

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

Word Count: 3000

Structural & Fluid Analysis (ACHA6010)

Luke Anthony Dyas

Student ID: 2104531

December 2024

Table of Contents

Introduction	1
CAE Tools Discussion	2
Validation Example 1 – Bimetallic Strip.....	2
Validation Example 2 – Bending Cantilever with Axial Force	4
Kinematic and Structural Analysis.....	5
Inertia Calculations	5
Materials	8
Environmental Considerations	9
Thermal Analysis	10
Combustion Load under Thermal Conditions	12
Tension Load under Thermal Conditions	14
Boundary Conditions Investigation	16
Mesh Sizing Investigation	18
Optimisation Possibilities	20
Conclusion.....	21
References.....	22

Introduction

This investigation utilises ANSYS Workbench to analyse and optimise the structural performance of a connecting rod under realistic operating conditions. Validation of the ANSYS software results against Dlubal Software examples with acceptable accuracy margins.

The study examines dynamic and thermal loads, incorporating inertia forces, combustion pressure and temperature conditions across varying materials, such as Steel, Aluminium, Titanium Alloys and Cast Iron. The thermal analysis highlights material suitability, while mesh sizing and boundary condition studies ensure simulation accuracy and efficiency.

Finally, through iterative optimisation, this investigation balances mass reduction with structural integrity, demonstrating the capabilities and limitations of CAE tools for enhancing component performance.

CAE Tools Discussion

Dlubal Software has developed their own advanced structural analysis and design software. It is considered extremely precise and reliable, used across 132 countries with 13,000 companies utilising it for structural and engineering tasks (Software, 2021). A perfect validation partner for this investigation. Dlubal provide simplified examples to demonstrate the accuracy of the software. To compare the accuracy of ANSYS Workbench, these examples have been repeated below.

Validation Example 1 – Bimetallic Strip

Dlubal's Description

A bimetallic strip is composed of invar and copper. The left end of the bimetallic strip is fixed, and the right end is free, loaded by temperature difference T_c . While neglecting self-weight, determine the deflection of the bimetallic strip (free end).

Table 1: Dlubal's Provided Information for the Validation Example

Material	Invar	Modulus of Elasticity	E_i	137000	MPa
		Poisson's Ratio	γ_i	0.280	—
		Coefficient of Thermal Expansion	α_i	$1.200X10^{-6}$	$^{\circ}C^{-1}$
	Copper	Modulus of Elasticity	E_c	130000	MPa
		Poisson's Ratio	γ_c	0.354	—
		Coefficient of Thermal Expansion	α_c	$2.000X10^{-5}$	$^{\circ}C^{-1}$
Geometry		Cross-Section Width	w	5	mm
		Layer Thickness	$t_i = t_c = t$	1	mm
		Length	L	100	mm
Load		Thermal Loading	T_c	100	$^{\circ}C$

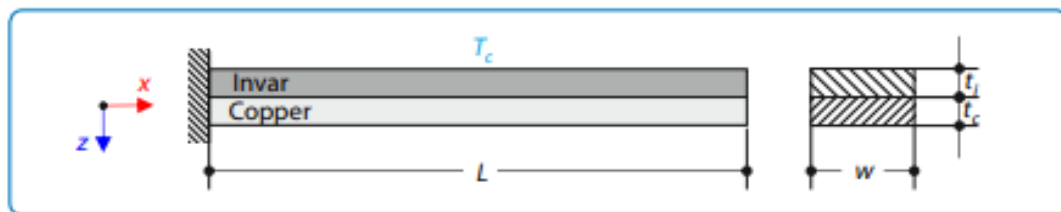


Figure 1: Dlubal's Diagram of the Example

Analytical Solution

Dlubal also provides the analytical solution for each example. This further validates both Dlubal's simulation, as well as the ANSYS Software results.

Equation 1: Calculating the dimensionless coefficient K_1

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

Equation 2: Calculating maximum deflection

$$\text{Deflection} = \frac{6(\alpha_i - \alpha_c)(t_i + t_c)T_c}{t_s^2 K_1}$$

ANSYS Solution

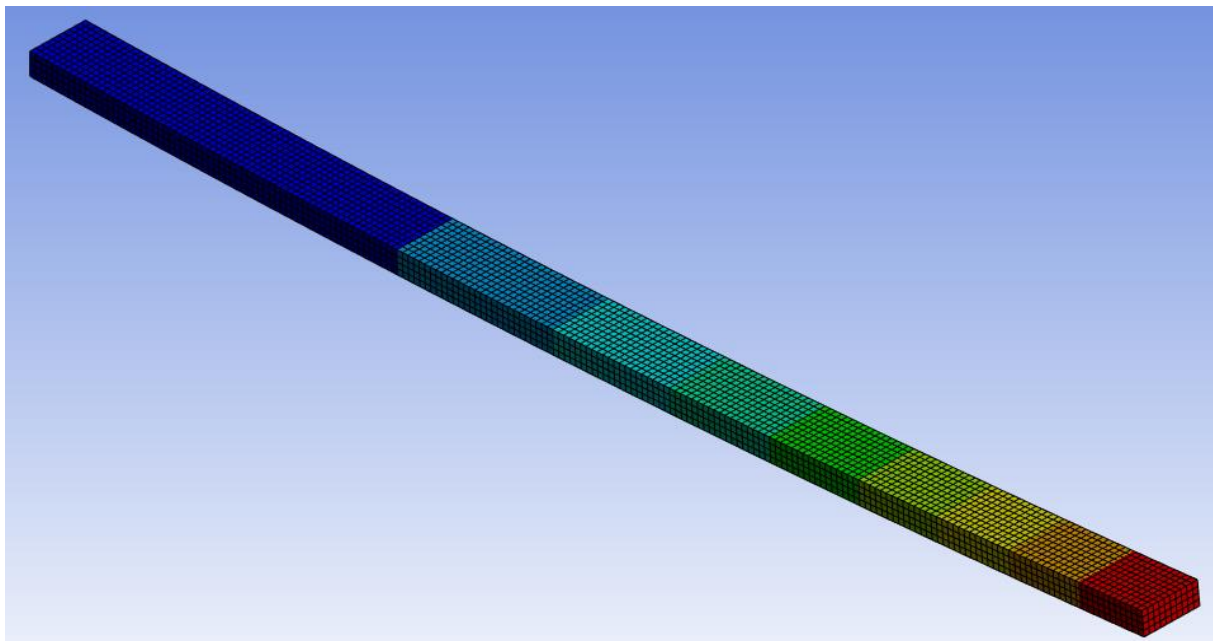


Figure 2: Showing an isometric view of the ANSYS Solution of the Validation Example Scenario

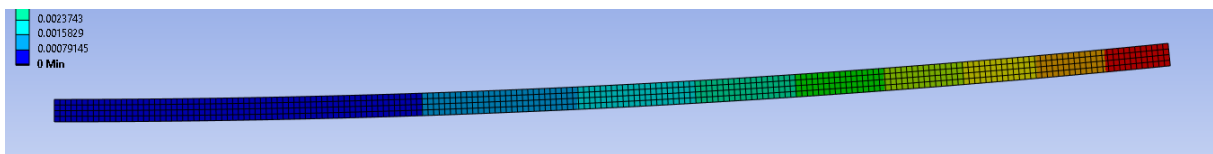


Figure 3: Showing a side on view of the ANSYS Solution of the Validation Example Scenario

Table 2: Comparison between each result for the Validation Example 1

Solution	Result	% Accuracy
Analytical	7.047 mm	0.00
Dlubal	7.065 mm	0.26
ANSYS	7.121 mm	1.05

Even with Dlubal using their own example there is still an accuracy error, so for ANSYS to be $\approx 1\%$ with unknown simulation factors (mesh sizing), this is an accuracy level that is suitable for the investigation.

Validation Example 2 – Bending Cantilever with Axial Force

Dlubal's Description

A cantilever is loaded by a transversal and an axial force on the right end and it is fully fixed on the left end. While neglecting self-weight, determine the deflection of the cantilever beam (free end).

Table 3: Dlubal's Provided Information for the Validation Example

Material	Steel	Modulus of Elasticity	E	210000.00	MPa
		Poisson's Ratio	γ	0.300	—
Geometry	Cantilever	Length	L	10.00	m
		Cross-section Width	w	0.08	m
Load		Transversal Force	F_z	7.65	kN
		Axial Force	F_x	1.60	kN

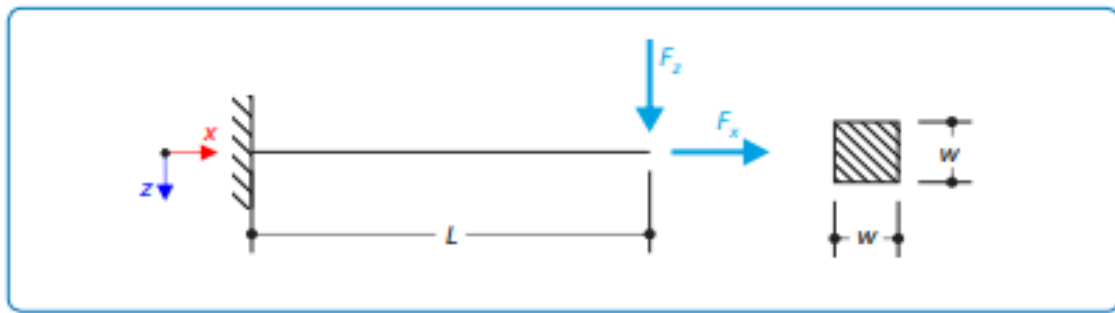


Figure 4: Dlubal's Diagram of the Example

ANSYS Solution



Figure 5: Showing an side on view of the ANSYS Solution of the Validation Example Scenario

Table 4: Comparison between each result for the Validation Example 2

Solution	Result	% Accuracy
Analytical	3557.478 m	0.000
Dlubal	3557.515 m	0.001
ANSYS	3555.700 m	-0.050

This result reinforces the use of ANSYS for this investigation with an accuracy percentage of ≈ 0.05

These validations have been selected as they are similar to the load applications experienced by the connecting rod.

Kinematic and Structural Analysis

The ANSYS software uses a linked workflow, where outputs from one analysis directly feed into another. This modularity ensures flexibility, allowing parameter isolation. The utilised modules include material properties, rigid dynamics, a steady-state thermal and static structural analysis.

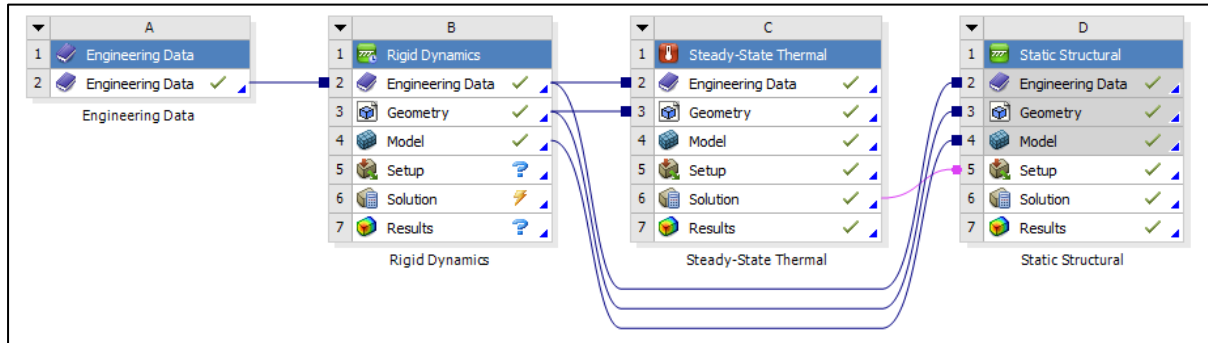


Figure 6: An overview of the module layout in ANSYS Workbench

Inertia Calculations

For this investigation the first key aspect that is analysed is the Inertia Calculations from the engine block assembly. This allows the first level of forces that the Connecting Rod will experience during its cycle. The choice of material across all these components will affect the inertia forces experienced by the Connecting Rod however, to simplify the investigation, during the inertia analysis, that each component is structural steel (the base material in ANSYS Workbench).

Rigid Dynamics Assumptions:

- Components do not deform, externally or internally (Young's modulus etc not considered)
- Perfect contacts – surfaces considered frictionless, with no imperfections
- No thermal effect (this is considered later)
- External influences such as vibration is not considered
- Additional forces such as combustion pressure is not considered (this is resolved later)

These assumptions reduce the accuracy of the modelling and produce a simplified result. However, allow the investigation to continue with these considerations. During the Rigid Dynamics various load/force points were recorded across the Connecting Rod. In figure 7 is the 3 that assess the forces between the gudgeon pin and the crankshaft.

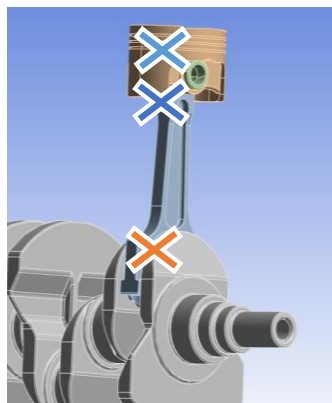


Figure 7: A view of the overall assembly and specific recorded probe load locations

During the Rigid Dynamics simulations, the connecting rods angular acceleration about the simulation Z axis. The inertia forces were calculated at 7000rpm as this investigation is ensuring that the connecting rod modifications can withstand the highest forces within this theoretical engine.

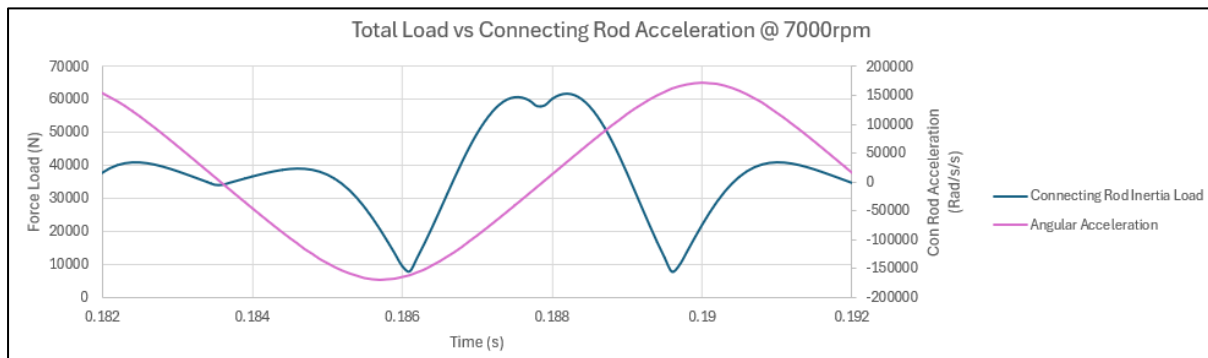


Figure 8: A graph created in Excel based on the exported ANSYS Data

Within figure 8 the total load (ignoring direction) experienced by the connecting rod due to the inertia forces, as well as the connecting rods angular acceleration during that time.

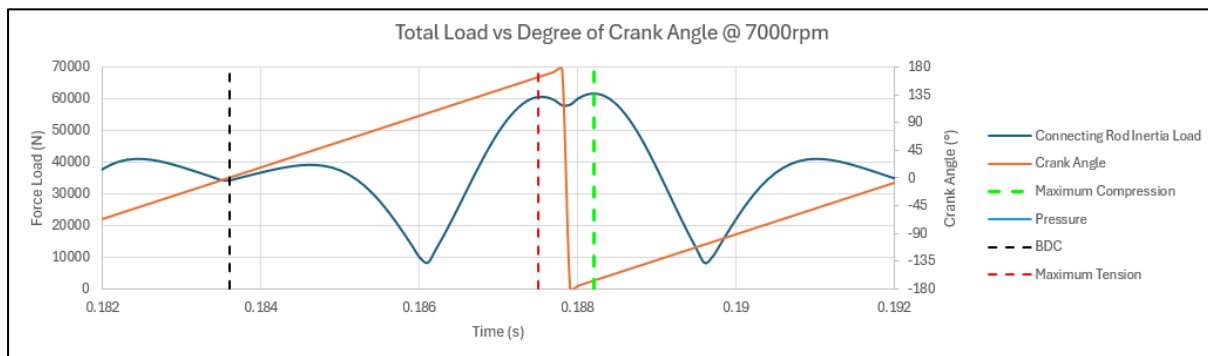


Figure 9: A graph created in Excel highlighting Maximum Figures

Figure 9 shows were the maximum compression and tension (directional) forces experienced by the connecting rod due to the inertia forces.

Due to the known connecting rod acceleration, the crank angle can be calculated and plotted. Now that the forces experienced at each crank angle are known, the X & Y forces experienced by the connecting rod can be calculated. Before this is done, the forces caused by combustion within the cylinder need to be accounted for. For this investigation known pressure data from a McLaren engine

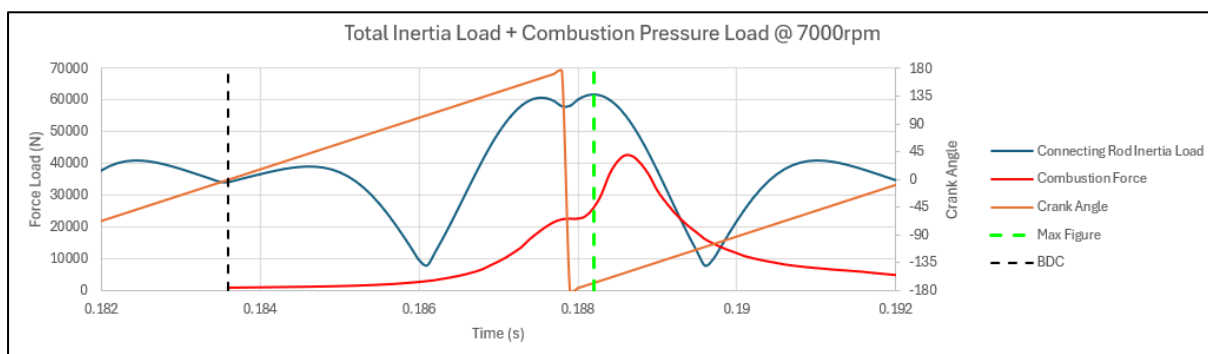


Figure 10: A graph created in Excel comparing the inertia forces and the forces caused by combustion at each crank angle.

These forces are then added together to show the total forces being applied to the connecting rod system. Some assumptions are made:

- The compression forces acting within the cylinder are 100% carried through the piston and gudgeon pin.
- Although the inertia forces maybe reduced by remaining gas forces in the cylinder before combustion, this has been ignored, assuming the cylinder is ambient pressure. This does mean a higher load is applied within the simulations however allows for a factor of safety to be naturally built in.

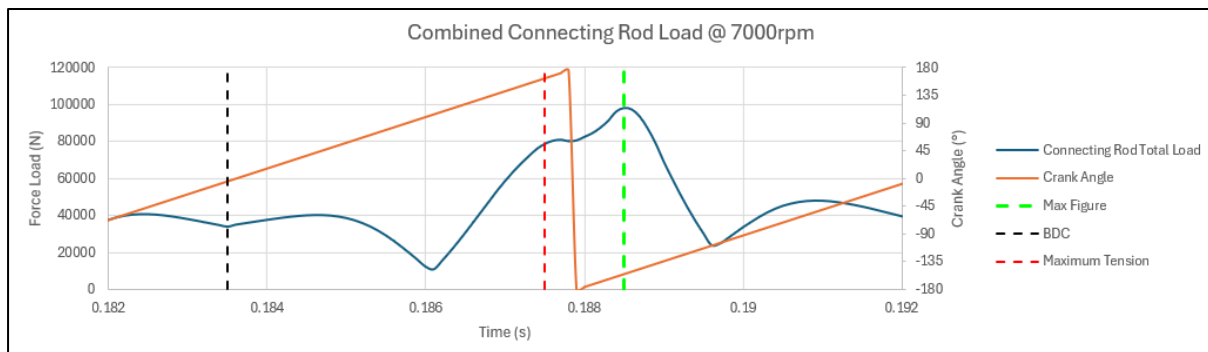


Figure 11: showing the total connecting rod forces per crank angle. As well as the new higher points of compression and tension load forces.

Now that the total rigid dynamic forces are known at each crank angle, through trigonometry the X and Y of these largest forces can be calculated and applied into the rest of the investigation.

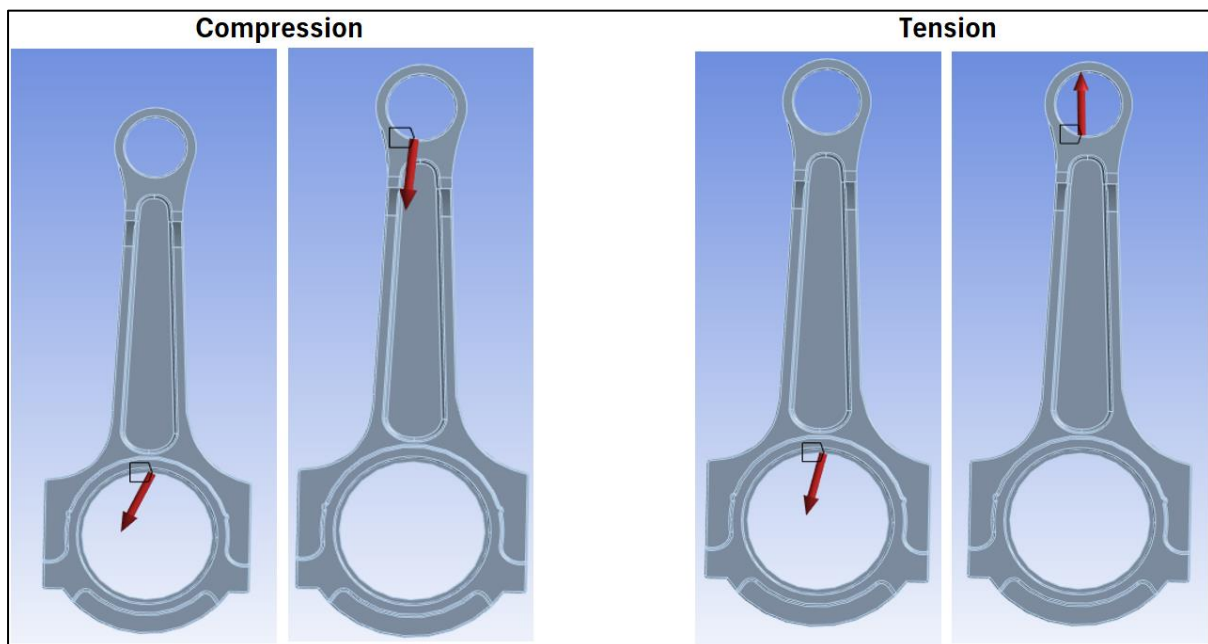


Figure 12: showing the net direction of each force causing compression and tension

Materials

The first step of the ANSYS Workbench flow allows the choice of material to be applied. Using another ANSYS programme, ANSYS Granta allows huge amounts of materials with huge amounts of data to be imported into ANSYS Workbench. A selection of Alloys has been selected for comparison.

These materials have been selected through research however are not guaranteed to be suitable, which allows the investigation to continue.

Table 5: A table created in Excel based on information from ANSYS Granta

Material Investigation			
1 - Steel Alloy	AISI 4340		Comments:
Oil Quenched & Tempered at 540c	Tensile Strength (Mpa)	1050-1300	High toughness and fatigue strength, commonly used in high-performance applications.
	Yield Strength (Mpa)	965-1190	
	Elastic Modulus (GPa)	205-213	
	Density (kg/m ³)	4.8e ³	
	Recycling (%)	49.4-54.6	
	Max Service Temp (C)	500	
	Price (GBP/kg)	1.04	
2 - Aluminium Alloy	7075 - T6		Comments:
	Tensile Strength (Mpa)	530-580	Lightweight with good machinability and corrosion resistance.
	Yield Strength (Mpa)	460-530	
	Elastic Modulus (GPa)	69-76	
	Density (kg/m ³)	2.77E+03	
	Recycling (%)	42.8-47.3	
	Max Service Temp (C)	150	
	Price (GBP/kg)	3.5	
3 - Titanium Alloy	Ti-6Al-4V		Comments:
	Tensile Strength (Mpa)	828-970	Excellent strength-to-weight ratio and corrosion resistance, but expensive
	Yield Strength (Mpa)	760-900	
	Elastic Modulus (GPa)	110-117	
	Density (kg/m ³)	4.41e ³	
	Recycling (%)	21.8-24.1	
	Max Service Temp (C)	350	
	Price (GBP/kg)	20.9	
4 - Cast Iron	Gray Cast Iron		Comments:
	Tensile Strength (Mpa)	300-400	High damping capacity and wear resistance, though brittle.
	Yield Strength (Mpa)	195-228	
	Elastic Modulus (GPa)	121-140	
	Density (kg/m ³)	7200	
	Recycling (%)	49.4-54.6	
	Max Service Temp (C)	350-450	
	Price (GBP/kg)	0.458	

(ANSYS, 2023)

Environmental Considerations

In modern engineering, there is a growing responsibility to minimise the environmental impact of the materials used. Environmental considerations, such as resource consumption, carbon emissions and the end-of-life recyclability are crucial for achieving sustainable development. (Aabid and Baig, 2023)

From table 5 the Steel Alloy has one of the highest recycling % rate with around 50%, with a well-established infrastructure for reuse means the overall energy demands are reduced. The Titanium Alloy has the lowest recycling % rate from the selected list, meaning its life cycle production energy is highest.

As well as the ability to recycle the material, the life cycle/durability of the component has an impact on the overall environmental impact. Within ANSYS it is possible to conduct a life cycle test on the component, this has not been conducted in this investigation.

Thermal Analysis

For the chosen materials to be tested through the force loads, the factor of temperature needs to be accounted for as temperature changes the characteristics of how the materials will respond. Through the Steady-State Thermal analysis the required temperature responses can be carried into the overall model.

Figure 13 shows a thermal analysis of another 4-stroke engine temperature distribution. From this along with other research a chosen range of thermal conditions can be applied to the connecting rod materials.

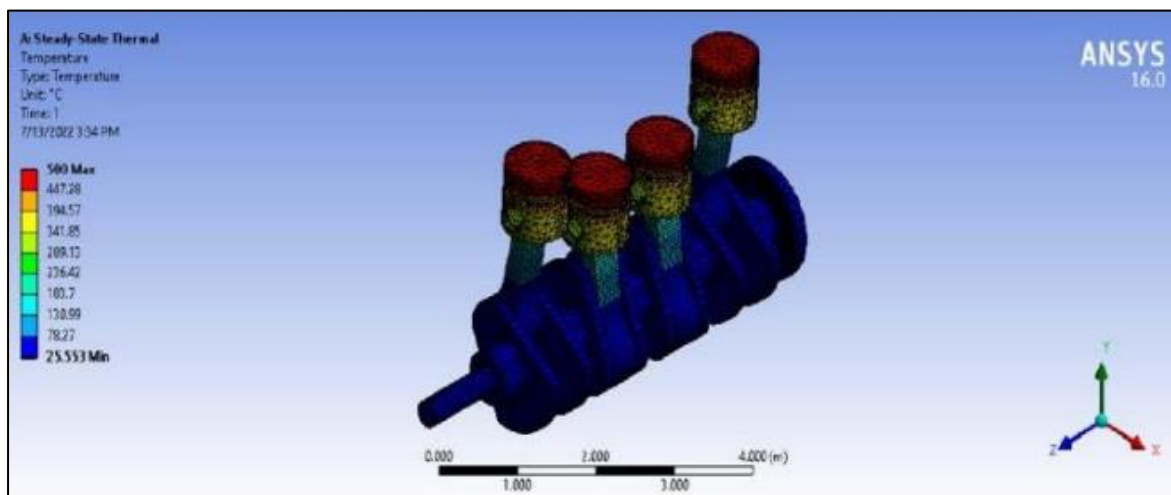


Figure 13: Thermal analysis of a 4-stroke engine entire system conducted by (Mohammed Mousa Azzawi, Auday Shaker Hadi and Atheer Raheem Abdullah, 2023)

Table 6: Showing the expected temperatures applied to the Connecting Rod

Applied Environment Temperatures	Connecting Rod Crankshaft	Connecting Rod Gudgeon Pin
°C	23	200

(Lewis, 2024) also conducted a study on the thermal temperatures within a 4 stroke engine and ensured the connecting rod was required to operate from -30° up to 180°C in most cases.

Table 6 shows the expected environmental temperatures, these are then applied to the connecting rod under each material and its properties to determine how they react to this thermal loading.



Figure 14: showing the temperature profiles of each material based on expected temperatures

Across every analysis in this investigation the visual representations use the same appropriate key. Additionally for the steady-state thermal analysis the mesh has been created by ANSYS system, the impact of mesh sizing is discussed later.

Initial analysis shows both the Steel and Titanium can withstand higher thermal loads due to their higher service temperature limits, while Steel provides superior thermal durability, Titanium compensates with its lightweight and high strength characteristics. The Aluminium Alloy has the best heat dissipation, ideal for lightweight applications, however as shown in figure 14 its low service temperature could limit its effectiveness in this investigation. Finally Cast Iron demonstrates better heat dissipation than the Steel and Titanium Alloys, however not as effective as the Aluminium Alloy.

Combustion Load under Thermal Conditions

The final module in the ANSYS Workbench brings together all the previous analysis' and applies each parameter together. This provides an accurate representation of how the connecting rod will react under the thermal and dynamic loads simultaneously. 2 main factors are recorded below:

- Deformation – this shows how much and where the geometry changes based on the applied loads. This is a vital factor for this investigation as the intricacies of the engine, requires minimal uncontrolled movement.
- Strain – this shows more precisely where the load is acting with the geometry, a vital tool when looking to optimise the component.

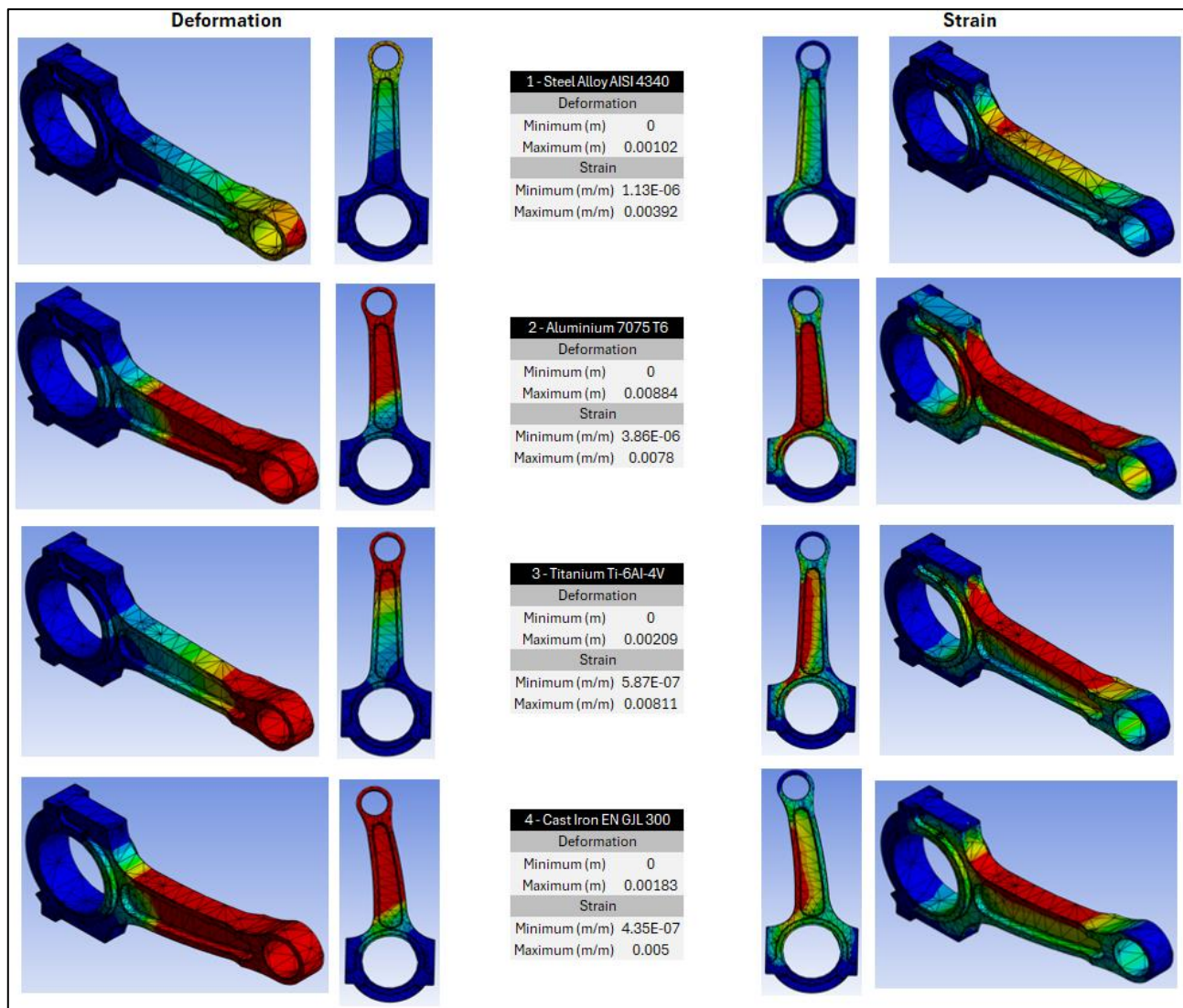


Figure 15: Showing the reactions of each material under expected thermal and dynamic compression loads

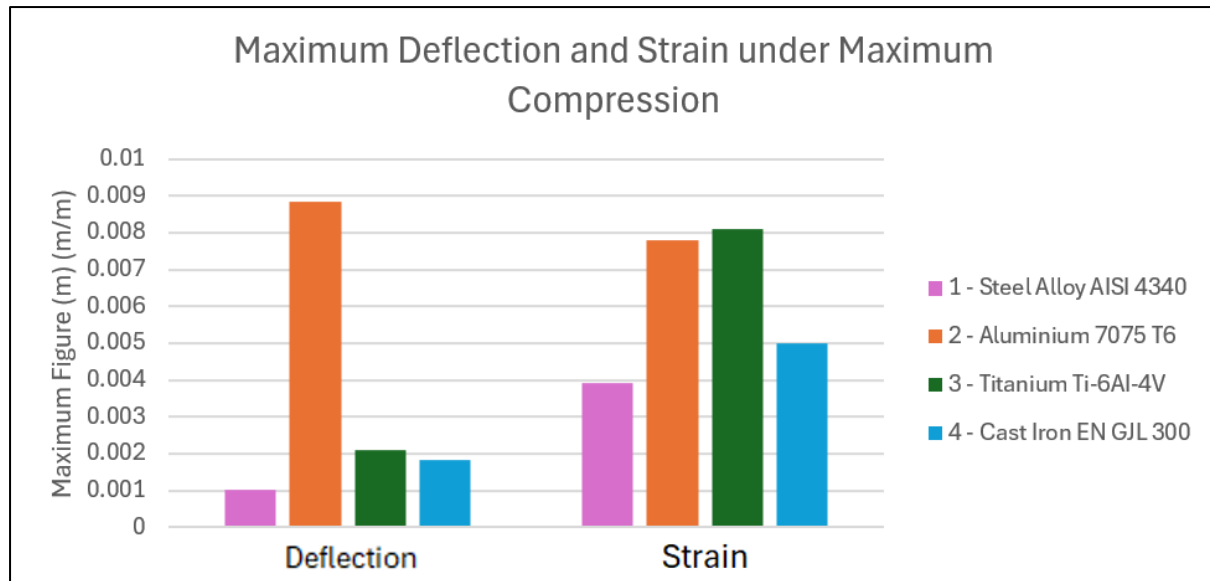


Figure 16: A graph created in Excel to graphically show the results in figure 15

The Steel Alloy outperforms the other materials in this analysis, with better strain handling characteristics and ultimately minimal deflection. However, also to note that the Aluminium Alloy has a lower peak strain than Titanium Alloy, yet a much larger overall deformation. This acts as expected due to the Aluminium Alloy having a lower yield strength & elastic modulus than the Titanium Alloy.

Tension Load under Thermal Conditions

Additionally, to the material and geometry being able to withstand the compression loads, it needs to be able to withstand the high tension loads.

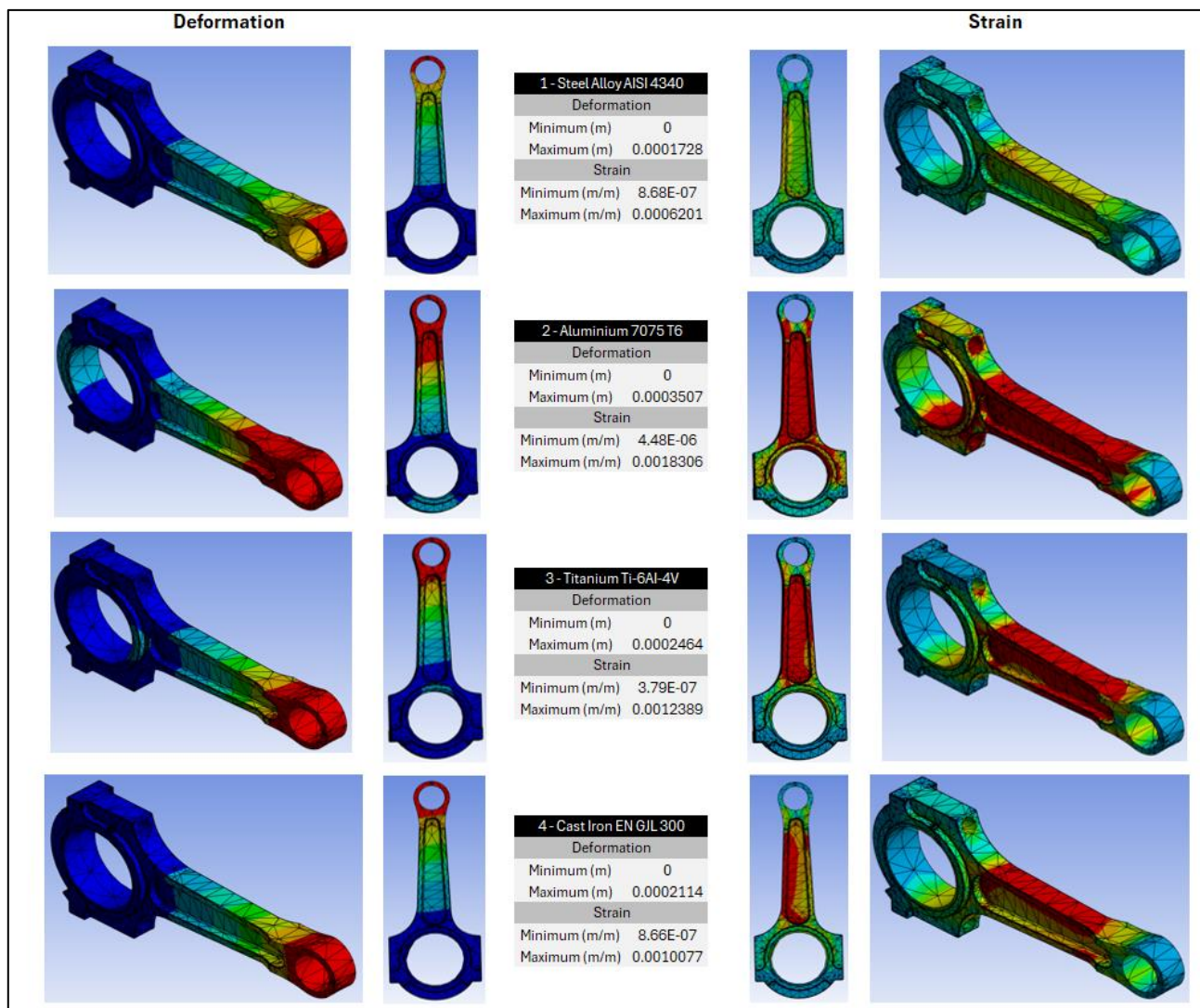


Figure 17: Showing the reactions of each material under expected thermal and dynamic tension loads

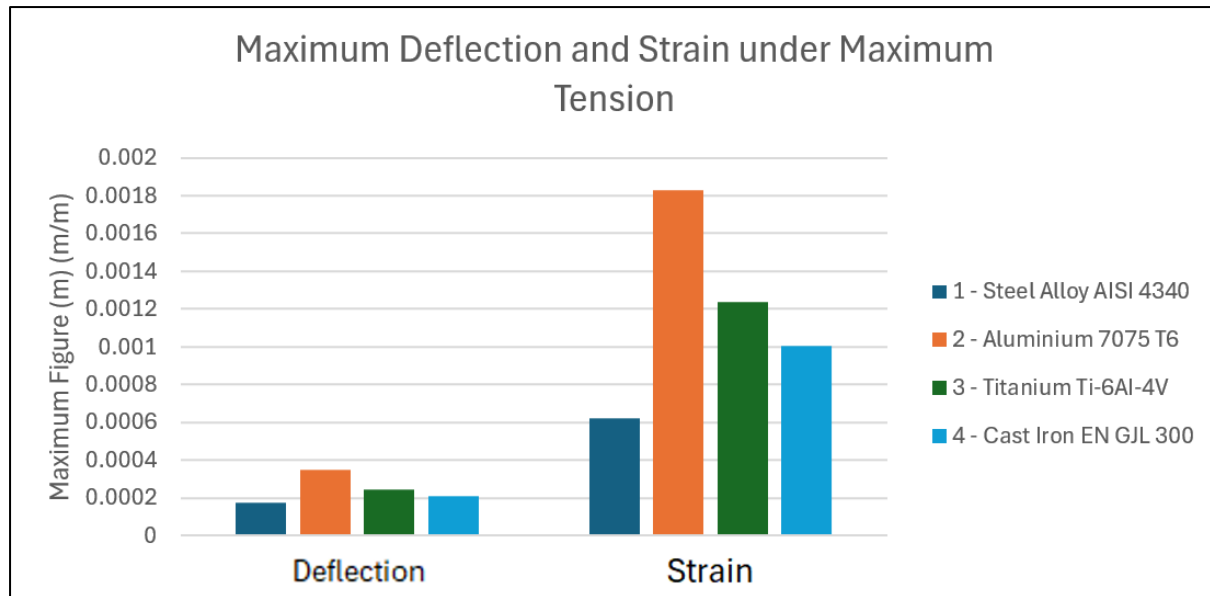


Figure 18: A graph created in Excel to graphically show the results in figure 17

Both the Steel Alloy and Cast Iron show minimal deflection, however the Steel once again outperforms due to the Cast Iron's brittleness. The Aluminium is again showing unwanted reactions for the connecting rod's requirements.

Going forward in the assignment for both the investigations and optimisation, the Steel Alloy AISI 4340 is used.

Boundary Conditions Investigation

This section investigates the impact of applying boundary conditions in different ways. The comparison of 3 different fixed points has been used:

- Bolt Fixing: the two bolt holes are locked in place, allowing the remainder of the component, including the big and small ends to react and deform as they would.
 - o This fixing doesn't account for any motion/thermal dynamics within the bolts that are holding the big end together around the crankshaft, reducing accuracy.
- Lower Fixing: the big end of the connecting rod is held in place, allowing the remainder of the component, including the small end to react and deform.
 - o This fixing assumes that the big end isn't moving/deforming around the crankshaft, as well as negating any loads applied from the crankshaft onto the connecting rod.
- Upper Fixing: the small end of the connecting rod is held in place, allowing the remainder of the component, including the big end to react and deform from this point.
 - o This fixing assumes that the small end isn't moving/deforming around the gudgeon pin, as well as negating any loads applied onto this part of the connecting rod.

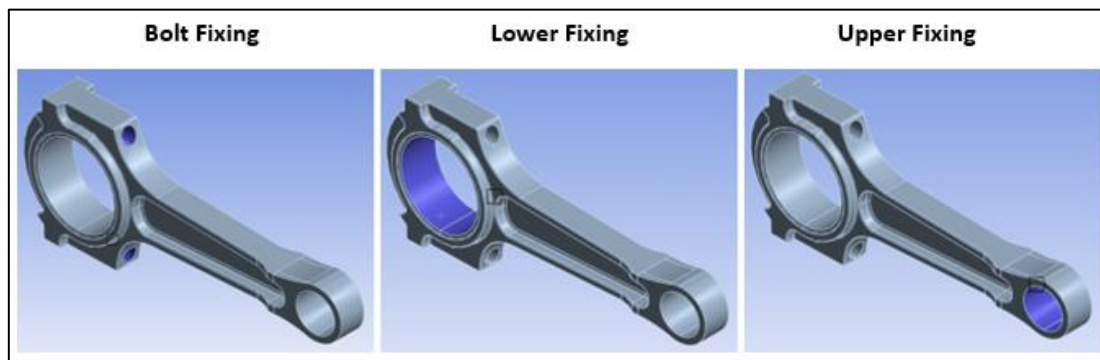


Figure 19: Showing the different Fixed Points

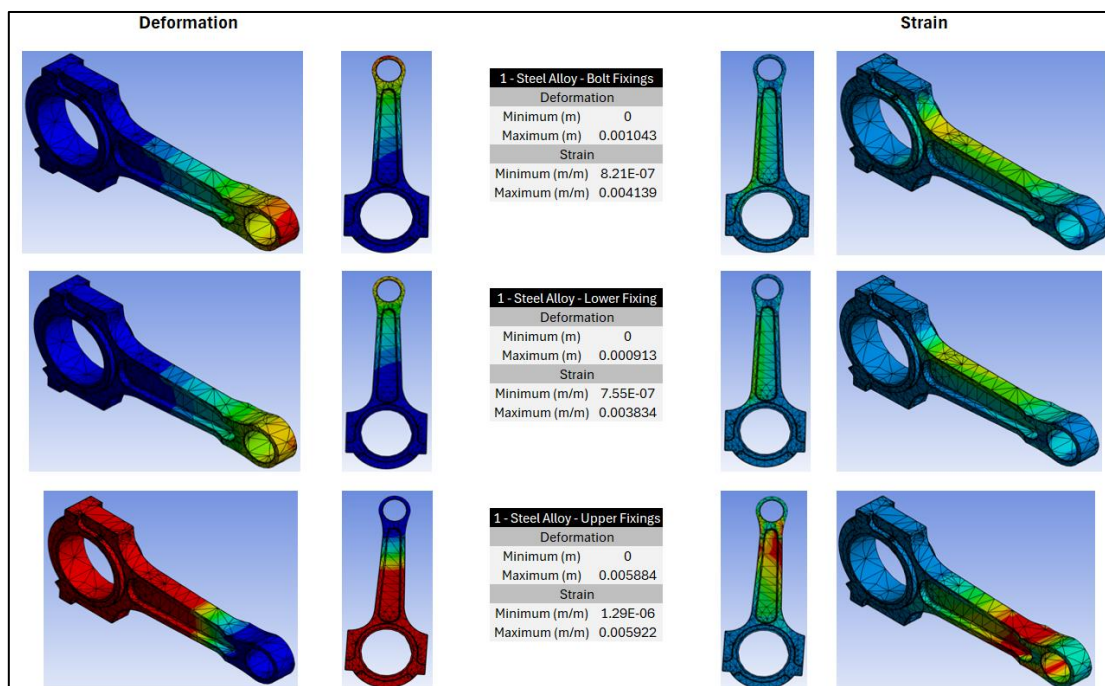


Figure 20: ANSYS results based on varied Fixed Points

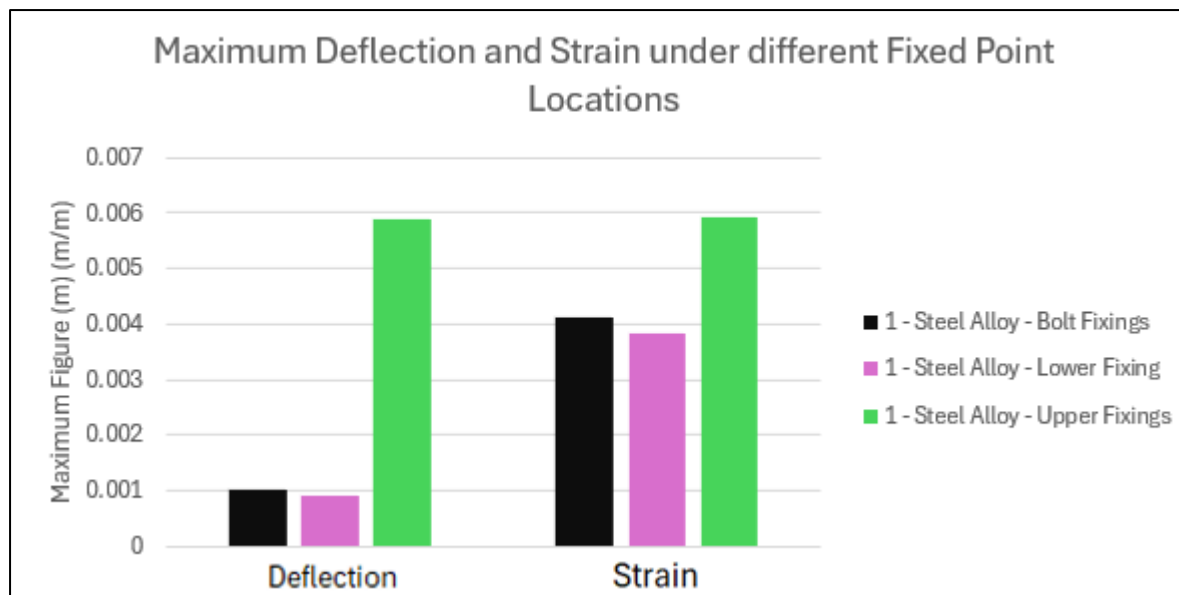


Figure 21: A graph created in Excel to graphically show the results in figure 20

The only variable changed in this specific investigation is the fixing locations, causing varying points of leverage and this change has a drastic effect on how the components reacts.

Table 7: A table showing the variance in results

	Fixing	% Variance
Deflection	Bolt Fixing	-
	Lower Fixing	-12
	Upper Fixing	464
Strain	Bolt Fixing	-
	Lower Fixing	-7
	Upper Fixing	43

Inaccurate assumptions about the models fixing locations can lead to misrepresentation of the actual load paths and stress concentrations. As table 7 shows this can lead to under and over estimations. The loads acting on the connecting rod act as a moment, if the pivot/leverage point is moved then the results change.

Mesh Sizing Investigation

Within ANSYS it is possible to control the size of the mesh (elements) that make up the geometry of the component.

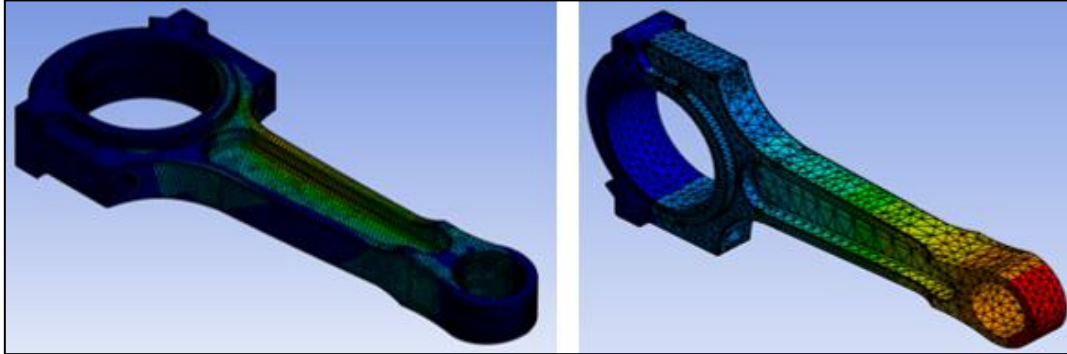


Figure 22: ANSYS representation of different mesh sizes

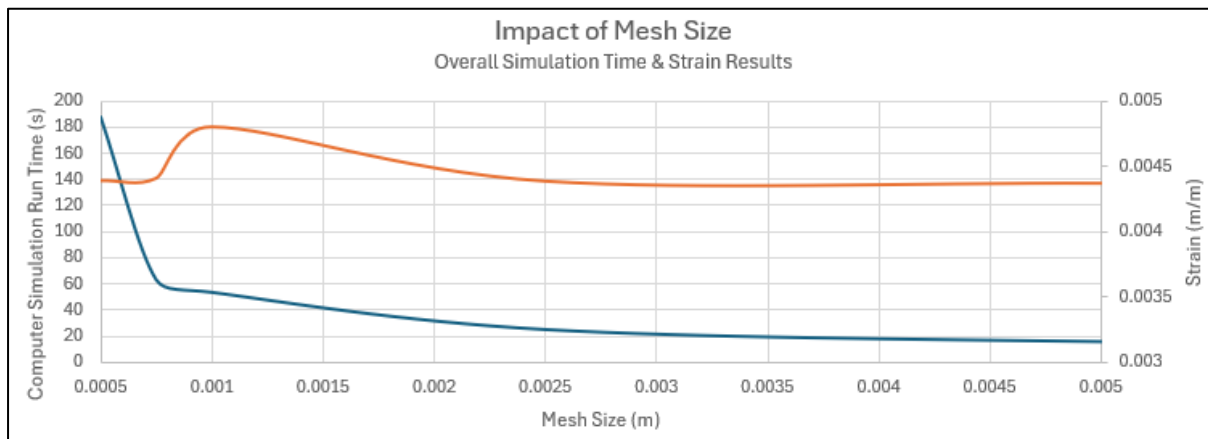


Figure 23: An Excel created graph plotting the impact of mesh size against results accuracy and simulation timing

Figure 23 shows how using a mesh size of 0.003m is adequate for this investigation. This provides a balance between simulation accuracy and computational efficiency.

The strain results plateau around this size, ensuring the model captures the critical behaviors of the component without unnecessary refinement, as well as avoiding over complicating areas.

The computational time remains manageable, making it practical for multiple iterations or design simulations, especially if this was applied to a much larger application.

The deformation results throughout each mesh size change less than 4%, indicating that the simulation is precise enough for the optimisation process.

However, the use of smaller mesh sizes allows for the location of the deformation and strain to be analysed more accurately. A compensation between using small mesh sizes and increasing accuracy while maintaining computational efficiency is through specific location mesh sizes.

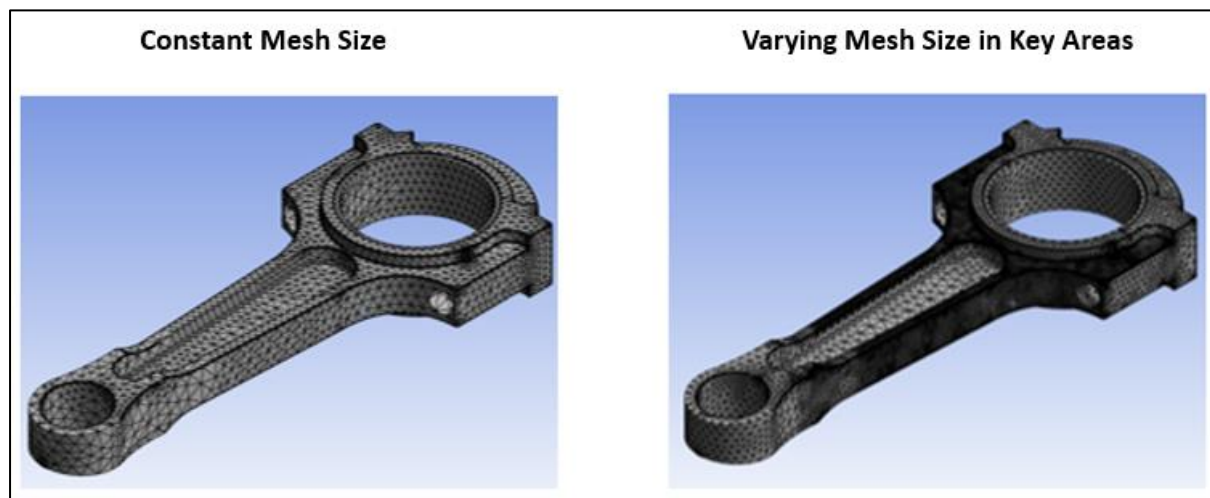


Figure 24: ANSYS representation of refined mesh sizes

This figure shows how different mesh sizes can be implemented into the ANSYS solutions. This means that specific areas of importance can be specifically analysed without impacting the whole simulation.

Optimisation Possibilities

By building up each module within the ANSYS Workbench allows varying iterations to be simulated and analysed. Each iteration looks to evaluate the design, adjust variables and re-evaluate until the objective function meets the desired level, within the constraints.

Common Types of Optimisations:

- Topology – prioritises the optimal material distribution within the given design space. If minimally restricted, topology can lead to innovative shapes.
- Material – Choosing the best material/s for the components/design. Discussed before.
- Size and Shape – Adjust geometry dimensions – length, thickness, cross-sectional area.

Common Optimisation Objectives:

- Weight/Mass
- Stiffness
- Stress or Strain
- Natural Frequencies

Figure 25 shows an example of an optimisation iteration. The objective to reduce the overall mass of the connecting rod while minimally impacting the deformation under these highest loads.

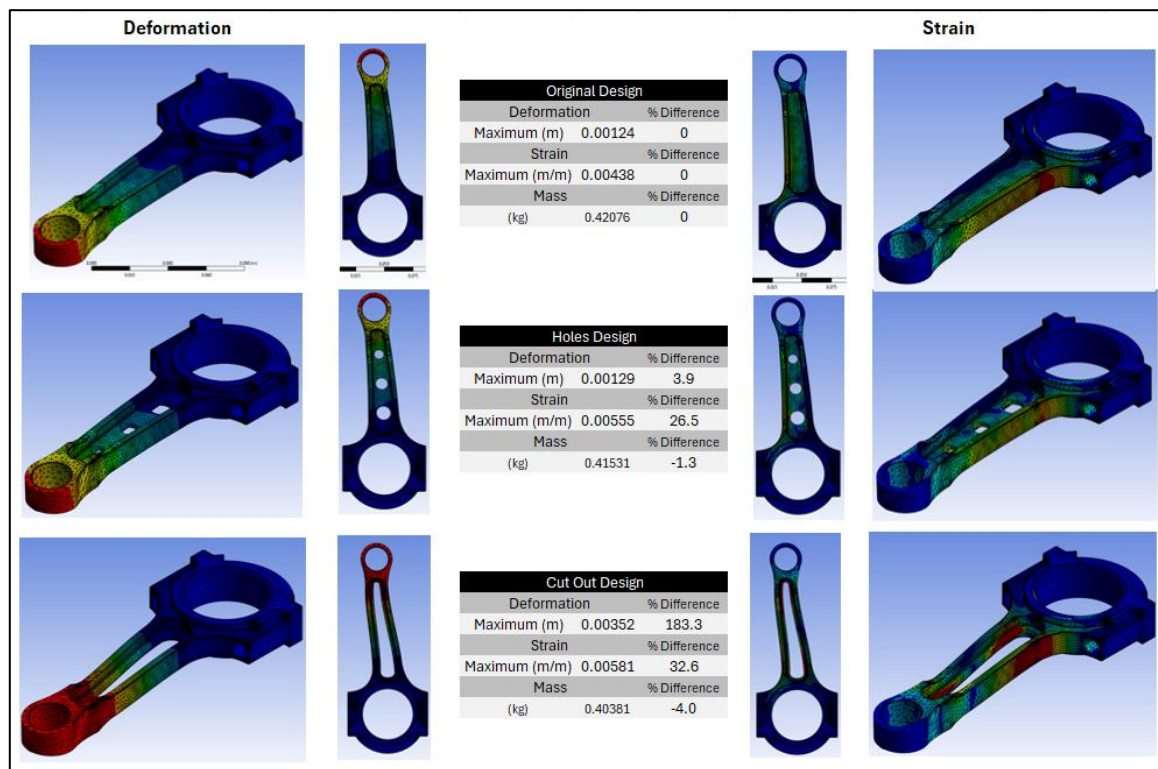


Figure 25: : ANSYS results based on varied iterations

These iterations have focused on the centre point of the connecting rod, which has produced minimal results. The relationship between these mass reductions has caused drastic increases in the connecting rods deformation and strain. It is clear to see that the original design had already conducted some optimisation in this area. For further optimisation other less strained areas, such as the big end could be worked on.

Conclusion

This investigation has demonstrated the effective use of ANSYS Workbench as a CAE tool to analyse, validate and optimise the performance of a connecting rod within a theoretical engine set-up, under realistic operational conditions. The accuracy of ANSYS for structural analysis was established through Dlubal Software examples.

The study highlighted the critical influence of dynamic and thermal loads, as well as material properties on the connecting rod performance. Steel Alloy AISI 4304 emerged as the optimal choice of material due to its superior balance of strength, thermal durability and minimal deformation under the predicted extreme conditions.

The further investigations into mesh sizing and boundary conditions demonstrated their impact of simulation accuracy and efficiency.

The findings reinforce the value of CAE tools such as ANSYS in facilitating efficient design processes, enabling engineers to make informed decisions to enhance performance and durability, while considering sustainability of critical components.

References

Aabid, A. and Baig, M. (2023) 'Sustainable Materials for Engineering Applications', *Materials (1996-1944)*, 16(18), p. 6085. Available at: <https://doi.org/10.3390/ma16186085>.

ANSYS (2023) *GRANTA EduPack, Formerly CES EduPack: Materials Education Support | Ansys*, www.ansys.com. Available at: <https://www.ansys.com/products/materials/granta-edupack>.

Khan, A. and Tripathi, A. (2023) 'International Journal of Research Publication and Reviews Modeling and Thermal Analysis of IC Engine's Connecting Rod by using CATIA and ANSYS Software', *International Journal of Research Publication and Reviews*, 4(9), pp. 1685–1690. Available at: <https://ijrpr.com/uploads/V4ISSUE9/IJRPR17264.pdf> (Accessed: 15 December 2024).

Lewis, G. (2024) *Prob. 5 - Connecting Rod*, *Utah.edu*. Available at: https://my.eng.utah.edu/~ma5090/week3p5.htm?utm_source (Accessed: 15 December 2024).

Mohammed Mousa Azzawi, Auday Shaker Hadi and Atheer Raheem Abdullah (2023) 'Finite Element Analysis of Crankshaft Stress and Vibration in Internal Combustion Engines Using ANSYS', *Mathematical modelling of engineering problems*, 10(3), pp. 1011–1016. Available at: <https://doi.org/10.18280/mmep.100335>.

Nkrumah Jacob Kwaku, Baba Ziblim, Sulemana Yahaya and Amedorme, S.K. (2023) 'Modal and Thermal Analysis of a Modified Connecting Rod of an Internal Combustion Engine Using Finite Element Method', *Modeling and Numerical Simulation of Material Science*, 13(03), pp. 29–49. Available at: <https://doi.org/10.4236/mnsms.2023.133003>.

Software, D. (2021) *Structural Analysis Software, Dlubal*. Available at: <https://www.dlubal.com/en>.