

University of Wales Trinity Saint David
School of Engineering



Mechanical and Manufacturing
Engineering

Structural & Fluid Analysis

Sheunesu John Ziki

2205423

14/12/25

Table of Contents

1	Introduction	4
2	Literature review	5
3	Methodology.....	7
3.1	Assembly modelling and definitions	7
3.2	Joint definitions and rigid dynamics	8
3.3	Steady state thermal study	8
3.4	Static structural analysis	9
3.5	Fatigue assessment	10
3.6	Design iteration for mass and performance	10
3.7	Mesh independence and validation	11
4	Results	11
4.1	Original piston results	11
4.1.1	Steady state thermal	11
4.1.2	Total deformation under load	12
4.1.3	Von Mises stress.....	12
4.2	New piston results	13
4.2.1	Steady state thermal	13
4.2.2	Total deformation.....	14
4.2.3	Von Mises stress.....	14
5	Discussion.....	15
6	Conclusion.....	16

Figure 1:Image showing the assembly used.	7
Figure 2: Shows assigned material properties.	7
Figure 3:Shows the mesh created.....	9
Figure 4:Shows the fully linked system created.	9
Figure 5:Image showing the different mesh sizes used.	10
Figure 6:Shows temperature distribution from steady state analysis.	11
Figure 7: Shows the total deformation of the piston.....	12
Figure 8: Show von Mises stress distribution.	13
Figure 9: Shows new piston steady state thermal results.	13
Figure 10: shows the total deformation of the new piston.	14
Figure 11: Shows von Mises stress for the new piston.	15

1 Introduction

Computer aided engineering (CAE) is a useful tool used in engineering design as it allows for the analysis of components before their prototypes are produced. The simulation tools have allowed for detailed analysis of stresses, deformation, thermal loads, fatigue and kinematic behaviours under realistic working conditions. Although CAE has become widely used in industry it still is an approximation of real working conditions and results are influenced by modelling assumptions, boundary conditions, mesh quality and the accuracy of input data such as material properties. Due to it being an approximation a gap is left for engineers to fill by running several simulations and validating the results using theoretical and experimental work to ensure that the results are reliable (Bahman and Iannuzzo, 2018).

This report will look at the application of CAE processes to analyse and optimise a single component (piston) from a slider crank assembly consisting of a piston, gudgeon pin, connecting rod and crankshaft using ANSYS. The assembly will be subjected to a kinematic study to establish loading conditions that reflects its operational motion. Relevant thermal conditions and the results from the kinematic study will then be used in a static structural study of the piston to allow for the identification of deflection patterns, stress concentrations, and fatigue problems. The results from all the studies will be the basis of the pistons redesign to improve its strength, fatigue life, and reduction in mass to show how CAE can be used to improve engineering decisions during a components development.

To evaluate the reliability of the numerical predictions a simplified test will be conducted using an analytical solution and will be compared to the ANSYS results. This solution will be supported by a mesh independence study to demonstrate numerical stability. The combination of simulation, optimisation of components and validation of results will be used to discuss the capabilities and limitations of CAE tools and how they influence engineers in decision making.

2 Literature review

Computer aided engineering (CAE) tools based on finite element analysis (FEA) are now widely used in piston and slider crank designing due to being able to predict stress, deformation, temperature and fatigue before manufacturing (Karem and Ismail, 2022). Studies have shown that 3D thermo-mechanical FEA is used to optimise piston's crown thickness, pin boss size and skirt design to keep stresses below the materials limits while reducing mass. These tools make it possible to evaluate complicated loading combinations such as gas pressure, inertia and thermal gradients that would be difficult to solve using hand calculations (Havale and Santosh Wankhade, 2017).

Research also emphasises on how CAE results are only as reliable as the modelling assumptions, material data and validation. Several papers have shown that pistons operate in a nonlinear setting with temperature dependent material properties, frictional contact and potential elasticity under extreme conditions. This means that linear elastic models that have simplified boundary conditions can misinterpret real stress (Subbarao and Vart Gupta, 2020). Research on the slider crank simulations has also shown that assumptions about joint stiffness, clearances and lubrication have a big impact on the predicted side thrust and pin loads that are applied to the piston model. This show that CAE should be used with awareness of how simplifications generate throughout the analysis tree (Wakshume and Sufe, 2025).

Recurring subjects in piston FEA research papers is the need for cautious treatment of the thermal loading and thermos mechanical coupling. Researchers usually start with a steady state thermal analysis by adding convection and radiation coefficients on the piston crown, ring belt, piston skirt and surfaces under the crown to approximate heat transfer to gases, coolant and oil (Mahajan and Deshmukh, 2016). The temperature field is then shaped into a structural analysis to return both the thermal and mechanical stresses. In comparative studies it is shown that disregarding temperature dependence can under predict the peak von Mises stresses and miss tensile regions on the piston crown, ring belt fillet and pin boss (Mahajan and Deshmukh, 2016). Thermal boundary conditions are often altered using experimental measurements showing that purely numerical predictions need calibration to show real operating conditions (G.V.N Kaushik, May2019).

Meshing strategy and model validation are seen as the key factors that affect CAE results reliability. Many research papers for piston design mention mesh refinement studies where the element size is reduced until the changes in peak stress fall below a tolerance. This shows that the solutions are mesh independent (Havale and Santosh Wankhade, 2017). Validation is usually carried out using simplified geometries with analytical solutions, such as flat planes or beams to ensure that the numerical results can be checked against closed form stress and deflection equations. Researchers also compare FEA predictions with strain gauge readings on rig tests which show good trend agreement but also show differences in significance due to doubts in loads, boundary conditions and material properties (Karem and Ismail, 2022). These studies show that CAEE models should always be validated and not taken at face value.

Optimisation and fatigue studies have added to these ideas by using CAE to push geometry changes while knowing the limits of numerical prediction. Research has applied S-N curve data for aluminium alloys to stress histories taken from FEA and used it to change piston geometry to improve fatigue safety and reduce mass (Singh and Yadav, 2021). The topology approach has also been used to remove material from that crown and under the crown where considered low utilisation which produces a lighter piston without a large increase in stress (Khyat, 2020). Research also notes that calculated fatigue is sensitive surface finish, residual stress and thermal cycling which are mainly approximated in CAE showing the need for experimental validation (Khyat, 2020).

3 Methodology

3.1 Assembly modelling and definitions

The study was conducted on a cylinder slider crank assembly that consists of a piston, gudgeon pin, connecting rod and crankshaft as seen in Figure 1 below. The assembly is a simplified version of an internal combustion engine where the piston is constrained to move linearly while the crankshaft rotates at a prescribed speed. The components were designed and assembled in SolidWorks and imported into ANSYS as a .STEP file.

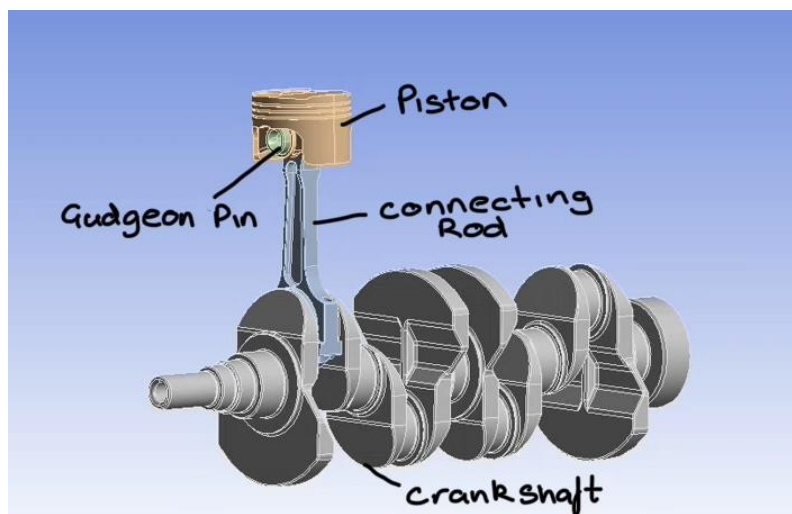


Figure 1: Image showing the assembly used.

The piston was assigned with the material 6061-T6 aluminium due to it being the analysed component while the rest of the components were assigned with structural steel. 6061-T6 aluminium was created in the engineering data tab in the ANSYS workbench where properties such as temperature dependency, density, Poisson's ratio, thermal conductivity and Young's modulus at different temperature's were defined as seen in Figure 2 below.

6061-T6 Aluminium Material Properties					
Young's Modulus			Poisson's Ratio		
Temp (°C)	Value	Unit	0.33		
			Coefficient Thermal Expansion 22°C		
22	67	Gpa	2.36E-05 1/°C		
100	70	Gpa	Thermal Conductivity		
150	68	Gpa	167 W/m-K		
200	64	Gpa	Specific Heat Capacity		
250	56	Gpa	0.896 J/g-°C		
300	48	Gpa			
350	41	Gpa			
400	32	Gpa			
450	21	Gpa			
500	12	Gpa			

Figure 2: Shows assigned material properties.

This combination of materials ensures that we see the realistic inertial behaviour when performing the rigid dynamics model, due to steel having an appropriate density for rotating the components. Other components/factors such as piston rings and oil clearances were not considered to keep the model simple and to help keep the mesh quality high without affecting the load transfer.

3.2 Joint definitions and rigid dynamics

A rigid dynamics analysis was conducted to find the realistic forces acting on the piston during the assembly's operation. Within the rigid dynamics model all the pre-established contact points created after importing the assembly were replaced mechanical joints. A revolute joint was used between the crankshaft's main journal and the ground to allow for the crankshaft to rotate. Another revolute joint was used between the crankpin and connecting rod to allow for the connecting rods linear motion. The final revolute joint was between the gudgeon pin and the piston bosses. The piston was constrained using a general joint to move up and down one axis to replicate it being fixed within a cylinder bore. These joints provided appropriate freedom for the assembly's motion. The bodies got their mass and inertia from the densities from the assigned materials.

A constant rotational velocity of 4000rpm was applied to the crankshaft as a joint load using the revolute joint between the crankshaft and ground creating the joint probe to replicate the operating speed. The analysis settings were set up to run through one full revolution at the crankshaft with a time step of 0.05 seconds to see the changes in the dynamic loading. After solving the peak total force values between the piston and the gudgeon pin and were exported as a .txt file which were imported into the static structural model.

3.3 Steady state thermal study

To capture the influence that temperature has on the behaviour of the piston a steady state thermal analysis was done. The engineering data and geometry from the rigid dynamics study was linked with the steady state study to ensure the correct material and geometry was imported into the study. All other components except the piston were then suppressed as the focus was to create a temperature profile for the piston. A mesh with an element sizing of 4mm was then created as needed for ANSYS to generate a solution as seen in Figure 3 below:

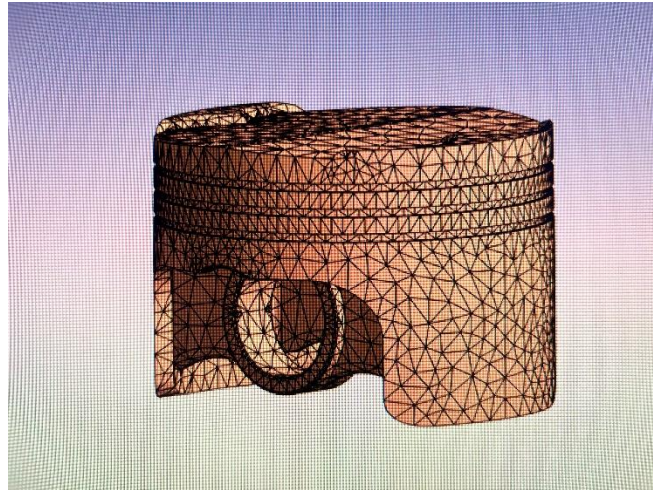


Figure 3:Shows the mesh created.

Convective boundary conditions were applied to the piston crown and sides of the crown to represent hotter combustion gases. The bottom of the crown, skirt and the pin bosses were assigned a cooler convection coefficient to approximate heat transfer. The solution from the steady state thermal study were then linked in the setup of the static structural study ensuring that temperature dependant decline of the material strength is shown. This also allowed for a completion of the fully linked system as seen in Figure 4 bellow:

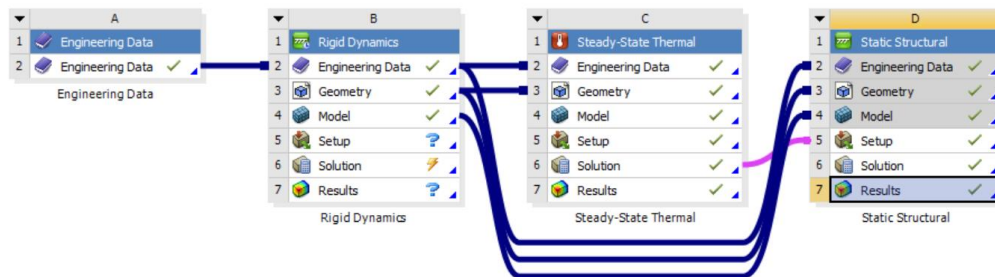


Figure 4:Shows the fully linked system created.

3.4 Static structural analysis

A static structural analysis was then created using the thermal and total force values established in the earlier studies. This study focuses on the piston meaning all other components were suppressed for this study. A mesh with an element sizing of 2mm was created for the piston crown and pin bosses whilst the rest was left with 4mm element sizing as shown in Figure 5 below.

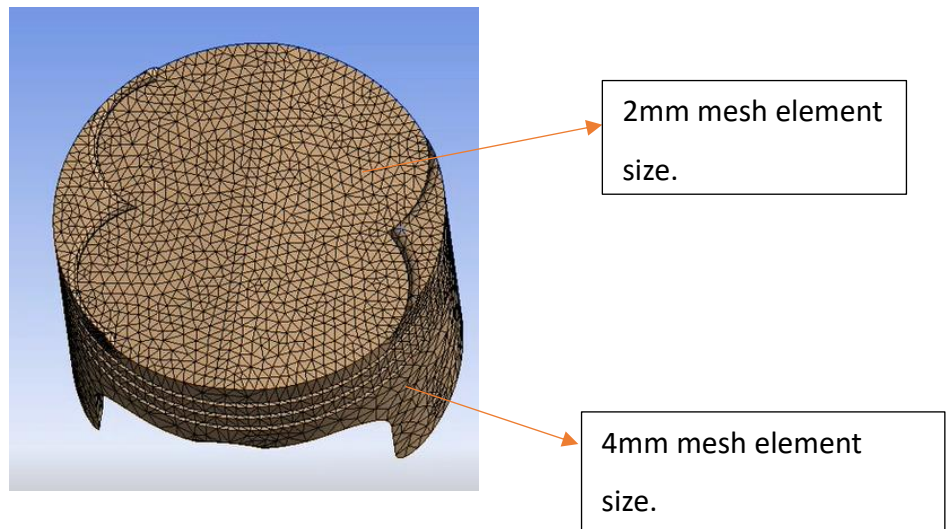


Figure 5: Image showing the different mesh sizes used.

The weak springs constraint was then applied to the piston as it allows for the piston to be supported while being fully flexible to allow for the total deformation to be seen without affecting the results. A pressure of 105Bar (10.5MPa) was also added to the piston crown to simulate combustion pressure in a cylinder. The solution produced a von Mises stress distribution and total deformation.

3.5 Fatigue assessment

The static structural solution provided the von Mises stress and total deformation for the piston under both thermal and mechanical loads which were used to estimate the alternating stress in key areas such as the piston crown and the pin bosses. These values were then used to estimate the fatigue safety factors in ANSYS at the mentioned key areas to assess if the piston would be able to withhold the imposed cyclic loads without failure.

3.6 Design iteration for mass and performance

Based on the results achieved, modifications were made to the pistons geometry to improve its stress distribution, fatigue and mass reduction. The modifications included adjusting the crown thickness, changing the fillet radii and removing material from low stress regions along the skirt. After each modification, a revaluation was done under the same boundary conditions and loads to allow for a comparison of stress, deformation, fatigue and mass. The modified design with the most effect was selected for achieving a balance between mass reduction, improved fatigue and keeping stress within good limits.

3.7 Mesh independence and validation

In order to validate the reliability of the results a mesh independence study was done by progressively refining the mesh and checking for any changes in the stress and displacement values. This ensured that the chosen mesh density produced accurate results.

4 Results

4.1 Original piston results

4.1.1 Steady state thermal

Figure 6 below shows the temperature distribution from the steady state thermal model of the piston before any changes. The piston crown is the hottest region with a maximum temperature of $\sim 307.85^{\circ}\text{C}$, while the piston skirt and pin boss areas were much cooler at a minimum of $\sim 162.2^{\circ}\text{C}$. the temperature decreases steadily from the combustion surface through to the ring belt and into the piston skirt. This shows that heat is conducted from the crown to the linear and lower piston.

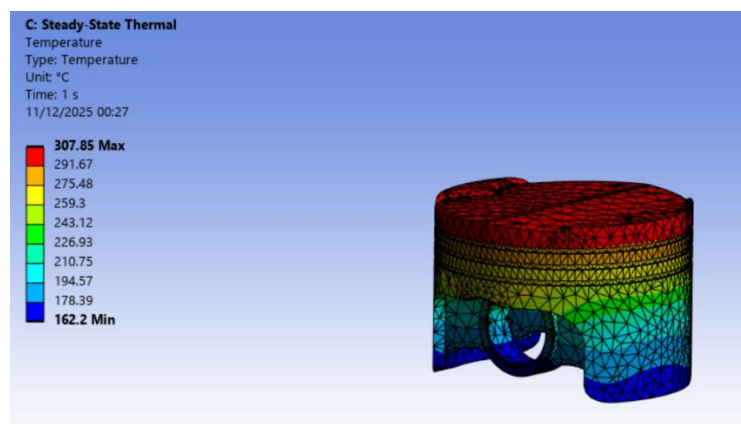


Figure 6: Shows temperature distribution from steady state analysis.

The gradient shown in Figure 6 is normal for aluminium pistons and implies that the upper areas of the piston experiences greater thermal expansion and a more noticeable reduction in yield strength compared to the piston skirt. The skirt which shows a temperature range of $\sim 160\text{-}190^{\circ}\text{C}$ is less affected by thermal softening but is prone to differential expansion similar to the hot crown. These temperature differences are important to the structural analysis due to generating important thermos elastic stresses when put together with mechanical loading conditions. The steady state temperature distribution from the thermal model was mapped

into the structural analysis. The imported distribution kept the same temperature range of $\sim 162\text{-}308^\circ\text{C}$ showing successful coupling.

4.1.2 Total deformation under load

Figure 7 shows the total deformation of the piston when it's subjected to pressure and the imported steady state temperature field. The maximum displacement seen is $\sim 3.46 \times 10^{-4}\text{m}$ which takes place at the centre of the piston crown where pressure and temperature reach their peak values. The minimum deformation of $\sim 2.55 \times 10^{-6}\text{m}$ is located in the skirt area where its cooler.

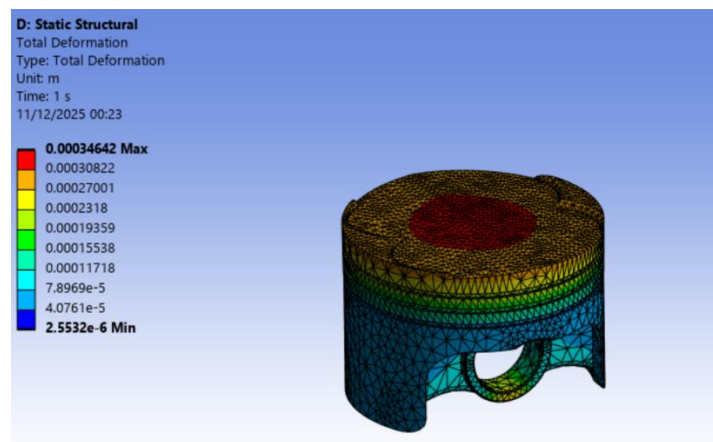


Figure 7: Shows the total deformation of the piston.

The deformation seen in Figure 7 is normal for aluminium pistons under strong thermal gradients. The piston crown bulges towards the combustion chamber while the skirt has small radial expansion. The importance of the predicted displacement is small compared with piston head and ring clearances mentioned in the literature review. The results show the need to consider crown thickness and ring belt geometry during the design stages.

4.1.3 Von Mises stress

The von Mises stress for the piston is shown in Figure 8. The model predicts a maximum von Mises stress of $\sim 2.2096 \times 10^9\text{Pa}$ and a minimum of $\sim 1.51 \times 10^6\text{Pa}$. The crown shows very low stress which shows that the pressure load is uniformly spread. The highest stresses are around the upper part of the piston near the ring belt transition and the pin boss region which shows the piston transfers combustion forces to the connecting rod.

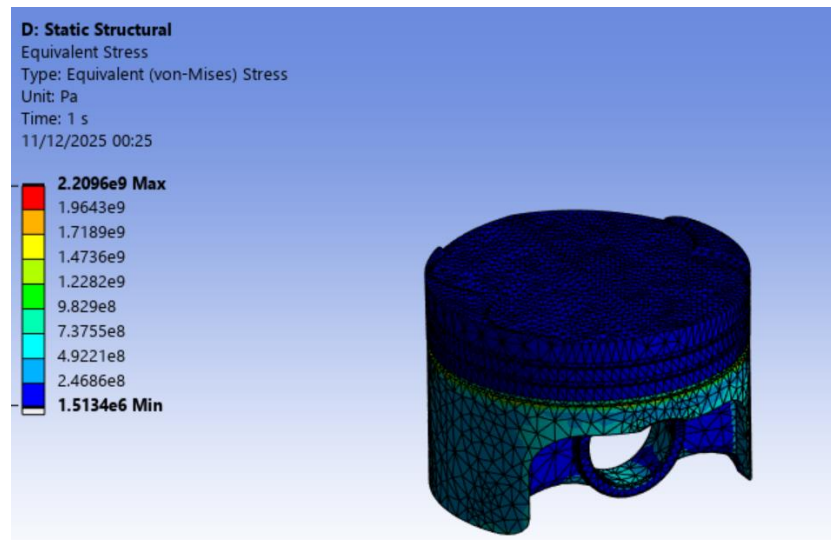


Figure 8: Show von Mises stress distribution.

4.2 New piston results

Two main changes were made to the original piston a dished crown made by cutting a recess in the crown and a reduction in the skirt by 0.5mm which both reduce the pistons mass. The same boundary conditions were used to see the influence that geometrical changes have.

4.2.1 Steady state thermal

Figure 9 below shows the temperature distribution from the steady state thermal model of the new piston. The piston crown remains the hottest area with new temperatures of 334.59 °C and a minimum of 168.06°C at the lower skirt area. Compared with the original piston the peak temperature has increased by ~9%.

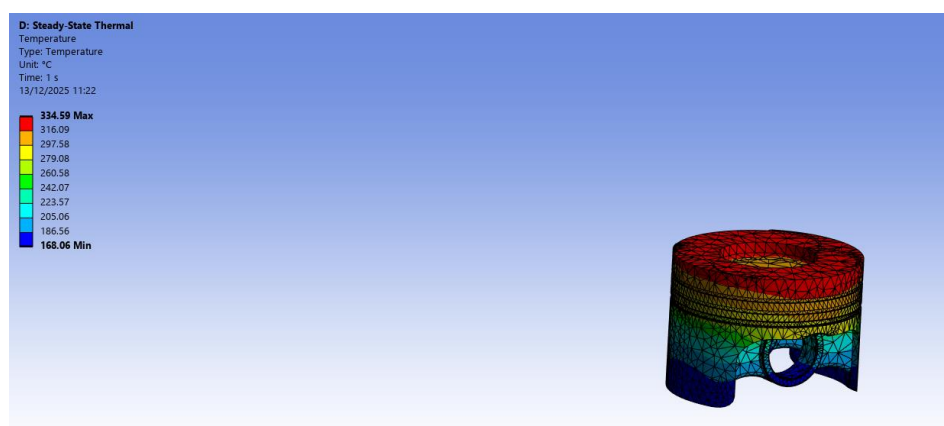


Figure 9: Shows new piston steady state thermal results.

The dished crown redistributes material and reduces the local cross section which increases thermal resistance between the hot piston crown and skirt. This causes more heat to be held in the piston crown while the skirt temperature changes slightly.

4.2.2 Total deformation

Figure 10 shows the total deformation of the new piston when it's subjected to pressure and the imported steady state temperature field. The maximum deformation is $\sim 3.588 \times 10^{-4}m$ with a minimum of $\sim 7.44 \times 10^{-6}m$. This shows an increase of $\sim 3\%$.

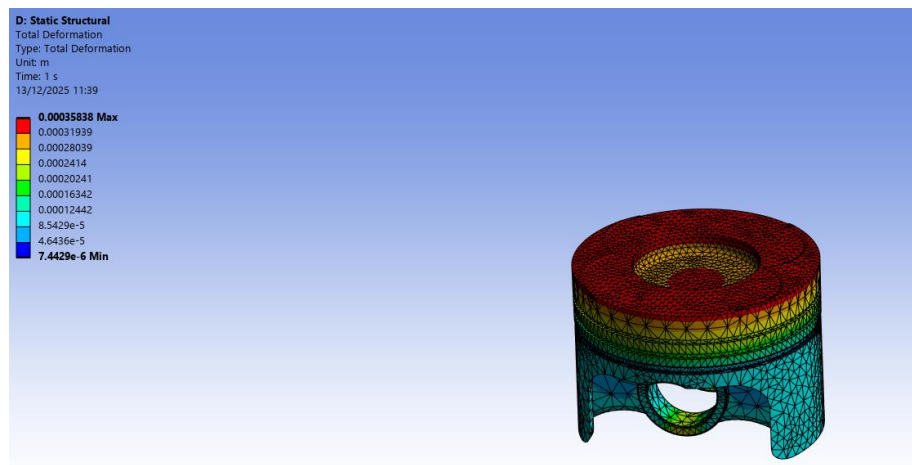


Figure 10: shows the total deformation of the new piston.

The deformation is similar to that of the original piston where the piston crown bends upwards under gas pressure and thermal expansion. The skirt and pin boss areas have smaller displacement. The larger deformation is caused by the higher crown temperature and reduction in thickness due to the dish. The overall displacement remains similar to the original piston meaning the new design does not have a big stiffness penalty.

4.2.3 Von Mises stress

The von Mises stress for the new piston is shown in Figure 11. This shows a maximum stress of $\sim 2.1026 \times 10^9 Pa$ and a minimum of $\sim 3.66 \times 10^6 Pa$. This shows a reduction in peak stress of $\sim 5\%$ compared to that of the original piston.

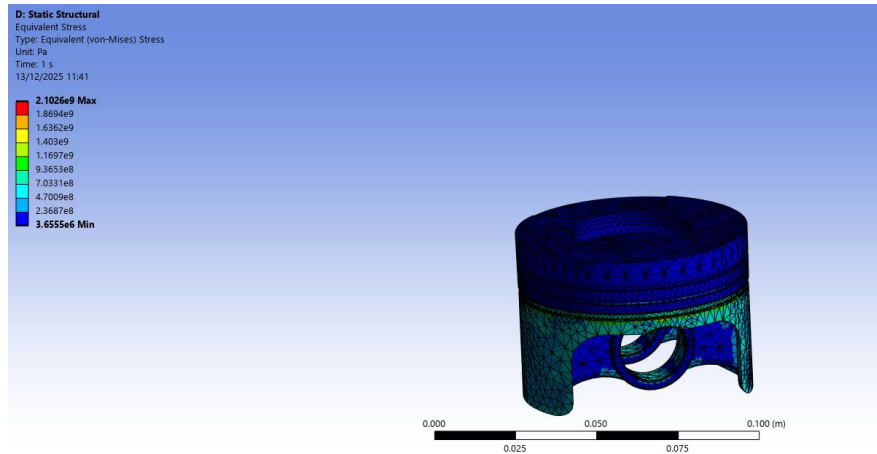


Figure 11: Shows von Mises stress for the new piston.

Similar to the original design we see the crown shows very low stress where the highest stresses are around the upper part of the piston near the ring belt transition and the pin boss region.

From the new piston design, we have seen a ~9% increase in crown temperature, a ~3% increase in maximum deformation, a ~5% reduction in peak von Mises stress and a reduction in mass due to the dished crown and reduction in skirt material. This shows the new piston trades a small increase in temperature and deformation for a reduction in stress and mass.

5 Discussion

The results from both the original piston and new piston models show how CAE tools can show structural and thermal behaviours that are not easily seen when doing hand calculations. Although good they also show the strong dependence of FEA predictions on modelling assumptions, material specifications, boundary conditions and mesh quality which should be considered when interpreting the output results.

The original piston showed a peak crown temperature of ~308°C and a maximum deformation of $\sim 3.46 \times 10^{-4}m$. The new piston design had an increased crown temperature of ~335°C and deformation of $\sim 3.588 \times 10^{-4}m$. These changes match the modifications made. The dish removes material from the piston crown which reduces the heat conduction to the piston skirt. The reduction in skirt material leads to greater expansion which matches research from piston behaviour. The thermal values depend on convection assumptions which are an approximation in CAE models. In real conditions there are other factors such as transient heat flux, combustion gas motions, oil cooling and non-

uniform temperature distributions which are not seen in this case. This shows the limitations in steady state thermal models where they can show thermal trends between different designs but can not reproduce the temperatures seen in real operating conditions.

The structural study results show noticeable numerical sensitivity. Both the original and new piston models gave von Mises stress peaks which exceed the yield strength of 6061-T6 aluminium. These values reflect stress singularities at sharp geometric transitions. Such singularities appear when linear elastic models and boundary conditions cause the stress to bend toward infinity as the element size decreases. This behaviour shows that the location of stresses is more important than the numerical magnitude.

The new piston had a small reduction in peak stress of ~5% despite having higher temperatures and deformation. This suggests that the dish redistributes load better through the crown and ring belt which improves the stress flow and reducing localised concentration zones. The removal of skirt material reduces mass without sacrificing the stiffness. Due to the model not including plasticity or temperature dependant material behaviour the results should be as comparative design assessments rather than predictions of mechanical strength.

Another consideration is the mesh dependence. Both pistons show strong gradients in the crown where element size influences localised stresses. A full mesh independence study is essential for validating future refinement. This shows that simulations offer meaningful insights, but reliability of results improves when mesh convergence, boundary condition sensitivity and analytical verification are demonstrated.

This study shows both the strengths and limitations of CAE. The tools have shown where design changes improve or worsen the pistons behaviour, but the accuracy of the results is limited by model simplifications. Engineers should use CAE outputs critically by relying on numerical trends rather than absolute values to improve design decisions.

6 Conclusion

This report applied CAE to analyse and optimise a piston subjected to both thermal and mechanical loading. The original piston model showed crown, ring belt and piston boss regions as structurally dominant features which experienced high thermal gradients and

stresses. the new piston design had a dished crown and less skirt material which led to increased temperatures and deformation but had a reduction in the peak stress and a reduction in mass.

The study also showed limitations of CAE including simplified boundary conditions, use of linear elastic material models and absence of temperature dependant behaviour and plasticity. The stress peaks seen in both pistons emphasise that FEA results should be interpreted quantitatively unless supported by mesh convergence and validation using analytical and experimental methods. The analysis proved effective for comparing and supports the use of CAE as a decision-making tool at the early stages of design.

7 References

G6017058719.

Bahman, A.S. and Iannuzzo, F. (2018) '8 - Computer-aided engineering simulations', in Suganuma, K. (ed.) *Wide Bandgap Power Semiconductor Packaging* Woodhead Publishing, pp. 199–223.

Havale, S.A. and Santosh Wankhade *DESIGN, THERMAL ANALYSIS AND OPTIMIZATION OF A PISTON USING ANSYS*.

Karem, M.H. and Ismail, A.E. (2022) *Finite Element Analysis of Piston for Structural Assessments* Penerbit UTHM.

Khyat, J. (2020) *Design Analysis and Optimization of Piston using CATIA and ANSYS*.

Mahajan, K.S. and Deshmukh, S.H. (2016) *Structural and Thermal Analysis of Piston*.

Subbarao, R. and Vart Gupta, S. (2020) 'Thermal and structural analyses of an internal combustion engine piston with suitable different super alloys', *Materials Today: Proceedings*, 22, pp. 2950–2956 Available at: <https://doi.org/10.1016/j.matpr.2020.03.429>.

Tech Scholar, M., Singh, R., Brijendra, K. and Yadav *A Review on Thermal Design Analysis of Piston Using RSM Method*.

Wakshume, E. and Sufe, G. (2025) *Finite element analysis of engine pistons: comparative assessment of aluminium and titanium alloys under combined thermal-structural loading* Springer Science and Business Media LLC.