



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

School of Engineering
Motorsport Engineering BEng

Application of CAE Processes
Structural & Fluid Analysis ACHA6010

Catriona Burn

2203239

November 2025

Word count: 3,283

Contents

1	Abstract.....	1
2	Introduction.....	1
3	Literature Review	2
4	Validation.....	3
4.1	Analytical case study.....	3
4.1.1	ANSYS replication	3
4.1.2	Analytical Solution.....	4
4.2	Mesh study.....	4
4.3	Boundary Condition Sensitivity	5
5	Assembly and Setup	6
5.1	Workflow overview	6
5.2	Materials	7
5.3	Contacts and constraints	7
6	Kinematic and Motion Loads	9
6.1	Kinematics Method.....	9
6.2	Results	9
7	Static Structural and Thermal Analysis.....	10
7.1	Steady State Thermal Setup	10
7.2	Static Structural Setup	10
7.3	Results	10
8	Optimisation and Improvement	12
9	Discussion and limitations	14
10	Conclusion.....	15
11	References	16
	Appendices	17

[Figure 1]	Bimetallic Strip deformation	3
[Figure 2]	Cantilever beam with pressure applied.....	5
[Figure 4]	Part 5: End bearing	6
[Figure 3]	Selected component and assembly.....	6
[Figure 5]	Workflow in ANSYS Workbench	6
[Figure 7]	AA6061-T6 material data (ANSYS Granta, 2024).....	7
[Figure 6]	AA6061-T6 Yield Strength graph	7
[Figure 8]	Piston with 4mm mesh	9
[Figure 9]	Motion Loads Graph	9
[Figure 10]	Motion Loads.....	9

[Figure 11] Inertial load points.....	10
[Figure 12] Stress distribution.....	11
[Figure 13] Total deformation.....	11
[Figure 14] High stress region	11
[Figure 15] Thermal distribution	11
[Figure 16] Design 1	12
[Figure 17] Design 2	12
[Figure 18] Design 3	12
[Figure 19] Design 3 deformation	13
[Figure 20] Design 2 deformation	13
[Figure 22] Design 3 Stress.....	13
[Figure 21] Baseline stress	13
[Figure 24] Design 1 Thermal	13
[Figure 23] Baseline thermal.....	13

1 Abstract

This study validates and applies CAE processes to piston analysis, integrating kinematic, thermal, and structural simulations. Results reveal deformation and stress trends consistent with literature, while optimisation strategies improved stiffness and reduced mass. The work demonstrates CAE's role in efficient design refinement and underscores limitations linked to mesh quality, boundary conditions and material behaviour.

2 Introduction

Computer-Aided Engineering (CAE) tools, such as ANSYS, are fundamental within automotive design and development, providing the capability to predict deformation, stress distribution and fatigue response prior to physical testing. The integration of these tools into modern workflow enables rapid refinement of complex components and offers valuable insight into performance limitations (Barbosa et al., 2022; Bhuvaneswari et al., 2024).

This report investigates the structural behaviour of a piston, focusing on its response to static loads and fatigue while also considering where mass reduction can be achieved without compromising performance. Within the piston-crankshaft assembly, components are subjected to considerable mechanical and thermal stress throughout each engine cycle, making structural reliability essential for maintaining efficiency and durability. Given the piston's central role in the engine operation, even minor weaknesses in its design can have significant consequences for overall performance.

To address these challenges, a series of potential design modifications are explored with the aim of improving strength and fatigue life, alongside targeted mass optimisation. CAE simulations are employed throughout to guide and justify design decisions, highlighting both the practical strengths of these tools and the inherent limitations that can influence interpretation of the results. Where appropriate, the findings are cross checked against hand calculations to ensure technical robustness. Particular attention is given to the inevitable trade-offs between structural integrity and reduced mass, which form a key part of the design evaluation.

3 Literature Review

The structural and thermal behaviour of pistons has been extensively studied due to the severe mechanical and thermal loads experienced during engine operation. Consequently, CAE methods are commonly used to predict stress, deformation and temperature distributions under representative loading conditions. However, the reliability of CAE results is highly dependent on modelling assumptions, such as material properties, mesh density, and boundary condition definition. This literature review therefore focuses on piston design considerations, CAE validation practices, and the influence of meshing and boundary conditions on simulation reliability.

Pistons are subjected to high cyclic loads arising from combustion pressure, inertial forces and thermal gradients. The piston crown experiences the highest pressures and temperature during combustion, making it a critical region for structural integrity. Piston design therefore aims to distribute these loads as evenly as possible to minimise stress concentrations and prevent failure. Mahle International GmbH and Springer Fachmedien Wiesbaden GmbH (2018) discuss the influence of crown geometry and thickness on compression ratio, strength, and durability, highlighting the importance of geometric optimisation in piston design.

Material selection is also a key factor, as it directly affects strength, stiffness, thermal performance and mass. Aluminium alloys are widely used due to their low density and high thermal conductivity, although their mechanical properties degrade at elevated temperatures. Chakravarti and Kumar Biswas (2024) compared AA6061-T6 with AA7050 for piston applications, concluding that AA7050 offers improved high-temperature strength, while AA6061 provides a balance between performance and cost. Zheng et al. (2025) further demonstrated that targeted geometric modifications can improve piston performance without increasing manufacturing complexity.

CAE tools are extensively used to support such design decisions; however, confidence in simulation results depends on appropriate validation. Simplified benchmark problems are commonly used to verify numerical accuracy prior to analysing complex components. The use of bimetallic strip models for coupled thermo-structural validation is well established, with Čepón et al. (2017) demonstrating verification through mesh convergence and analytical comparison. Mesh density and boundary condition definition are recognised as major sources of uncertainty in finite element analysis. Jadhav and BedajangsagarKishor (2014) showed that appropriate mesh refinement is essential for accurately capturing stress concentrations, while Wang, Liu and Shi (2010) demonstrated that pressure-based loading more accurately represents combustion paths in piston analysis than equivalent force-based approaches.

4 Validation

This section presents the validation of the CAE methodology and simulation setup. The purpose of this validation is to demonstrate the credibility of CAE tools used and show how their accuracy can be improved through appropriate modelling choices. Validation is carried out using analytical calculations alongside mesh and boundary condition experiments, to verify that the selected settings are appropriate and that the results are as expected. The analytical cases are used to identify potential modelling errors, while the mesh and boundary condition studies address discretisation, constraint and load application effects.

4.1 Analytical case study

The first analytical study considers a bimetallic strip composed of Invar and Copper. One end of the strip is fully constrained, while the opposite end is free. A thermal load is applied, causing differential thermal expansion between the two materials. The material properties and geometric parameters used in this study are provided in the table below.

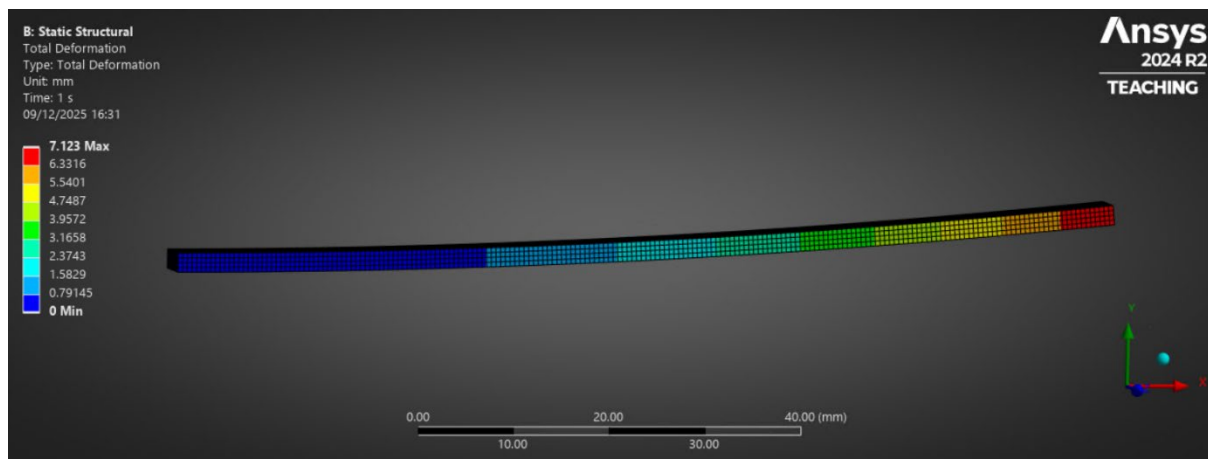
[Table 1] Bimetallic strip parameters

Material	Invar	Modulus of Elasticity	E_i	137000	MPa
		Poisson's Ratio	ν_i	0.280	-
		Coefficient of Thermal Expansion	α_i	1.2×10^{-6}	$^{\circ}\text{C}^{-1}$
	Copper	Modulus of Elasticity	E_c	130000	MPa
		Poisson's Ratio	ν_c	0.354	-
		Coefficient of Thermal Expansion	α_c	2.0×10^{-5}	$^{\circ}\text{C}^{-1}$
Geometry	Cross-section width		w	5	mm
	Layer Thickness		$t_i = t_c = t$	1	mm
	Length		L	100	mm
Load	Thermal Loading		T_c	100	$^{\circ}\text{C}$

4.1.1 ANSYS replication

The material data was defined within the Engineering Data module in ANSYS, and the geometry was modelled in SolidWorks before being exported as a STEP file for use in ANSYS Workbench.

The model was imported into a Static Structural study in ANSYS workbench. Each section of the strip was assigned its corresponding material, after which the mesh was defined. A global element size of 0.5mm was applied and the mesh was generated. Boundary conditions were then applied, with one end of the beam fixed and a uniform thermal load of 100°C is applied to both materials.



[Figure 1] Bimetallic Strip deformation

4.1.2 Analytical Solution

The analytical solution for the bimetallic strip is based on classical bimetallic beam theory (Čepon et al., 2017). The theory models curvature and deflection arising from differential thermal expansion between bonded layers, assuming linear elastic material behaviour, perfect bonding and small deflections. These assumptions are consistent with the ANSYS static structural model used, making the theory suitable for comparison. The following equations are from Dlubal Software (2018), verification example.

The **equivalent bending stiffness** of the beam is calculated using:

$$K_1 = 4 + 6 \frac{t_c}{t_i} + 4 \left(\frac{t_c}{t_i} \right)^2 + \frac{E_c}{E_i} \left(\frac{t_c}{t_i} \right)^3 + \frac{E_i t_i}{E_c t_c} = 4 + 6 \frac{1}{1} + 4 \left(\frac{1}{1} \right)^2 + \frac{130000}{137000} \left(\frac{1}{1} \right)^3 + \frac{137000 \times 1}{130000 \times 1} = 16.002 \quad [\text{Equation 1}]$$

The **maximum tip deflection** is then defined by:

$$u_{z,max} = \frac{6(a_i - a_c)(t_i + t_c)T_c L^2}{t^2 K_1} = \frac{6((1.2 \times 10^{-6}) - (2.0 \times 10^{-5}))(1 + 1) \times 100}{1^2 \times 16.002} \frac{100^2}{2} \approx -7.049 \text{ mm} \quad [\text{Equation 2}]$$

[Table 2] Bimetallic strip analytical vs simulation

	Max tip deflection (mm)
Analytical	-7.049
Simulation	7.123
% difference	1.01%

The simulated maximum tip deflection shows good agreement with the analytical solution, with a 1.01% difference. The negative sign in the analytical result reflects the assumed deflection direction based on the chosen sign convention, while the simulation reports the deflection magnitude.

4.2 Mesh study

Following verification of the analytical solution, a mesh sensitivity study was conducted on the beam to assess the influence of element size on the predicted maximum deformation. A range of global mesh sizes were investigated, from relatively coarse to fine, while all other model parameters were held constant. The maximum deformation of the beam was used as the primary convergence metric, as it is directly comparable with the analytical solution and sensitive to discretisation effects.

As shown in table 3 the predicted deformation increases with mesh refinement before approaching a stable value. Beyond a mesh size of approximately 1mm, further refinement results in negligible changes in the deformation, indicating that mesh independence has been achieved. Based on this behaviour, a mesh size of 0.5mm was selected for subsequent analyses as it provided a suitable balance between numerical and computational cost.

[Table 3] Mesh study comparison with analytical

Mesh size (mm)	Max deformation(mm)	% difference from analytical
4.5	7.1194	1.00999
4	7.1198	1.01004
3	7.1213	1.01026
2.5	7.1219	1.01034
1	7.1233	1.01054
0.5	7.123	1.01050
0.25	7.1213	1.01026
0.1	7.1203	1.01011

While full mesh independence was not performed for the piston due to computational cost, comparison of relative element size showed that a 4mm piston mesh represents a similar proportion of component area to the validated beam mesh. This provides justification that the selected mesh density is reasonable for the subsequent structural analysis.

4.3 Boundary Condition Sensitivity

A simplified cantilever beam was used to assess the sensitivity of CAE results to boundary condition and load application choices. The beam was fully constrained at one end and subjected to the same resultant load applied using two different methods: a concentrated transverse force and an equivalent distributed pressure applied over the beam end face. All other parameters were kept constant.

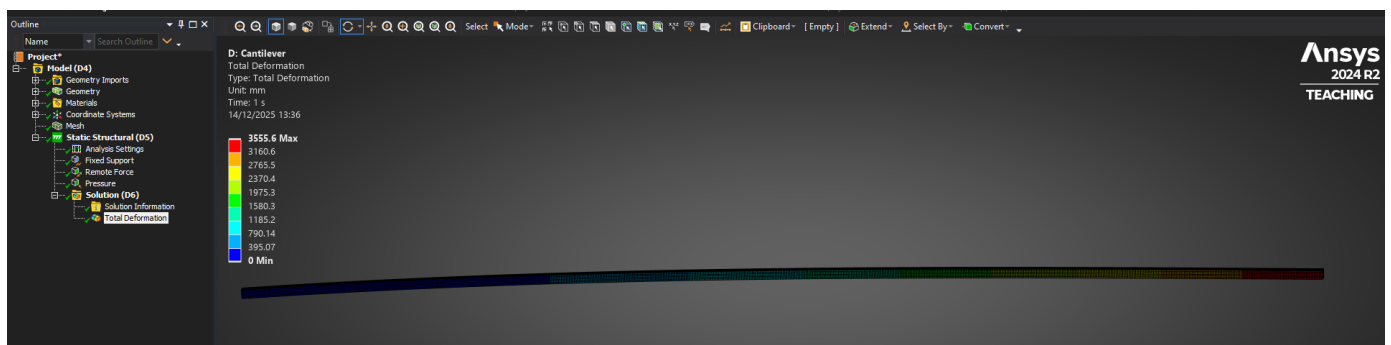
[Table 4] Cantilever beam parameters

Material	Steel	Modulus of Elasticity	E	210000	MPa
		Poisson's Ratio	ν	0.3	-
Geometry	Cantilever	Length	L	10	m
		Cross-sectional width	w	0.08	m
Load		Transversal Force	F_z	7.65	kN

Applied pressure calculated from force:

$$P_z = \frac{F_z}{A} = \frac{7.65 \times 10^3}{(0.8 \times 0.8)} = 11,953.13 Pa \quad [Equation 3]$$

Although the applied load magnitude was equivalent in both cases, differences were observed in the predicted deformation and stress distribution. The pressure-based loading produced a more distributed stress field, while the concentrated force resulted in higher local stresses near the constraint. This demonstrates that CAE results are sensitive to how loads are applied, not just their magnitude. This highlights the importance of applying boundary conditions that accurately represent real load paths and reinforces the need for engineering judgement when interpreting CAE results.



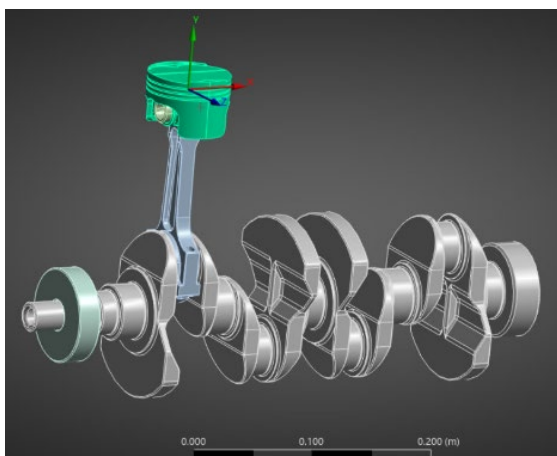
[Figure 2] Cantilever beam with pressure applied

5 Assembly and Setup

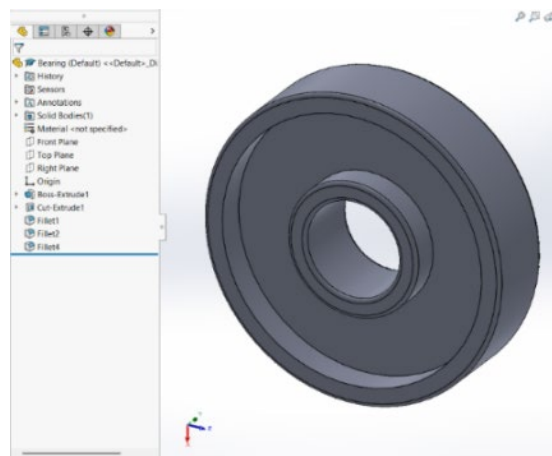
The model used for this simulation is a simplified 4 cylinder system, reduced to a single operating piston to reduce complexity while maintaining kinematic behaviour. The 5 components:

1. Piston.
2. Gudgeon Pin.
3. Connecting-rod.
4. Crankshaft.
5. End bearing.

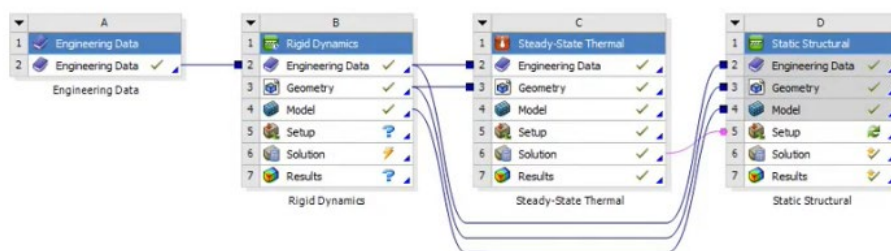
The assembly was provided as a pre-assembled model, excluding the end bearing that was added to provide a realistic crankshaft support. The model was used to simulate the motion loads acting on the piston during an engine cycle, as well as realistic reaction forces for structural analysis. Figure 3 shows the assembled five-component model.



[Figure 3] Selected component and assembly



[Figure 4] Part 5: End bearing



5.1 Workflow overview

[Figure 5] Workflow in ANSYS Workbench

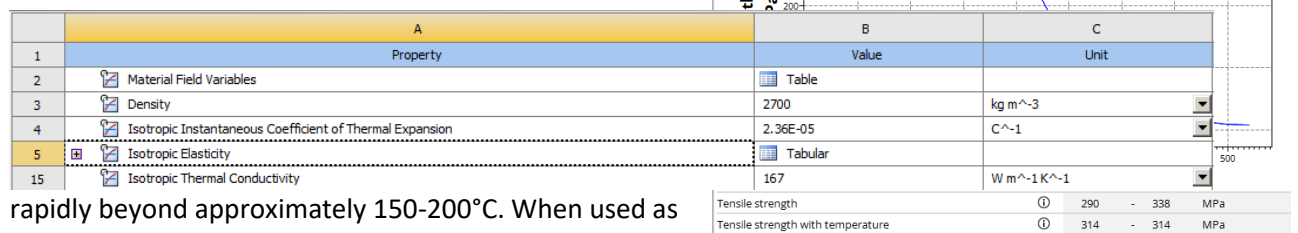
Material properties were defined in the Engineering Data component and applied consistently across all analyses. Geometry and material data were first used in the rigid dynamics model, which provided motion-induced loads for subsequent simulations. The steady-state thermal analysis was

then conducted with all components except the piston suppressed, allowing temperature distributions to be obtained. These temperature results, along with motion loads from the rigid dynamics analysis, were imported into the static structural model to evaluate stress and deformation.

The analysis workflow consisted of validation through mesh independence and analytical comparison, followed by rigid dynamics (kinematics), steady-state thermal, static structural and finally geometry optimisation.

5.2 Materials

The material used for the piston baseline study is 6061-T6 Aluminium, selected due to its availability of reliable temperature-dependent material data from (ANSYS Granta, 2024). Aluminium alloys are widely used for pistons because of their low density and high thermal conductivity, which support reduced reciprocating mass and effective heat dissipation. AA6061-T6 exhibits temperature-dependant behaviour, with reductions in Young's Modulus and yield strength as temperature increases, alongside increased deformation and wear at elevated temperatures. Experimental and numerical studies show that material performance improves up to approximately 100°C, before degrading beyond this point due to thermal softening (Safdarian and Parente, 2024). This behaviour is consistent with the temperature-strength relationship shown in figure 6, where yield strength decreases



rapidly beyond approximately 150-200°C. When used as a piston material, AA6061-T6 therefore offers

favourable strength-to-weight performance at moderate temperatures, however, elevated operating temperatures significantly reduce allowable stress margins in critical regions such as the crown and ring lands, making thermal degradation an essential consideration in structural analysis.

The remaining engine components are assigned structural steel for simplification purposes, as variations in their material properties have a minimal influence on overall inertia in the kinematic analysis. Structural steel was selected due to its high stiffness, strength and fatigue resistance, which are appropriate for components subjected to cycle loading, particularly the repeated high-cycle loading and journal wear resistance extended service life.

Constraint	Reference	Mobile
Body-to-Ground: Revolute	Z-axis	Crank
Body-to-Ground: General	Free in z-axis	Piston
Body-to-Body: Revolute	Crank	Con Rod
Body-to-Body: Revolute	Gudgeon Pin	Con Rod
Body-to-Body: Fixed	Gudgeon Pin	Piston
Body-to-Body: Revolute	Crankshaft	Bearing

5.3 Contacts and constraints

The components are fixed by the following joints:

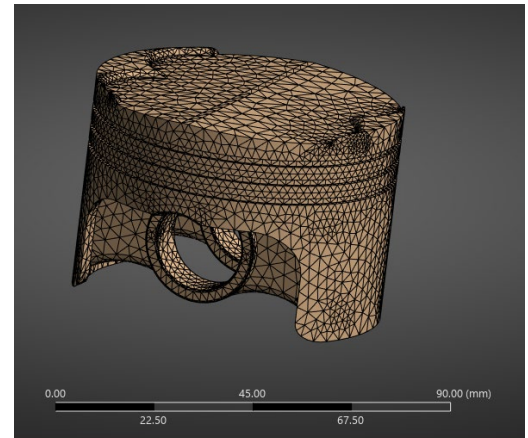
[Table 5] Model constraints

These constraints are needed for the assembly to function as a crank assembly would. Allow piston to move linearly and the crank to rotate around the z-axis. Once these constraints were applied, geometric checks using the configure function were performed to ensure that the piston and other parts moved correctly through 360°.

6 Kinematic and Motion Loads

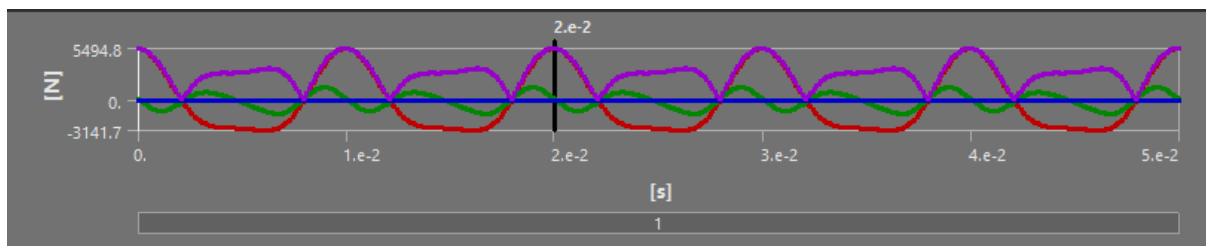
6.1 Kinematics Method

The kinematics study was performed to obtain realistic inertia and reaction forces over an engine cycle. Friction effects were excluded and all components were modelled as rigid bodies. Engineering data and imported geometry were used to define the model, with constraints applied as described in section 5.2 to ensure correct connectivity and motion. A 4mm quadratic mesh was generated using slow transition settings and a medium span angle centre; smaller element sizes were trialled but resulted in excessive computational cost. Analysis settings were adjusted to output only essential data. A step end time of 0.05s was used to capture five crankshaft revolutions with a step time of 0.001s to resolve peak accelerations at TDC and BDC (Figure 9). The crankshaft was driven at a constant speed of 6500rpm (Previous coursework data, 2024).



6.2 Results

The results from this are shown below in Figures 9 and 10. These results are critical study as they provide the time-varying inertial and reaction loads acting at the gudgeon-pin-piston joint throughout the engine cycle, with peak loads occurring near top and bottom dead centre. The components were modelled as rigid bodies, as the objective of this analysis was load extraction rather than deformation prediction. The motion loads generated from this study are subsequently imported into the static structural analysis to enable realistic evaluation of piston deformation and von-Mises stress.



[Figure 9] Motion Loads Graph

Tabular Data								
	Time [s]	☒ Joint Probe (Total Force X) [N]	☒ Joint Probe (Total Force Y) [N]	☒ Joint Probe (Total Force Z) [N]	☒ Joint Probe (Total Force Total)			
201	2.e-002	5481.4	157.84	-3.5549e-013	5483.7			
202	2.01e-002	5443.5	-24.281	-3.5351e-013	5443.5			
203	2.02e-002	5370.9	-201.94	-3.4928e-013	5374.7			
204	2.03e-002	5264.7	-371.46	-3.4284e-013	5277.7			
205	2.04e-002	5125.7	-529.38	-3.3424e-013	5153.			
206	2.05e-002	4955.5	-672.5	-3.2358e-013	5001.			
207	2.06e-002	4755.7	-797.95	-3.1094e-013	4822.1			
208	2.07e-002	4528.	-903.28	-2.9644e-013	4617.2			
209	2.08e-002	4274.5	-986.45	-2.807e-013	4386.9			

[Figure 10] Motion Loads

7 Static Structural and Thermal Analysis

7.1 Steady State Thermal Setup

The next stage of component analysis is a steady-state-thermal study, which evaluates the heat transfer during an engine cycle and assesses the thermal behaviour of the piston under representative operating conditions. This analysis is important during the design process due to the high thermal loads experience by the piston, which must maintain structural integrity and durability over its service life.

To perform the thermal analysis, the model geometry and boundary conditions were appropriately defined. As thermal and static structural analyses focus solely on the piston, all other components were suppressed and the piston was modelled as a flexible body to allow thermal deformation. The thermal convection boundary conditions applied in this study are summarised in table 6. During combustion, the piston crown is subjected to the highest thermal loading while the lower regions of the piston remain significantly cooler due to heat transfer to the lubricating oil.

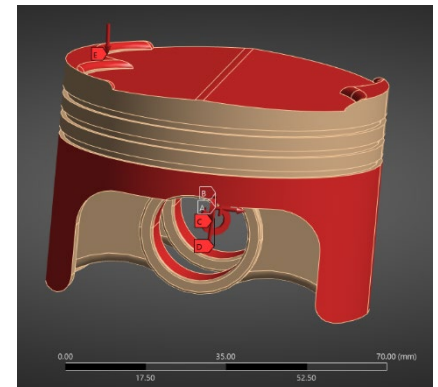
[Table 6] Thermal boundary conditions

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

The applied thermal loads shown above (table 6) are consistent with published piston thermal analysis for combustion engines (Hu, Yang and Dong, 2021).

7.2 Static Structural Setup

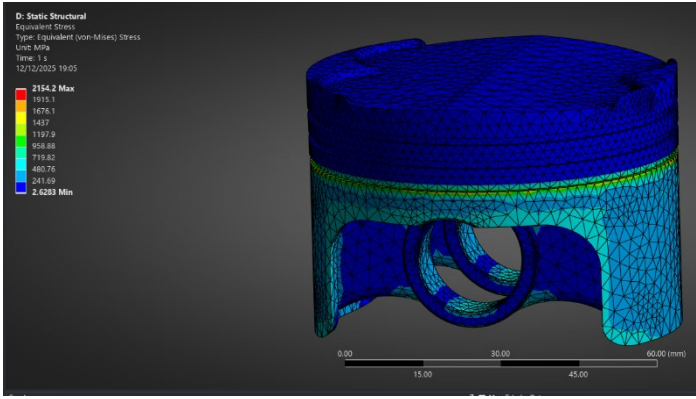
Similarly to the thermal analysis, all components except the piston were suppressed and the piston was modelled as a flexible body. Motion loads exported from the kinematics analysis were imported to represent inertial loading throughout the engine cycle, including loads acting on the gudgeon-pin bore, skirt and crown (Figure 11). As shown in the workflow in figure 5, the thermal solution is directly coupled to the static structural analysis to account for temperature effects. A peak combustion pressure of 72.631Bar, obtained (Previous coursework data, 2024), was applied to the piston crown to represent combustion loading.



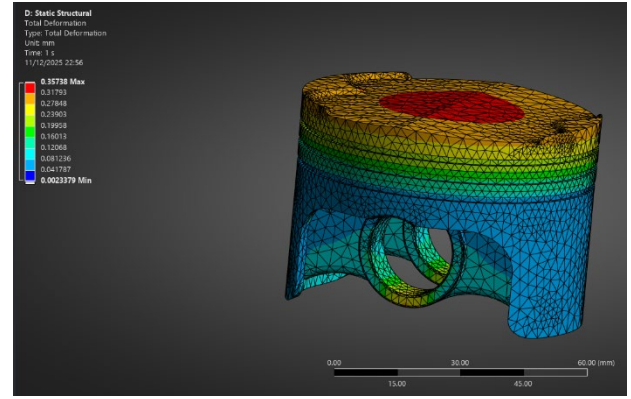
[Figure 11] Inertial load points

7.3 Results

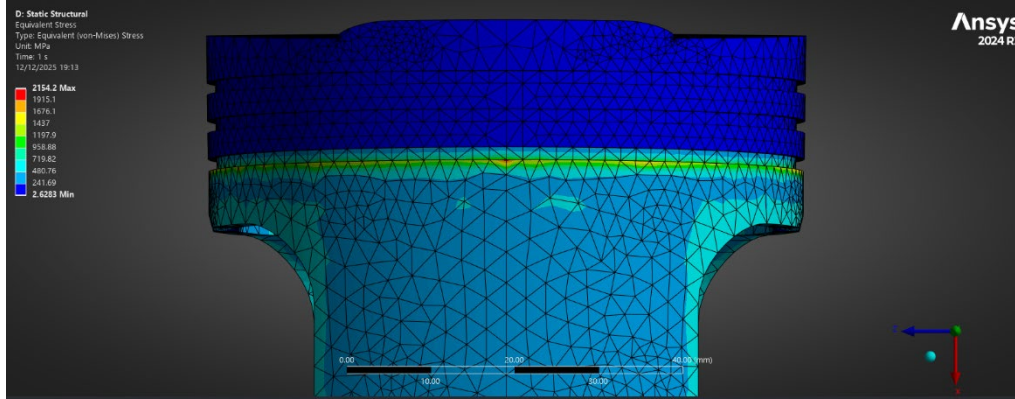
The results show the total deformation, von-Mises stress distribution and temperature field in the piston under combined inertial loading and peak combustion pressure. Figure 13 indicates that the maximum deformation of 0.36mm occurs at the piston crown, which is expected due to the combined effects of high combustion pressure and elevated temperature, consistent with findings reported in literature by (Singh Gahir and Kumar, 2021). Additional deformation is observed around the gudgeon-pin bore, which is attributed to high inertial forces transmitted through the piston-pin interface.



[Figure 12] Stress distribution



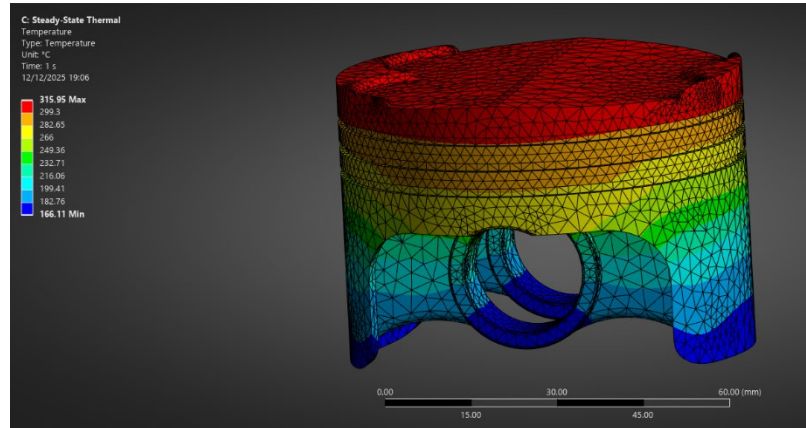
[Figure 13] Total deformation



[Figure 14] High stress region

The von-Mises stress distribution shows elevated stresses around the oil groove region. However, closer inspection indicates that this region contains locally irregular and relatively coarse mesh elements (figure 14), suggesting that the peak stresses of 2154.2MPa in this area are influenced by mesh concentration rather than true physical behaviour. Excluding this region the highest stresses are ~800MPa and observed around the gudgeon-pin-bore and crown, which aligns with expected piston loading patterns reported by Zhao (2021). Across the remainder of the piston, stress levels remain below 800MPa.

The thermal results show the highest temperatures of 315.95°C at the piston crown, where combustion heat is applied, with temperatures decreasing to 166.11°C at the base of the piston due to distance from the combustion zone and heat transfer to the lubricating oil. This temperature distribution is consistent with expected piston thermal behaviour and provides the basis for thermally coupled structural analysis.



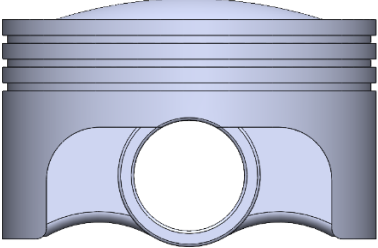
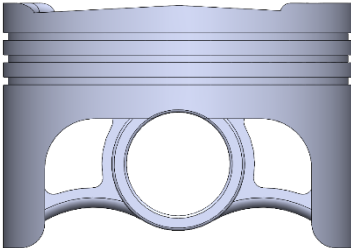
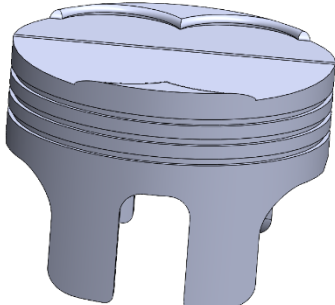
[Figure 15] Thermal distribution

The maximum von-Mises stress predicted was 2154.2MPa at 315.95°C, which exceeds the temperature-dependent yield strength of AA6061-T6. This peak occurs in a localised region associated with mesh irregularity and is therefore not considered physically representative. Excluding this region, stresses remain significantly lower but still indicate limited suitability of AA6061-T6 at these temperatures. Consequently, while AA6061-T6 is retained as a baseline material to demonstrate the CAE workflow, higher-strength aluminium alloys such as AA7050 (Chakravarti and Kumar Biswas, 2024) would be more appropriate for practical piston applications.

8 Optimisation and Improvement

Using the FEA results, the piston geometry was modified to improve its structural performance, with a primary focus on reducing total deformation while maintaining a low mass-and therefore low inertia. Geometric parameters were adjusted based on stress distributions observed in the baseline analysis, targeting regions of high stress and low structural efficiency. Guidance from established piston design practice was used to inform these modifications (Mahle International GmbH and Springer Fachmedien Wiesbaden GmbH, 2018), resulting in the development of the following design variants.

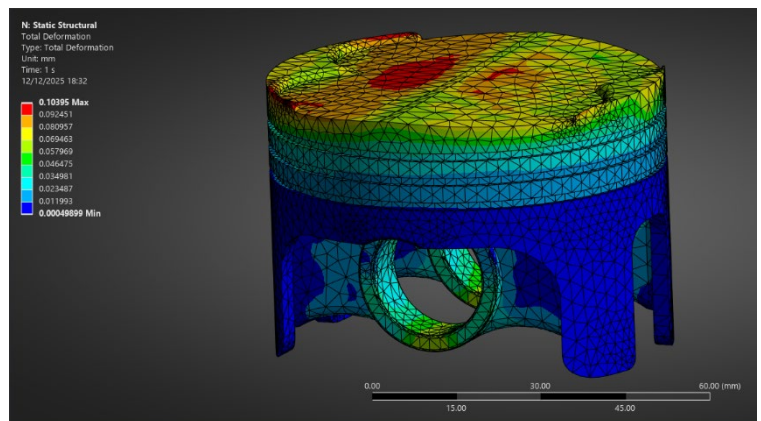
[Table 7] Optimisation designs

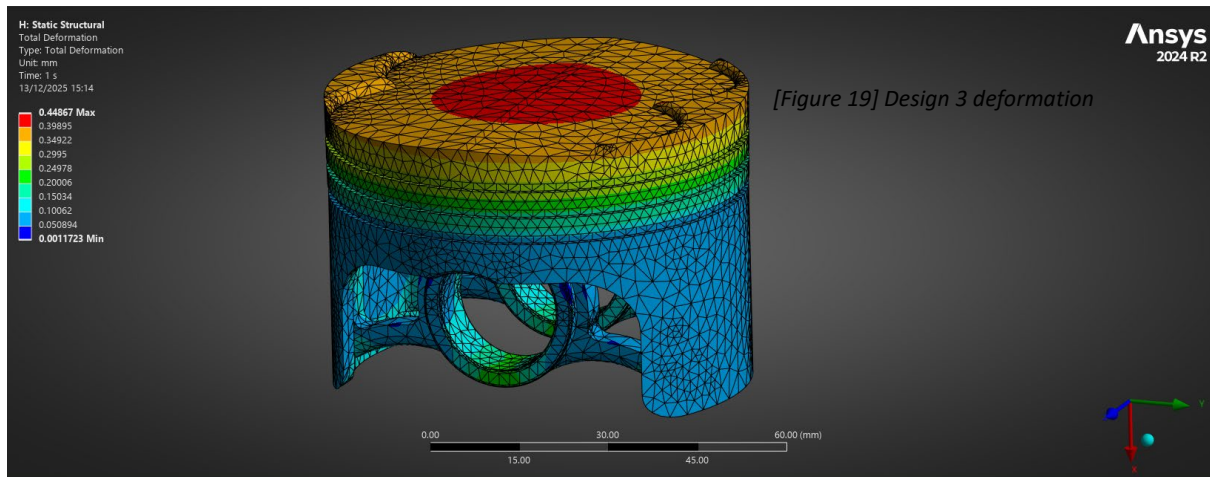
Design 1	Design 2	Design 3
 [Figure 16] Design 1	 [Figure 17] Design 2	 [Figure 18] Design 3
Increased crown surface area to promote more uniform heat distribution. The skirt length was reduced to decrease mass while maintaining structural integrity.	Material was removed from regions experiencing lower stress and thermal loading in attempt to reduce overall piston mass.	An alternative mass reduction strategy was implemented by removing material in low-stress regions while maintaining load-bearing features.

[Table 8] Design result comparison

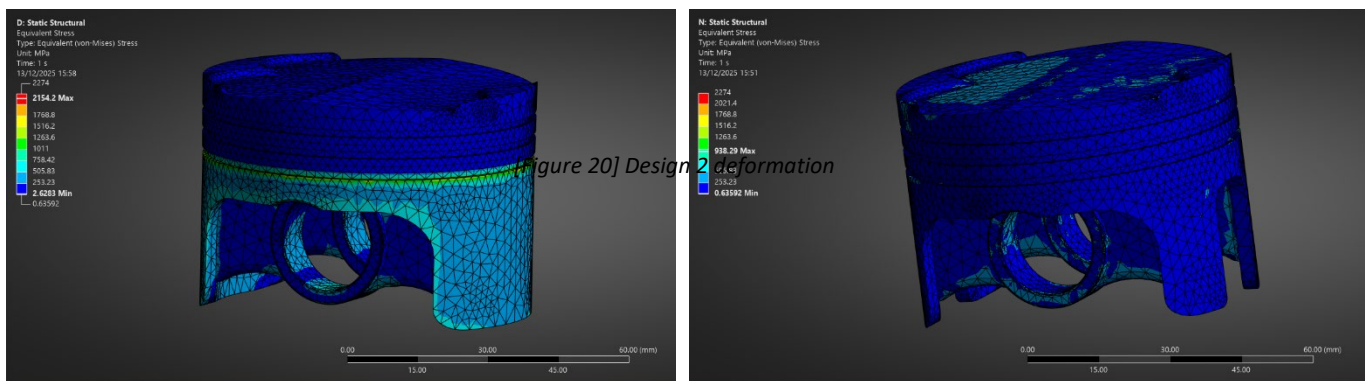
		Deformation		Stress		Temperature	
	Mass (kg)	Max (mm)	Min (mm)	Max (MPa)	Min (MPa)	Max (°C)	Min (°C)
Baseline	0.25482	0.35738	0.0023379	2154.2	2.6283	359.8	225.35
Design 1	0.24734	0.36779	0.0005056	2139.6	2.0029	312.8	166.87
Design 2	0.24426	0.44867	0.0011723	2274	3.3271	371.84	218.94
Design 3	0.24863	0.10395	0.000499	938.29	0.63592	366.37	234.94

Table 8 compares the baseline piston to the three design variants under the same combined inertial, thermal and combustion-pressure loading. In terms of deformation, the baseline piston reaches a maximum displacement of 0.357mm at the crown, with design 1 showing a similar response (0.368mm). Design 2 produces the highest deformation (0.449mm), consistent with reduced underside support lowering overall stiffness. In contrast, design 3 shows the lowest deformation (0.104mm); the crown displacement pattern appears less uniform, which is consistent with stiffness redistribution from the modified skirt geometry and gudgeon-pin load path.



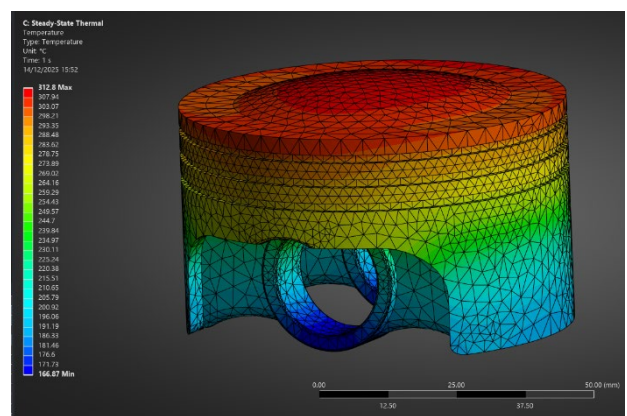
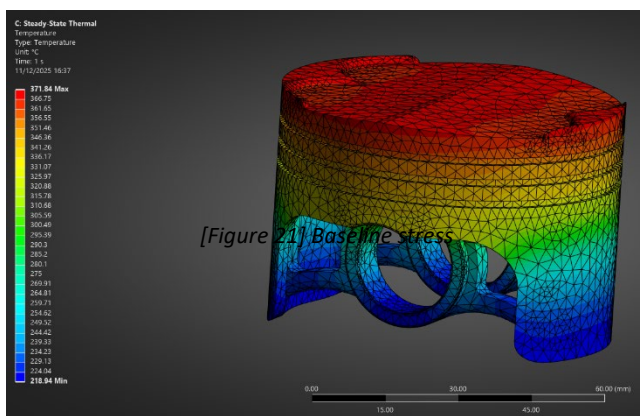


For equivalent (von-Mises) stress, the baseline and designs 1-2 show high peak values (~ 2.1 - 2.3 GPa). The peak in baseline is associated with a localised region around the ring/oil-groove edge where the mesh irregularity can amplify stress, so this maximum is treated cautiously. Design 3 reduces the maximum stress subsequently to 938 MPa, and removed the previously observed edge hotspot, indicating improved stress distribution and a more realistic peak location within the primary



load path (crown and pin-boss region). Design 2 shows the highest stress (2274 MPa), which aligns with reduced torsional stiffness from the underside of the material.

Thermally, all designs show the expected trend of highest temperature at the crown and lowest temperatures towards the piston base. Design 2 exhibits the highest crown temperature (371.8°C),



likely due to reduced material available for heat spreading, while design 1 shows the lowest maximum temperature (312.8°C). Minimum temperatures vary only moderately across designs (~167-235°C), indicating that the design changes primarily influence the crown region rather than the cooler lower sections.

9 Discussion and limitations

The results from the thermal and structural analyses show generally realistic behaviour in terms of stress, temperature and deformation trends across the piston designs. With the exception of design 3, the predicted responses were consistent given the engine loading conditions. The behaviour observed in all but design 3 highlights the sensitivity of stress results to mesh quality, where local mesh inconsistency led to artificially high stress concentrations. Excluding this high value result, the designs demonstrated stable and physically reasonable behaviour under engine forces.

Although peak stresses in some regions exceeded the material yield strength, the results remain suitable for comparative assessment between designs. Validation against analytical calculations showed a small percentage difference, indicating that the simulation provides a reliable representation of the dominant physical behaviour and supports its use for component evaluation.

Despite this, several limitations exist. Material behaviour was idealised, neglecting manufacturing defects and residual stresses. Geometric simplifications were applied for computational efficiency, and materials were assumed to be homogeneous and isotropic. Boundary conditions and operating conditions were assumed steady and exact, and contact interactions were simplified. Finally the mesh was deemed adequate; however a full mesh convergence study was not performed.

10 Conclusion

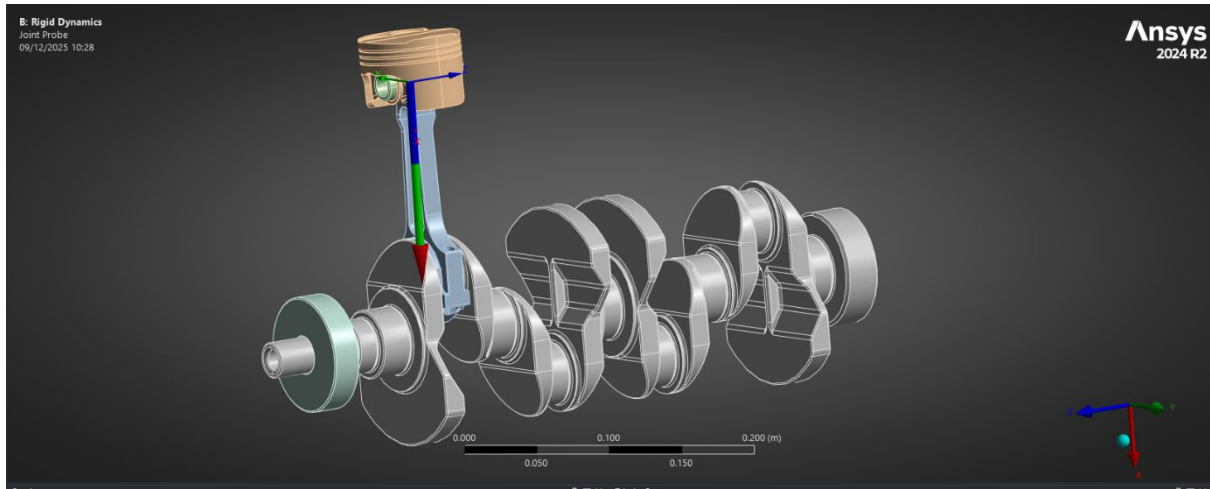
This report assessed the applicability of finite element methods for predicting piston thermo-mechanical behaviour through numerical and analytical comparison. Careful selection of mesh quality and boundary conditions enabled a reasonably reliable simulation to be produced, and with further work could achieve high accuracy. The results from this study demonstrate the value of CAE in automotive design by allowing performance trends to be evaluated efficiently. Reducing piston mass was found to have limited influence on inertial loads but a significant effect on structural stiffness. Future work could investigate alternative materials to improve accuracy and correlation with real-world behaviour.

11 References

- ANSYS Granta (2024) *Granta EduPack 2024*. Cambridge: ANSYS Granta. Available at: ANSYS Granta database (Accessed: 10 December 2025)
- Čepon, G., Starc, B., Zupančič, B. and Boltežar, M. (2017). Coupled thermo-structural analysis of a bimetallic strip using the absolute nodal coordinate formulation. *Multibody System Dynamics*, 41(4), pp.391–402. doi:<https://doi.org/10.1007/s11044-017-9574-7>.
- Chakravarti, S. and Kumar Biswas, A. (2024). Design and Structural Analysis of Engine Piston. *International Journal of Scientific Research in Science and Technology*, 11(18). doi:<https://doi.org/10.32628/IJSRST>.
- Dlubal Software (2015). *Verification Example 0042 -Bending Cantilever with Axial Force*.
- Dlubal Software (2018). *Verification Example 0078 -Bimetallic Strip Description*.
- Hu, L., Yang, J. and Dong, F. (2021). Evaluation and sensitivity analysis of a piston's thermal load in a high-power diesel engine. *International Journal of Engine Research*, 23(6), pp.927–939. doi:<https://doi.org/10.1177/1468087421998633>.
- Jadhav, N.P. and BedajangsagarKishor (2014). Optimization of Geometrical Parameters of Four Stroke Engine Piston. *International Journal of Engineering Research and Applications*, 4(2).
- Mahle International GmbH and Springer Fachmedien Wiesbaden GmbH (2018). *Pistons and engine testing*. Wiesbaden Springer Fachmedien Wiesbaden GmbH Springer Vieweg.
- Safdarian, R. and Parente, M.P. (2024). Investigating the formability of 6061-T6 aluminum alloy sheets at elevated temperatures using experimental and numerical methods. *Proceedings of the Institution of Mechanical Engineers, Part L: Journal of Materials: Design and Applications*, 238(11), pp.2245–2258. doi:<https://doi.org/10.1177/14644207241243304>.
- Singh Gahir, G. and Kumar, S. (2021). *FINITE ELEMENT ANALYSIS OF SOME PISTON MATERIALS OF IC ENGINE*.
- University of Wales Trinity Saint David (2024) Thermodynamics Data: Provided by Owen Williams. Ford Sigma Cylinder Pressure data. Unpublished student coursework material.
- Wang, Y., Liu, Y. and Shi, H. (2010). Simulation and Analysis of Thermo-Mechanical Coupling Load and Mechanical Dynamic Load for a Piston. Second International Conference on Computer Modeling and Simulation. pp.106–110. doi:<https://doi.org/10.1109/iccms.2010.131>.
- Zhao, F. (2021). Modeling and Thermal-Mechanical Coupling Analysis of Piston in Car Engines. *Annales de Chimie - Science des Matériaux*, 45(1), pp.83–92. doi:<https://doi.org/10.18280/acsm.450111>.
- Zheng, B., Shui, Q., Luo, Z., Hu, P., Yang, Y., Lei, J. and Yin, G. (2025). Optimization of Engine Piston Performance Based on Multi-Method Coupling: Sensitivity Analysis, Response Surface Model, and Application of Genetic Algorithm. *Materials*, 18(13), p.3043. doi:<https://doi.org/10.3390/ma18133043>.

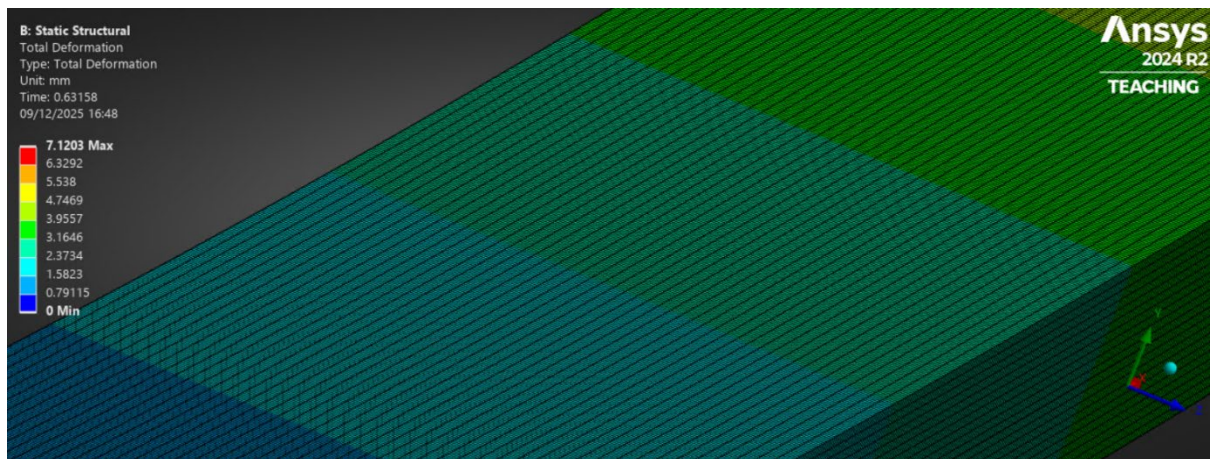
Appendices

Motion load force direction:

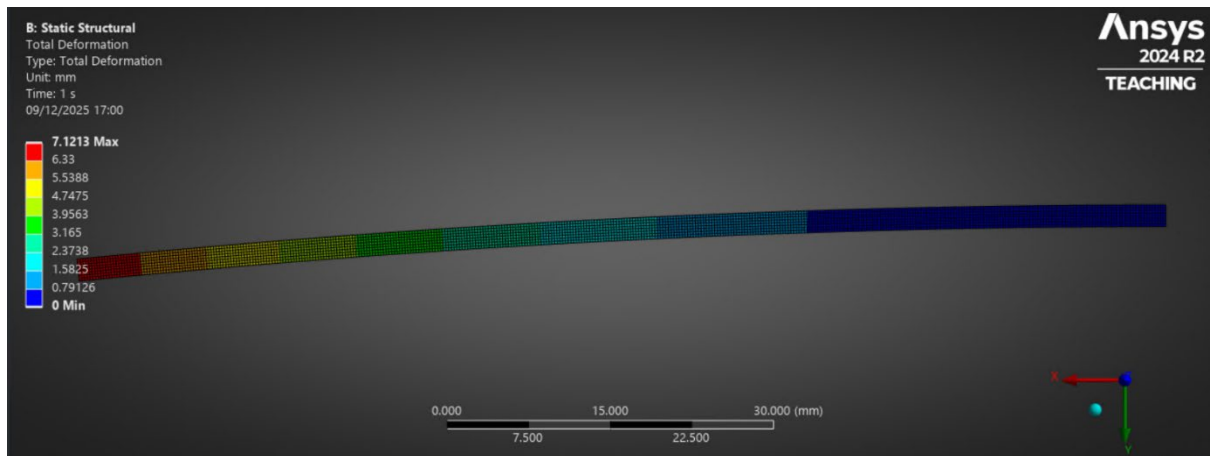


Validation:

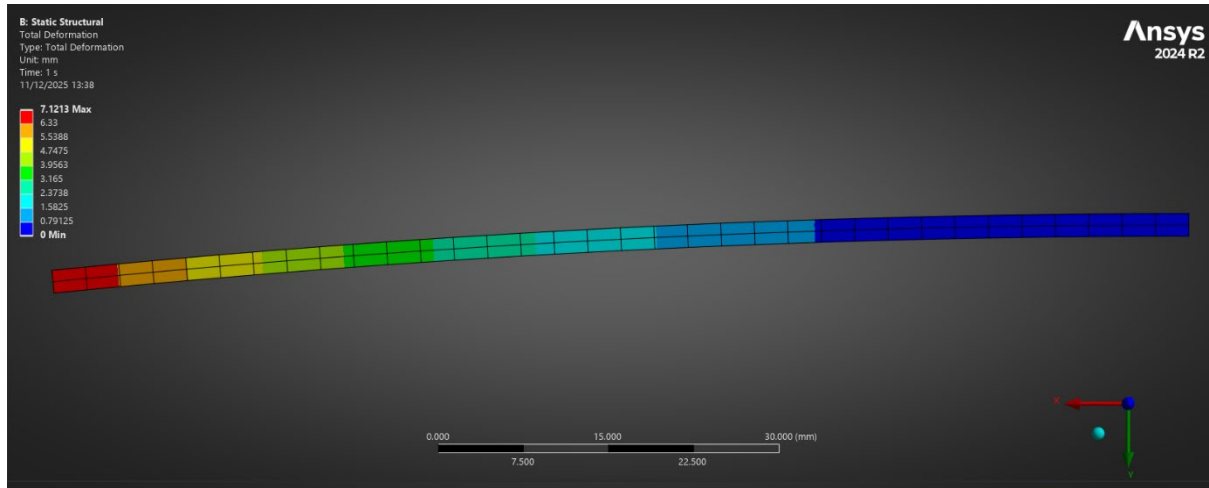
0.1mm mesh:



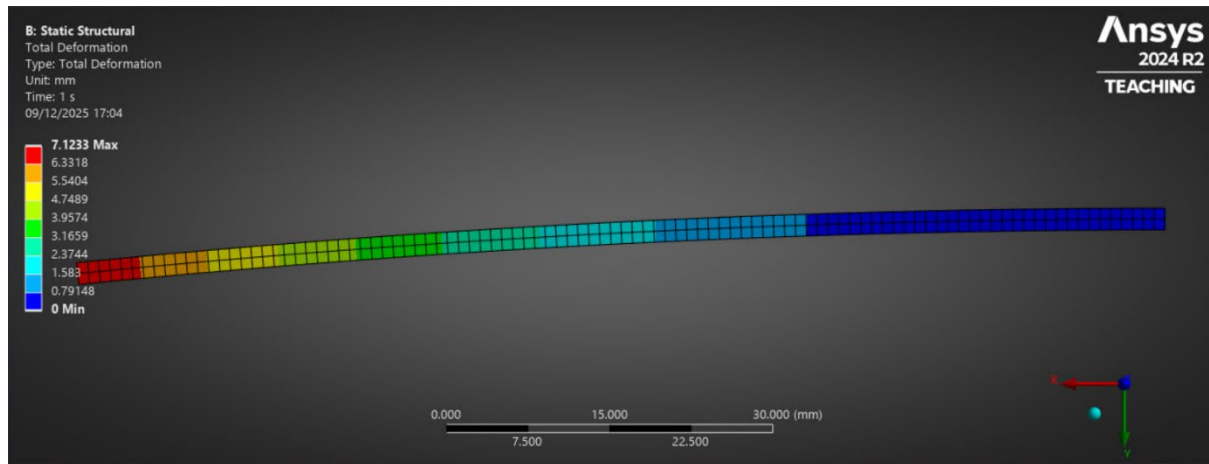
0.25mm mesh:



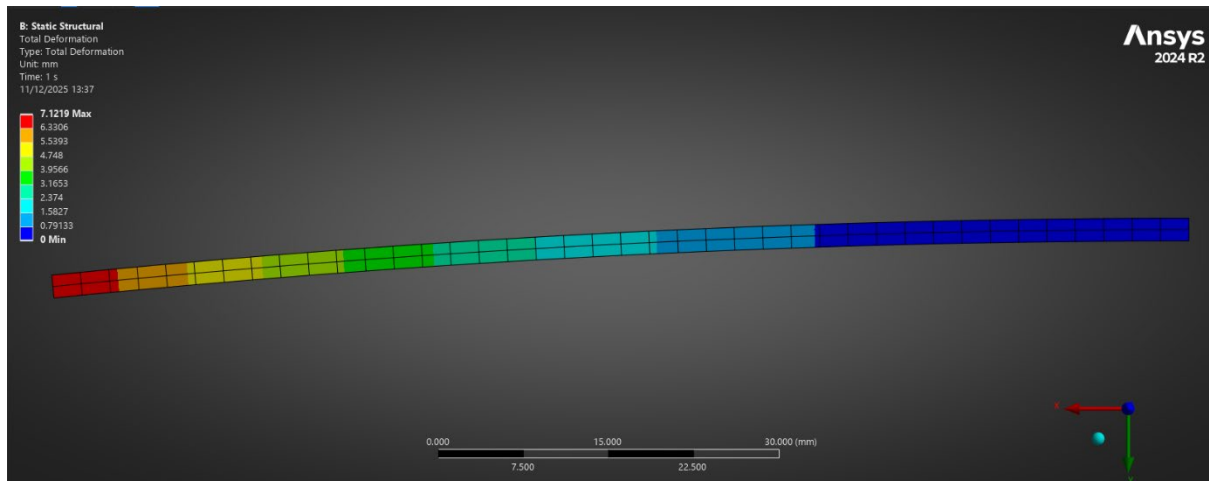
0.5mm mesh:



1mm mesh

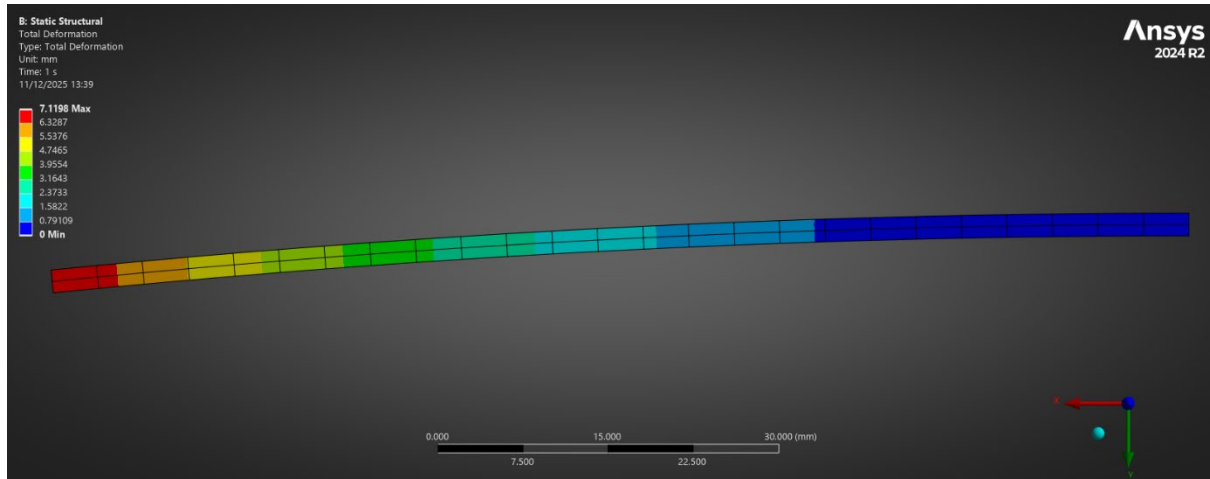


2.5mm mesh

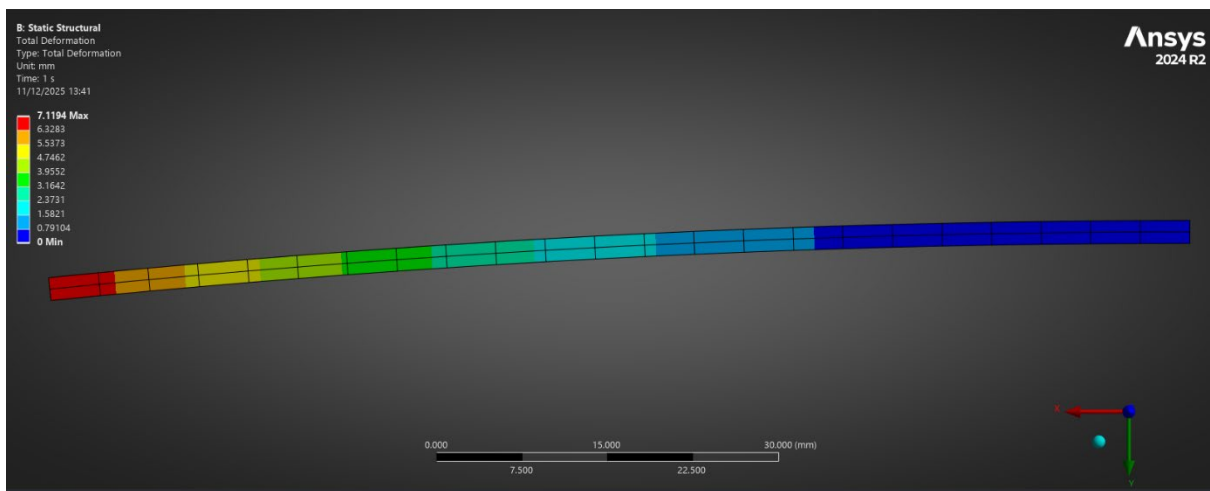


3mm mesh

4mm mesh

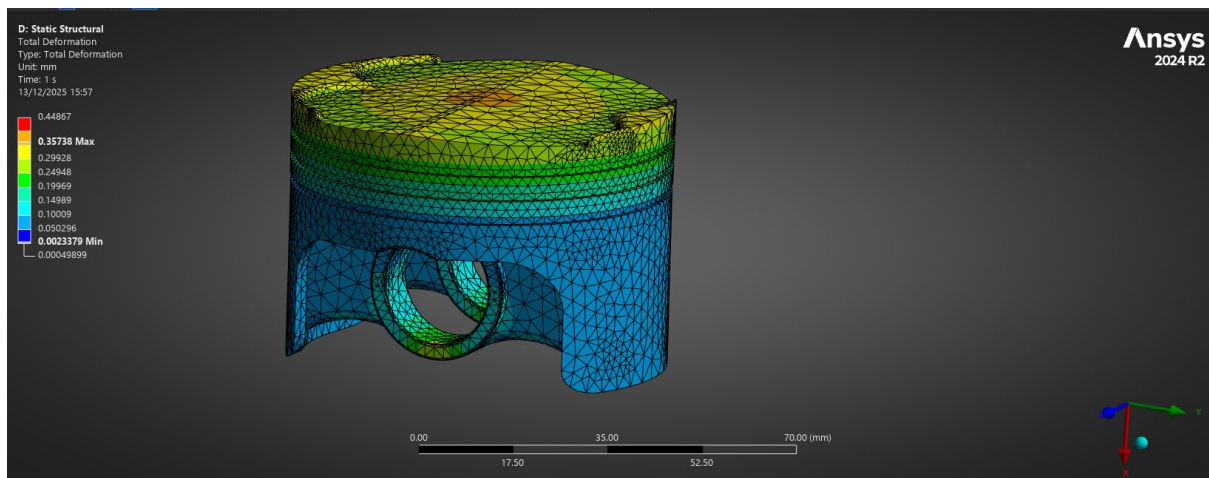


4.5mm mesh

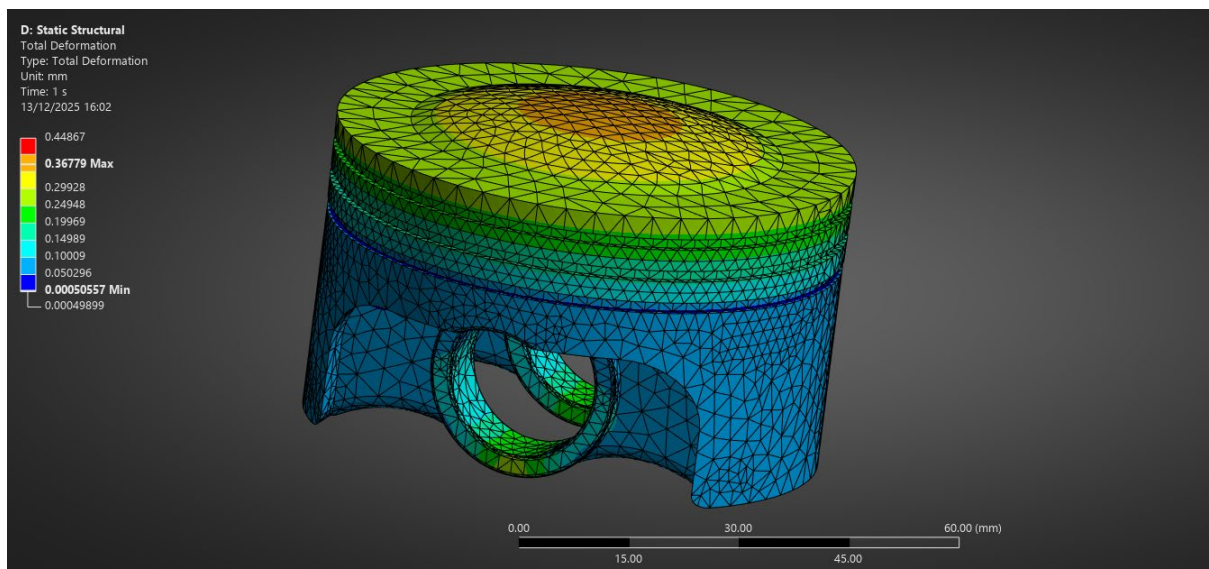


Deformation:

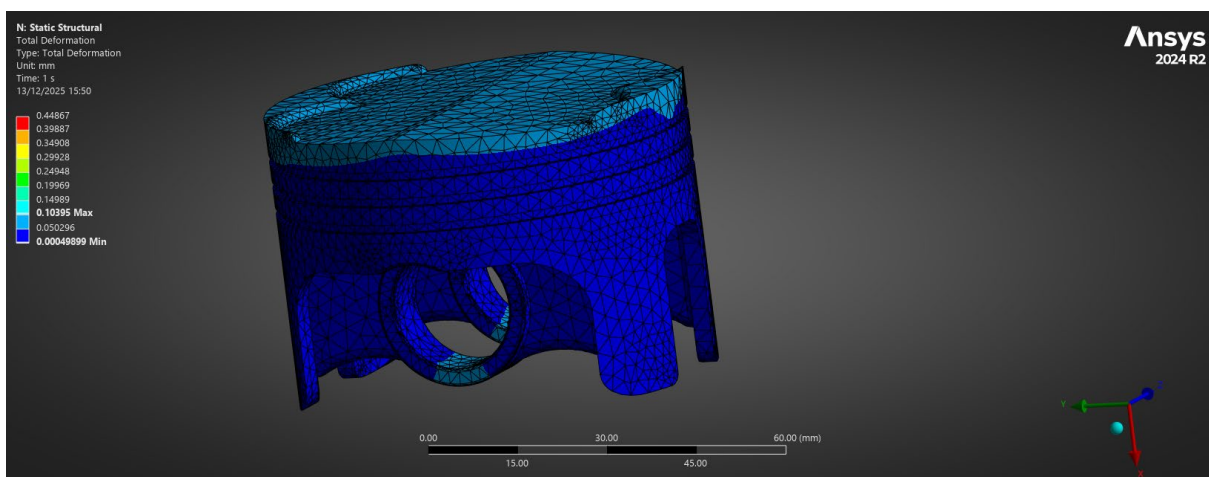
Piston 1



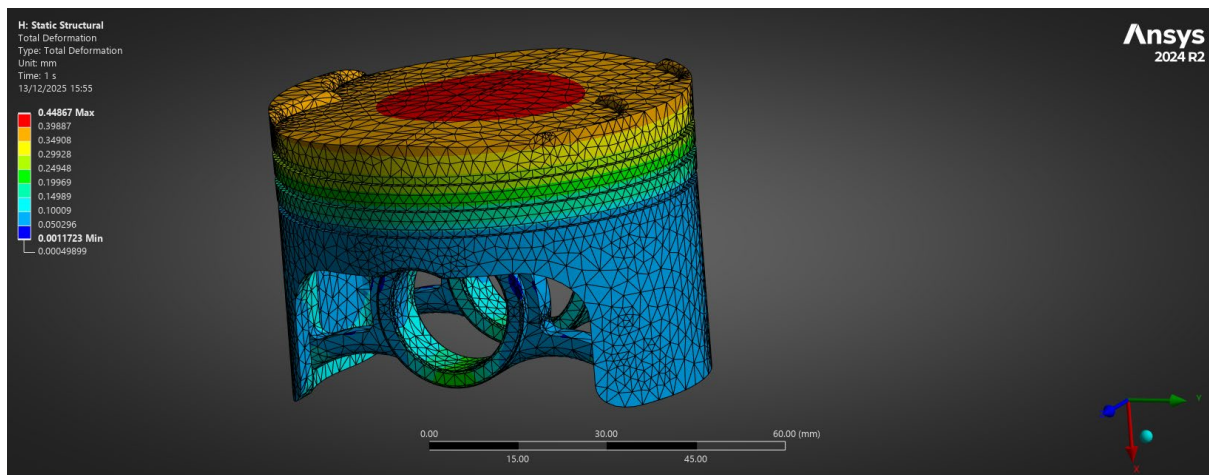
Piston 2



Piston 3

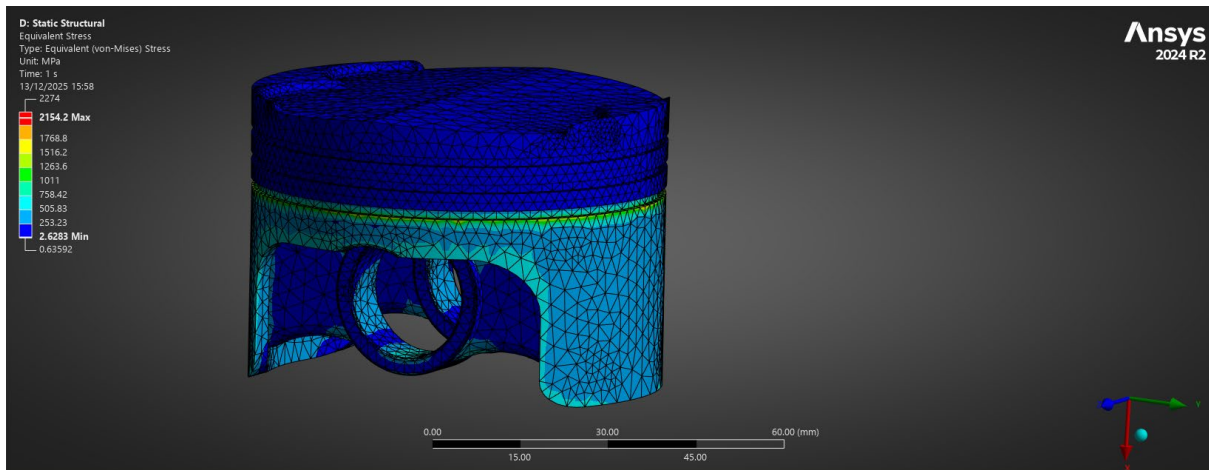


Piston 4

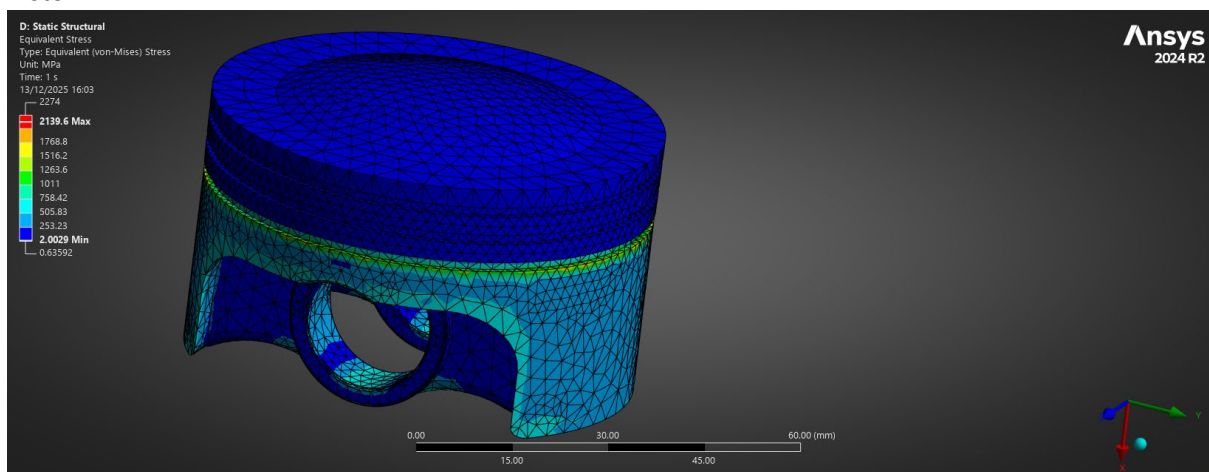


Stress:

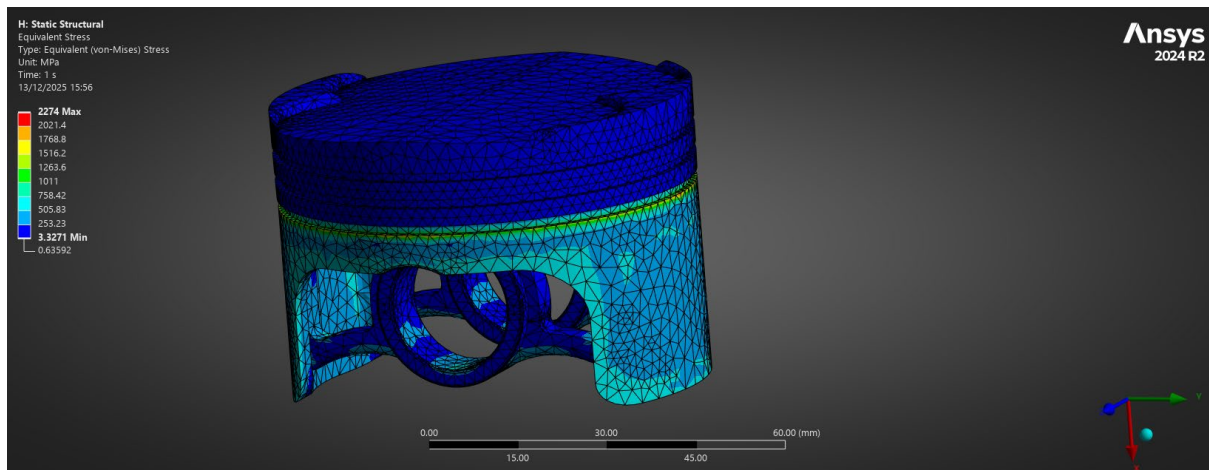
Piston 1



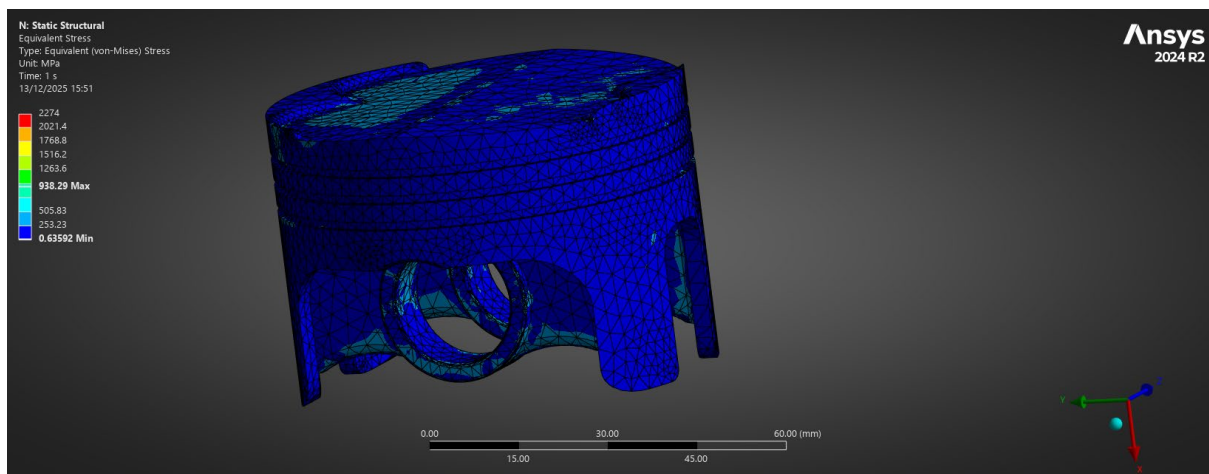
Piston 2



Piston 3

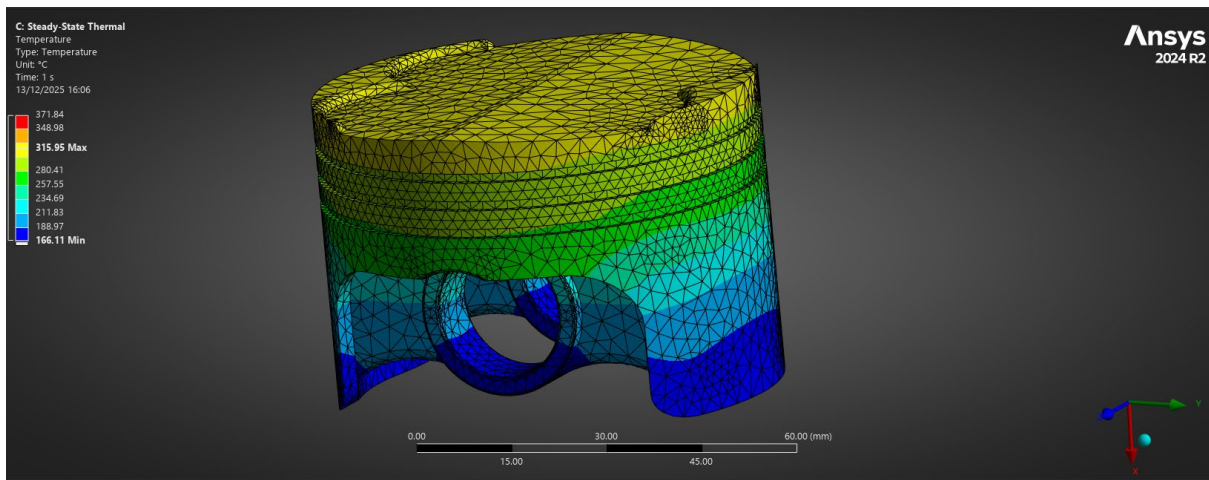


Piston 4

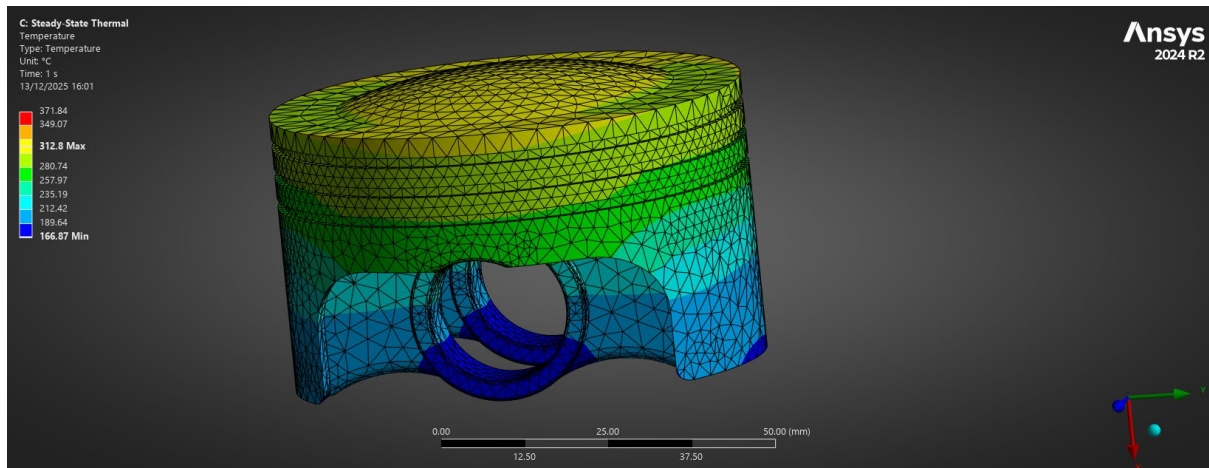


Thermal:

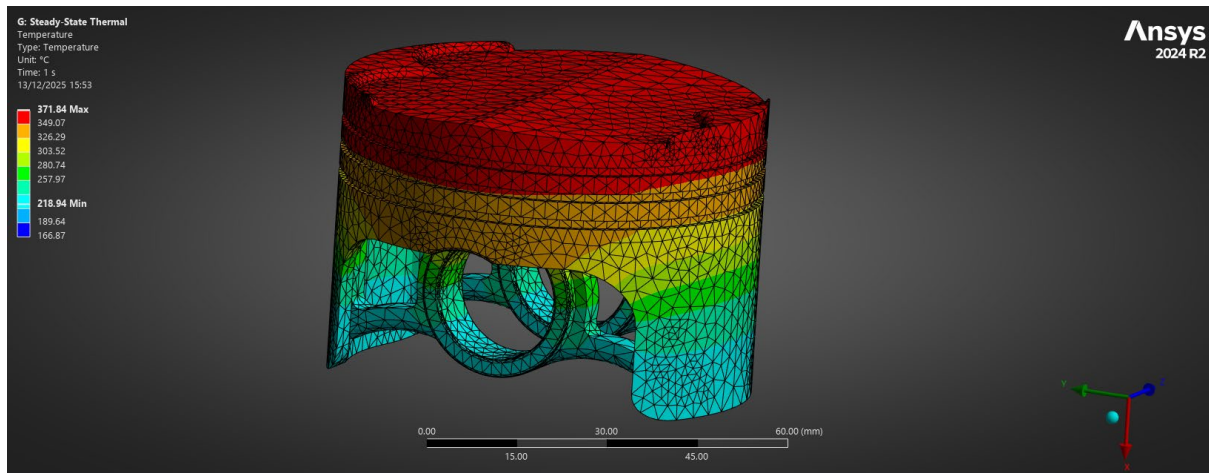
Piston 1



Piston 2



Piston 3



Piston 4

