

University of Wales Trinity Saint David

Wales Institute of Science and Art



Prifysgol Cymru

Y Drindod Dewi Sant

University of Wales

Trinity Saint David

Application of CAE Processes

Structural & Fluid Analysis | ACHA6010

Joao Ferreira

1902427

December 2024

Table of Contents

Table of Figures	2
1 Introduction	3
2 Modelling and Assembly	4
3 Validation Testing	7
3.1 Analytical Solution	7
3.2 Software Solution	8
3.3 Mesh Independence Study	9
3.4 Boundary Condition Influence	10
4 Kinematic Simulation and Structural Analysis.....	12
4.1 Rigid Dynamics.....	12
4.2 Steady-State Thermal	15
4.3 Static Structural Study	17
5 Conclusions	21
6 References	21

Table of Figures

Figure 1 - Modelled Brake Disc	4
Figure 2 - Modelled Brake Pad	4
Figure 3 - Modelled Piston	5
Figure 4 - Brake Calliper v1.0	5
Figure 5 - All Components in Assembly	6
Figure 6 - Cantilever Beam Diagram.....	7
Figure 7 - Beam Cross Section.....	7
Figure 8 - Ansys Verification Solution.....	8
Figure 9 - Ansys Verification Solution v2	8
Figure 10 - 'Coarse' Mesh Solution.....	9
Figure 11 - 'Fine' Mesh Solution.....	9
Figure 12 - Fixed Support Beam Simulation	10
Figure 13 - Force Applied Beam Simulation	11
Figure 14 - Rigid Dynamics Study Screenshot 1	12
Figure 15 - Rigid Dynamics Study Screenshot 2.....	12
Figure 16 - Rigid Dynamics Study Screenshot 3.....	13
Figure 17 - Rigid Dynamics Study Screenshot 4.....	13
Figure 18 - Rigid Dynamics Study Screenshot 5.....	14
Figure 19 - Rigid Dynamics Study Screenshot 6.....	15
Figure 20 - Steady-State Thermal Study Screenshot 1.....	15
Figure 21 - Steady-State Thermal Study Screenshot 2.....	16
Figure 22 - Steady-State Thermal Study Screenshot 3.....	16
Figure 23 - Steady-State Thermal Study Screenshot 4.....	17
Figure 24 - Static Structural Study Screenshot 1	17
Figure 25 - Static Structural Study Screenshot 2	18
Figure 26 - Static Structural Study Screenshot 3	18
Figure 27 - Static Structural Study Screenshot 4	19
Figure 28 - Static Structural Study Screenshot 5	19
Figure 29 - Static Structural Study Screenshot 5	20
Figure 30 - Static Structural Study Screenshot 6	20

1 Introduction

This report explores the principles of Finite Element Analysis (FEA) and its application in engineering design and analysis. FEA is an essential tool in modern engineering, allowing for the simulation of physical phenomena and the prediction of how materials and structures will respond to various forces, temperatures, and other environmental conditions. The report begins with the creation of a disc brake system model to demonstrate the application of these principles in a real-world context. To validate the accuracy and reliability of the simulation results, a simple beam deflection example is used, providing a basis for comparison with analytical solutions. The knowledge gained from this validation process is then applied to the disc brake system, showcasing how FEA can be leveraged to optimize designs, enhance performance, and reduce the need for costly physical prototyping.

2 Modelling and Assembly

For this study, a disc brake system was selected for analysis. The brake disc, pads, and piston were modelled using SolidWorks.

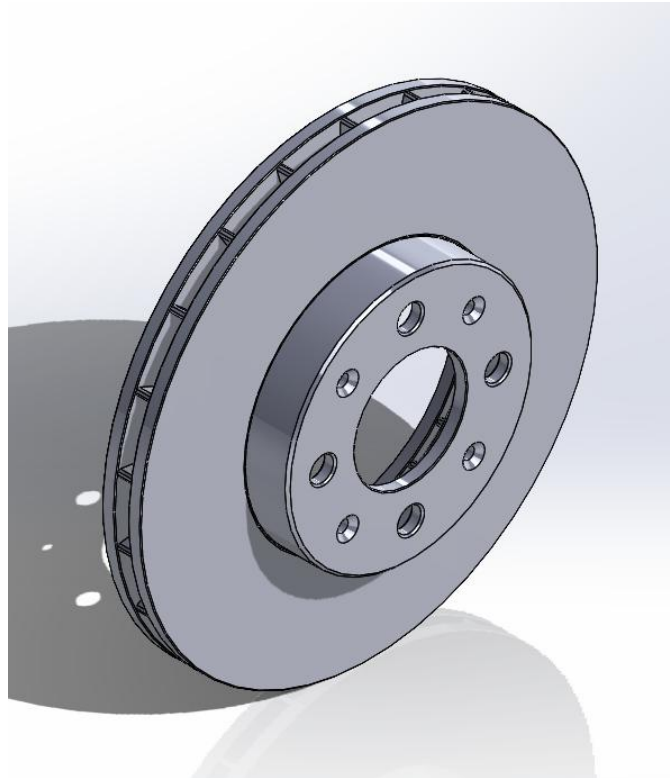


Figure 1 - Modelled Brake Disc

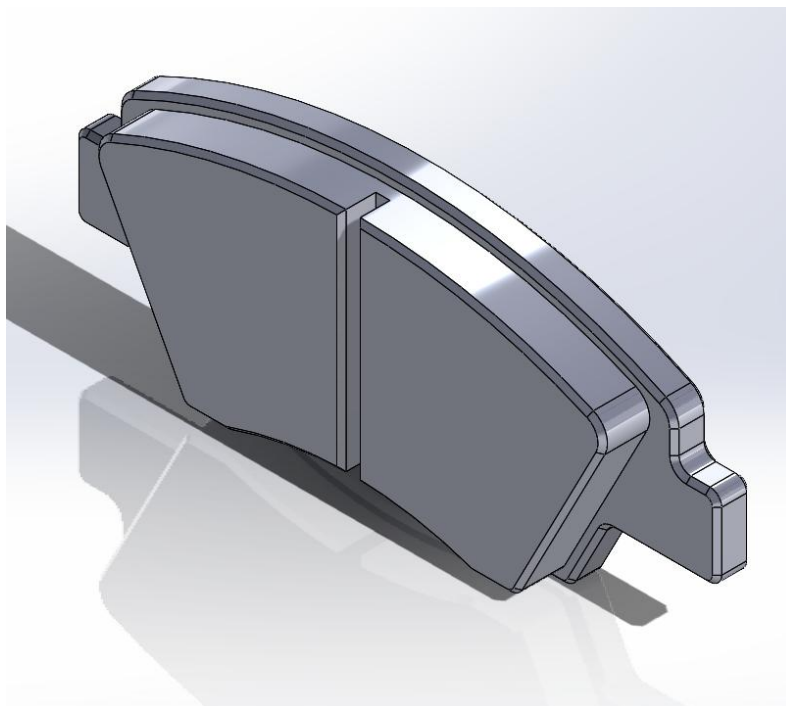


Figure 2 - Modelled Brake Pad

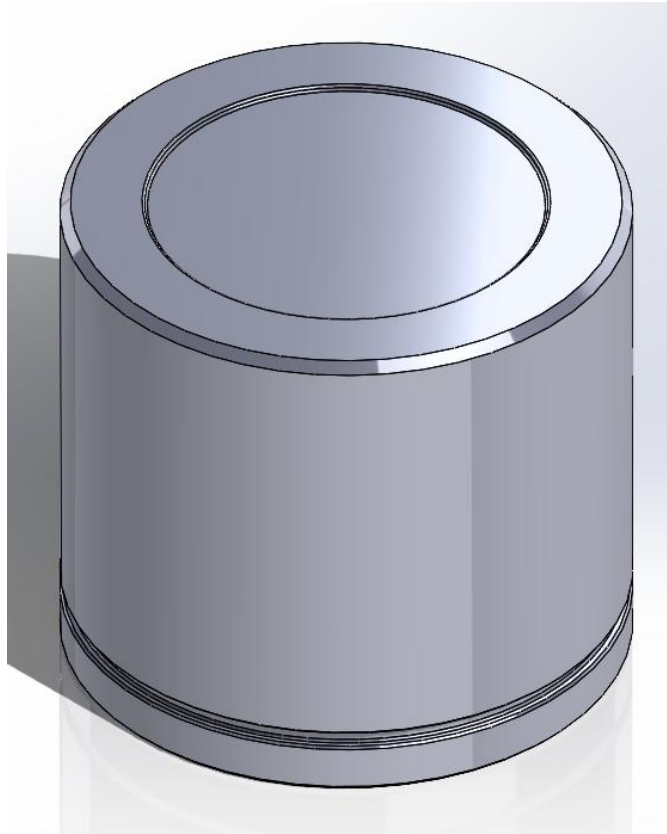


Figure 3 - Modelled Piston

The brake calliper was selected as the focus of analysis. Instead of using a finalized design, arbitrary bounding dimensions were modelled to enable topology optimization in later stages.

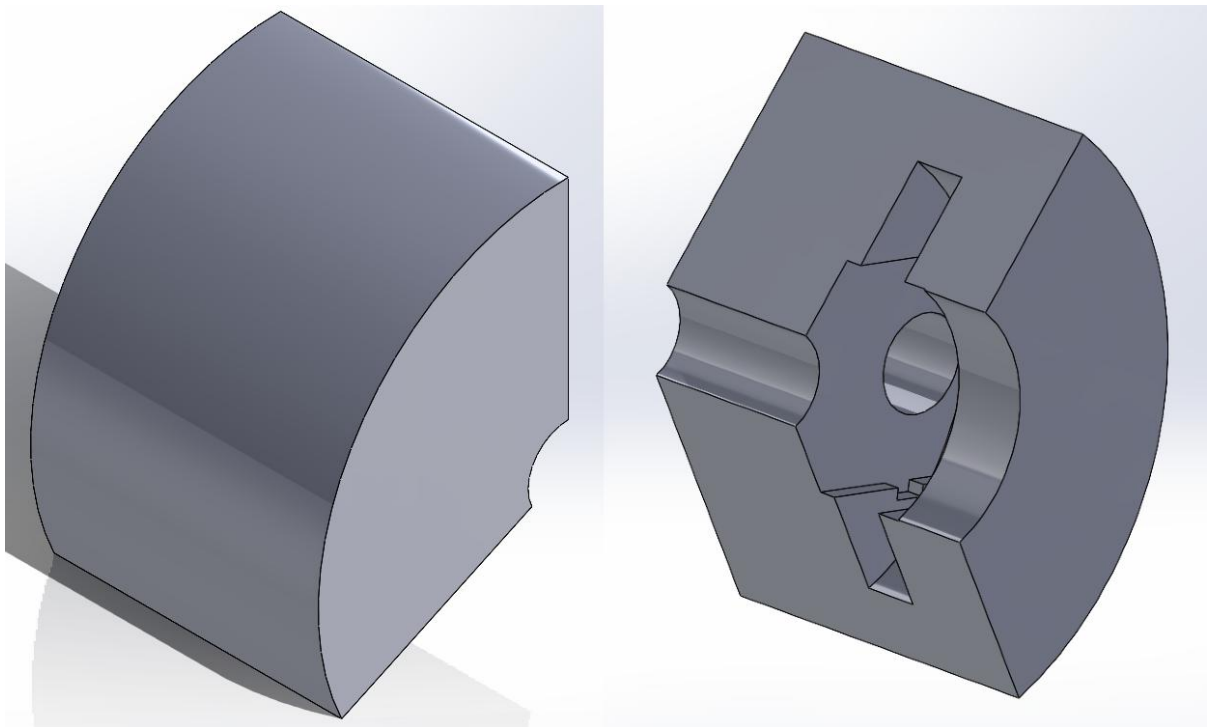


Figure 4 - Brake Calliper v1.0

All components were assembled and prepared for import into ANSYS.

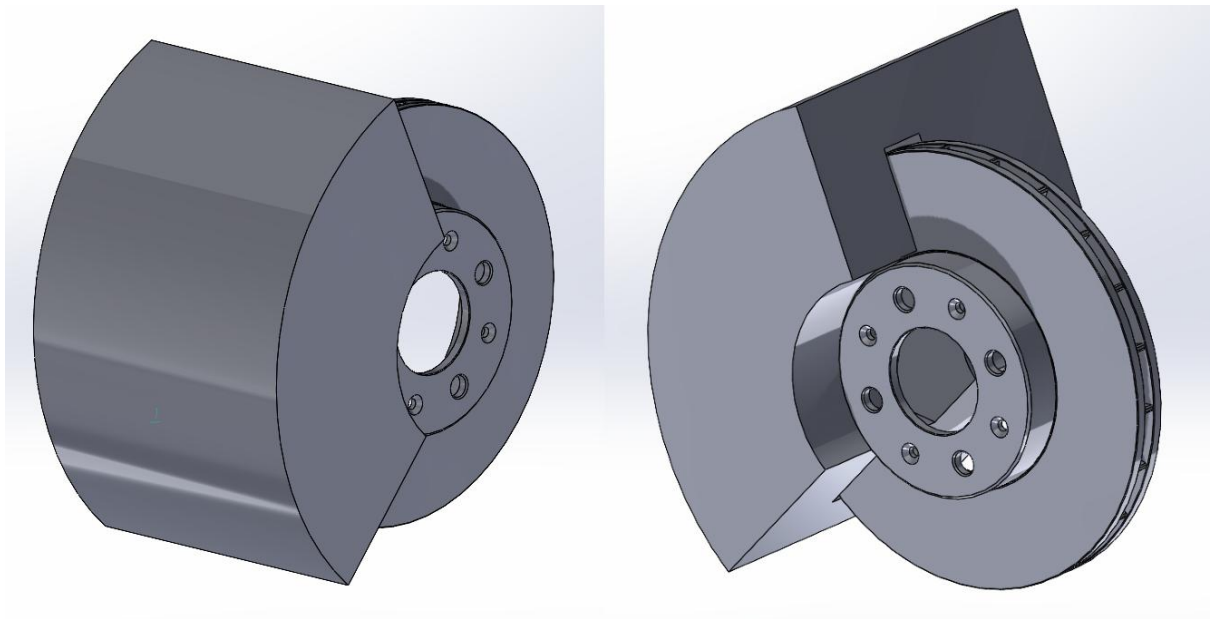


Figure 5 - All Components in Assembly

3 Validation Testing

3.1 Analytical Solution

For validation purposes, a simple cantilever beam scenario was used.

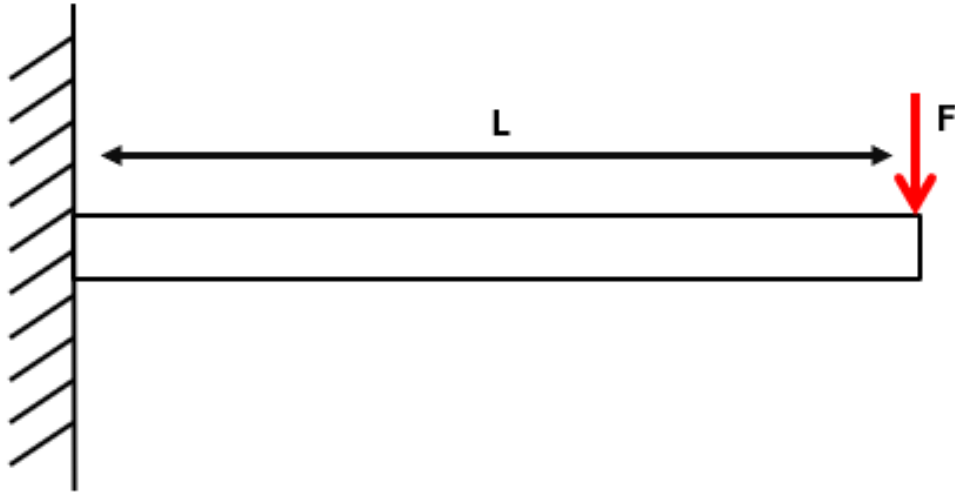


Figure 6 - Cantilever Beam Diagram

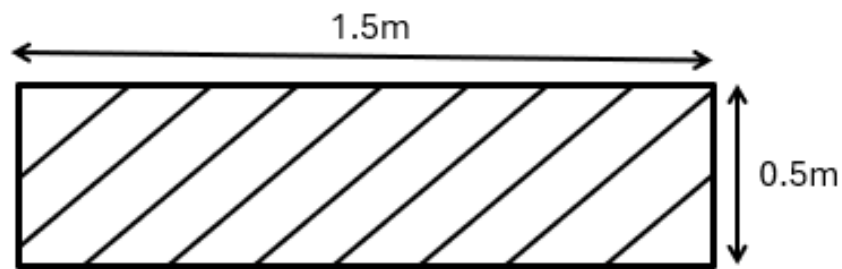


Figure 7 - Beam Cross Section

Analytically the maximum displacement can be obtained by:

$$\delta_{max} = \frac{PL^3}{3EI}$$
$$\delta_{max} = \frac{500 \times 5^3}{3 \times 20 \times 10^{11} \times 15.63 \times 10^{-3}}$$
$$\delta_{max} = 6.67 \times 10^{-7} \text{ m}$$

3.2 Software Solution

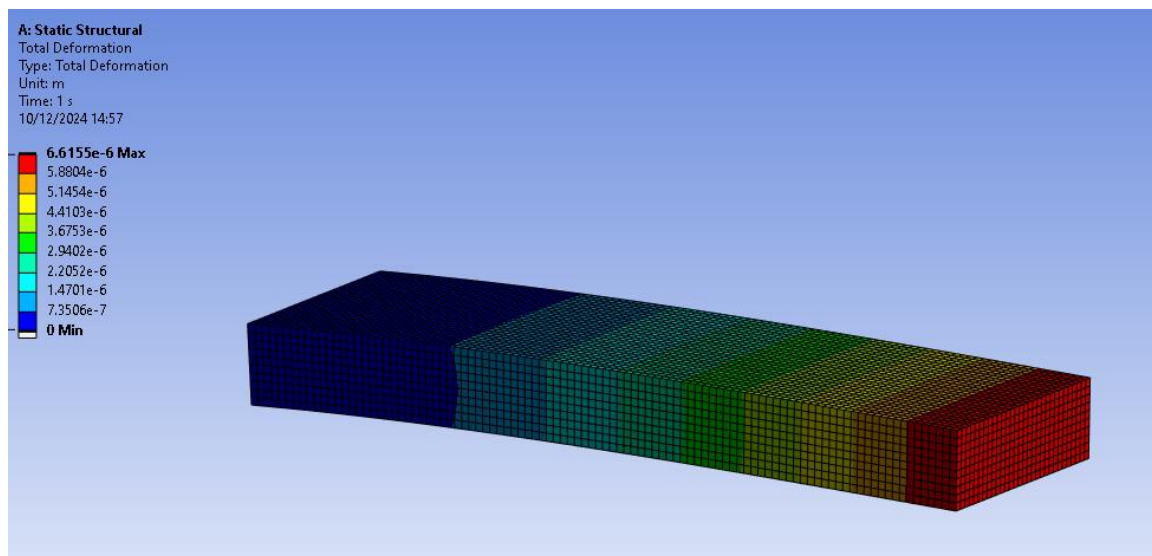


Figure 8 - Ansys Verification Solution

The results show that the analytical solution yields values ten times greater than those provided by Ansys. However, this discrepancy corresponds to only $5.953 \mu\text{m}$.

This difference arises primarily from the beam's dimensions and material properties. The combination of a relatively large cross-sectional area and the high stiffness of the 'Structural Steel' material used in Ansys ensures that the deflection remains extremely small, even under a 500 N load applied 5 meters from the fixed end.

When the applied force is increased to 10,000 N, the analytical solution predicts a deflection of 1.33×10^{-5} m, while Ansys provides the following result:

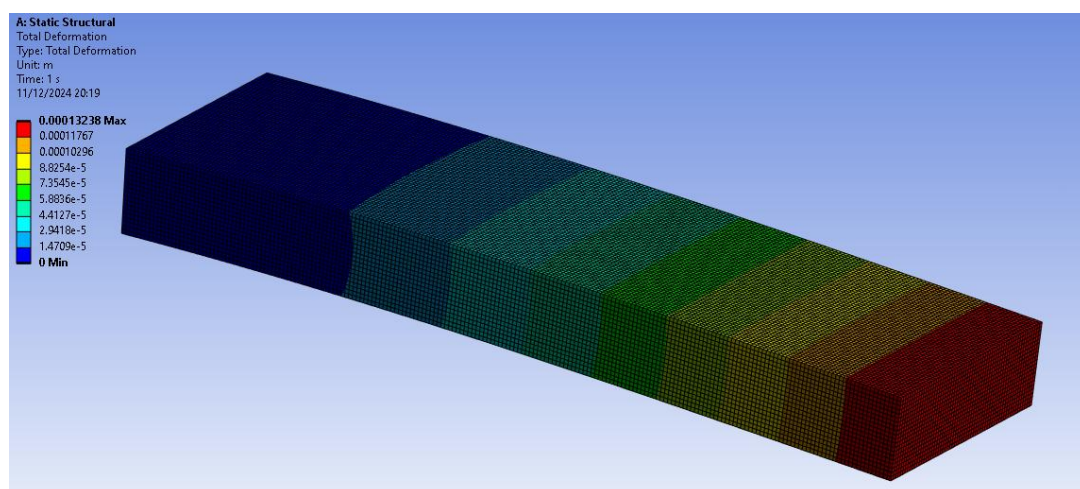


Figure 9 - Ansys Verification Solution v2

The simulation, after adjustments, now predicts a result of 1.32×10^{-5} – 51.32×10^{-5} m. This represents an error of approximately 0.75%, which is well within the standard deviations typically observed between analytical and simulation solutions.

3.3 Mesh Independence Study

In Figure 9, a mesh size of 0.025m was initially used. To assess the influence of mesh sizing on the results, the solver was rerun using a coarser mesh with a size of 0.1m and a finer mesh with a size of 0.001m.

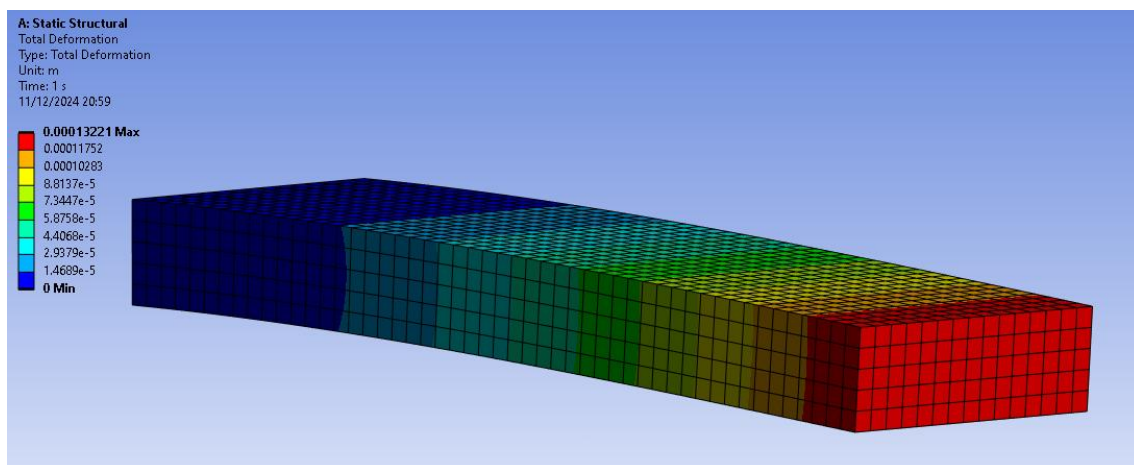


Figure 10 - 'Coarse' Mesh Solution

In this case, the coarser mesh size of 0.1m resulted in a deflection of 1.3221×10^{-5} m, while the initial mesh size of 0.025m produced 1.3238×10^{-5} m. The minor discrepancy of approximately 0.13% indicates that the coarser mesh is still able to capture the primary behaviour of the system. This aligns with best practices in FEA, where a balance is sought between computational efficiency and result accuracy. Mesh refinement studies, such as this one, are critical for identifying an optimal mesh size that ensures both precision and feasibility. (Plevris, et al, 2018)

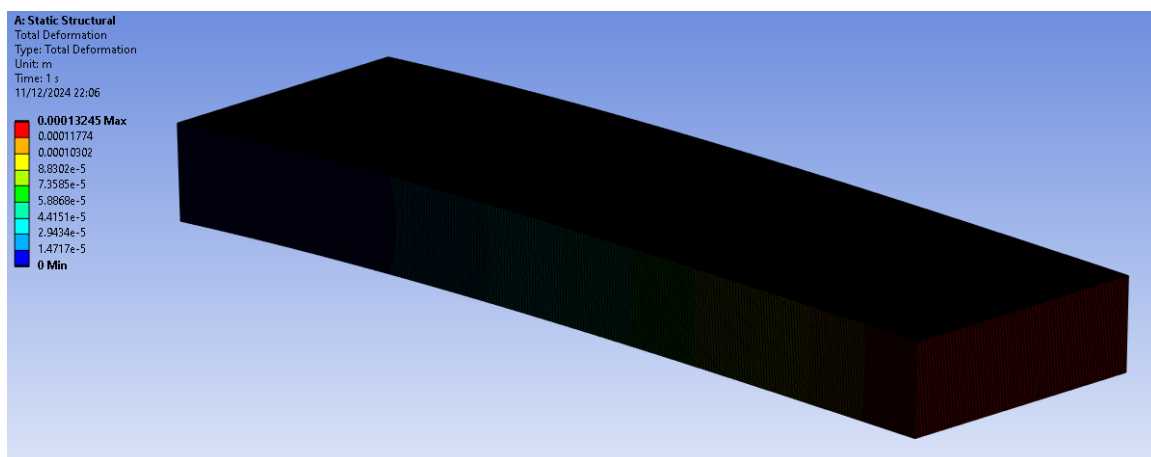


Figure 11 - 'Fine' Mesh Solution

With the fine mesh size of 0.0001m, the solver produced a result of 1.3245×10^{-5} m. This is very close to the results from the initial (0.025m) and coarser (0.1m) meshes, which gave 1.3238×10^{-5} m and 1.3221×10^{-5} m, respectively. The difference between the fine mesh and the coarser meshes is minimal (approximately 0.08% and 0.18%, respectively), which suggests that the results are converging, indicating that further mesh refinement might not significantly improve the accuracy for this particular model.

3.4 Boundary Condition Influence

Adequate selection and application of boundary conditions is essential in FEA studies to ensure that the simulation accurately represents the physical behaviour of the system. Inappropriate or oversimplified boundary conditions can lead to significant errors, as they directly influence the displacement, stress, and strain distributions calculated in the model. By accurately reflecting the real-world constraints and interactions, boundary conditions provide the framework for a valid and reliable analysis. (Moga, et al, 2024)

The following boundary conditions were applied for this simulation:

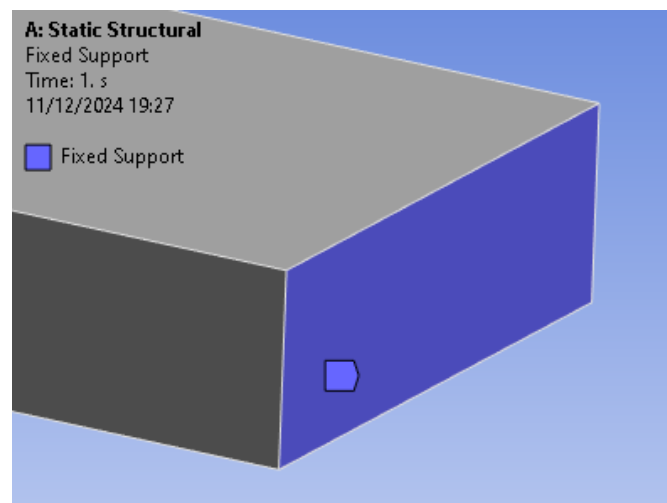


Figure 12 - Fixed Support Beam Simulation

A fixed support was applied to the fixed end of the beam, specifically constraining the side face of the beam. While using a fixed support was appropriate for this simulation, such conditions are rarely representative of real-world scenarios. In practical applications, completely fixed faces are uncommon, as there is typically some form of fixing or an interface with other components, such as bolted joints, clamps, or connections through adhesives. These interactions introduce additional complexities and deformations that may not be fully captured by assuming a rigidly fixed face.

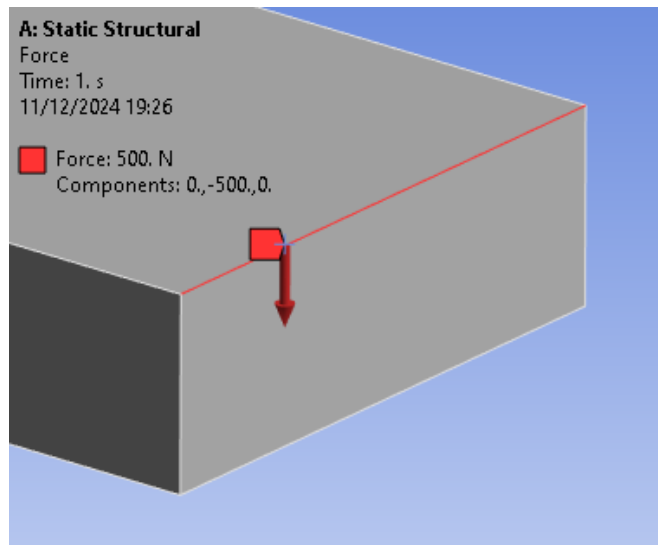


Figure 13 - Force Applied Beam Simulation

In this simulation, the force was applied in Ansys to the edge opposite the fixed support. The direction of the force was set perpendicular to this edge to simulate the desired loading condition effectively. While there are alternative methods to apply this load, no other approach was identified that could distribute the force evenly along the edge. This uniform distribution was necessary to achieve a deflection equivalent to the analytical solution, ensuring consistency between the simulation and theoretical predictions.

4 Kinematic Simulation and Structural Analysis

4.1 Rigid Dynamics

The initial step involved importing the relevant Engineering Data into the system. Stainless steel, structural steel and brake friction material were imported for use later.

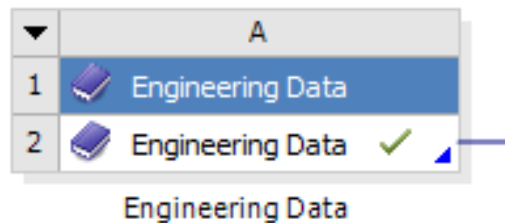


Figure 14 - Rigid Dynamics Study Screenshot 1

Subsequently, a Rigid Dynamics module was incorporated into the system, and the assembly geometry was imported.

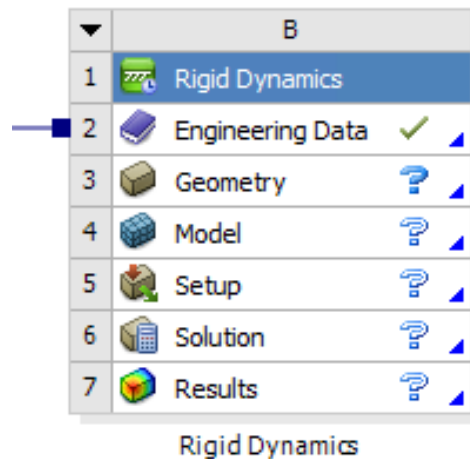


Figure 15 - Rigid Dynamics Study Screenshot 2

Materials were assigned to the individual components, with the 'Stiffness Behaviour' option set to rigid. This approach was chosen to better understand how the forces dynamically influence the system.

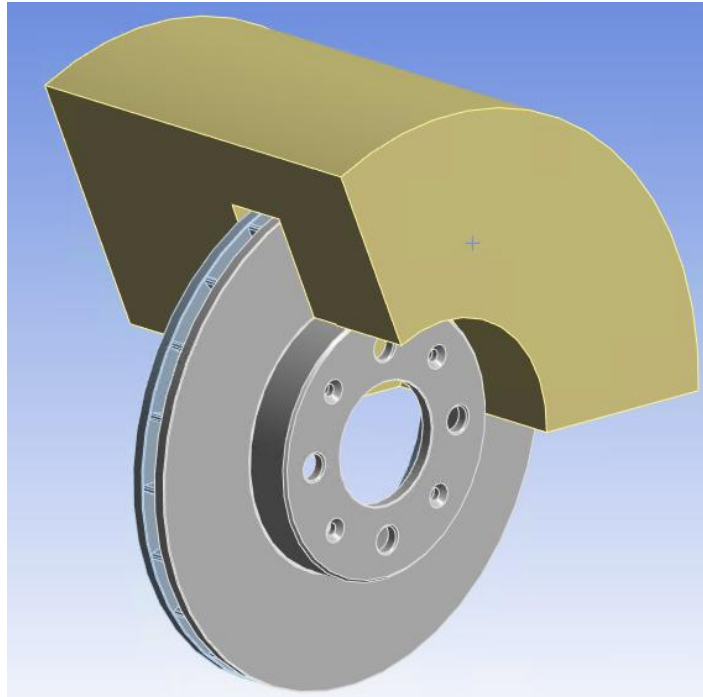


Figure 16 - Rigid Dynamics Study Screenshot 3

Due to the way the assembly was imported into ANSYS, the disc was divided into two parts. To prevent independent movement between these parts, a Fixed connection was applied. Translational joints were also used to model the floating action of the brake caliper perpendicular to the surface of the disc.

Scope	
Scoping Method	Geometry Selection
Contact	1 Face
Target	2 Faces
Contact Bodies	brakedisc Boss-Extrude1
Target Bodies	brakepad Chamfer7[2]
Protected	No
Definition	
Type	Forced Frictional Sliding
☒ Friction Coefficient	0.4
Advanced	

Figure 17 - Rigid Dynamics Study Screenshot 4

To simulate the interaction between the brake pads and the disc, a Force Frictional contact was implemented with a coefficient of friction of 0.4, typical for a disc brake system.

A 350 N force was applied to the piston-to-caliper joint to represent the braking pressure. Additionally, a joint probe was placed at the opposite brake pad-to-caliper joint to measure the forces transmitted through the caliper during the actuation of the braking system.

Upon analysing the results, it became apparent that there was an issue with the connection setup, as the maximum force calculated in the axis perpendicular to the friction surfaces was an unreasonably low value of $1.98 \times 10^{-29} \text{ N}$.

This discrepancy could be attributed to boundary conditions that did not allow for representative movement, as well as the absence of reaction forces typically exerted by hydraulic fluid on the piston chamber.

Given these limitations, the current setup was deemed unviable. Consequently, the analysis will proceed using the example assembly provided during class for simplicity.

With the updated assembly plan, revolute joints were implemented for the crank-to-rod, rod-to-pin, and pin-to-piston interfaces. Additionally, body-to-ground joints were applied to both the crank and the piston constraining their degrees of freedom, restricting the crank to rotation along its own axis and the piston to linear translation along its axis, thereby replicating their behaviour within an engine block.

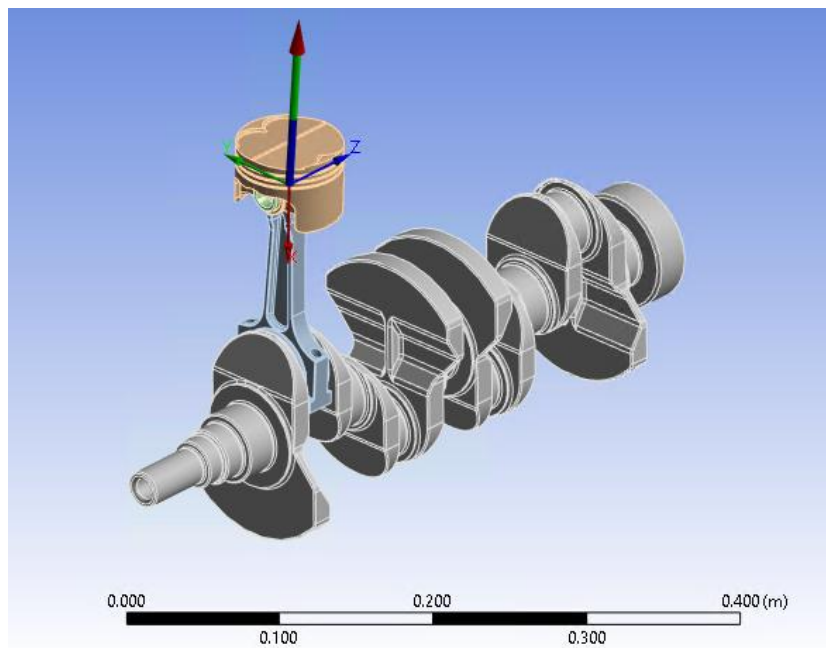


Figure 18 - Rigid Dynamics Study Screenshot 5

A rotational velocity of 4000 RPM was applied in the simulation, and the highest load point in the loading profile (illustrated in Figure 19) was identified and exported for subsequent use in the static

structural study. Since the static structural analysis can only examine a single point from the transient simulation, the worst-case scenario was chosen for this purpose.

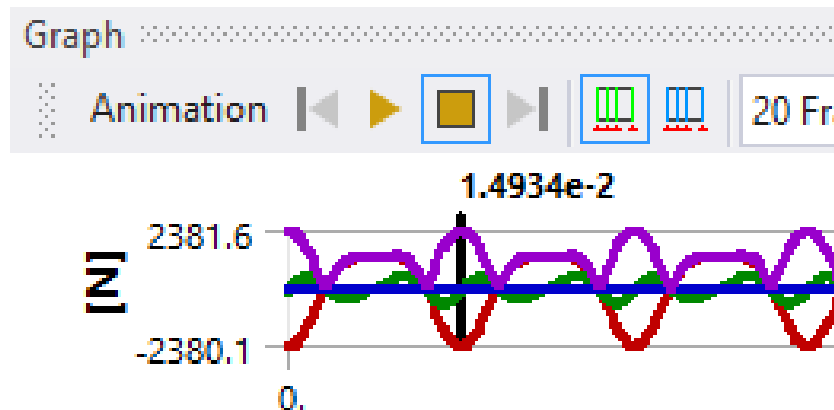


Figure 19 - Rigid Dynamics Study Screenshot 6

4.2 Steady-State Thermal

The steady-state thermal study was added to the system bringing in the engineering data and geometry from the previous study.

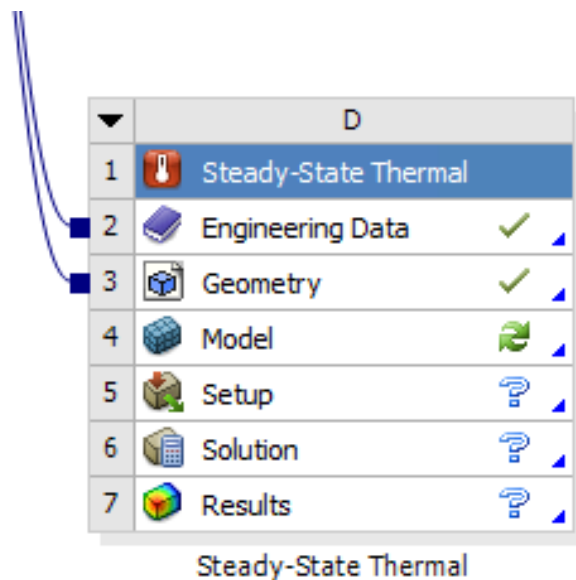


Figure 20 - Steady-State Thermal Study Screenshot 1

A mesh with a 4 mm element size was subsequently generated for the geometry. This mesh size was selected to balance computational efficiency with the level of detail required for accurate results, ensuring sufficient resolution to capture the thermal and structural behaviour of the components.

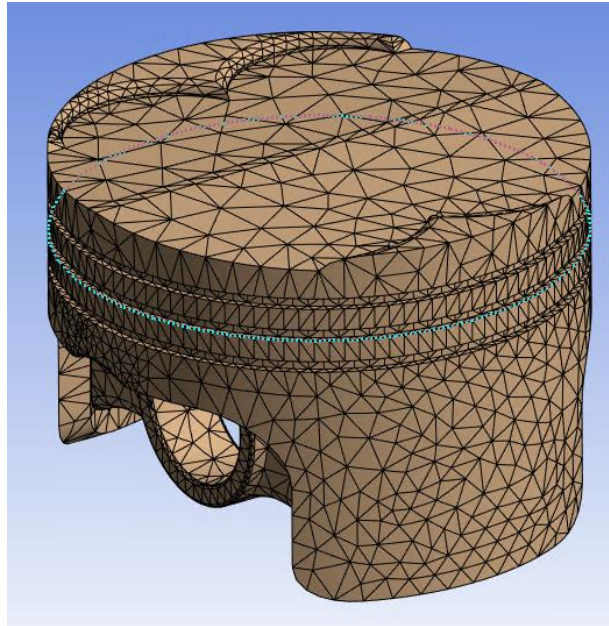


Figure 21 - Steady-State Thermal Study Screenshot 2

Six convection elements were added to the study to simulate heat transfer at critical surfaces of the piston. These elements detailed the film coefficient and ambient temperature for each specific region, including the top surface of the piston, the piston sides, the piston ring grooves, the oil control ring groove, the skirt, and the underside of the piston. This setup allowed for a realistic representation of the thermal boundary conditions, capturing the heat dissipation effects across different areas of the piston exposed to varying thermal environments.

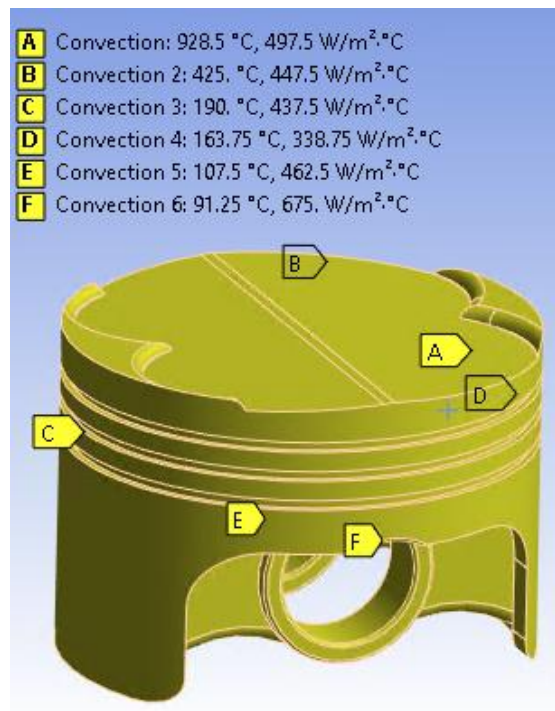


Figure 22 - Steady-State Thermal Study Screenshot 3

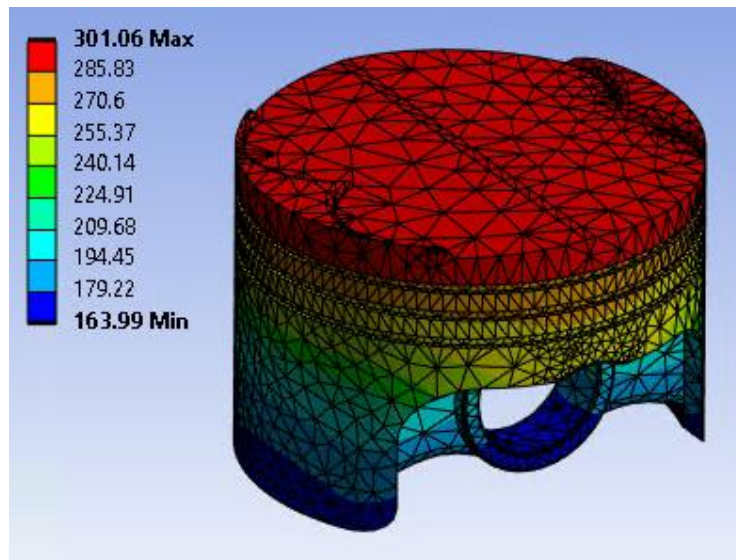


Figure 23 - Steady-State Thermal Study Screenshot 4

Temperature profiles were obtained from the analysis, as shown in Figure 23. The crown of the piston, which is in direct contact with the combustion gases, exhibited temperatures of approximately 300°C. In contrast, the lower regions of the piston, such as the skirt and underside, experienced much lower temperatures, averaging around 160°C. These results are consistent with expected thermal behaviour in a piston, where the crown endures higher temperatures due to direct exposure to combustion, while the lower sections remain relatively cooler.

4.3 Static Structural Study

The first step in the static structural study is to suppress all other bodies in the assembly, as the focus is solely on the piston. The piston is then set to "flexible" to enable the analysis of its deformation under the applied load.

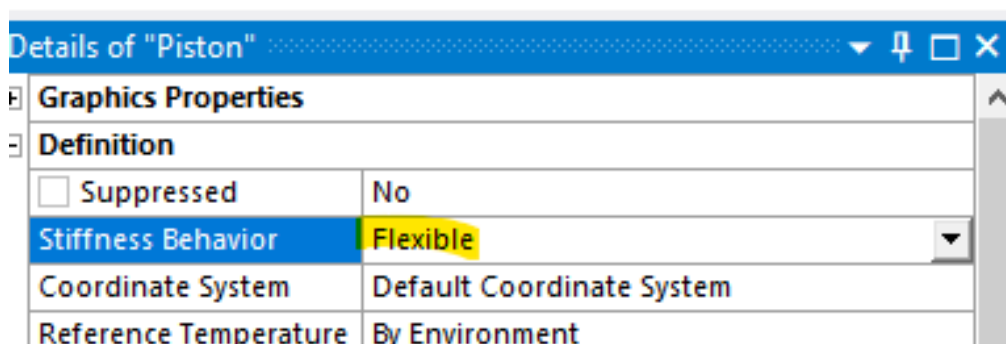


Figure 24 - Static Structural Study Screenshot 1

The motion loads, as shown in Figure 25, were subsequently imported into the analysis.

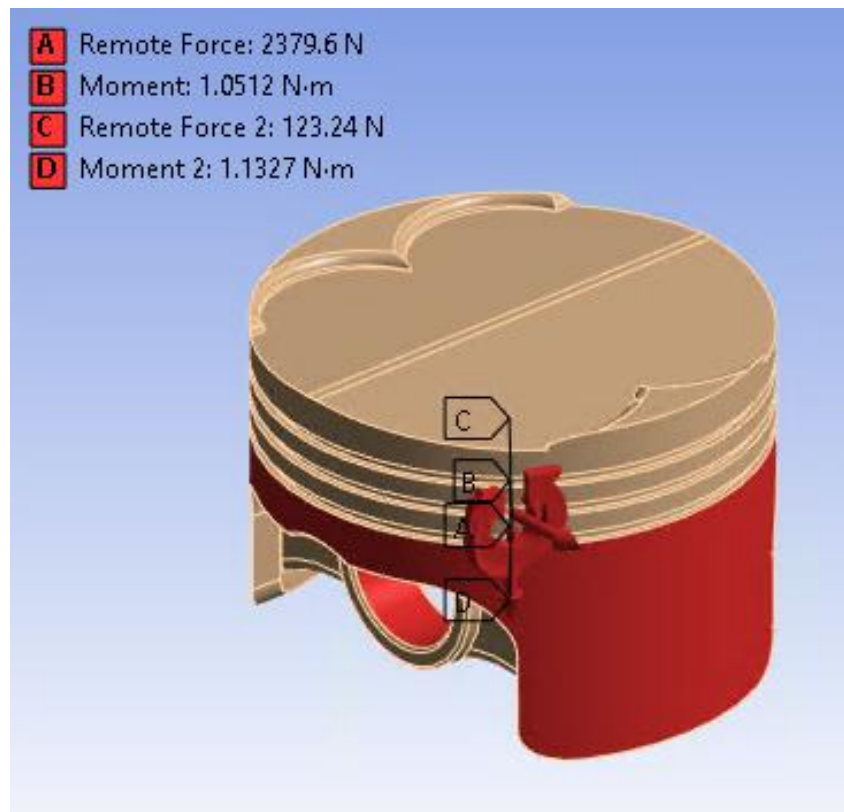


Figure 25 - Static Structural Study Screenshot 2

The weak springs system was then applied to stabilize the model. This was necessary due to the under-constrained nature of the piston, which was isolated in the simulation and could have resulted in uncontrolled movement or convergence issues. Conversely, applying fixed constraints at this stage would have influenced the final deformation results, so weak springs were used instead to provide minimal stiffness and maintain stability without affecting the accuracy of the deformation analysis.

Auto Time Stepping	Program Controlled
Solver Controls	
Solver Type	Program Controlled
Weak Springs	On
Spring Stiffness	Program Controlled
Solver Pivot Checking	Program Controlled

Figure 26 - Static Structural Study Screenshot 3

A uniform pressure of 105 bar was applied to the top surfaces of the piston, representing the forces exerted by the combustion gases during engine operation.

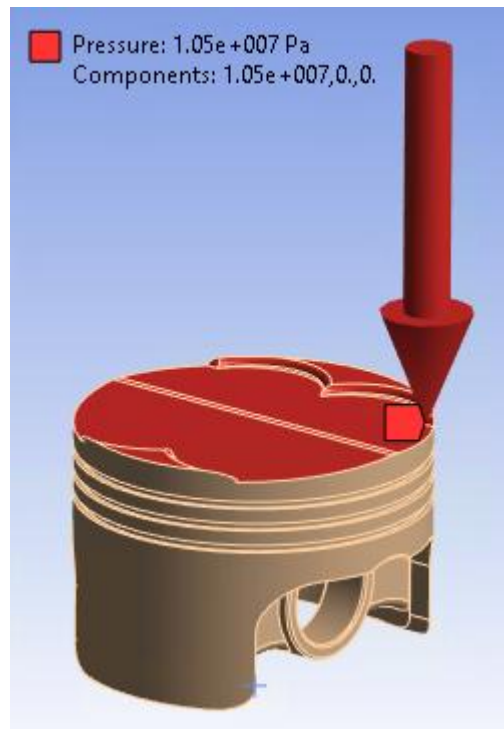


Figure 27 - Static Structural Study Screenshot 4

While running the solution, several error messages were encountered that could not be resolved despite attempts to troubleshoot. These errors were likely due to issues with the setup or constraints applied during the simulation. Nevertheless, the simulation was completed, and the results obtained were as follows:

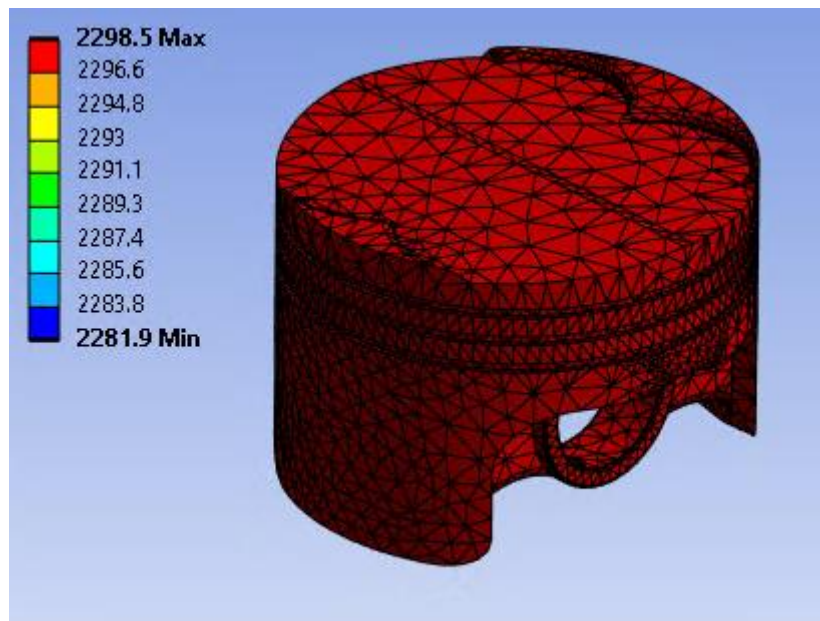


Figure 28 - Static Structural Study Screenshot 5

Figure 28 illustrates the total deformation calculated by the simulation, which is shown to be nearly 3000 meters, or 3 km. This result is clearly implausible, as such a significant deformation would imply that the piston is moving far beyond its expected range of motion. This unrealistic outcome suggests that there may be issues with the model setup, such as incorrect boundary conditions or inappropriate material properties, which are causing exaggerated deformation values that do not reflect the true behaviour of the piston under typical operating conditions.

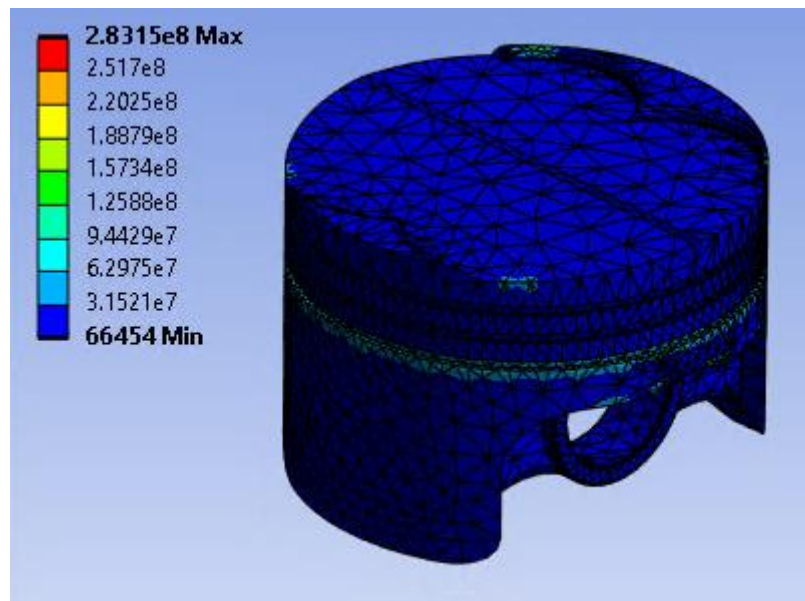


Figure 29 - Static Structural Study Screenshot 5

Similarly, the Von-Mises stress values obtained from the simulation are clearly incorrect.

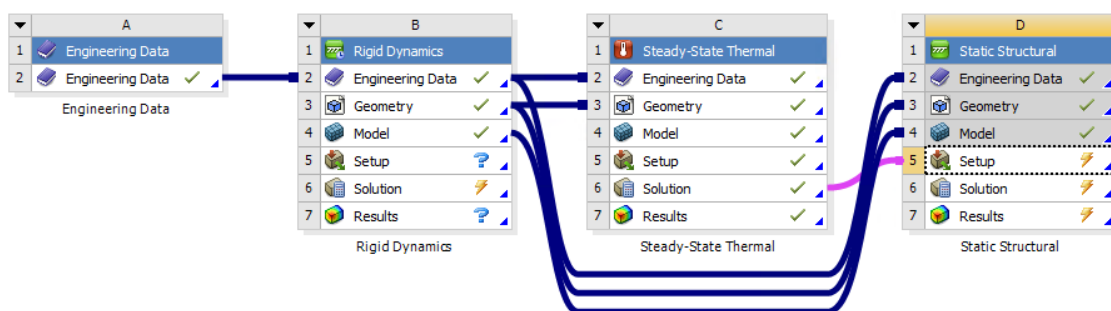


Figure 30 - Static Structural Study Screenshot 6

Figure 30 presents the final system setup, showcasing the fully configured system along with all of its dependencies.

5 Conclusions

In conclusion, this study has provided valuable insights into the principles and applications of Finite Element Analysis (FEA). Although the initial goal was to model and analyse a disc brake system, complications related to the boundary conditions hindered a successful simulation. As a result, the analysis shifted to the example provided in class. However, even this simplified example led to an error, with the final static structural solve being clearly incorrect. These challenges highlight the complexity of FEA and the importance of a solid understanding of the underlying principles when applying this powerful tool. While FEA is an invaluable resource in engineering, offering detailed insights into system behaviour and performance, its effectiveness relies heavily on accurate model setup, appropriate boundary conditions, and expert interpretation of results. This experience reinforces the notion that FEA is a powerful tool, but it must be employed by those with the requisite expertise to ensure reliable and meaningful outcomes.

6 References

Moga, R. A., Olteanu, C.D. and Delean, A.G. (2024). The Importance of Boundary Conditions and Failure Criterion in Finite Element Analysis Accuracy—A Comparative Assessment of Periodontal Ligament Biomechanical Behavior. *Applied Sciences*, [online] 14(8), p.3370. doi:<https://doi.org/10.3390/app14083370>.

Plevris, V. and Markeset, G. (2018). Educational Challenges in Computer-based Finite Element Analysis and Design of Structures. *Journal of Computer Science*, 14(10), pp.1351–1362. doi:<https://doi.org/10.3844/jcssp.2018.1351.1362>.