



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

**ACHA6010**  
**Application of CAE Processes**  
**Structural Analysis**

The University of Wales Trinity Saint David

School of Engineering

Harvey Hodge

2305884

# Contents

|       |                                                                      |                                     |
|-------|----------------------------------------------------------------------|-------------------------------------|
| 1.0   | Introduction .....                                                   | 2                                   |
| 1.1   | Background & Project Rational .....                                  | 2                                   |
| 1.2   | Aims & Scope .....                                                   | 2                                   |
| 2.0   | Literature Review & Methodology .....                                | 3                                   |
| 2.1   | General Piston Characteristics .....                                 | 3                                   |
| 2.2   | Literature Review .....                                              | 3                                   |
| 2.3   | Methodology .....                                                    | 4                                   |
| 2.3.1 | Validation Overview .....                                            | 4                                   |
| 2.3.2 | Component Modelling & Load Definition Overview .....                 | 5                                   |
| 2.3.3 | Structural Assessment & Optimisation Overview .....                  | 5                                   |
| 3.0   | Model Validation .....                                               | 6                                   |
| 3.1   | Bimetallic Strip Linear + Thermal/Thermomechanical Deformation ..... | 6                                   |
| 3.2   | Bending Cantilever with Axial Force .....                            | 8                                   |
| 4.0   | Baseline Piston Analysis .....                                       | 9                                   |
| 4.1   | Kinematic Simulation .....                                           | 9                                   |
| 4.2   | Static Structural Analysis .....                                     | 11                                  |
| 4.3   | Steady State Thermal Analysis .....                                  | 12                                  |
| 4.4   | Material Comparison Study .....                                      | 12                                  |
| 5.0   | Piston Topology-guided Refinement Study .....                        | 13                                  |
| 6.0   | Piston Geometry Refinement & Optimisation .....                      | 14                                  |
|       | Discussion of CAE Tools & Limitations .....                          | 15                                  |
|       | Conclusion .....                                                     | 16                                  |
|       | References .....                                                     | <b>Error! Bookmark not defined.</b> |
|       | Appendix .....                                                       | 17                                  |

# 1.0 Introduction

## 1.1 Background & Project Rational

Engineering design increasingly depends on Finite Element Analysis (FEA) to evaluate component behaviour under realistic operating conditions and to optimise performance before physical testing. In this assignment, a multi-component engine assembly is developed and analysed using ANSYS 2024. To ensure confidence in the numerical methods applied later in the study, two validation exercises, linear + thermal and solely linear deformation tests are first conducted. These tests allow a systematic investigation into the influence of mesh size, mesh generation and free-face mesh types on simulation accuracy. Each deformation mode is benchmarked against published data with established characteristics, enabling direct comparison and verification of results. With a robust understanding of these fundamentals, the most reliable modelling approach can then be applied to the piston assembly, allowing realistic loading conditions to be assessed through detailed static structural and thermal analysis, including the evaluation of stress fields, deformation behaviour, and material response.

## 1.2 Aims & Scope

- Validate FEA methodology through a linear + thermal analysis and a solely linear analysis, assessing the influence of mesh size, mesh generation and free-face meshing types on simulation accuracy.
- Develop and prepare a multi-component engine assembly in ANSYS 2024 with structural and thermal conditions applied.
- Apply validated methodology techniques to the assembly to evaluate the total deformation, stress distribution and material response under realistic mechanical and thermal loading.
- Analyse and interpret CAE results, identify limitations and reliability to reduce piston total deformation and enhance piston performance.

## 2.0 Literature Review & Methodology

### 2.1 General Piston Characteristics

A piston is a key mechanical component used in a variety of different systems such as, pneumatic cylinders, internal combustion engines, reciprocating pumps and gas compressors. It operates with a reciprocating motion inside a cylinder which maintains a gas-tight seal to allow pressure changes to be effectively utilised. In engine applications, the piston works in conjunction with a connecting rod and crankshaft which transfers the force generated by the expanding combustion gases into rotational motion which drives the crankshaft.

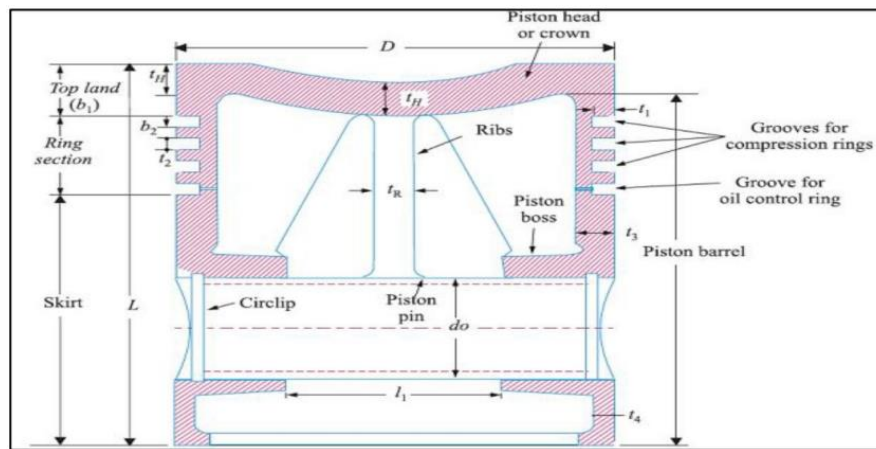


Figure 1 - (IJRASET, 2021) Piston Internals

Figure (1) shows the general design characteristics for a piston; the main considerations should be:

- As light as possible in relation to mass.
- Requires high strength to resist the high gas pressure and inertia forces.
- Material properties should be resistant to heat and mechanical deformation.
- Requires a suitable bearing surface to avoid wear.
- Should provide enough support for the Gudgeon Pin.
- To effectively channel combustion induced heat to the cylinder walls to minimise thermal loading.

### 2.2 Literature Review

In recent years, a variety of different studies have been conducted upon the strength and failure mechanisms of pistons. For example, (Lu, 2017) carried out the analysis of thermal temperature fields and thermal stress under a steady temperature field for diesel engine pistons. (S. Kashyap, 1999) Demonstrates that combining FEA with optimisation techniques significantly reduces the deformation in components similar to the piston head; The paper shows how accurate load application, mesh quality and fixed support placement influences the overall structural behaviour therefore emphasising the need for a validated FEA methodology when assessing model deformation to improve the overall model reliability.

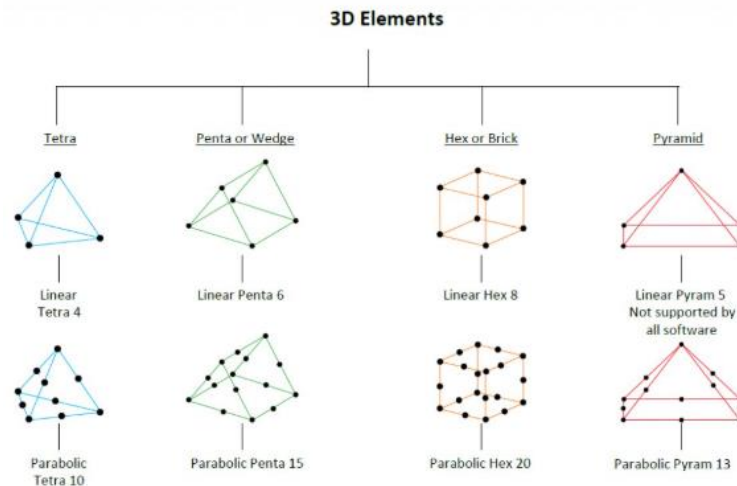


Figure 2 - (Link, 2021)

In the investigation of simulation influences, the chosen Finite Element Analysis (FEA) solution methods, particularly element types and node formulations, significantly affect the output. Figure (2) displays different element types like Tetrahedrons and Hexahedrons used in ANSYS FEA, where tetra meshing is a commonly used, fast method for generating preliminary solutions in 2D and 3D. While easy to implement on simple geometries, this method often results in a very high number of nodes and elements with limited control over mesh flow. Conversely, hexahedral elements are generally more accurate and computationally efficient, although they are significantly harder to mesh, especially in complex components. Furthermore, the use of quadratic elements, which incorporate mid-side nodes, is preferred over linear elements as they provide a better stress gradient capture, thereby enhancing the overall simulation accuracy.

## 2.3 Methodology

The methodology for the piston modelling analysis is divided into three main phases:

- Replication of Computational FEA for Validation
- Component modelling & Load Definition
- Structural Assessment & Model Optimisation
- 

### 2.3.1 Validation Overview

To ensure the reliability of the FEA results, the validation process is a critical part ensuring the data collected from the piston analysis is correct. To do this, 3 different studies have been replicated in ANSYS 2024 which are simplified test cases where the solution is derived from fundamental engineering principles. In this study (DLUBAL, 2021) provides two validation examples, one being an isotropic linear elasticity test which is temperature dependant, conducted upon a bimetallic strip using copper and invar and the other is an independent linear analysis with forces in the x and y direction applied around a fixed bending moment.

### 2.3.2 Component Modelling & Load Definition Overview

Next the creation of a realistic Computer-Aided Design (CAD) assembly is made using SolidWorks 2025, with four distinct components. Below shows the CAD assembly in which the kinematic analysis will be conducted upon to generate a representative and realistic set of loading conditions that the system experiences during operation.

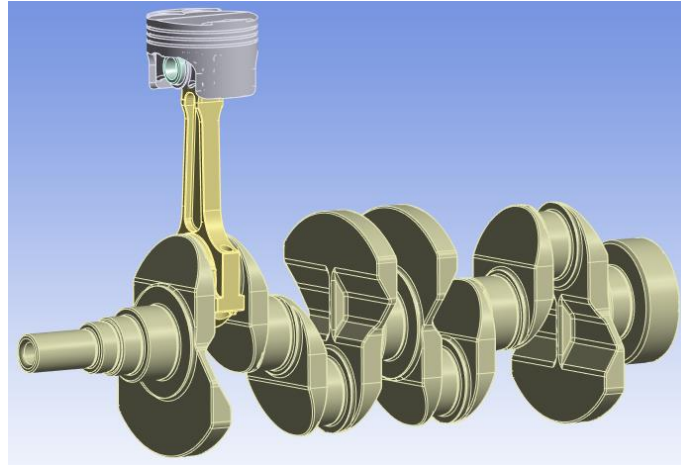


Figure 3 – Crank Shaft Assembly

Figure (3) shows the crank shaft, conrod, gudgeon pin and piston head after being assembled in SolidWorks. The main area of concern is the piston head, therefore once the loads are gained through the complete assembly all parts except the piston head are suppressed for a more focused analysis.

### 2.3.3 Structural Assessment & Optimisation Overview

This phase focuses upon the results provided by ANSYS FEA as well as the conduction of a mesh study to refine the results in relation to the validation study to ensure the data is as representative as possible to what would be expected in a realistic environment. For this phase the simulation setup requires the defining of material properties such as the strength, hardness, density and thermal conductivity; as well as setting boundary conditions to replicate mechanical movements.

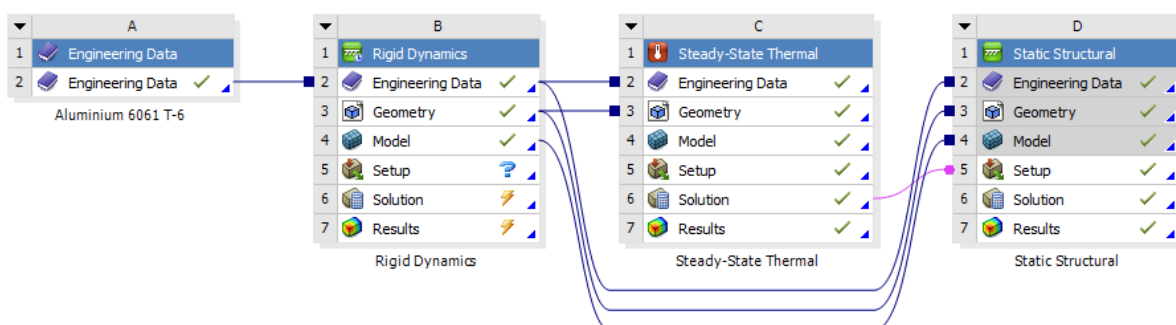


Figure 4 - Schematic of the overall simulation showing the progression from using the engineering data to gain results in the rigid dynamics, steady-state thermal and static structural where all results are combined for final analysis.

## 3.0 Model Validation

### 3.1 Bimetallic Strip Linear + Thermal/Thermomechanical Deformation

To validate the linear and thermal analysis for the piston assembly, a comparative study using a bimetallic strip from (DLUBAL, 2021) was replicated in ANSYS. This study provided essential properties (Copper and Invar) and sizing for CAD model creation. Figure (5) illustrates the expected total deformation under the combined thermal and linear loads. Prior to the ANSYS replication, hand calculations were performed using the relevant equations to determine the expected total deformation value.

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

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

Figure 5- Bimetallic Strip Hand Total Deformation Equations

The corresponding hand calculations are shown in figure (6) where the values provided in figure (6) are calculated to show the method of gaining a value of ‘-7.049mm’ for the total deformation.

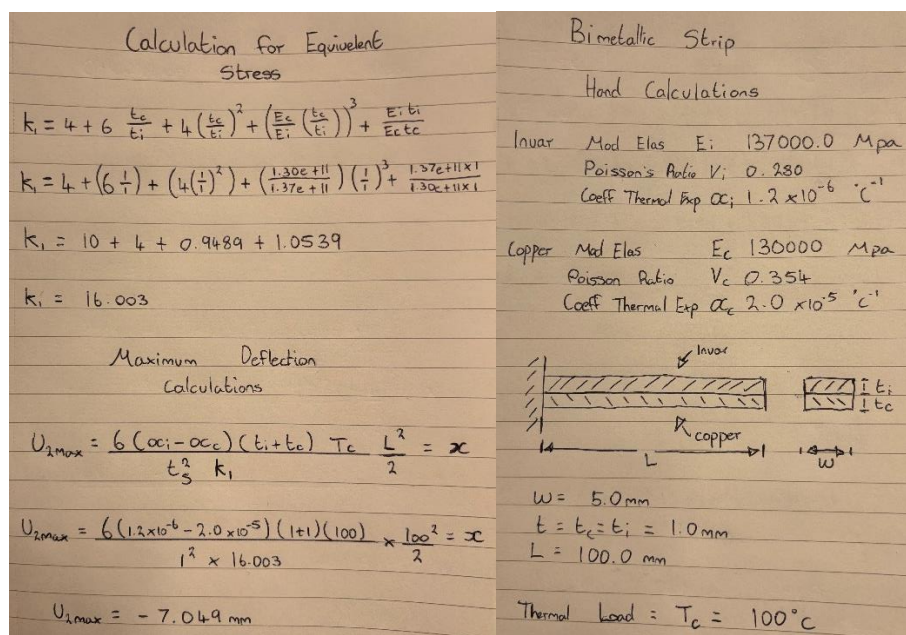


Figure 6 – Hand Calculations

The two SolidWorks beams were imported as an assembly into ANSYS Static Structural, enabling the assignment of distinct material properties for combined evaluation. After defining properties and contact, the assembly was automatically meshed. One end of the assembly was fixed to simulate a bending constraint and a thermal load of  $122^\circ\text{C}$  was applied. The core result is the expected bending response, produced because the differing thermal expansion coefficients of the two metals cause them to deform at varying rates.

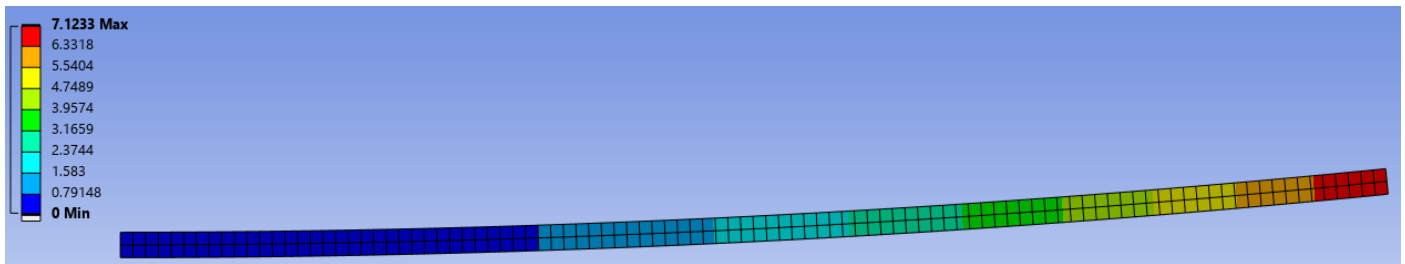


Figure 7 – Bimetallic Beam Deformation

Above shows figure (7) which is the baseline output of the simulation without any refinement in mesh size or meshing methods. Although the output of -7.1233 is close to the desired deformation, further refinement is applied to try and meet the calculated output. Figures (8+9) show a comparison between a Quadratic mesh generation and a Linear mesh generation with the corresponding total deformation when using the automatic, tetrahedron and hex dominant methods.

Figure 8- Linear Mesh Generation Comparison

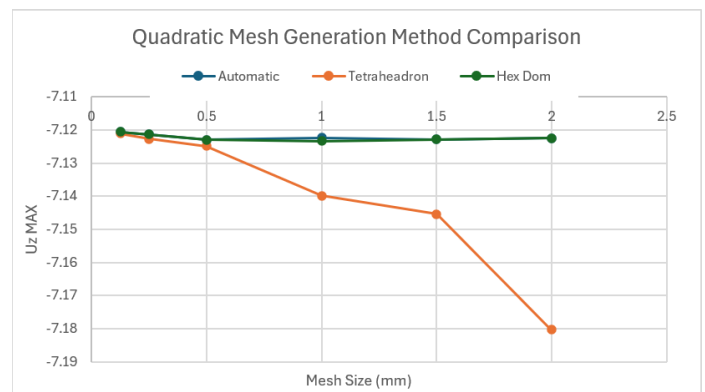
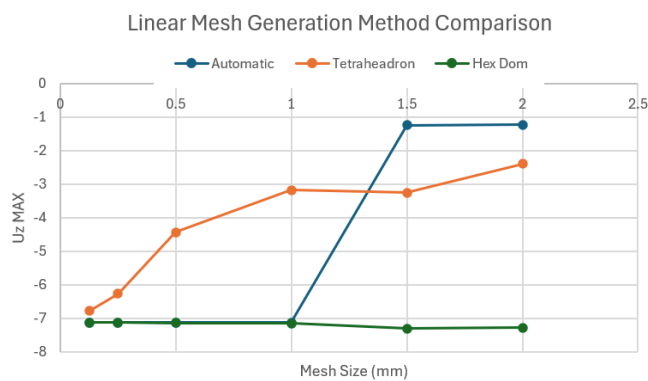


Figure 9 – Quadratic Mesh Generation Method Comparison

As demonstrated in Figure (9), the automatic and hex dominant mesh methods yielded highly similar load outputs, suggesting the automatic method may default to a quadratic element formulation. Both showed minimal change upon mesh refinement. Conversely, the tetrahedron mesh accurately converged, with refinement bringing the load output closer to the validation paper and hand calculations. The linear method showed significant size-dependent deviation. While hex dominant results tracked the quadratic method, the automatic generation method notably underpredicted the load output when a larger element size was initially used.

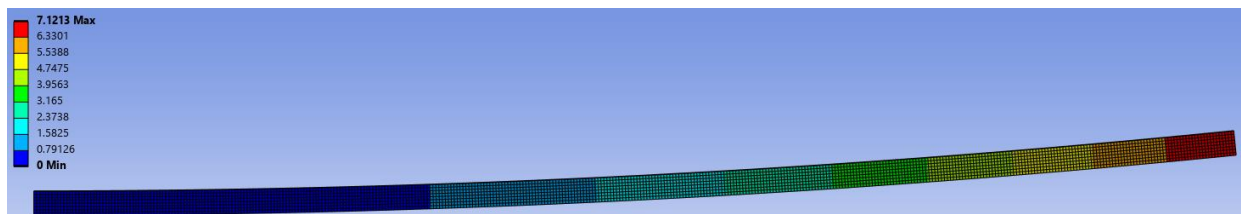


Figure 10 – Refined Mesh Total Deformation Beam Study

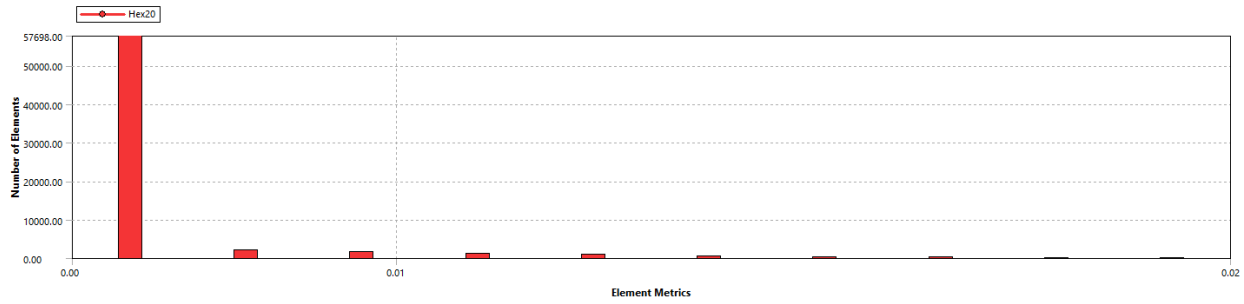


Figure 11 -Cell Skewness Bar Chart for Bimetallic Strip Refined Mesh Study

Figure (11) shows the refined simulation and the resultant mesh skewness where the calculated deformation is <1% from the validation paper. A target for the piston assembly is to reach a similar low mesh skewness to gain accurate deformation and stress figures with the same meshing strategy.

### 3.2 Bending Cantilever with Axial Force

To verify the accuracy of the linear static solver and the meshing approach, a cantilever beam subjected to simultaneous axial and transverse loading was analysed. This case was chosen because it provides a closed-form analytical solution for deflection, enabling a direct comparison between theoretical and numerical results. The study also allowed for analysis on the influence of mesh quality parameters, specifically skewness and element distortion, on accuracy. As supported by (Cook et al., 2002), the Euler-Bernoulli beam model assumes small displacements and neglects shear deformation, which is ideally suited for validating linear static FEA outputs.

|          |            |                       |       |            |     |
|----------|------------|-----------------------|-------|------------|-----|
| Material | Steel      | Modulus of Elasticity | $E$   | 210000.000 | MPa |
|          |            | Poisson's Ratio       | $\nu$ | 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  |

Figure 12 – Bending Cantilever with Axial Force CAD Geometry & Loading Parameters

Using Euler-Bernoulli beam theory the maximum transverse deflection at the free end is given by:

$$U_{z,max} = \frac{F_z L^3}{3EI_y}$$

Implementing the values from figure (12) provides a theoretical displacement of 3557.478mm.

Figure (13) shows the deformation predicted by the linear FEA model, with the maximum displacement calculated as 3554.9mm. This represents a deviation of <1% from the analytical value. This confirms that the model accurately captures the bending deformation. The displacement profile shows the smooth curvature with no numerical outliers while the stress contours follow the expected linear distribution through the cross section shown in the appendix as figure (27).

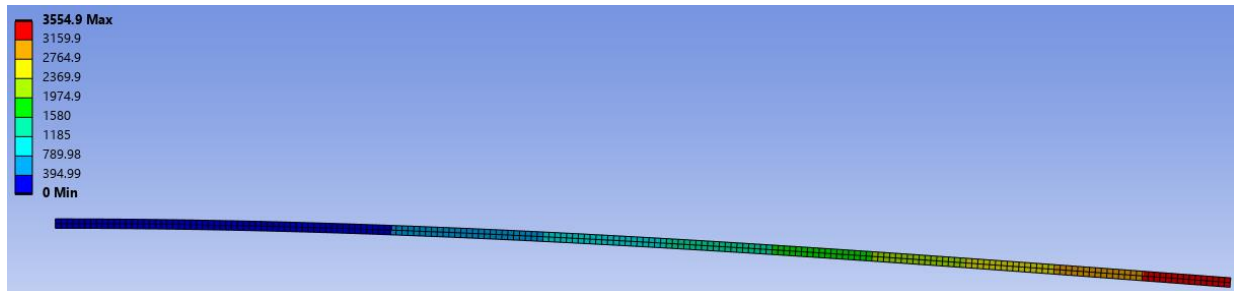


Figure 13 – Cantilever Beam Deformation

To verify mesh quality, both element skewness and Jacobian ratios were assessed. Skewness values remained well below recommended limits (<0.5), indicating minimal distortion and reliable strain accuracy, while acceptable Jacobian ratios confirmed that no elements were excessively warped. The skewness distribution shown in Figure (14) demonstrates that the mesh is suitable for structural analysis. A comparative coarse, medium, fine mesh study, included in appendix Figure (28), further justifies the chosen meshing parameters.

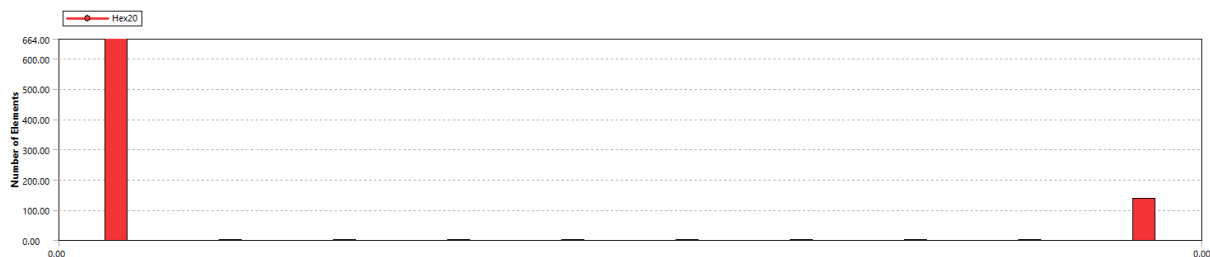


Figure 14 – Cell Skewness Distribution

## 4.0 Baseline Piston Analysis

### 4.1 Kinematic Simulation

The kinematic simulation setup begins with defining the material properties within the Engineering Data section. For this analysis, Aluminium 6061-T6 was selected from ANSYS Granta; this is because it is a widely used piston material in internal combustion engines due to its favourable balance of strength, machinability, and corrosion resistance. The relevant mechanical and thermal characteristics implemented in the model are summarised in figures (15). Figure (16) shows a temperature dependant Young's Modulus so the material can be represented correctly in the later study.

| Property                                                 | Value    | Unit                              |
|----------------------------------------------------------|----------|-----------------------------------|
| Density                                                  | 2700     | kg m <sup>-3</sup>                |
| Isotropic Instantaneous Coefficient of Thermal Expansion | 2.36E-05 | C <sup>-1</sup>                   |
| Isotropic Thermal Conductivity                           | 167      | W m <sup>-1</sup> C <sup>-1</sup> |

Figure 15 – Material Characteristics

| Temperature | Youngs Modulus (Gpa) | Poissons Ratio | Bulk Modulus (Pa) | Shear Modulus (Pa) |
|-------------|----------------------|----------------|-------------------|--------------------|
| 22          | 67                   | 0.33           | 6.58E+10          | 2.52E+10           |
| 100         | 70                   | 0.33           | 6.86E+10          | 2.63E+10           |
| 150         | 68                   | 0.33           | 6.66E+10          | 2.55E+10           |
| 200         | 64                   | 0.33           | 6.27E+10          | 2.41E+10           |
| 250         | 56                   | 0.33           | 5.49E+10          | 2.11E+10           |
| 300         | 48                   | 0.33           | 4.70E+10          | 1.81E+10           |
| 350         | 41                   | 0.33           | 4.02E+10          | 1.54E+10           |
| 400         | 32                   | 0.33           | 3.13E+10          | 1.20E+10           |
| 450         | 21                   | 0.33           | 2.05E+10          | 7.89E+09           |
| 500         | 12                   | 0.33           | 1.18E+10          | 4.51E+09           |

Figure 16 – Temperature Dependant Youngs Modulus Table

To establish realistic loading, the piston assembly was analysed in a rigid dynamics environment. This step is crucial as it accurately models the time-dependent motion of the piston, connecting rod and crankshaft, surpassing the fidelity of simplified analytical assumptions. By defining appropriate joint constraints (like revolute & general joints), the assembly's movement is restricted to physically realistic DOF.

Because the components are treated as rigid bodies, meshing is not required at this stage, and no deformation is calculated. A transient simulation was run over 0.05s, representing engine operation at 4000 rpm, with a timestep of 0.001s to ensure adequate resolution of the piston kinematics. This allowed accurate computation of inertial forces, reaction loads, and side thrust throughout the full 720° engine cycle.

To extract the piston-specific loading, the joint between the gudgeon pin and piston was included in the solution setup, enabling direct export of the reaction forces for subsequent thermal and structural analysis. Figure (17) presents the resulting force history, showing the variation in X, Y, Z, and total forces over the cycle.

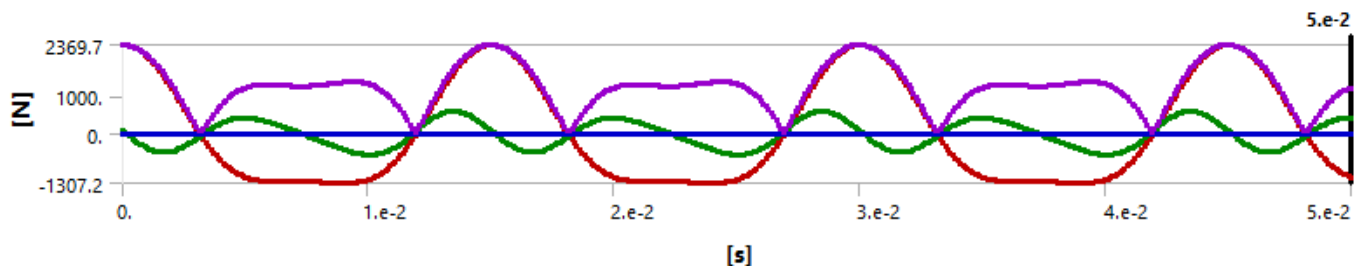


Figure 17 – Time-dependant Forces

## 4.2 Static Structural Analysis

Accurate piston dynamics modelling requires analysing numerous factors. This study applies the maximum load from preliminary rigid dynamics to the static structural and thermal analysis to simulate the resulting maximum deformation and equivalent stress. Key analysis areas are the piston crown, gudgeon pin holes, and piston ring grooves, as failure in these regions compromises performance. The simulated total deformation results are then directly compared against validation data from (Singh, 2019)

Figure 18 - (Singh, 2019) Piston

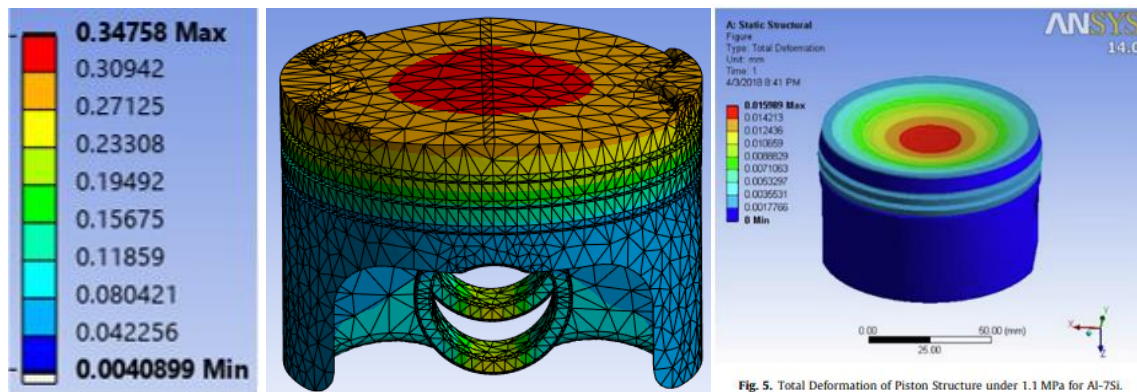


Figure 19 – Study Piston Analysis

Weak springs were applied solely for numerical stabilisation, providing improved solver convergence while ensuring that no artificial stiffness was introduced into the model or affected the accuracy of the results.

The comparison is to show the general correlation between the total deformation between the figure (19) which is from the study and figure (18) which is from (Singh, 2019). Both simulations have different materials and variables applied yet the general trends are consistent throughout.

Utilising the Hex Dominant meshing strategy, a mesh sensitivity study was conducted by systematically reducing mesh sizes from 0.5mm to 5.0mm in 0.5mm increments. The resulting change in total deformation was plotted against the mesh size to quantify the effect of discretization refinement on the simulation results. This convergence data for the materials is detailed in appendix Figure (29). Below shows the final meshing images of the piston head with a refined mesh on the crown and gudgeon pin holes for increased accuracy when displaying total deformation and other outputs.

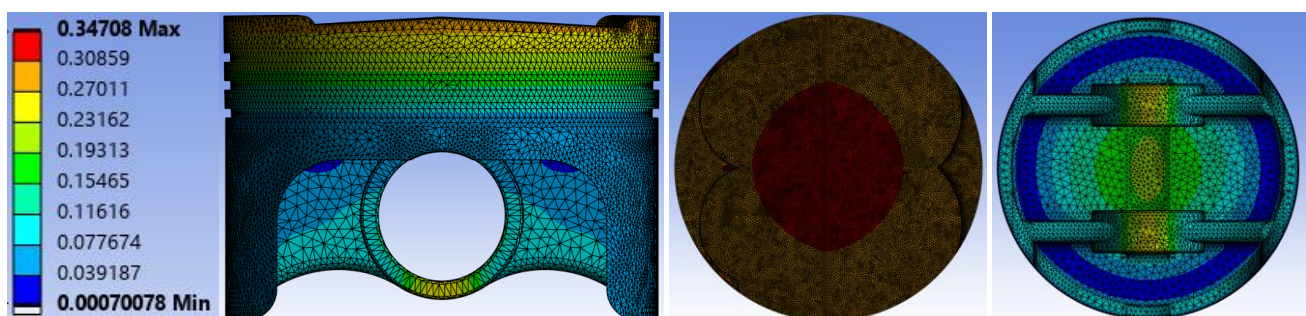


Figure 20 – Different Plane View Piston Baseline Total Deformation Images

With a refined mesh the cell skewness is taken into consideration this is because highly distorted elements compromise the numerical solutions accuracy and stability. It measures deviation from the ideal element shapes. High skewness introduces significant interpolation errors, especially in high stress areas which will result in a slower convergence and increased computational time.

### 4.3 Steady State Thermal Analysis

Integrated into the simulation is a steady state thermal analysis to gain an estimate upon the temperature distribution of the piston during operation; this allows for the static and rigid analysis to incorporate the data from this section to provide enhanced accuracy on the final results.

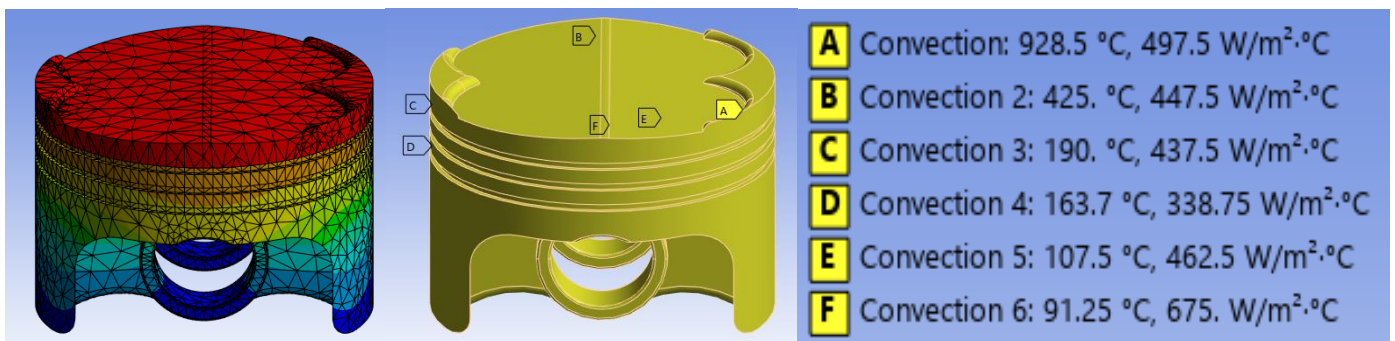


Figure 21 – Temperature Distribution on Baseline Piston

Different heat transfer convections were applied to create a realistic representation of the changes in temperature the piston experiences during operation. Consistent convection coefficients were used throughout each study and geometry enhancement to ensure a direct comparison between simulations are available. Furthermore, the baseline mesh values were applied to gain suitable results for comparison.

### 4.4 Material Comparison Study

To investigate the influence of material selection upon piston performance, a comparison between 3 different materials, Aluminium 6061 T-6 (baseline material), Cast Iron ASTM A48 and Titanium Alloy Ti-6Al-4V were simulated under the same load conditions, boundary conditions, mesh settings and thermal coefficients to compare the structural response, specifically in relation to deformation and stress. The selection of these materials are due to aluminium and titanium both having a high strength to weight ratio while cast iron has a high stiffness and wear resistance.

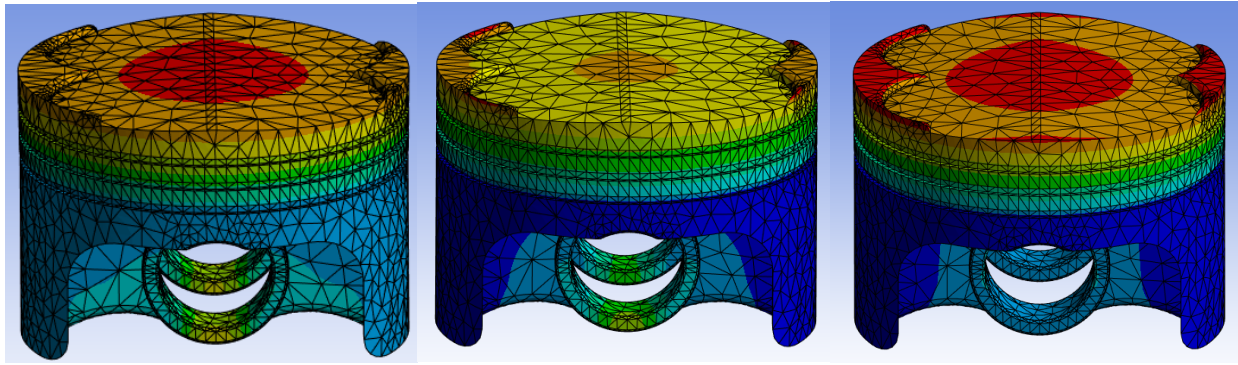


Figure 22 – Different Material Total Deformation Images

As expected, based on a lower Young's Modulus, Aluminium 6061-T6 exhibited the largest structural response showing a deformation of 0.347mm as well as the highest Von Mises stress (2207.61MPa). This data indicates that although the aluminium is lightweight, it experiences the most displacement and highest stress intensity under the maximum applied load.

Cast Iron (ASTM A48) provided the best structural resistance, achieving the lowest deformation and Von Mises stress due to its high stiffness. This material would be the ideal choice if component mass was not a primary design concern.

Overall, the Titanium Alloy proved to be the best material due to its optimal strength to weight ratio. Providing a balance between cast iron and aluminium where it is much lighter and able to retain the high strength and durability capabilities against variable operating temperatures. This balance would reduce inertia forces, enable higher RPMs resulting in a direct boost in overall efficiency.

|                          | Total Deformation | Stress (Von-mises) |
|--------------------------|-------------------|--------------------|
| Cast Iron ASTM A48       | 0.186             | 890.54             |
| Aluminium 6061 T-6       | 0.347             | 2207.61            |
| Titanium Alloy Ti-6Al-4V | 0.217             | 1474.21            |

Figure 23 – Material Results Comparison

## 5.0 Piston Topology-guided Refinement Study

Moving forward, Titanium was selected to implement into the topology guided piston enhancement; this is due to its key qualities of stiffness, weight and strength in relation to the other materials. Using ANSYS a topology optimisation study was conducted to assess the key areas which the software identifies as having low structural significance resulting in areas where weight can be removed with compromising the overall piston characteristics.

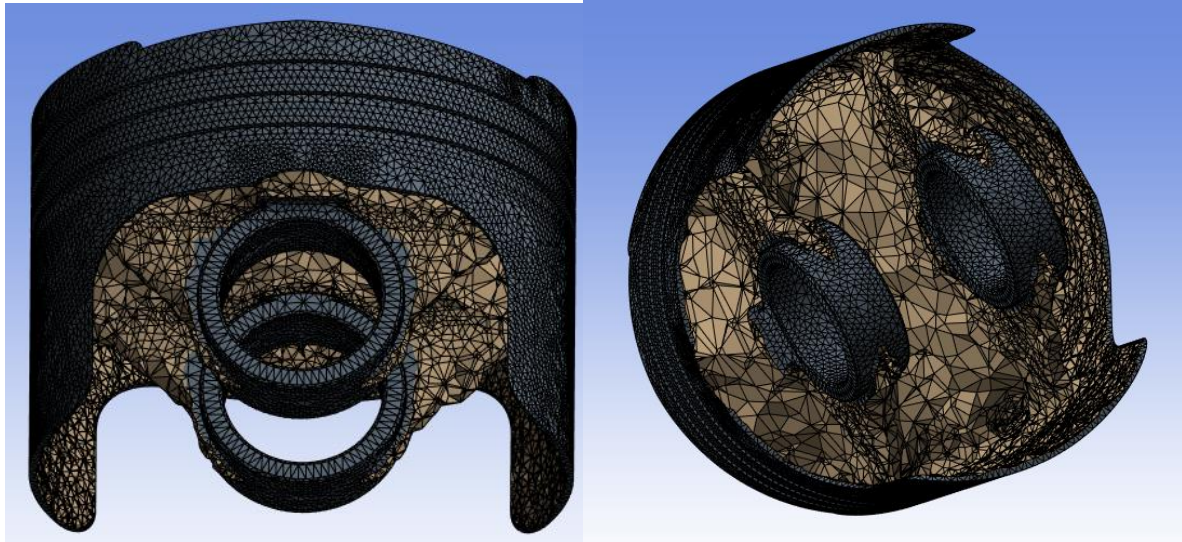


Figure 24 – Piston Baseline Topology 40% Refinement

Figures (24) show the refined piston geometry after boundary constraints were applied to preserve critical features like the outer surfaces and gudgeon pin holes. A refined mesh was necessary to prevent significant geometric skewing and ensure accurate mass reduction calculations.

A 40% mass reduction factor was targeted to represent a realistic optimized piston. The resulting software guidance confirmed that significant material could be removed from the underside of the piston, specifically away from the main load path.

## 6.0 Piston Geometry Refinement & Optimisation

The next part of the study looked into optimising the piston geometry less aggressively by removing and adding material in specific areas of interest to reduce the overall total deformation and stress regions. The CAD geometry was then inputted back into the simulation using Titanium Ti-6Al-4V with the same parameters to ensure a direct comparison can be made between models.

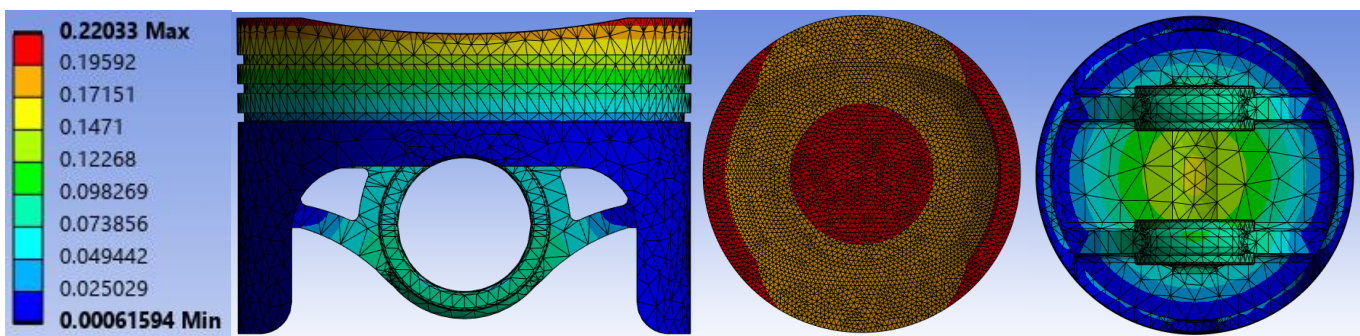


Figure 25 – Optimised Titanium Piston Total Deformation

Analysing figure (25), the enhanced piston compared to the baseline piston shows a reduction in maximum deformation as well as overall stress showing the effects of changing the geometry from the simulation output. A key focus for the piston optimisation was reducing the deformation around gudgeon pin holes where the main force is induced upon the piston, to mitigate this, more material was added on the underside to increase the thickness of the material to support the load transfer.

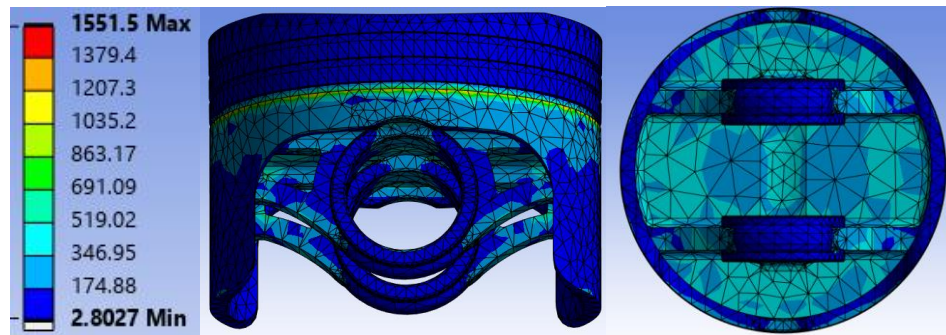


Figure 26 – Von Mises Stress Distribution Optimised Piston

Further analysis of the Von Mises stress distribution shows the refined piston geometry achieves a more uniform stress profile compared to the baseline design.

In the original piston, the peak stress was concentrated around the gudgeon pin bosses, reaching approximately 1550 MPa. After refinement, this maximum stress was reduced to approximately 1400 MPa with fewer highly stressed elements around the pin boss features.

Crucially, the refined geometry also exhibits a smoother stress gradient across the underside of the crown. This indicates improved load transfer and successfully reduced localised stress concentrations under the same loading conditions.

## 7.0 Discussion of CAE Tools & Limitations

Overall, the CAE workflow which is implemented in this study provides a useful insight into assessing a piston's characteristic within an engine assembly for analysis upon structural behaviour for geometry design optimisation to enhance overall engine dynamics; however, the simulation can be refined in areas for a more accurate representation of the piston dynamics specifically in areas where assumptions and solver limitations arise throughout the simulation.

The provided CAE results are indicative, not fully predictive, due to multiple simplifications. The mechanical analysis used only peak forces from rigid dynamics, neglecting elastic deformation and dynamic pressure cycling. The thermal analysis relied on a steady-state approximation, which failed to capture transient thermal cycling. Crucially, the structural solver assumed linear elastic material behaviour and small deformations, making it insufficient for predicting critical failure scenarios like high-cycle fatigue or plasticity, an issue compounded by using simplified material data. Consequently, the output values for stress, strain, and deformation represent a comparative guide rather than a fully predictive representation of the engine assembly.

## 8.0 Conclusion

This study has demonstrated the effective use of Finite Element Analysis (FEA) to evaluate and optimise the structural and thermal performance of a piston within a multi-component engine assembly. The validation studies confirmed the suitability of the meshing strategy applied to the piston head as well as the applied solver settings and modelling techniques, therefore providing confidence in the resultant residuals from the overall analysis.

A rigid dynamics kinematic simulation established realistic loading for the static and thermal analyses. The baseline piston showed critical stress and deformation concentrations around the gudgeon pin bosses and the crown-to-skirt region. A comparative material study determined that Titanium (Ti-6Al-4V) offered the best stiffness to weight ratio for performance enhancement, while Cast Iron provided high stiffness but increased mass, and Aluminium 6061 T-6 resulted in the highest deformation. Finally, geometry refinement, guided by topology optimization, successfully achieved a more uniform stress distribution and reduced the peak Von Mises stress from approximately 1550MPa to 1400MPa under identical conditions.

Moving forward, further work should focus on improving model realism and predictive accuracy. This involves refining the topology-driven geometry through more representative boundary conditions (e.g., gudgeon pin constraints) and conducting a formal mesh convergence study in critical regions to improve stress accuracy. Crucially, capturing cyclic engine loading effects requires incorporating non-linear material behaviour and a fully coupled transient thermo-mechanical analysis. Additionally, extending the material study to include advanced aluminium alloys and composite materials is necessary for a more comprehensive assessment of real-world piston design trade-offs.

## 9.0 Bibliography

Cook et al. (2002). *Concepts and Applications of Finite Element Analysis*. 4th ed. USA: Malkus, DS.

DLUBAL. (2021, 09 05). *Verification Examples*. Retrieved from [https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/fmaterial-3?f=\\_](https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/fmaterial-3?f=_)

Genisten, V. (2024, 01 2015). *Cantilever with Moment Loading at Free End – Structural Analysis*.

Retrieved from Cantilever with Moment Loading at Free End – Structural Analysis:

<https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/008852>

Gensichen, G. (2015, 01 15). *Cantilever with Moment Loading at Free End – Structural Analysis*.

Retrieved from <https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/008852>

IJRASET. (2021). *Design, Structural and Thermal Analysis of Piston*. India: IJRASET.

Link, S. (2021, 10 19). *3D Tetra Meshing*. Retrieved from 3D Tetra Meshing: <https://skill-lync.com/student-projects/week-8-3d-tetra-meshing-challenge-129>

Lu, Y. (2017). *Analysis of thermal temperature fields and thermal stress under steady*. China: Xing Zhang.

Lumpe, G. (2024, 01 15). *Cantilever with Moment Loading at Free End – Structural Analysis*. Retrieved from <https://www.dlupal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/008852>

S. Kashyap. (1999). *Finite Element Analysis and Optimisation in Fixture Design*. NY USA: W.R DeVies.

Singh, U. (2019). *Comparative analysis of different materials for piston and justification*. Inid: Jatin Lingwal.

## 10.0 Table of Figures

|                                                                                                                                                                                                                                                  |    |
|--------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------------|----|
| Figure 1 - (IJRASET, 2021) Piston Internals.....                                                                                                                                                                                                 | 3  |
| Figure 2 - (Link, 2021).....                                                                                                                                                                                                                     | 4  |
| Figure 3 – Crank Shaft Assembly .....                                                                                                                                                                                                            | 5  |
| Figure 4 - Schematic of the overall simulation showing the progression from using the engineering data to gain results in the rigid dynamics, steady-state thermal and static structural where all results are combined for final analysis. .... | 5  |
| Figure 5- Bimetallic Strip Hand Total Deformation Equations .....                                                                                                                                                                                | 6  |
| Figure 6 – Hand Calculations .....                                                                                                                                                                                                               | 6  |
| Figure 7 – Bimetallic Beam Deformation .....                                                                                                                                                                                                     | 7  |
| Figure 8- Linear Mesh Generation Comparison .....                                                                                                                                                                                                | 7  |
| Figure 9 – Quadratic Mesh Generation Method Comparison .....                                                                                                                                                                                     | 7  |
| Figure 10 – Refined Mesh Total Deformation Beam Study.....                                                                                                                                                                                       | 7  |
| Figure 11 -Cell Skewness Bar Chart for Bimetallic Strip Refined Mesh Study .....                                                                                                                                                                 | 8  |
| Figure 12 – Bending Cantilever with Axial Force CAD Geometry & Loading Parameters.....                                                                                                                                                           | 8  |
| Figure 13 – Cantilever Beam Deformation.....                                                                                                                                                                                                     | 9  |
| Figure 14 – Cell Skewness Distribution .....                                                                                                                                                                                                     | 9  |
| Figure 15 – Material Characteristics .....                                                                                                                                                                                                       | 10 |
| Figure 16 – Temperature Dependant Youngs Modulus Table .....                                                                                                                                                                                     | 10 |
| Figure 17 – Time-dependant Forces .....                                                                                                                                                                                                          | 10 |
| Figure 18 - (Singh, 2019) Piston .....                                                                                                                                                                                                           | 11 |
| Figure 19 – Study Piston Analysis .....                                                                                                                                                                                                          | 11 |
| Figure 20 – Different Plane View Piston Baseline Total Deformation Images.....                                                                                                                                                                   | 11 |
| Figure 21 – Temperature Distribution on Baseline Piston.....                                                                                                                                                                                     | 12 |
| Figure 22 – Different Material Total Deformation Images .....                                                                                                                                                                                    | 13 |
| Figure 23 – Material Results Comparison.....                                                                                                                                                                                                     | 13 |
| Figure 24 – Piston Baseline Topology 40% Refinement.....                                                                                                                                                                                         | 14 |
| Figure 25 – Optimised Titanium Piston Total Deformation .....                                                                                                                                                                                    | 14 |
| Figure 26 – Von Mises Stress Distribution Optimised Piston .....                                                                                                                                                                                 | 15 |
| Figure 27.....                                                                                                                                                                                                                                   | 18 |
|                                                                                                                                                                                                                                                  | 17 |

11.0 Appendix

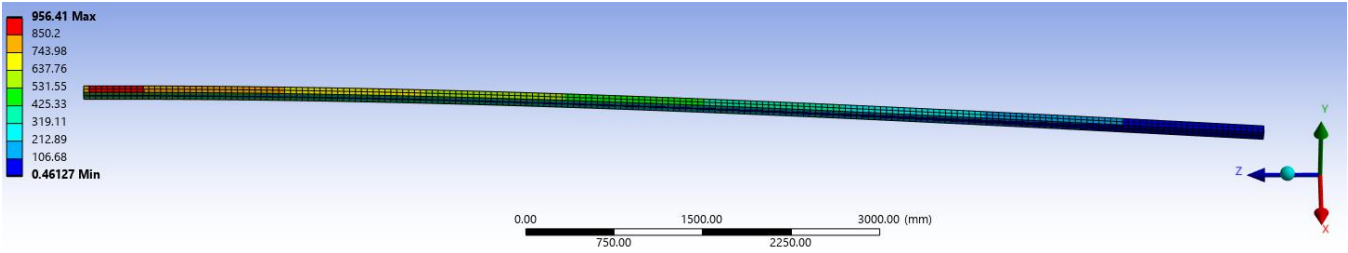


Figure 27

|        | Mesh Size (mm) | Total Deformation Stress (von-mises) |        |
|--------|----------------|--------------------------------------|--------|
| Fine   | 3.125          | n/a                                  | n/a    |
|        | 6.25           | 3555.7                               | 1354.5 |
| Medium | 12.5           | 3555.7                               | 1085.3 |
|        | 25             | 3555.5                               | 921.13 |
| Coarse | 50             | 3554.9                               | 956.41 |

Figure 28

| Mesh Size | Hex Dom    |           |
|-----------|------------|-----------|
|           | Total Defo | Eq Stress |
| 5         | 0.34757    | 2239      |
| 4.5       | 0.34758    | 2176.4    |
| 4         | 0.34758    | 2209      |
| 3.5       | 0.34754    | 2215.1    |
| 3         | 0.34788    | 2039.8    |
| 2.5       | 0.34751    | 2305.7    |
| 2         | 0.34753    | 2122.8    |
| 1.5       | 0.34732    | 2146.6    |
| 1         | 0.34707    | 2207.6    |

Figure 29