

University of Wales Trinity Saint David

Wales Institute of Science & Art



Prifysgol Cymru

Y Drindod Dewi Sant

University of Wales

Trinity Saint David

Application of CAE Processes

Structural & Fluid Analysis (ACHA6010)

Tim James

2112656

December 2024

Contents

1	Introduction	1
2	CAE Tools Discussion	2
2.1	Design Process	2
2.2	Limitations	2
3	Modelling and Assembly	3
4	Kinematic, Structural and Thermal	5
4.1	Kinematic Analysis	5
4.2	Static Structural Study	6
4.3	Thermal Considerations.....	7
5	Optimization and Improvement.....	10
	Improvement	10
	Optimization	10
6	Validation	11
6.1	Simplified Test Case.....	11
6.2	Analytical and Software Comparison	12
6.3	Mesh Independence Study.....	12
6.4	Boundary Condition Influence.....	13
7	Discussion.....	15
8	Conclusion	16
9	References.....	17

1 Introduction

This project focuses on simulating and analysing results of a crankshaft-piston assembly in ANSYS Workbench where a realistic assembly of individual components will be used to perform Kinematic, Thermal and Structural studies to optimize the performance of the piston. This assembly was chosen due to the critical nature in which this subsystem is used in an Internal Combustion Engine (ICE) to convert Rotational Motion from the Crankshaft into Reciprocating Motion of the Piston to complete an engine cycle(Ravi Kumar *et al.*, 2023). This subsystem is widely used within the automotive sector where materials and designs vary from platform to platform where components must endure a wide range of parameters from mechanical, thermal and dynamic loads which makes it a suitable choice for this comprehensive study.

This assignment aims to replicate the operating conditions of the chosen assembly, identify potential limitations of the model and to refine the design of the piston to improve overall performance. Additional validation techniques such as Analytical comparisons and mesh independence studies were conducted to verify the results achieved.

The assembly was supplied by Andrew Noble as well as accompanying Material properties for Aluminium 6061-T6 and Boundary conditions which were then used to perform an initial simulation to validate with existing results.

2 CAE Tools Discussion

2.1 Design Process

ANSYS is a Computer-Aided Engineering (CAE) package which is widely used in engineering design and analysis. It can conduct various simulation types from Structural, Thermal, Fluid and Crash Analysis and was utilized in this assignment to perform Kinematic, Structural and Thermal analyses of the crankshaft-piston assembly. The analysis of these components is critical for understanding the performance, stresses and failure points to ensure they can be used in real-world applications. ANSYS can be used to replicate conditions with high precision by applying boundary conditions, loads and constraints based on kinematic and thermal properties, this allows the modelling of accurate mechanical interactions of components and potential stress profiles that the subsystem would endure in realistic operations.

Material properties can also be incorporated into simulations and assigned to individual components which is crucial for accurate assessment of component performance. ANSYS allows iterative improvements to be made to assembly components by analysing stresses and deformation results which can direct modifications to geometry, design and material choices which can improve structural performance, reduce weight or reduce fatigue.

Analytical solutions can also be derived from fundamental principles and compared with results from a simulation to validate the accuracy of the model and to increase reliability of results.

2.2 Limitations

Limitations of ANSYS include the input accuracy which are heavily dependent on inputting precise data and boundary conditions to ensure the model produces accurate and realistic results. Complex assemblies may require fine mesh sizes which is computationally and timescale demanding as each node must be calculated individually and connected to produce an overall result. ANSYS may also require simplification of boundary conditions as they may not be practical or possible to fully capture real world complexities within systems which can reduce the accuracy of results.

3 Modelling and Assembly

Figure 1 below gives us a view of the Assembly of components that will be used to complete the Kinematic, Structural and Thermal Studies. The assembly consists of a Crankshaft, Connecting Rod, Gudgeon Pin and Piston. Each component is interconnected to make the subsystem where the choice of material is critical as it can negatively affect the durability, efficiency and performance of the system.

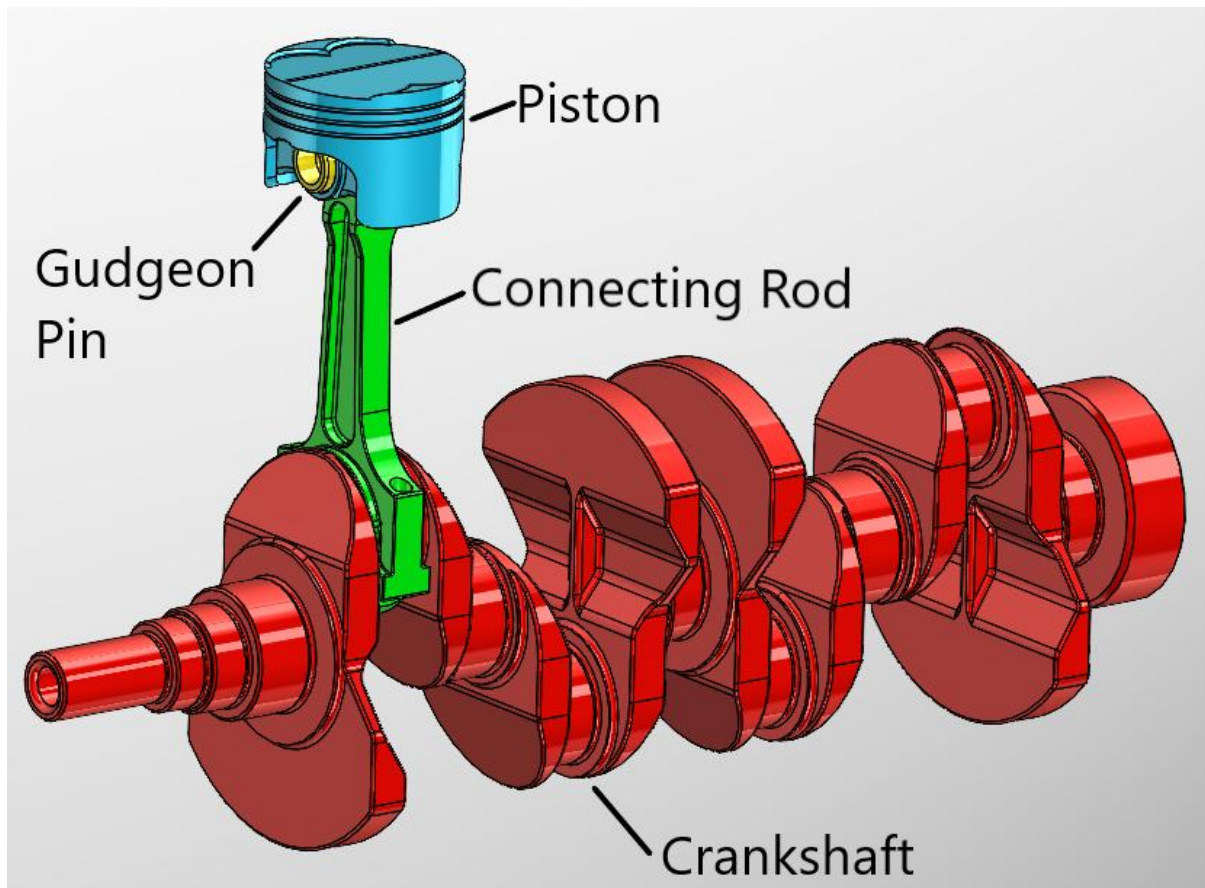


Figure 1 - Assembly of Simulation Components

The Piston is exposed to high temperatures and pressures within the cylinder during combustion making thermal conductivity and strength essential properties when selecting a material (Mahle, 2016). Aluminium alloys are commonly used for their lightweight and heat dissipation properties whereas cast iron or forged steel will be used for high performance or diesel applications due to superior strength.

The Gudgeon pin connects the piston to the connecting rod and will need to withstand high shear stresses and wear resistance (Bianco *et al.*, 2022) which makes steel alloys a common material choice.

The Connecting rod transfers the forces between the piston and the crankshaft which will be subject to high levels of fatigue (Witek and Zelek, 2019) which increases with engine load as the component will be stretched and compressed during the engine cycles. Forged steels are a general material used for connecting rods however Aluminium alloys are also an increasingly common choice (Godara *et al.*, 2022).

The Crankshaft is typically made from forged steel or cast iron due to their excellent strength and fatigue resistance properties. As the Crankshaft is subjected to various angular torque values it will be subjected to 'twisting' which can result in fatigue or a complete failure of the component which makes the material choice vital for its application requirements(Ravi Kumar *et al.*, 2023).

4 Kinematic, Structural and Thermal

Kinematic, Structural and Thermal Simulations were then performed on the Assembly specifically looking at the Structural and Thermal properties that the Piston is subject to, all boundary conditions used for the initial simulation were then used for a grey cast iron piston to keep consistency throughout the simulations to compare materials specifically.

4.1 Kinematic Analysis

An initial kinematic study was completed on a Crankshaft assembly that included the Crankshaft, Piston, Connecting Rod and Gudgeon Pin models to generate a realistic set of loading conditions that the assembly would be subjected to. Aluminium 6061-T6 was selected as the Piston material choice as data was provided to attribute a temperature range with Young's modulus as well as heat transfer coefficients and Temperatures at various points across the model. Structural steel was then used for the other components as the studies were not dependant on their characteristics or results foremost.

The first step in performing the Kinematic Analysis was to load the assembly and associated Engineering data to supply the models with the correct Material assignments as well as ensuring the models were a rigid stiffness behaviour. Connections in the form of Joints were then created to revolute the various components in relation to each other to simulate an engine cycle as shown in figure 2 below where the Gudgeon Pin is joined to the Piston collar. The piston was also set to be free in the X axis so that as the Crankshaft completed a rotation, the piston could only move in one domain.

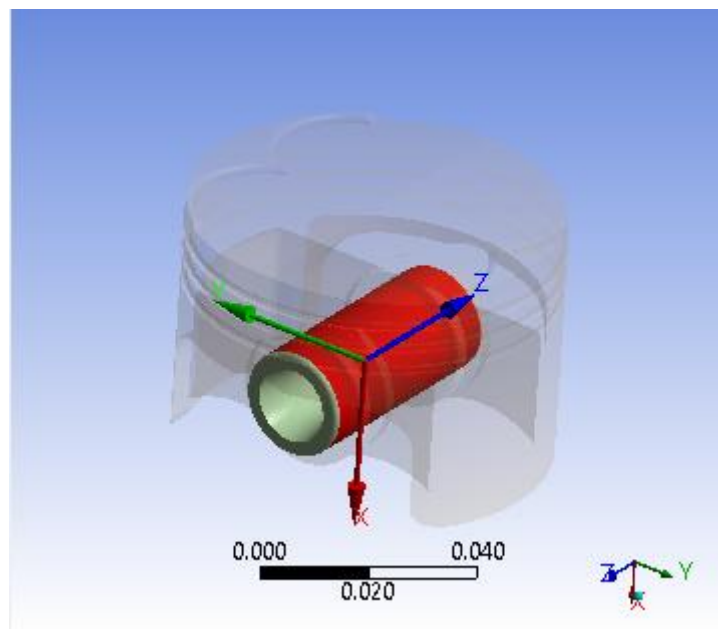


Figure 2 - Revolute Connection

The maximum value for force (Newtons) across the XYZ origins were then exported from a joint probe to be implemented into the Thermal and Structural analysis simulations. These measurements represent the actual loads experienced by the system due to motion, inertia and their interaction with other components. The static structural analysis will evaluate how this system withstands these applied forces to give accurate results in stress and deformation. Higher forces can increase the

contact pressures between the components leading to head generation due to friction, this can influence the forces that impact thermal behaviour.

4.2 Static Structural Study

The static structural analysis will evaluate how this system withstands the applied forces produced from the kinematic analysis to give accurate results in stress and deformation.

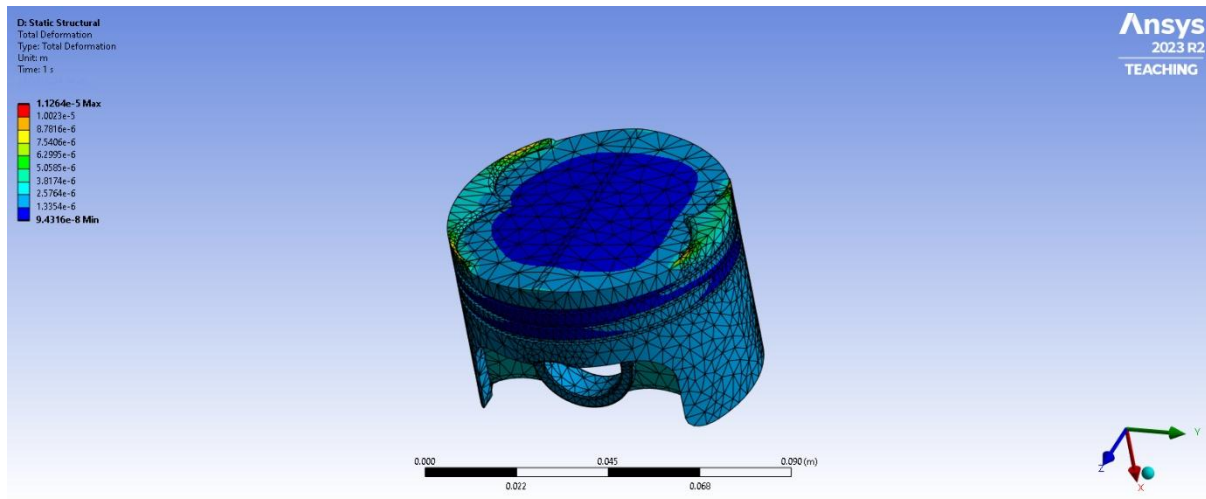


Figure 3 - 6061-T6 Total Deformation

Figure 3 above shows the total deformation of the piston under the applied pressure on the piston crown of 10.5MPa. The maximum deformation appears around the upper edges where the valve cutout profile exists, this typically happens due to this area experiencing a high stress concentration under the applied load such as when the combustion pressure is acting on the piston crown. Typically, high deformation around the ring grooves and piston crown might lead to excessive stress and failure of the component.

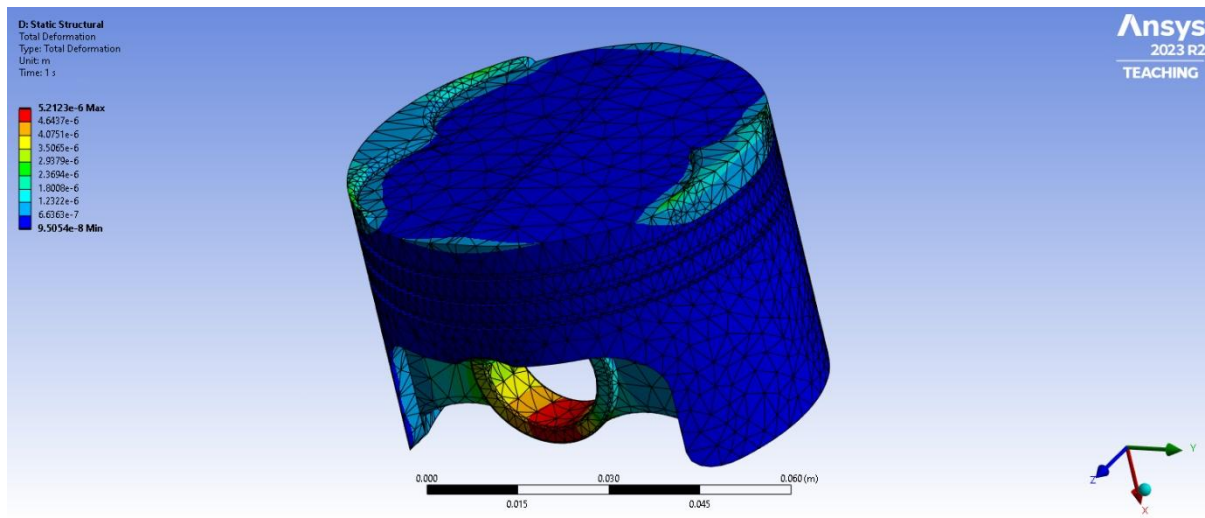


Figure 4 - Grey Cast Iron Deformation

The simulation where grey cast iron was used as the material with the same boundary conditions shows a higher overall deformation around the Gudgeon pin insert compared to that of the 6061 T6 Aluminium. The cast iron piston deforms less due to its generally higher elastic modulus which makes it stiffer. The cast iron has an overall less deflection however due to the lower tensile strength it makes the stress distribution more critical as seen around the pin insert which sees the largest stress concentration, this could result in a failure releasing the connection between the connection rod and piston.

4.3 Thermal Considerations

Thermal considerations such as applying accurate loads, realistic boundary conditions such as the thermal profile of the component, defining the film coefficients and fluid temperature with a correctly applied ambient temperature can help validate the results of the simulation. Material properties such as the thermal conductivity, specific heat capacity and density also play a crucial role in selecting the correct material for this application.

From the kinematic studies, higher forces can increase the contact pressures between the components leading to head generation due to friction, this can influence the forces that impact thermal behaviour.

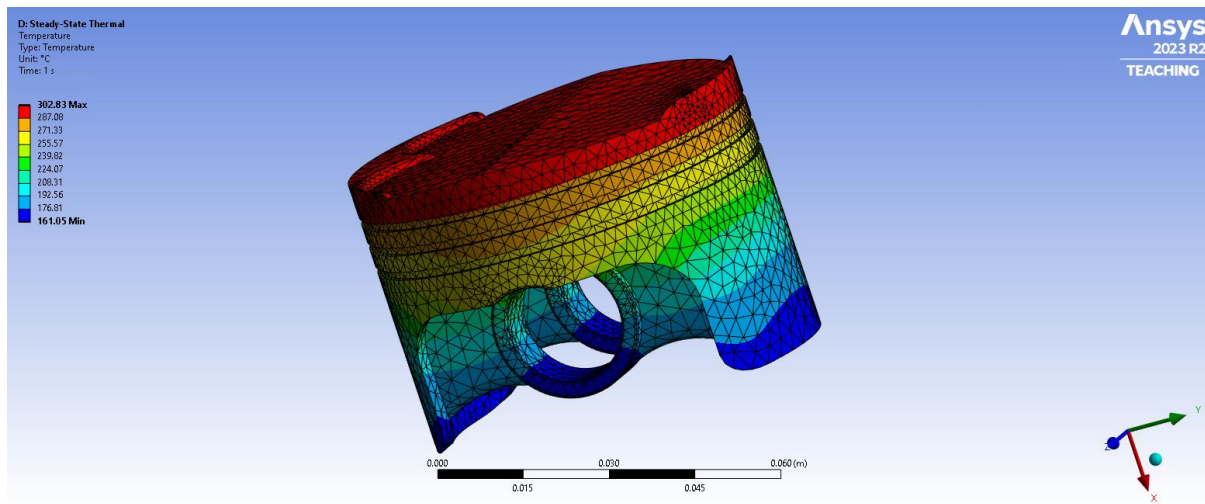


Figure 5 - 6061 T6 - Temperature Gradient

Figure 5 above shows the temperature gradient from the thermal simulation performed on the 6061 T6 material. Aluminium has a high thermal conductivity which allows heat to flow through the component more efficiently which can result in a more uniform temperature distribution and fewer hot spots forming. The maximum temperature reaches 302.83 Celcius which is similar to the reference simulation which resulted in 310 celcius, the difference may be from meshing choice and the selected areas for the various thermal boundaries.

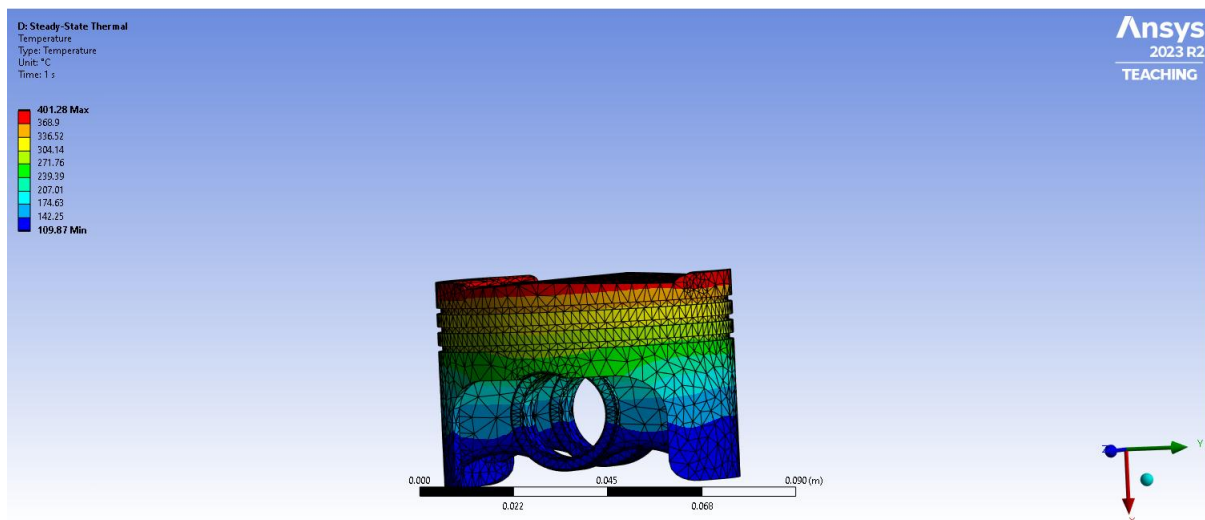


Figure 6 - Grey Cast Iron Temperature Gradient

Figure 6 shows us the comparative temperature gradient with the same boundary conditions applied to the 6061 T6. Grey cast iron typically has a moderate thermal conductivity but improved wear resistance, however due to being lower than the Aluminium the heat can accumulate more readily in the hot spot regions such as the crown, this can lead to steeper temperature gradients from the top of the crown down to the skirt and underside, although it has a lower minimum temperature than the Aluminium, the maximum reaches 401 Celsius which is significantly higher.

Comparing the materials, the grey cast iron shows a steeper thermal profile and generally a higher peak temperature at the crown, this is due to it having a lower thermal conductivity which limits

how quickly and effectively it can transfer heat through the component. The Aluminium having a superior heat dissipation property can transfer more heat away from the combustion surface which results in a lower peak temperature which is an advantage in preventing thermal stresses which can result in a longer lifespan of the component.

5 Optimization and Improvement

5.1 Improvement

The current simulations show the displacement and temperature gradients of the pistons with the material choice selected for comparison to identify potential stress hot spots under load. Below is an outline of recommended next steps emphasising a suitable balance between performance, fatigue and the overall mass.

5.2 Optimization

High regions of deformations around the piston crown edges and Gudgeon pin boss means that geometry refinement to reduce the stress concentration would be suitable for this component. Smoother transitions and fillets around the crown area could potentially reduce the peak stresses that the surface experiences.

Potential for removing material in regions that experience minimal load such as the skirt could reduce the overall weight of the component without affecting the performance as these regions show very little deformation or thermal effect under load.

Considering the engine operating temperature to ensure the thermal conductivity of the material selected is appropriate and able to dissipate the heat through the material more efficiently can reduce overall fatigue.

6 Validation

6.1 Simplified Test Case

A simplified test case involving a cantilever beam subjected to a uniform distributed load (UDL) of 2750Pa was analytically calculated to measure the resulting deflection. A simulation study was then performed to compare the analytical result with that of the simulation for validation of the results. By applying fundamental principles such as the Euler-Bernoulli beam theory a solution can be derived (Fudlailah, Allen and Cordes, 2024).

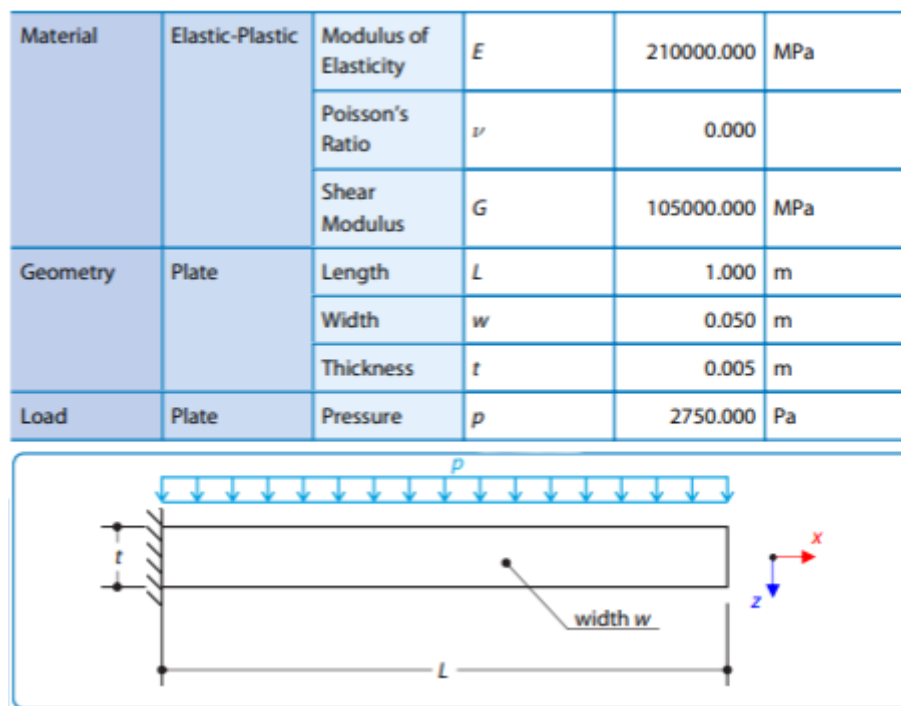


Figure 7 - Cantilever Properties

Figure 7 gives us the properties of the cantilever that we can use to analytically measure the maximum deflection of the beam.

6.2 Analytical and Software Comparison

Due to the pressure being applied to the top surface over length (L) the total vertical load is:

$$F_{total} = p \times (L \times w) = 2750Pa \times (1.0m \times 0.05m) = 137.5Nm$$

Calculating the second moment of area to then find the maximum deflection:

$$I = wt^3/12 = 12(0.05m)(0.005m)^3 = 5.21 \times 10^{-10}m^4$$

Using the standard formula for a cantilever beam with a UDL to calculate deflection:

$$\delta_{max} = QL^4 \div 8EI$$

$$\delta_{max} = 8 \times 2.10 \times 10^{11}Pa \times 5.21 \times 10^{-10}m^4(137.5N/m) \times (1.0m)^4 \approx 0.157m.$$

Total deflection is 0.157m

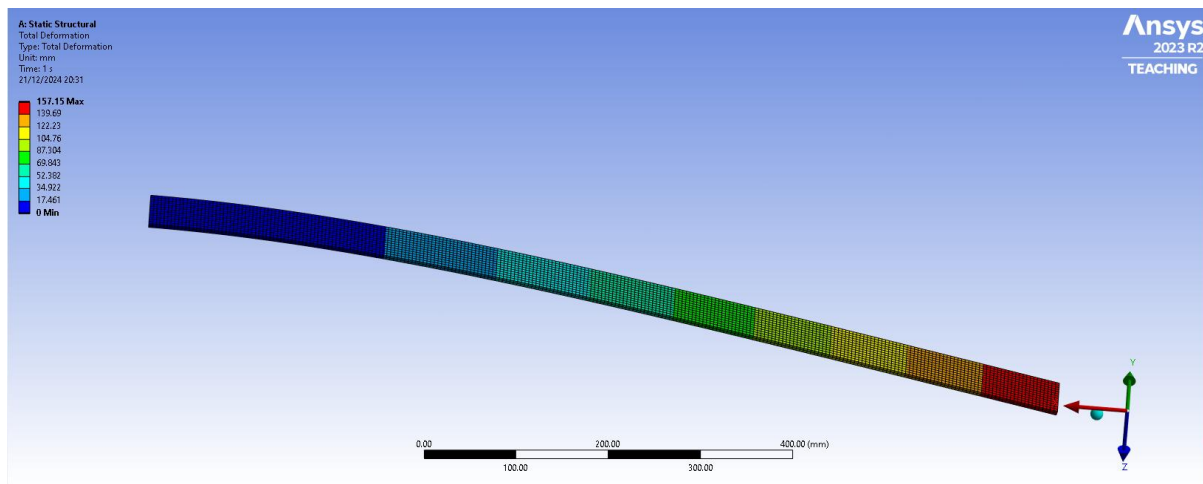


Figure 8 - Simulated Cantilever

Figure 8 above gives us the simulated cantilever UDL study which gives us a total deflection result of 157mm (0.157m) which is validated by the result produced from the analytical analysis.

6.3 Mesh Independence Study

The objective of this study is to investigate how the mesh density affects the accuracy of simulation and the computational costs involved. An optimal mesh can produce negligible changes in key results such as stress and deformation which can further validate the simulation ('Ansys Solution and Postprocessing', 2007). Once a point of convergence is reached, such as the mesh refinement changes producing insignificant results you then have mesh independence. Using that mesh then allows for all subsequent analysis to have a reliable balance between accuracy and computational

requirements.

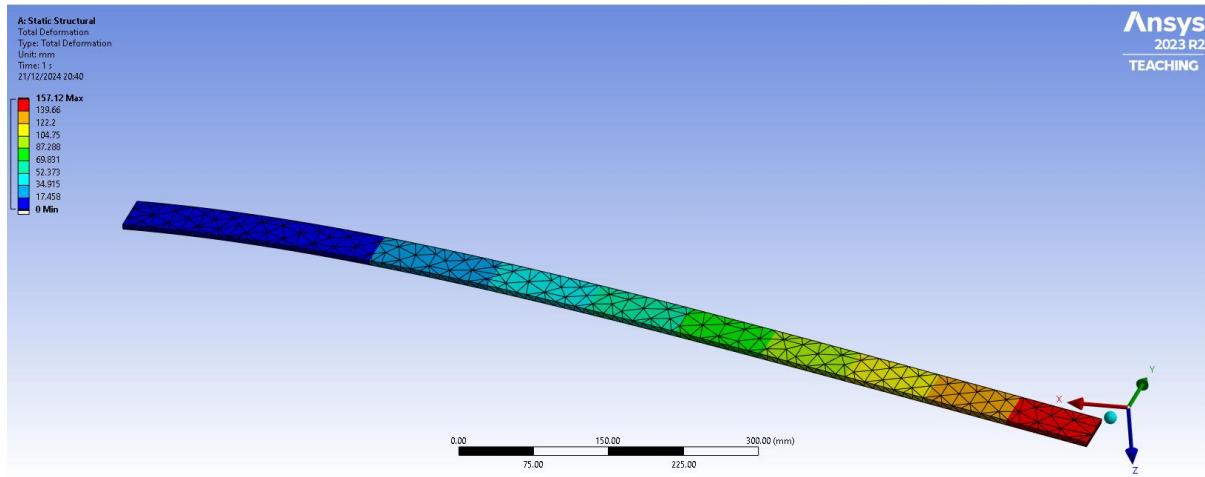


Figure 9- Tri Mesh Cantilever

As an example of a mesh study, the mesh style was changed to tetrahedral with the same element size to determine what effect this would have on the overall result of the study. The results show that even for a simple geometry such as the beam where a standard quad mesh system can efficiently mesh the component, the results have varied slightly. If the incorrect mesh type and size were used for a more complex component or system for analysis this could result in a larger deviation from the expected result which would then require a re-meshing and analysis resulting in more computational use, cost and time.

6.4 Boundary Condition Influence

Boundary conditions specify how a component or assembly is supported and loaded within a simulation. They directly influence the distribution of stresses, displacements and variables such as temperature. If constraints are overly restrictive the structure may appear stiffer than intended which could produce lower displacements or stresses of the component. Under-restraining a component may also not reflect real world behaviour due to having more degrees of freedom than intended.

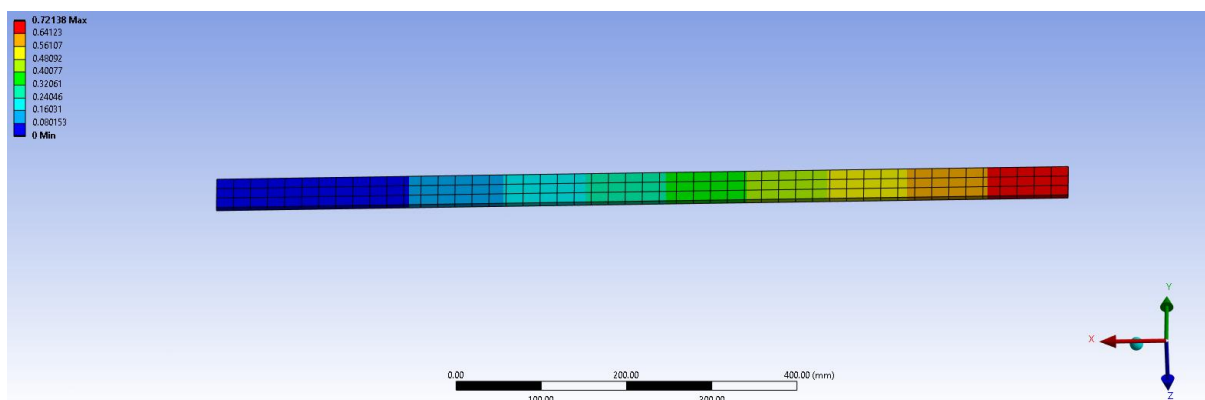


Figure 10 - Point Load Example

Figure 10 above shows us an incorrectly modelled load on the cantilever, where the forces acting upon the surface are intended to be a UDL, the force on the surface was set to a point location which has given an incorrect value for deflection than what the simulation had intended.

The method in which supports and loads are applied to the model simulation will change how the force flows through the model, this can alter the local stress concentration, deflection profiles and failure points. Validating boundary conditions against known physical properties or constraints is critical for confidence in the results produced from simulation.

7 Discussion

This project has demonstrated the key uses for using CAE for the design and analysis process within the engineering sector. It has demonstrated the use of techniques such as kinematic analysis, static structural assessment and thermal considerations to provide insight into mechanical behaviour under realistic real-world conditions. The assembly selection to produce the kinematic analysis provided a realistic set of loading conditions that simulated the environment in which that assembly operates. By implementing motion profiles and identifying key rotations between components it meant a valid structural and thermal study could be produced and demonstrates how kinematic outputs are essential inputs for further simulations.

Using the loading conditions generated in the kinematic study a detailed assessment was conducted on the piston revealing a stress distribution pattern for selected materials with consistent boundary conditions. The inclusion of thermal considerations allowed for further simulation and complex behaviours to arise in the simulation, reflecting real-world scenarios where material properties degrade under heat exposure. This shows the strengths of CEA in identifying potential design weaknesses without producing physical prototypes for testing.

The next steps for this project would be to optimize and create iterative improvements of the geometry and material selection by refining the component design based on the FEA results collected. It would then be possible to achieve better structural performance, improved fatigue life and a reduction in the overall mass.

8 Conclusion

The analytical and simulation steps completed within this report demonstrate the significance of CAE tools for analysing and predicting system performance for real-world application. However, it also emphasizes the importance of ensuring the correct boundary conditions, meshing type and method and optimization of design is to fully benefit from the tools. Implementing the correct material selection and appropriate properties will allow for further improvements of this component and address the limitations observed during simulation from computational requirements, time usage and the overall results.

9 References

'Ansys Solution and Postprocessing' (2007) *The Finite Element Method and Applications in Engineering Using Ansys®*, pp. 149–186. Available at: https://doi.org/10.1007/978-0-387-28290-9_5.

Bianco, L. *et al.* (2022) 'Influence of the thermal deformation on the lubricating performance of the piston-gudgeon pin interface in an internal combustion engine', *Tribology International*, 174, p. 107719. Available at: <https://doi.org/10.1016/J.TRIBOINT.2022.107719>.

Fudlailah, P., Allen, D.H. and Cordes, R. (2024) 'Verification of Euler–Bernoulli beam theory model for wind blade structure analysis', *Thin-Walled Structures*, 202, p. 111989. Available at: <https://doi.org/10.1016/J.TWS.2024.111989>.

Godara, S.S. *et al.* (2022) 'Design & analysis of connecting rod using ANSYS software', *Materials Today: Proceedings*, 56, pp. 1896–1903. Available at: <https://doi.org/10.1016/J.MATPR.2021.11.166>.

Mahle (2016) 'Pistons and engine testing'.

Ravi Kumar, L. *et al.* (2023) 'Investigations on multi cylinder four stroke diesel engine's crankshafts with different materials', *Materials Today: Proceedings* [Preprint]. Available at: <https://doi.org/10.1016/J.MATPR.2023.03.058>.

Witek, L. and Zelek, P. (2019) 'Stress and failure analysis of the connecting rod of diesel engine'. Available at: <https://doi.org/10.1016/j.engfailanal.2019.01.004>.