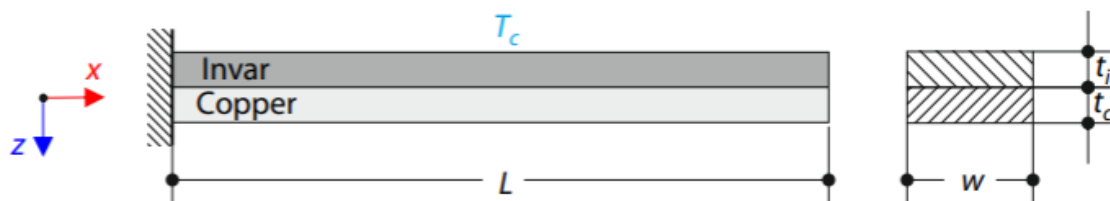
To completely refine these there would need to be extensive further work performed using methods like local mesh sizing to complex areas, or even slight geometry adjustments.

3.3 Bimetallic Strip Thermo-Mechanical Mesh Study

To validate the thermal solver within Ansys I was able to use an analytical study based on a bimetallic strip with Invar and Copper, thermally loaded to evaluate deformation with heat.

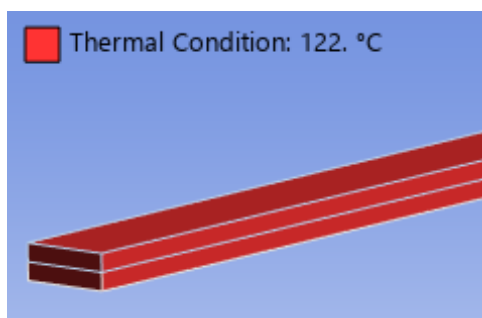
Material	Invar	Modulus of Elasticity	E_i	137000.000	MPa
		Poisson's Ratio	ν_i	0.280	—
		Coefficient of Thermal Expansion	α_i	1.200×10^{-6}	$^{\circ}\text{C}^{-1}$
	Copper	Modulus of Elasticity	E_c	130000.000	MPa
		Poisson's Ratio	ν_c	0.354	—
		Coefficient of Thermal Expansion	α_c	2.000×10^{-5}	$^{\circ}\text{C}^{-1}$
Geometry	Cross-section Width		w	5.000	mm
	Layer Thickness		$t_i = t_c = t$	1.000	mm
	Length		L	100.000	mm
Load		Thermal Loading	T_c	100.000	$^{\circ}\text{C}$

(Table 3: Bimetallic Strip Load Properties (Dlubal, 2025))

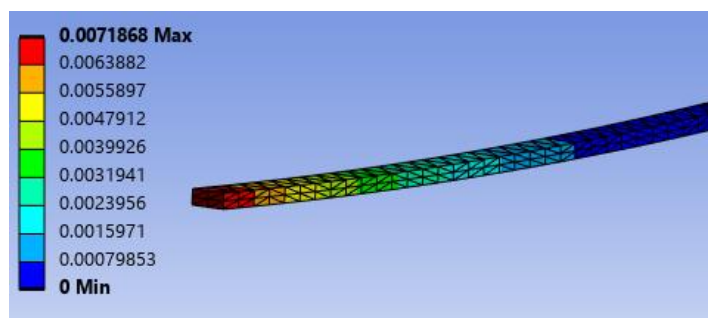


(Figure 9: Bimetallic Strip Load Conditions (Dlubal, 2025))

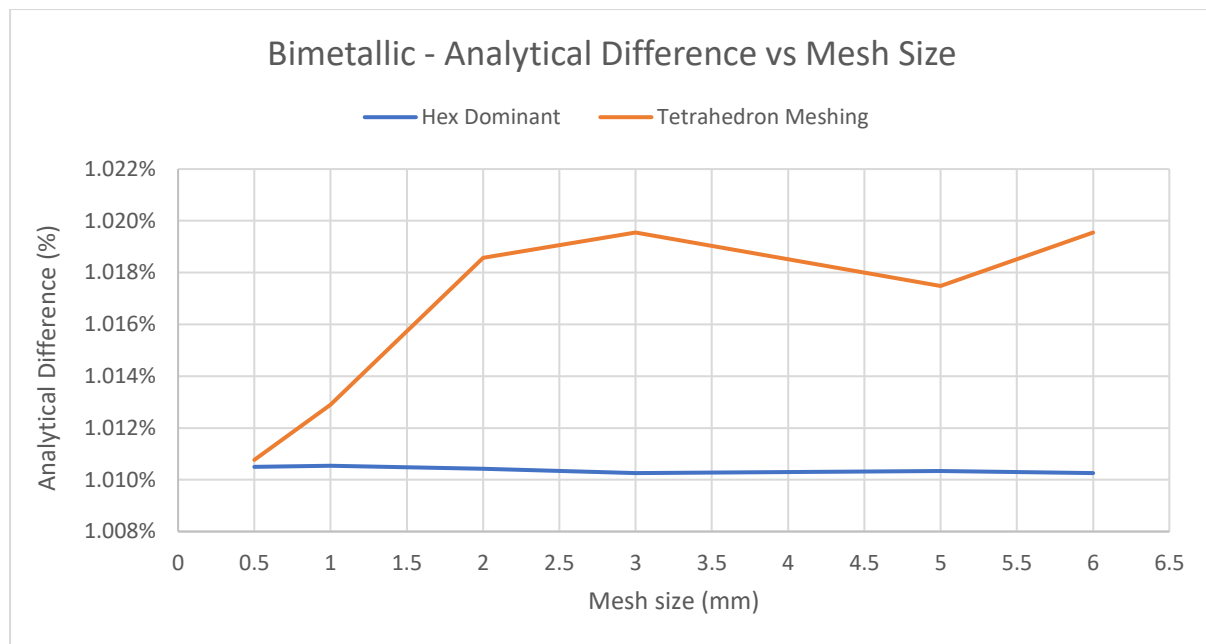
The base temperature that the solver works from is 22°C so after applying 100°C the temperature reached 122°C as shown below.



(Figure 10: Thermal Load)



(Figure 11: Total Deformation – Metres)

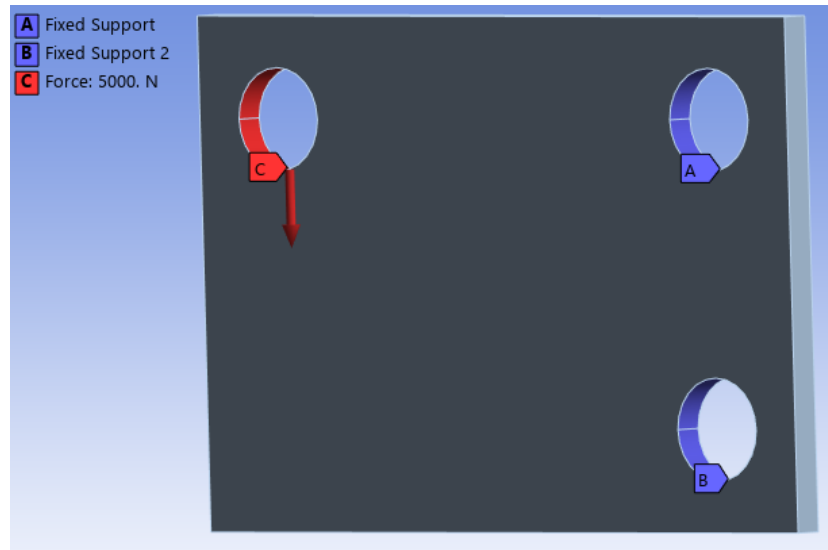


(Graph 7: Analytical Result Comparison – Bimetallic Deformation)

Graph 7 shows how the temperature load produced a wider variation of results for tetrahedron meshing and suggesting that hex dominant was more accurate to the analytical solution. The meshes tested were also much finer compared to previous validation testing, showing how only a millimetre of mesh change affected the results more than uniform or axial loads.

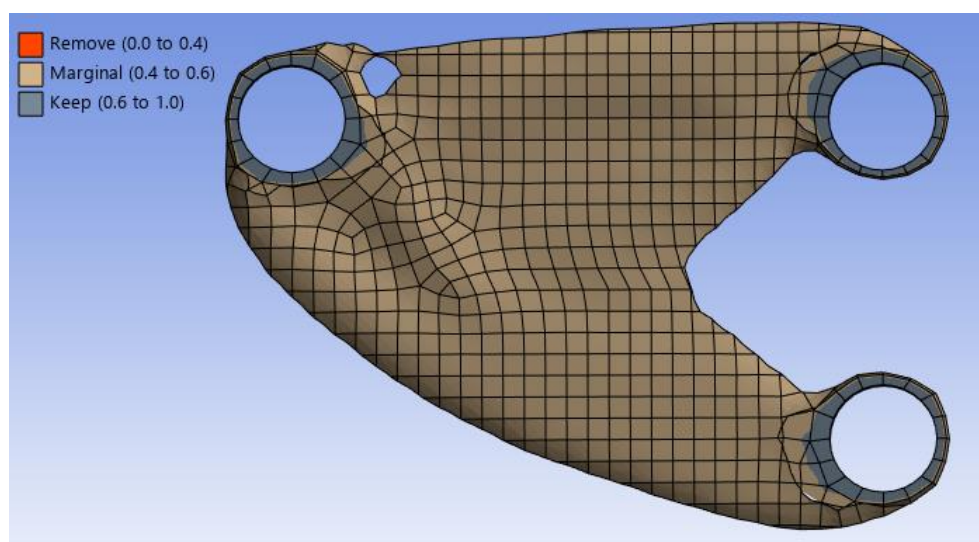
3.4 Topology Methods Mesh Study

Considering the use of topology later in this study, I decided to try and validate 2 of the solver methods using a simple plate design, recording weight removal accuracy and shape produced. The plate is made from Ansys default structural steel, as the material isn't hugely relevant with a focus on mass and mesh size, and requesting a 50% weight reduction/retention.



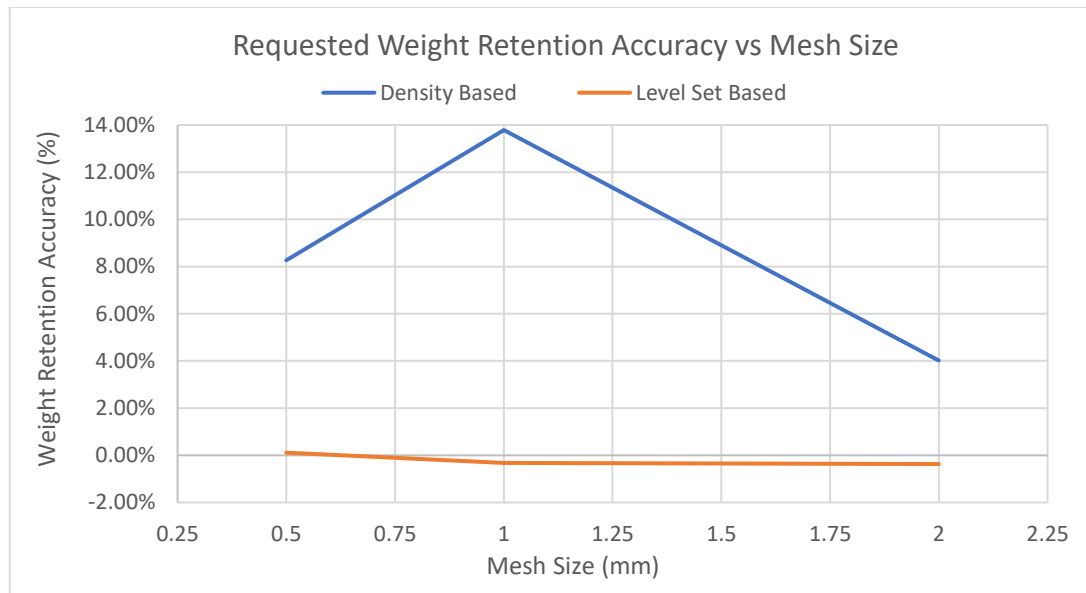
(Figure 12: Metal Plate Topology Load & Boundary Conditions)

Above are the load and boundary conditions, the region for mass optimisation within the solver is anywhere other than these condition areas. I utilised density based and level set based topology methods to understand which would be a better option for accuracy and structure, both were tested using auto meshing.



(Figure 13: Metal Plate Density Based Topology – 2mm Mesh Size)

Figure 13 above shows the first optimisation made with density based topology, there is a key in the top left to understand what material is marginal for removal. Interestingly, the solver classes a lot of the material as marginal, which could be a result of this method being known for outputting less polished shapes with a larger post-processing requirement. (Ansys, 2025a)



(Graph 8: Retention Accuracy vs Mesh Size)

The graph above details the accuracy of how close the solver method gets to the requested 50% reduction/retention. Density based is unable to reach the level set based accuracy, which gets as low as 0.11% at the finest mesh. This could be due to the level set based method being able to adjust this specific shape to a better degree of accuracy, as it focuses on moving the boundary of the design rather than the material within the elements. (Ansys, 2025a)

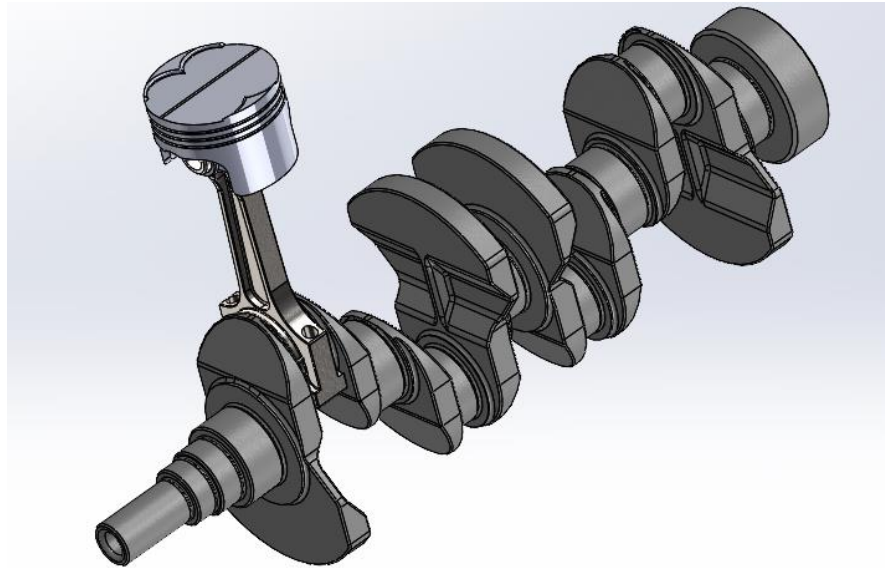


(Figure 14: Metal Plate Level Set Based Topology – 0.5mm Mesh Size)

As shown in figure 14, the output of the level set based approach seems to be the most polished and potentially would have easier manufacturability from that outcome. (Allaire, G., Jouve, F., & Toader, A.-M. 2004)

4 Kinematic Simulation and Structural Analysis

4.1 Application of Material and Boundary Conditions



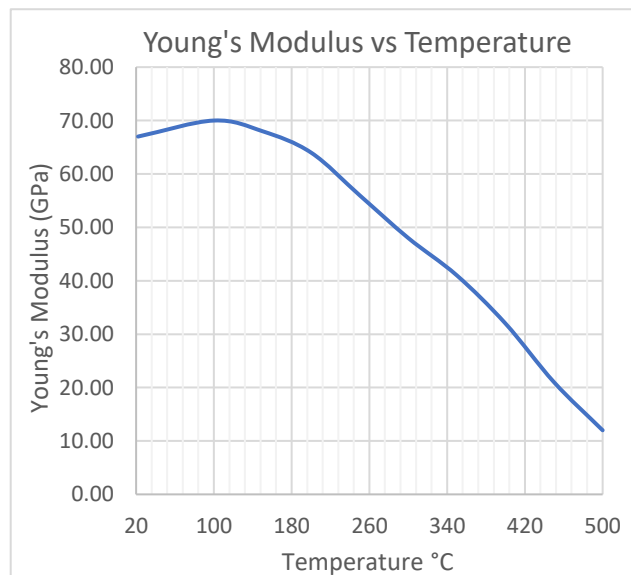
(Figure 15: Test Case Assembly in Solidworks CAD)

Figure 15 shows the assembly designed in Solidworks ready to be exported and be used as the geometry within ANSYS FEA.

The crankshaft, connecting rod, and gudgeon pin will be using the default ANSYS structural steel material. This isn't too important as I will be investigating only the piston after the kinematic analysis. The piston will be using 6061-T6 aluminium for this analysis, properties shown below.

Property	Value	Unit
Poisson's Ratio	0.33	-
Coefficient Thermal Expansion 22°C	2.36E-05	°C ⁻¹
Thermal Conductivity	167	W/m-k
Density	2700	Kg m ⁻³

(Table 4: 6061-T6 Aluminium Property List)
(Piston Thermal Study Data, 2025)



(Graph 9: 6061-T6 Aluminium Young's Modulus)
(Piston Thermal Study Data, 2025)

To be able to begin a kinematic analysis within the rigid dynamics solver, I must add some joints to constrain components to certain rotations.

- Revolute - Crankshaft to Connecting Rod
- Revolute - Connecting Rod to Gudgeon Pin
- Revolute - Gudgeon Pin to Piston
- Revolute - Crankshaft to Ground
- General - Piston to Ground

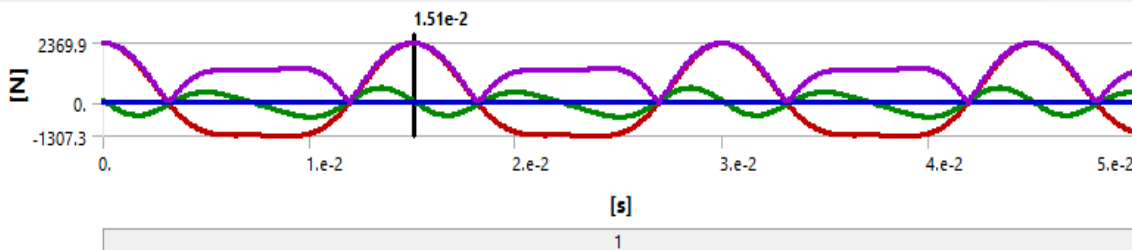
Shown to the left are the joints allowing rotation and also allowing only vertical piston movement.

From this, I was then able to create a joint probe from the gudgeon pin and piston joint, and also a rotational velocity input for the crankshaft.

(Figure 16: Joints for Assembly)

4.2 Kinematic Analysis - Motion Outputs & Loading Conditions

To get motion loads for realistic loading conditions, I applied a 4000rpm rotational velocity to the crankshaft and retrieved the joint probe results.

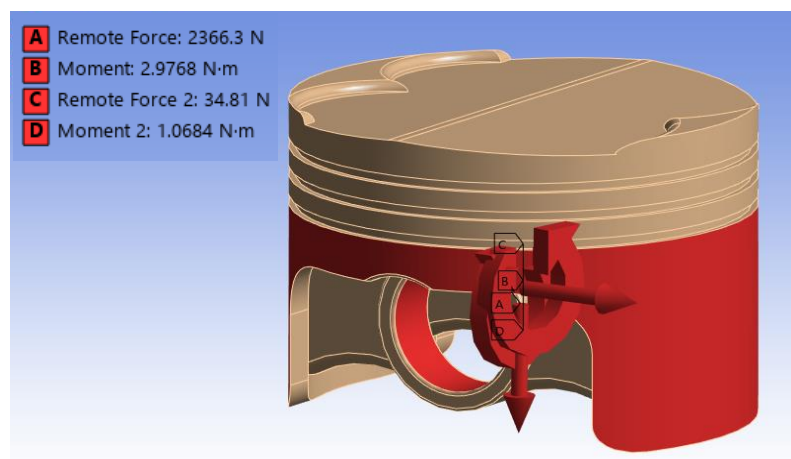


(Graph 10: Joint Probe Motion Forces)

The graph above shows the forces on the joint probe based on the gudgeon pin and piston joint. I highlighted the peak on the graph to export the motion results, in the highest load and 'worst case' scenario.

After exporting the motion loads, I imported them to use realistic loading conditions within the static structural study.

The moment and forces imported onto the piston are shown on the figure to the right.

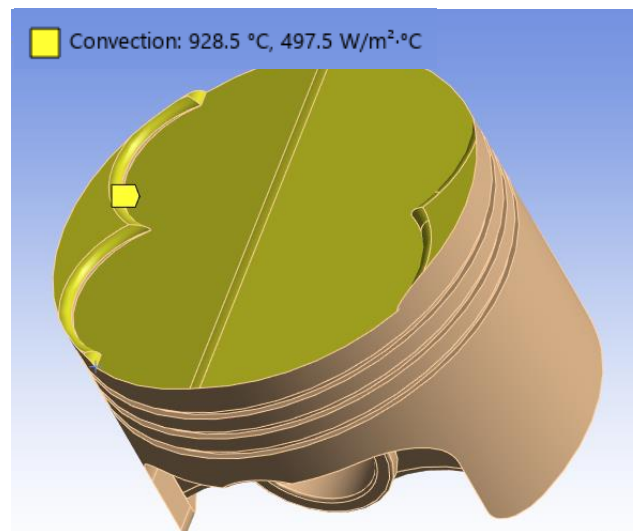


(Figure 17: Motion Loads on Piston)

4.3 Piston Steady State Thermal Study

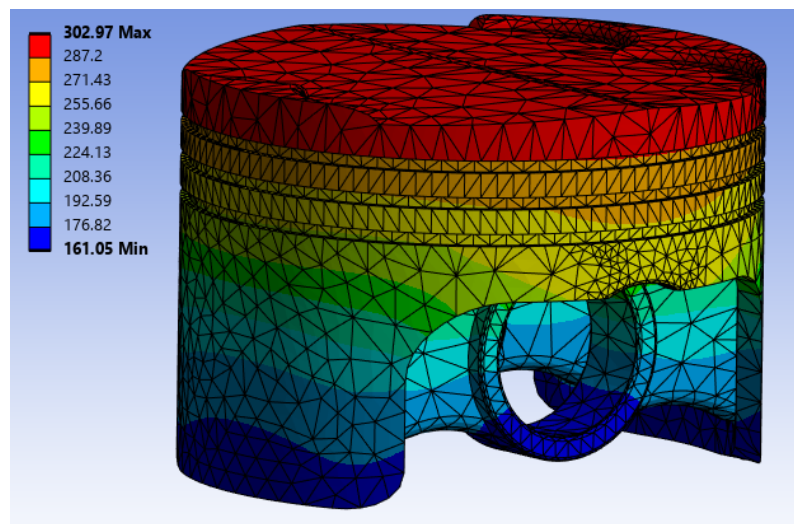
4.3.1 Convection Loads

Before moving onto the structural analysis of the piston, I applied some convection heat loads to mimic the working conditions that the piston is subject to. Figure 18 shown below displays the convection heat applied to the crown of the piston. The thermal study boundary conditions table for the rest of the piston is shown in Appendix B.



(Figure 18: Convection Load on Piston Crown)

4.3.2 Piston Temperature Profile

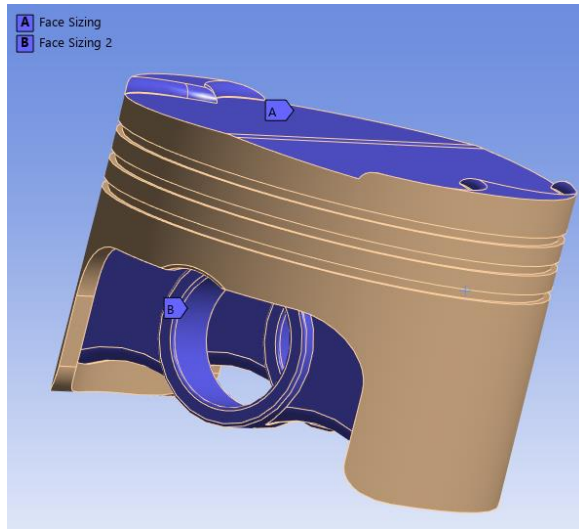


(Figure 19: Thermal Study Temperature Profile)

Shown above is the temperature profile output for the thermal study, showing the max on the piston crown and above the first piston ring. While there is no direct validation comparison, I was able to find similar temperature ranges within piston FEA testing (MAHLE GmbH, 2012). This information will be useful later during design improvement comparison.

4.4 Piston Static Structural Analysis

4.4.1 Face Sizing



(Figure 20: Local Face Mesh Sizing Selection)

Before running the stress and deformation calculations, I decided to apply smaller local mesh sizing to the load face and intricate gudgeon pin area. (MacDonald, B.J. 2021)

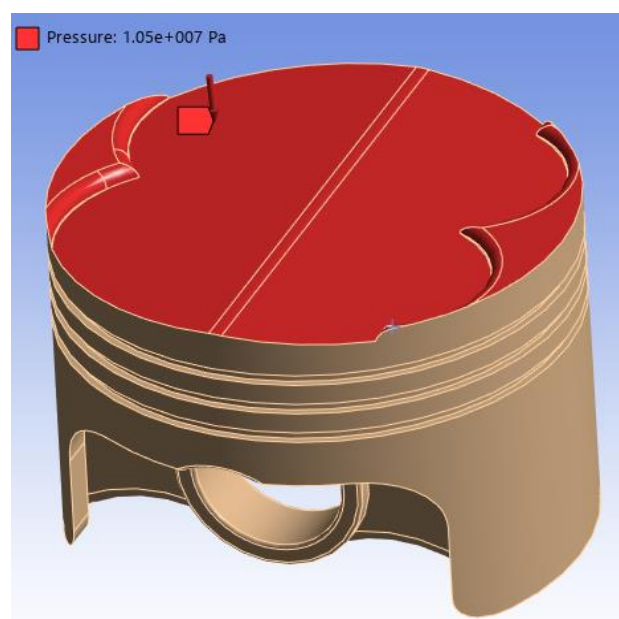
Figure 20 shows the local sized areas where the mesh was reduced from 4mm to 2mm.

I did this to hopefully increase the detail of stress and deformation results within these areas.

4.4.2 Boundary & Load Conditions

The piston requires further constraints to rigid body motion before being able to run the tests, but these constraints must not restrict appropriate loading and deformation. Therefore, the weak springs constraint was enabled which adds a very low stiffness, bounding the pistons movement. This could be considered as a limitation as ideally there would be appropriate constraints without this imaginary boundary. (Ansys, 2025b).

To complete the load conditions, I added a pressure load to the crown face of the piston to replicate the cylinder pressure the piston must resist.

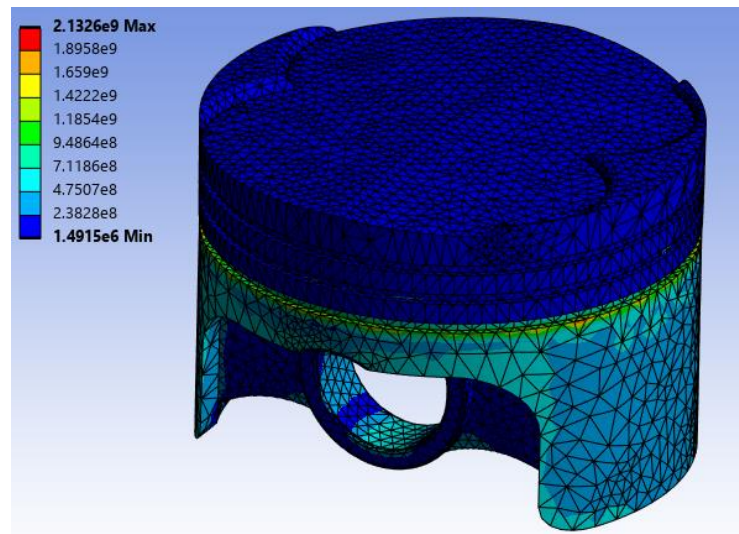


(Figure 21: Crown Pressure Load)

4.4.3 Equivalent Stress (Von Mises)

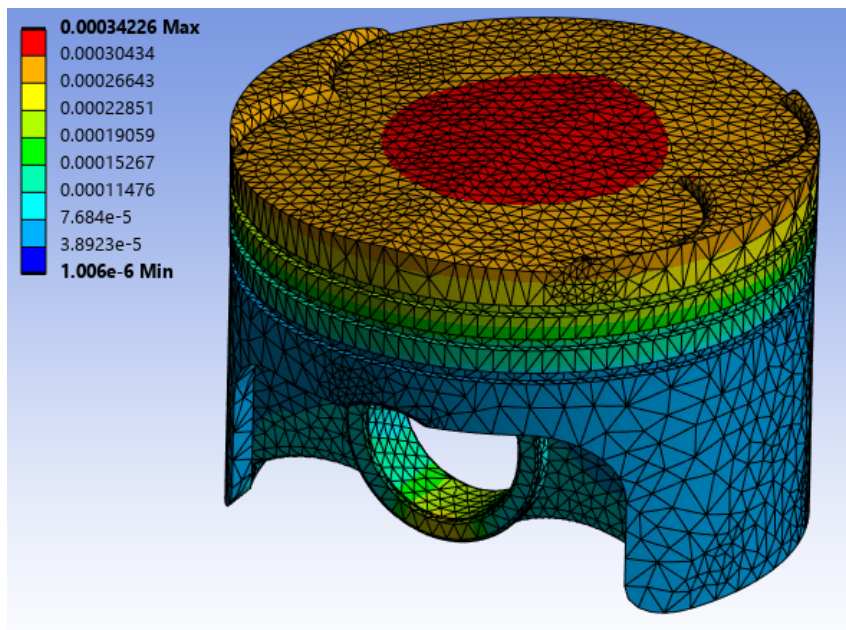
As shown by figure 22 on the right, the 6061-T6 piston crown has good stress distribution as this is a minimum stress area.

The main stress was seen on the gudgeon pin and areas of the skirt and surprisingly, on the oil control ring edge.



(Figure 22: Equivalent Von Mises Stress Profile)

4.4.4 Total Deformation



(Figure 23: Total Deformation of Thermo-Mechanical Loading)

Above is the deformation outcome of the static structural study on the piston, with relatively predictable results. The largest deformation is within the centre of the pressure load and there are also changes to the shape of the gudgeon pin holes. These were loaded and constrained areas which were working against each other therefore, I would consider these results to have a good degree of accuracy on basic principles. (MAHLE GmbH, 2016).

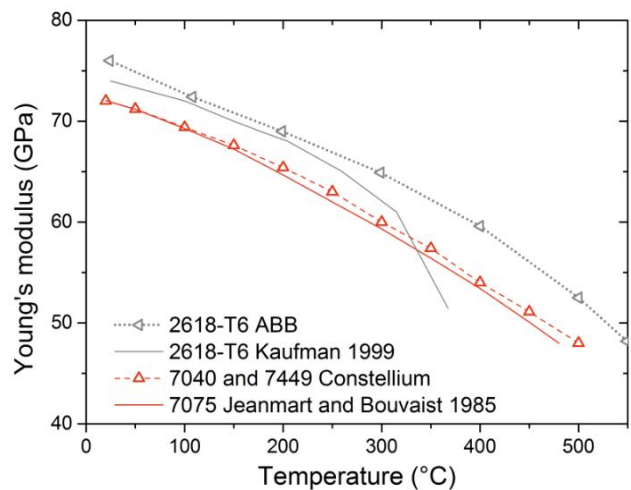
5 Optimisation and Iterative Improvement

To evaluate the purpose of FEA studies I will be iteratively testing different piston materials before a topology study with an aim to improve structural performance while optimising mass. The same test conditions will be used, as explained in the previous section, with the thermal study loads from Appendix B.

5.1 Material Iteration 1 – 2618-T6 Aluminium

Property	Value	Unit
Poisson's Ratio	0.3	-
Coefficient Thermal Expansion 22°C	1.79E-05	°C ⁻¹
Thermal Conductivity	124.9	W/m-k
Density	2814.49	Kg m ⁻³

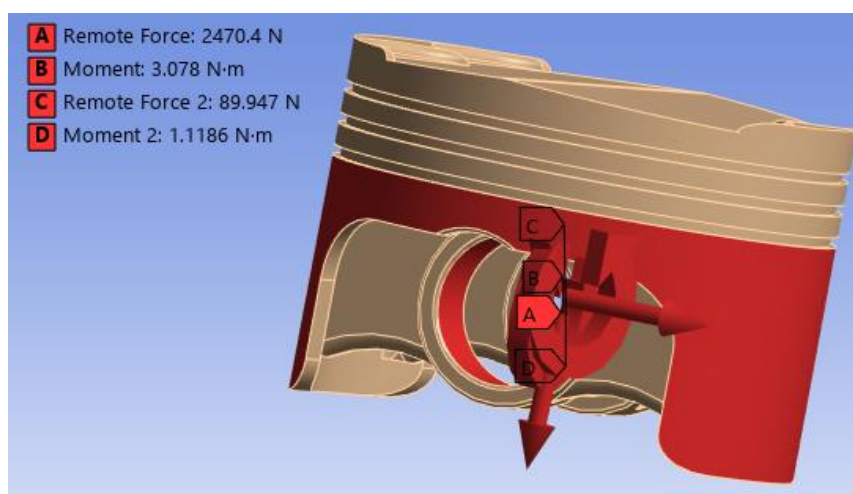
(Table 5: 2618-T6 Aluminium Property List)
(Fushun Special Steel, 2022)
(Granta EduPack, 2024)



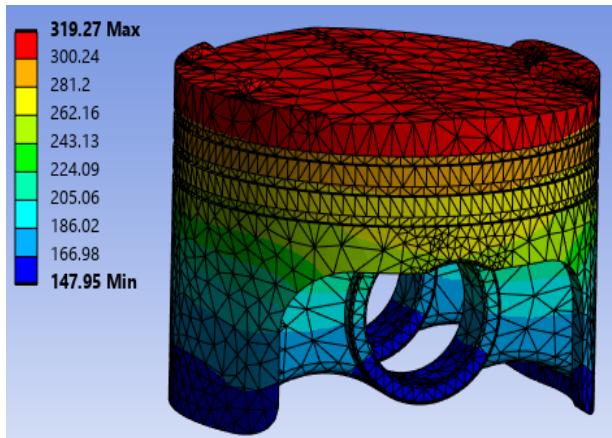
(Graph 11: 2618-T6 Aluminium Young's Modulus)
(Chobaut, N. 2015)

As shown above I was able to find a 2618-T6 aluminium in Granta EduPack to import into ANSYS, with a researched poisson's ratio (Fushun Special Steel, 2025) and young's modulus (Chobaut, N. 2015) shown on graph 11 above as 2618-T6 ABB.

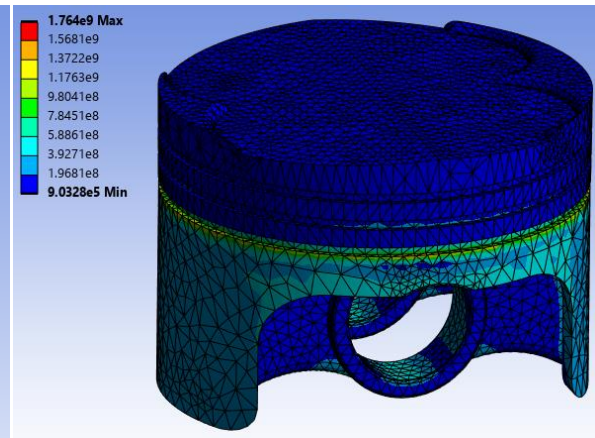
The joint constraints in rigid dynamics were identical between tests, but I recalculated the motion loads as the pistons were different weights.



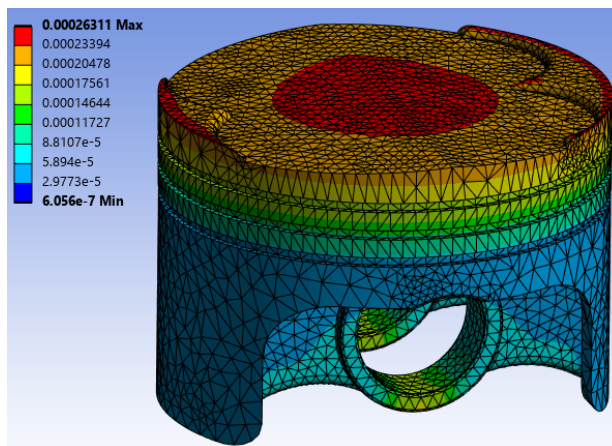
(Figure 24: 2618-T6 Motion Loads)



(Figure 25: 2618-T6 Temperature Profile)



(Figure 26: 2618-T6 Equivalent Stress)



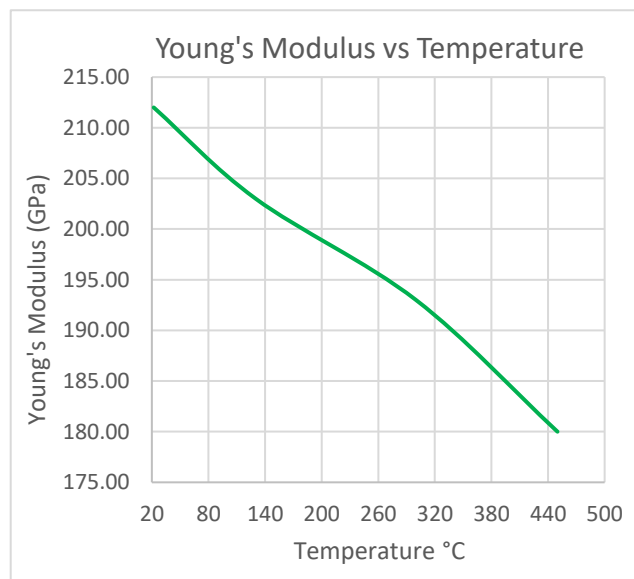
(Figure 27: 2618-T6 Total Deformation of Thermo-Mechanical Loading)

These figures show the same extracted profiles as carried out on the 6061-T6. The 2816-T6, being another aluminium alloy, had similar trends across the tests, showing repeatability. The results will be compared closer after the next material is tested.

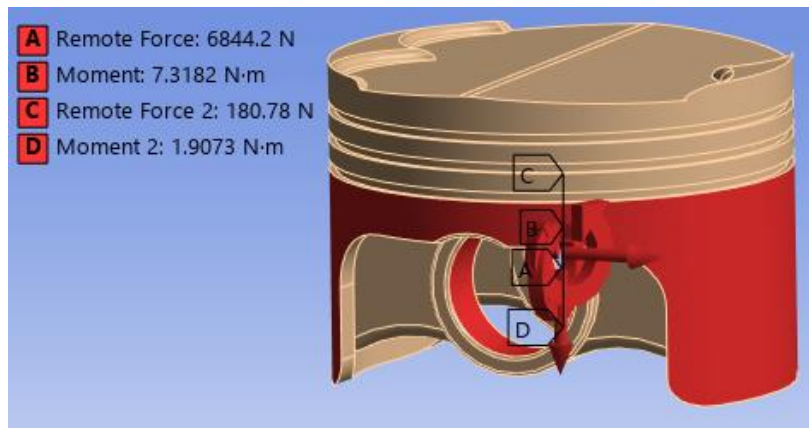
5.2 Material Iteration 2 – 42CrMo4 Steel

Property	Value	Unit
Poisson's Ratio	0.27	-
Coefficient Thermal Expansion 22°C	1.37E-05	°C ⁻¹
Thermal Conductivity	44	W/m-k
Density	7800	Kg m ⁻³

(Table 6: 42CrMo4 Steel Property List)
(MAHLE GmbH, 2016)

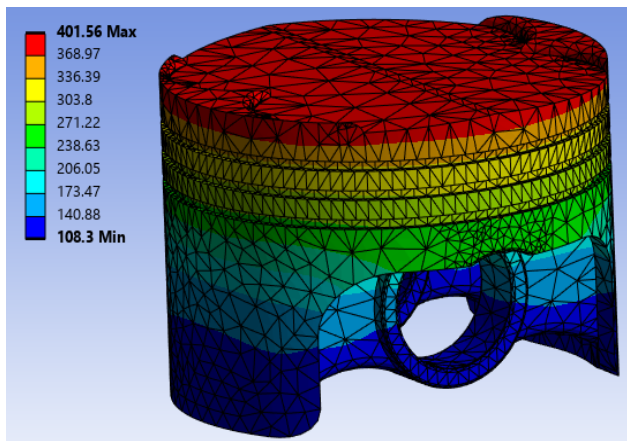


(Graph 12: 42CrMo4 Steel Young's Modulus)
(MAHLE GmbH, 2016)

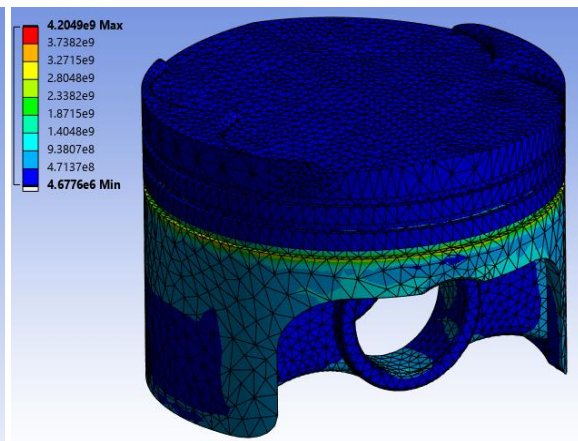


(Figure 28: 42CrMo4 Motion Loads)

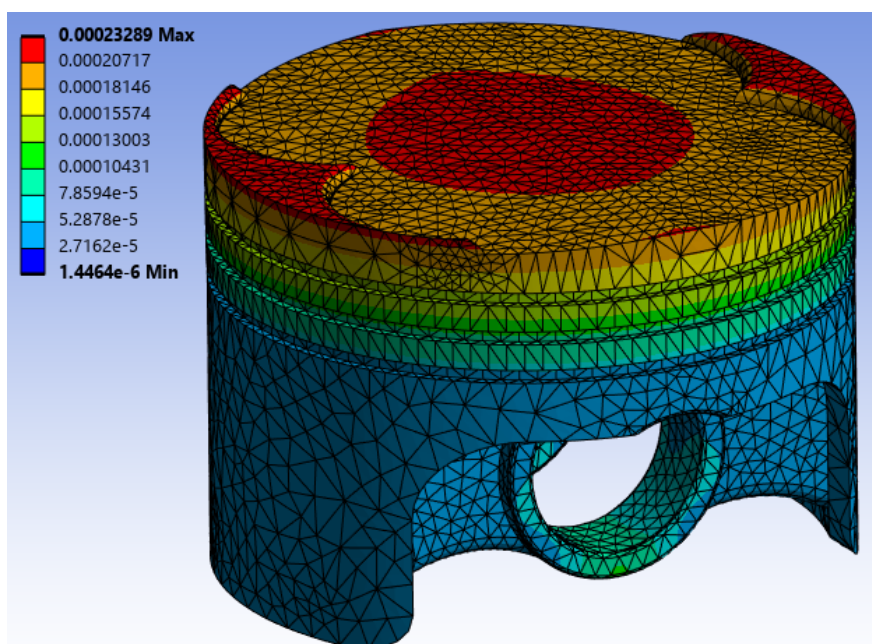
Due to the weight of the steel, the motion loads in figure 28 are visibly larger. This may amplify the loading conditions, and in the context of the other engine components (connecting rod, crankshaft etc.) could mean that this material is less suitable for this application.



(Figure 29: 42CrMo4 Temperature Profile)

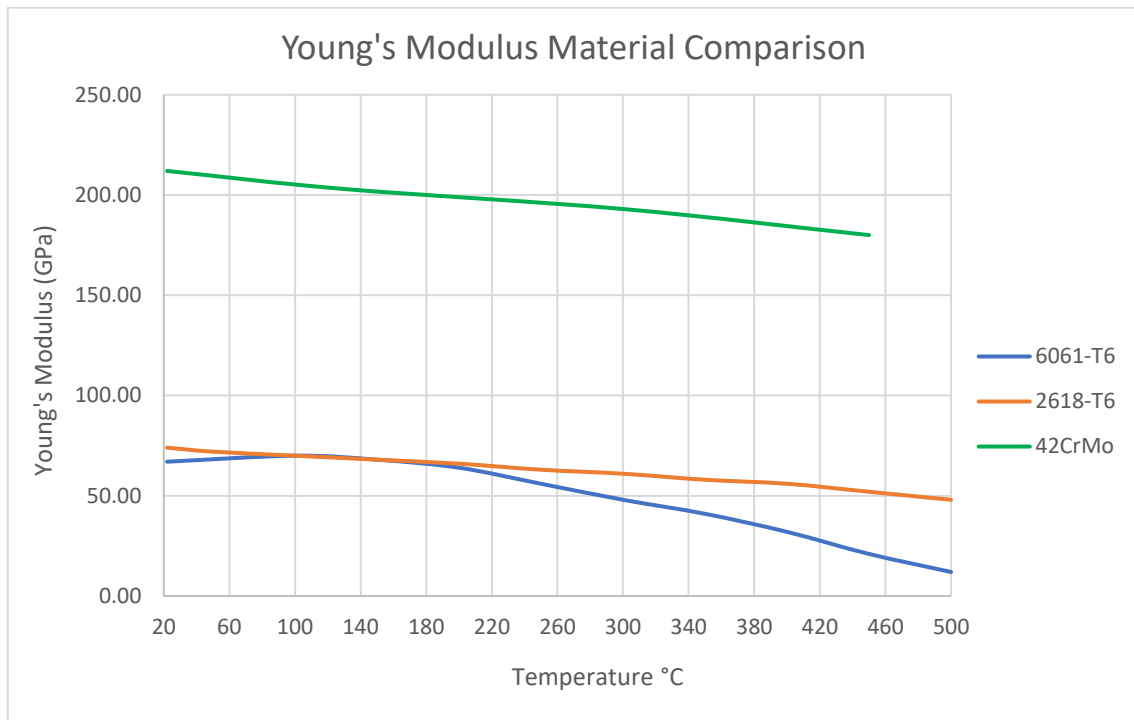


(Figure 30: 42CrMo4 Equivalent Stress)



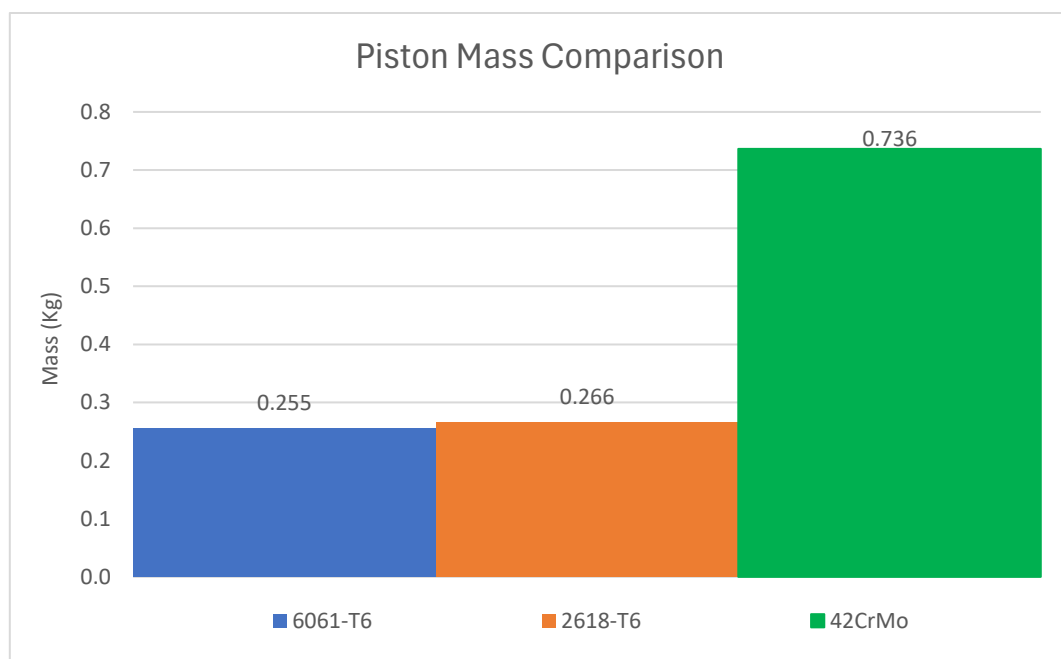
(Figure 31: 42CrMo4 Total Deformation of Thermo-Mechanical Loading)

5.3 Material Comparison & Optimisation Choice



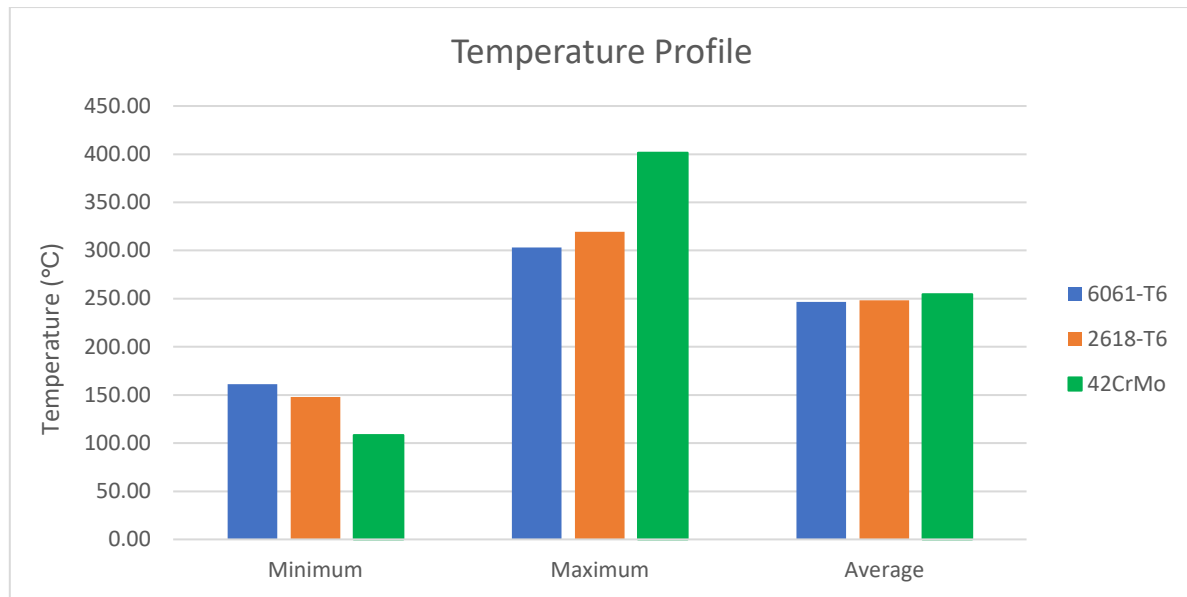
(Graph 13: Young's Modulus Material Temperature Comparison)

As shown by the graph above, the aluminium young's moduli are close, with the 2618-T6 having slightly more temperature resistance and the steel having a far higher young's modulus.



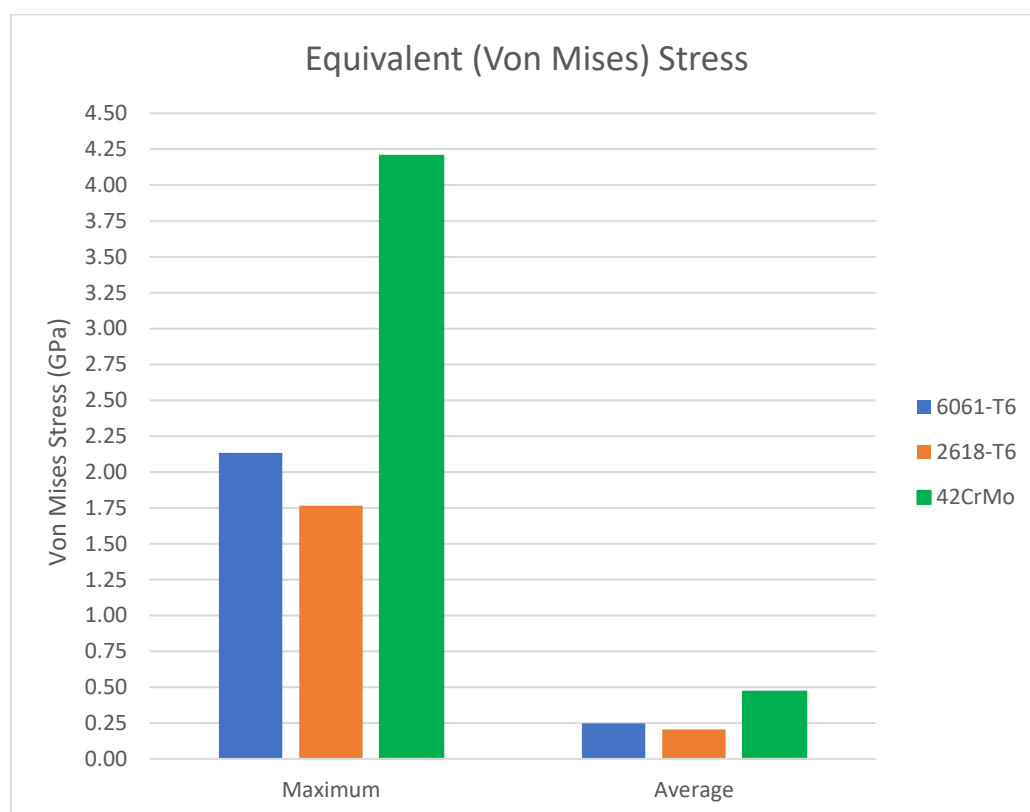
(Graph 14: Piston Material Mass Comparison)

With the steel being a much denser material, it proved to be quite a lot heavier, around 2.8 times the mass of the aluminium materials, which were predictably close. This will be an important consideration as there is a focus on mass optimisation within this re-design study.



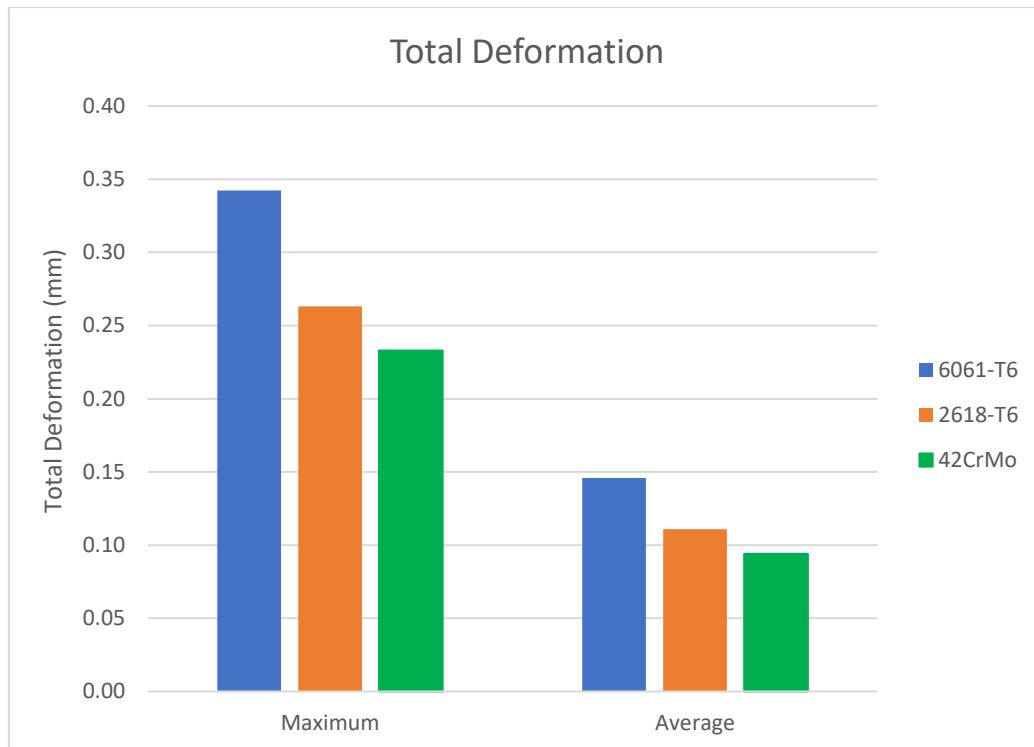
(Graph 15: Temperature Material Comparison)

Graph 15 shows how the minimum, maximum, and average temperature varied during the testing, with the steel showing the largest range but remains close on average.



(Graph 16: Equivalent Stress Material Comparison)

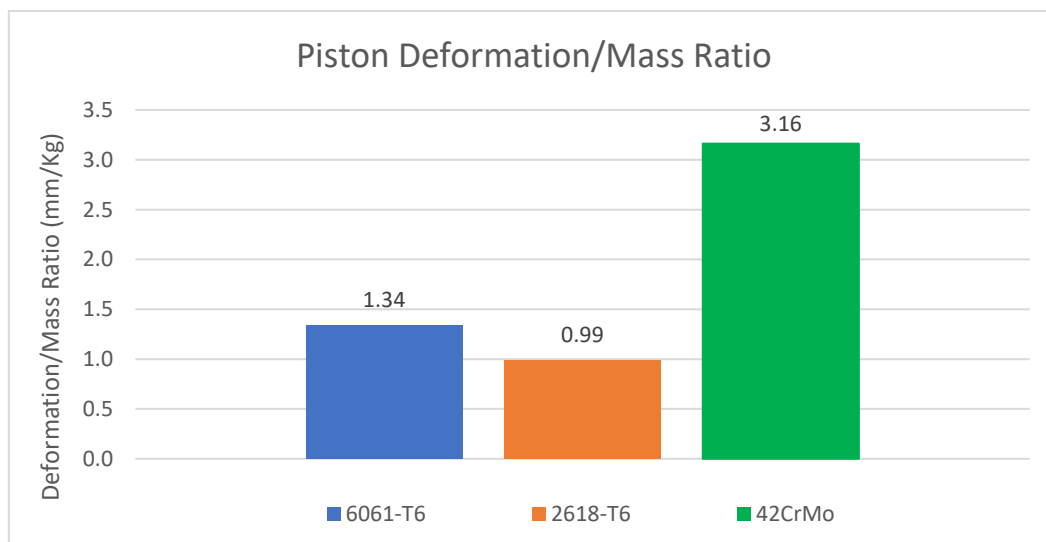
In the stress comparison above, the 2618-T6 provided the least stress by a considerable margin over the 6061-T6, while the steel ended up being the most stressed by close to double the aluminium. This would be due to the steel being much stiffer by young's modulus, therefore having a heavier resistance to elasticity than the other materials.



(Graph 17: Equivalent Stress Material Comparison)

The deformation results shown in graph 17 explains the results clearly, as the steel has the least deformation for being the strongest material chosen.

Considering that the optimisation objective was to improve structural strength while optimising mass, I would say that the 2618-T6 aluminium would be a good design change for the piston moving forward.



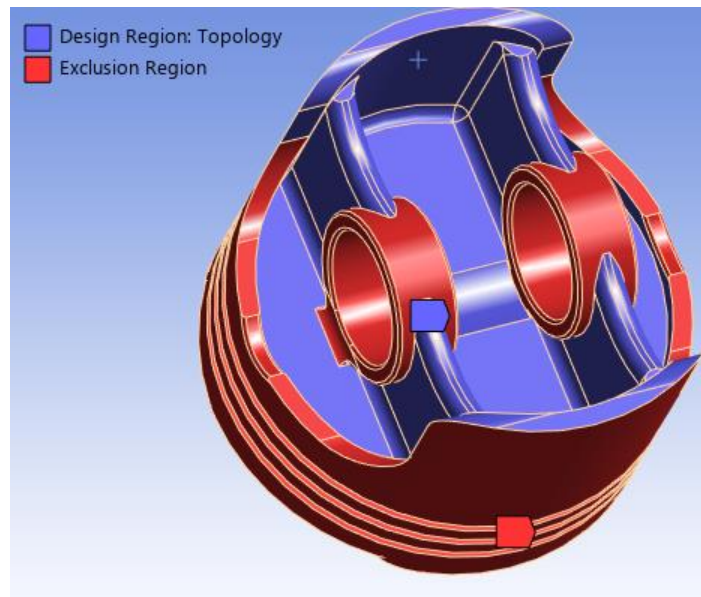
(Graph 18: Deformation/Mass Ratio Material Comparison)

As shown in the graph above, 2618-T6 aluminium was the only material tested with less deflection than the mass figure, proving why this would be a good improvement over 6061-T6. I will be choosing this material moving forward into the mass optimisation topology study.

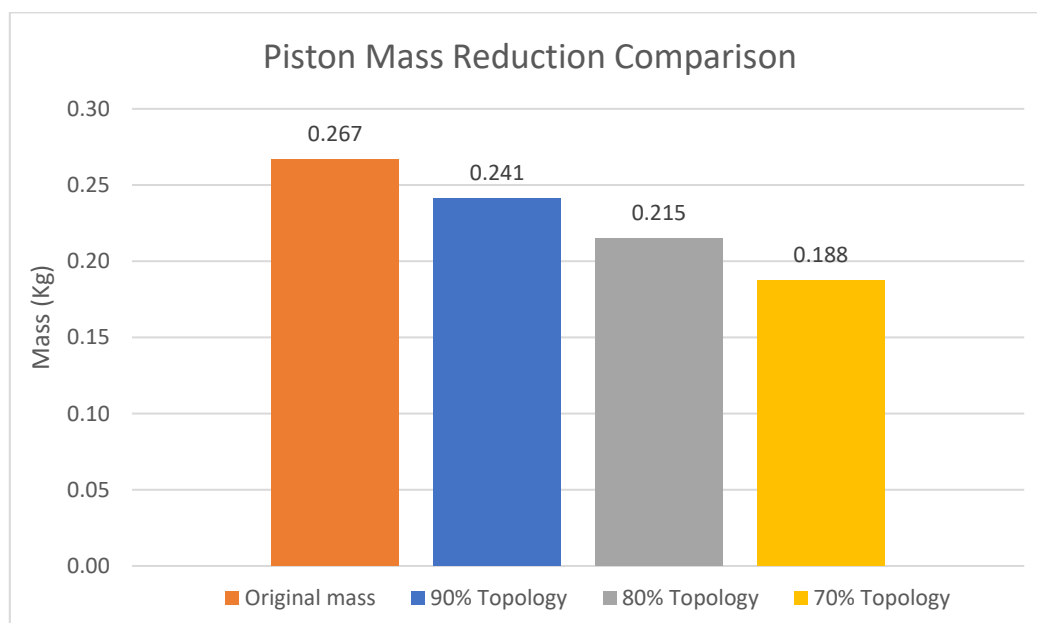
5.4 Design Mass Reduction – ANSYS Topology

To start the topology study, I first had to highlight what areas I wanted the solver to have design freedom of, these are shown as the design region in figure 32. I will be using only the current piston design as a basis for removing mass.

While this may be classed as an already optimised part, and using a more basic shape may provide more feasible topology results, there is also the risk that the solver would cut material away incorrectly, resulting in a shape that wouldn't work as a piston design. Unfortunately, this is a limitation of topology and computed mass reduction. (Bendsøe, M.P. and Sigmund, O. 2003)



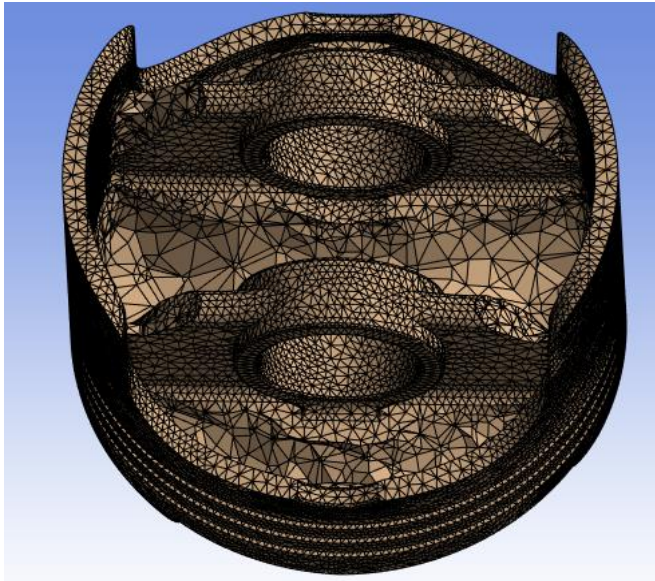
(Figure 32: Design & Exclusion Regions)



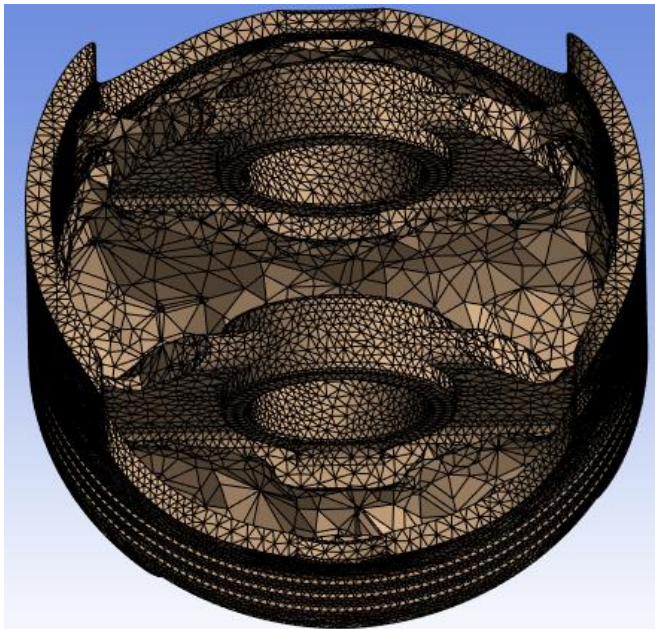
(Graph 19: Topology Reduction Comparison)

I used the previously evaluated level set based method, and it provided requested mass reduction accuracy within the range of 1%. Graph 19 above shows the testing I was able to run, with the retained masses (90% retained meaning a reduction in mass of 10%). It was possible to further remove mass from the design region but the range I tested is already a large chunk of mass to reduce from this piston.

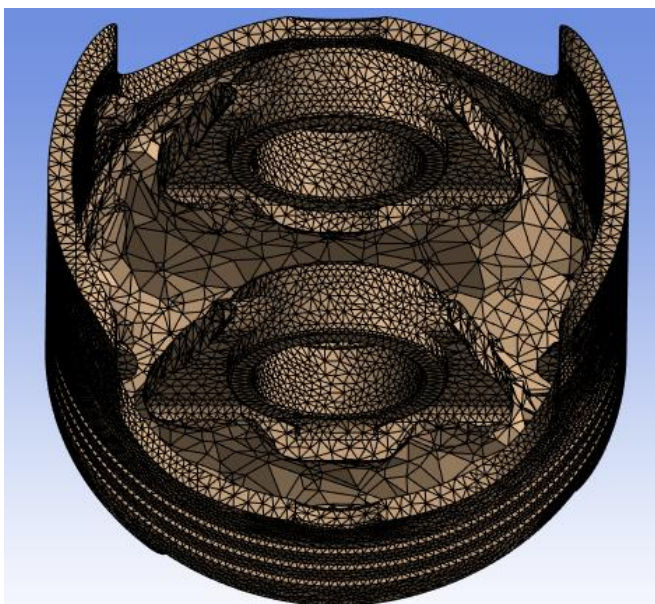
Another limitation I faced was that I couldn't input rigid dynamics motion loads and thermal loads into the solver, meaning there was only the static structural pressure load of 10.5 MPa on the crown and a fixed condition on the gudgeon pin holes.



(Figure 33: 10% Piston Mass Reduction)



(Figure 34: 20% Piston Mass Reduction)



(Figure 35: 30% Piston Mass Reduction)

Figure 33 shows the most structurally sound outcome, retaining most of the important features for gudgeon pin insert structure. In all the outcomes the material was mainly removed from the underside middle of the piston, which could have a negative impact on the thermo-mechanical test as this area may deform heavier with less material underneath the crown to dissipate heat.

At 30% mass reduction in figure 35 it is very clear to see an unrealistic structural result, as the side support for the gudgeon pin holes has been removed. This shows the limitation of not having appropriate loading conditions to this area, as there would be a need for this structure to avoid component failure.

5.5 Final Design Choice



(Figure 36: MAHLE 2618 Alloy Piston)
(JXL Performance, 2025)



(Figure 37: 10% Reduction Topology)

As shown in the figures above, my final design choice to round off this optimisation would be the 2618-T6 aluminium material for the piston, with a 10% mass reduction based on the underside. The limitations around these choices will be detailed in the next conclusion section.

6 Critical Conclusions of Results and Limitations

Within my testing, I was able to compare the temperature profile through thermal temperature and film coefficients around separate parts of the piston (Appendix B). However, a limitation from this would be that I am unsure of how the film coefficients were gathered, as if these were material-specific then to ensure full temperature profile accuracy, I would need these coefficients for each of my separate materials in an identical test.

Topology is a useful tool within FEA, but it is limited by the inputs, mesh quality and manufacturability of the final output, meaning it is best to use as an initial mass reduction stage with post-processing following. To ensure my reduction strategy is viable I would do further testing on a refined version of the raw 10% reduction output.

While I was able to conclude a piston material based on thermal capability, deformation and equivalent stress, to completely refine this decision I would need a fatigue analysis to ensure the material has longevity within this piston design.

To summarise, if this analysis was used in a real-world application further work would be needed, including but not limited to, a fatigue study, improved boundary conditions to release the need for a weak springs constraint, detailed mesh refinement to avoid skewness among other mesh quality factors and a post-processed and tested mass optimisation solution.

References

- Allaire, G., Jouve, F., & Toader, A.-M. (2004). *Structural optimization using sensitivity analysis and a level-set method*. *Journal of Computational Physics*, 194(1), 363–393.
<https://doi.org/10.1016/j.jcp.2003.09.032>
- Ansys (2024) *Skewness*, *ansys.com*. Available at:
https://ansyshelp.ansys.com/public/Views/Secured/corp/v242/en/wb_msh/msh_skewness.html
- Ansys (2025a) *Structural Optimization Analysis Guide*, *ansys.com*. Available at:
https://ansyshelp.ansys.com/public/Views/Secured/corp/v252/en/mech_struct_opt/mech_struct_opt.html
- Ansys (2025b) *Weak Springs*, *ansys.com*. Available at:
https://ansyshelp.ansys.com/public/account/secured?returnurl=/Views/Secured/corp/v252/en/add_ag/add_ag_objref_weak_spr.html?utm_source=chatgpt.com
- Bendsøe, M.P. and Sigmund, O. (2003) *Topology Optimization: Theory, methods and applications*. Berlin: Springer.
- Chobaut, N. (2015) *Young's modulus versus temperature for AA2618 in T6 state and for three... | Download Scientific diagram, Research Gate*. Available at: https://www.researchgate.net/figure/3-Youngs-modulus-versus-temperature-for-AA2618-in-T6-state-and-for-three-7xxx-alloys_fig89_283774913
- Dlupal (2025) *Elastic Bending – Continuous Load*
- Dlupal (2025) *Bending Cantilever with Axial Force*
- Dlupal (2025) *Bimetallic Strip*
- Fushun Special Steel (2022) *42CRMO4 1.7225 alloy steel: Fushun Special Steel, Fushun Special Steel Co., Ltd. - Professional Supplier of Special Steel, and Manufacturer of Tool Steel | manufacturer of tool steel, stainless steel, nickel alloy, alloy steel*. Available at:
https://www.fushunspecialsteel.com/42crmo4-1-7225-alloy-steel/?utm_source=chatgpt.com
- GridPro (2025) *Different types of meshes in CFD | GridPro, gridpro.com*. Available at:
<https://gridpro.com/different-types-of-meshes-in-cfd>
- JXL Performance (2025) *MAHLE Motorsport Forged Piston Set (2618 alloy) for Ford Focus St mk3, JXL Performance*.
- Khan, H., Zahra, F. T., Bukhari, S. M. A., Husnain, N., Zaman, M. F., Raza, H., Anwar, M. T., Tariq, A., Siddiqui, F. A., & Ahmed, W. (2025). *Effect of Metallic and Ceramic Coatings on Structural Behaviour of Heavy-Duty V8 Engine Using Finite Element Analysis*. *Advanced Theory and Simulations*, 8(9).
<https://doi.org/10.1002/adts.202500151>
- Lorenzini, M., Barbieri, S. G., Mangeruga, V., & Giacomini, M. (2023). *Development of an experimental/numerical validation methodology for the design of exhaust manifolds of high-performance internal combustion engines*. *Engineering Failure Analysis*, 152, Article 107526.
<https://doi.org/10.1016/j.engfailanal.2023.107526>

Mac Donald, B.J. (2021) *Practical stress analysis with finite elements*. Dublin, Ireland: Machdohnil Ltd.

MAHLE GmbH. editor. (2012). *Pistons and engine testing Springer Vieweg*.

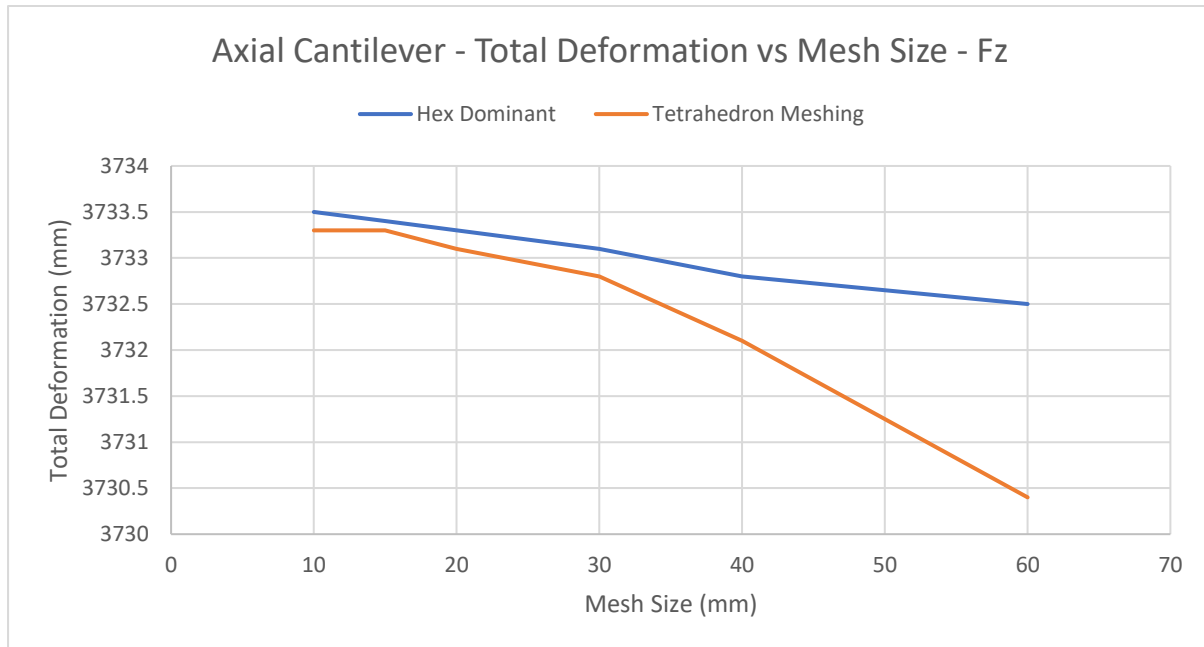
MAHLE GmbH. editor. (2016). *Pistons and engine testing (2nd edition.)*. Springer Vieweg.

Piston Thermal Study Data (2025)

Zienkiewicz, O.C., Taylor, R.L. and Zhu, J.Z. (2014) *The finite element method: Its basis and fundamentals*. San Diego (Calif.): Elsevier Science & Technology Books.

Appendices

Appendix A – Axial Cantilever Total Deformation Fz Graph



Appendix B – Piston Thermal Study Data

Piston Location	Temperature (°C)	Heat Transfer Coefficient (W/m ² .°C)
Crown	928.5	497.5
Crown Side	425	447.5
Top Groove	190	437.5
Middle Groove	190	437.5
Oil Groove	163.75	338.75
Underside	91.25	675
Skirt	107.5	462.5

(Piston Thermal Study Data, 2025)