



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

University of Wales Trinity St David – School of Engineering

ACMA6010 – Structural & Fluid Analysis
RHYS TREMELLING (2102909)

Contents

1.0	Introduction	4
2.0	Simulation	4
2.1	Kinematic Analysis.....	4
2.2	Thermal Analysis	5
2.3	Static Structural Analysis	6
2.3.1	Total Deformation	6
2.3.2	Equivalent Von-Mises Stress	7
3.0	Design Improvement.....	9
3.1	Material Selection and Optimisation	9
3.1	Topology Optimisation	11
3.2	Simulation of Topology Optimised Piston	13
3.2.1	Total Deformation Analysis.....	13
3.2.2	Equivalent Stress Analysis	15
4.0	Validation Testing	16
4.1	Elastic Bending - Continuous Load	16
4.1.1	Analytical Analysis	16
4.1.2	Simulation Analysis.....	18
4.1.3	Comparison	18
4.1.4	Mesh Independence Study	19
4.2	Bimetallic Strip	20
4.2.1	Analytical Analysis	20
4.2.2	Simulation Analysis.....	21
4.2.3	Comparison	21
4.2.4	Mesh Independence Study	22
4.3	Boundary Condition Influence	23

Figures

Figure 1 Assembly of kinematic Analysis.....	4
Figure 2 Temperature distribution on the top of the piston using aluminium alloy	5
Figure 3 Temperature distribution on the bottom of the piston using aluminium alloy	6
Figure 4 Results of the thermal simulation using aluminium alloy	6
Figure 5 Total deformation of the top of the piston using aluminium alloy	6
Figure 6 Total deformation of the bottom of the piston using aluminium alloy	7
Figure 7 Total deformation results of the static structural study using aluminium alloy.....	7
Figure 8 Equivalent Von-Mises Stress of the top of the piston using aluminium alloy	7
Figure 9 Equivalent Von-Mises Stress of the bottom of the piston using aluminium alloy	8
Figure 10 Equivalent Von-Mises Stress results of the static structural study using aluminium alloy	8
Figure 11 Total deformation of the top of the piston using aluminium-silicon carbide composite material.....	9
Figure 12 Total deformation of the bottom of the piston using aluminium-silicon carbide composite material.....	10
Figure 13 Total deformation results of the static structural study using aluminium-silicon carbide composite material	10
Figure 14 Equivalent Von-Mises Stress of the top of the piston using aluminium-silicon carbide composite material	10
Figure 15 Equivalent Von-Mises Stress of the bottom of the piston using aluminium-silicon carbide composite material	11
Figure 16 Equivalent Von-Mises Stress results of the static structural study using aluminium-silicon carbide composite material	11
Figure 17 Top of the topology optimised piston	12
Figure 18 Bottom of the topology optimised piston	12
Figure 19 Closer view of the bottom of the topology optimised piston	12
Figure 20 Results of the topology optimisation of the piston.....	13
Figure 21 A view of the material removed from the topology optimisation of the piston	13
Figure 22 Total deformation of the top of the topology optimised piston	14
Figure 23 Total deformation of the bottom of the topology optimised piston	14
Figure 24 Total deformation results of the static structural study using topology optimised piston ...	14
Figure 25 Equivalent Von-Mises Stress of the top of the topology optimised piston	15
Figure 26 Equivalent Von-Mises Stress of the bottom of the topology optimised piston	16
Figure 27 Equivalent Von-Mises Stress results of the static structural simulation using the topology optimised piston	16
Figure 28 Mechanical simulation of the Elastic Bending - Continuous Load validation example	18
Figure 29 Graph comparing mesh element size against calculated maximum deflection for the Elastic Bending - Continuous Load validation example	19
Figure 30 Mechanical simulation of the Bimetallic Strip validation example	21
Figure 31 Graph comparing mesh element size against calculated maximum deflection for the Bimetallic Strip validation example.....	22

1.0 Introduction

This report focuses on the structural analysis of a critical engine component, the internal combustion engine piston. As the piston endures significant mechanical stresses and thermal variations during engine operation, its design and analysis are vital for ensuring performance and reliability. ANSYS Mechanical simulation software will be used to conduct this analysis. ANSYS is renowned for its finite element analysis (FEA) capabilities, enabling precise evaluation of complex mechanical systems under real-world operating conditions. This report outlines the simulation process, including kinematic, thermal, and static structural analyses, iterative design improvements, and optimization. Validation testing and a mesh independence study further verify the accuracy and robustness of the simulation results.

2.0 Simulation

2.1 Kinematic Analysis

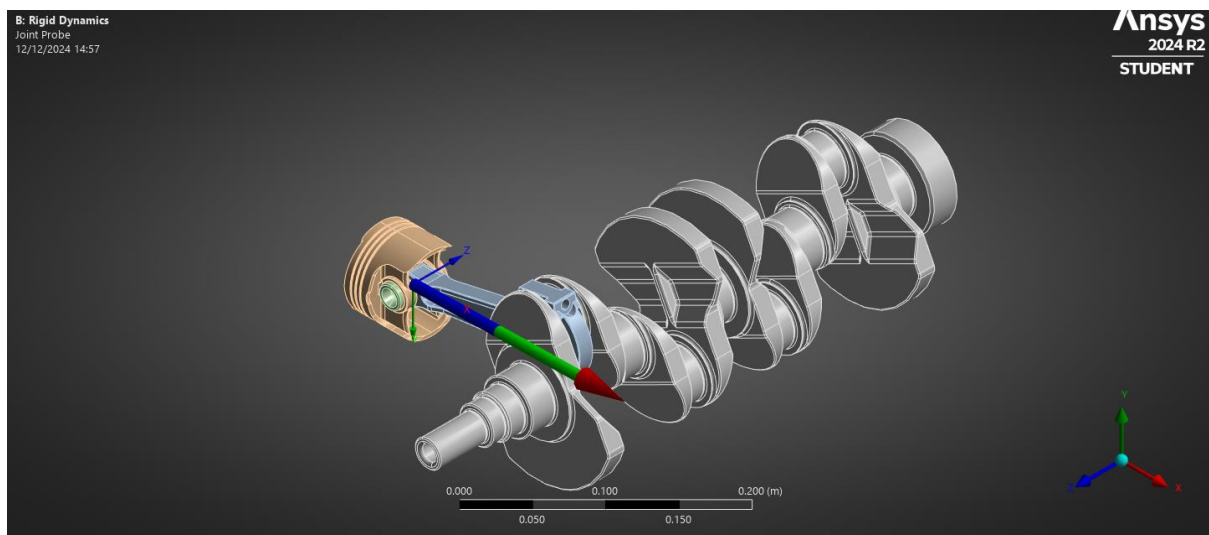


Figure 1 Assembly of kinematic Analysis

The rigid dynamics simulation analysed the internal combustion engine crankshaft under operational constraints to replicate realistic conditions. Constraints were applied to allow the connecting rod to rotate freely about the gudgeon pin, enabling the piston to rotate up against the gudgeon pin while also being fixed to move up and down, replicating the constraints of the engine block, mimicking its natural motion within the assembly. Additionally, the crankshaft was simulated to rotate at a constant speed of 4000 RPM, reflecting typical engine operating conditions. The forces acting along the X, Y, and Z axes were calculated, with the maximum force along the X-axis determined to be 2379.4. At the same time, the Y-axis experienced 600.81 N. Minimal forces were observed along the Z-axis (8.6892×10^{-14} N), resulting in a total force of 2381.1 N. These findings provide valuable insights into the dynamic behaviour of the crankshaft and associated components under high-speed rotation.

Maximum Value Over Time	
X Axis	2379.4 N
Y Axis	600.81 N
Z Axis	8.6892e-014 N
Total	2381.1 N

2.2 Thermal Analysis

The thermal simulation of the piston assembly was conducted to evaluate the temperature distribution and its effects under specified operating conditions. As outlined in the provided data, initial loading conditions included boundary constraints based on the piston material properties (6061-T6 aluminium) and thermal conductivities across different regions. The thermal conductivity varied with temperature. Specific heat capacity, Young's modulus, and the coefficient of thermal expansion were also considered to model the material's response to thermal loading accurately. The results, as summarized in the table, show a minimum temperature of 145.22°C, a maximum of 283.14°C, and an average of 227.41°C. These values correspond to the heat transfer conditions and highlight regions of significant thermal gradients. The simulation outcomes are critical for assessing thermal stresses, particularly in high-temperature zones, such as the crown and grooves, where material performance is critical to maintaining durability and reliability under operational loads.

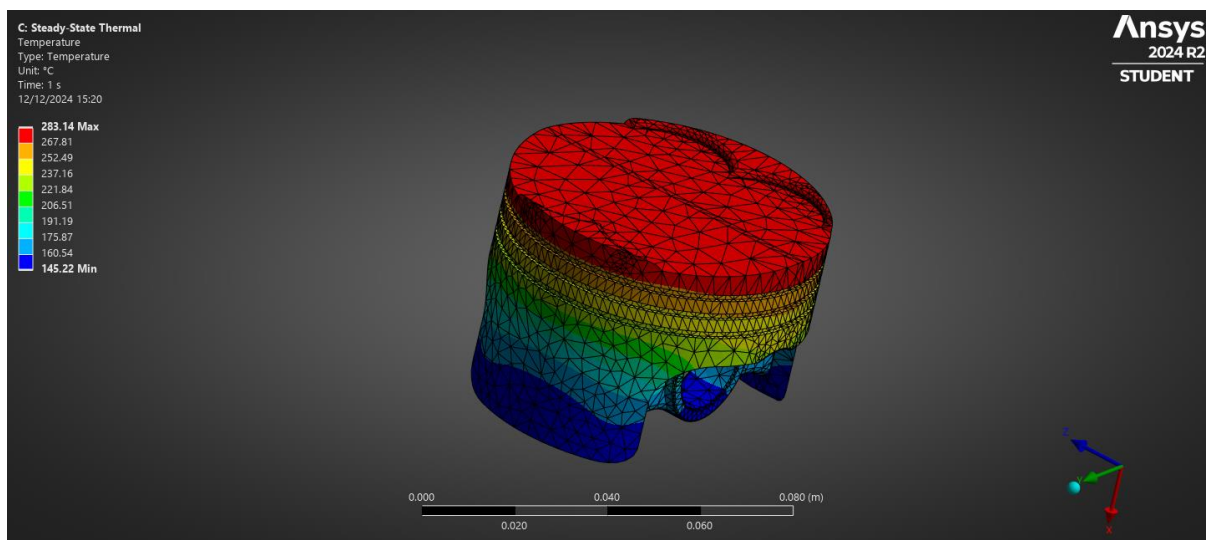


Figure 2 Temperature distribution on the top of the piston using aluminium alloy

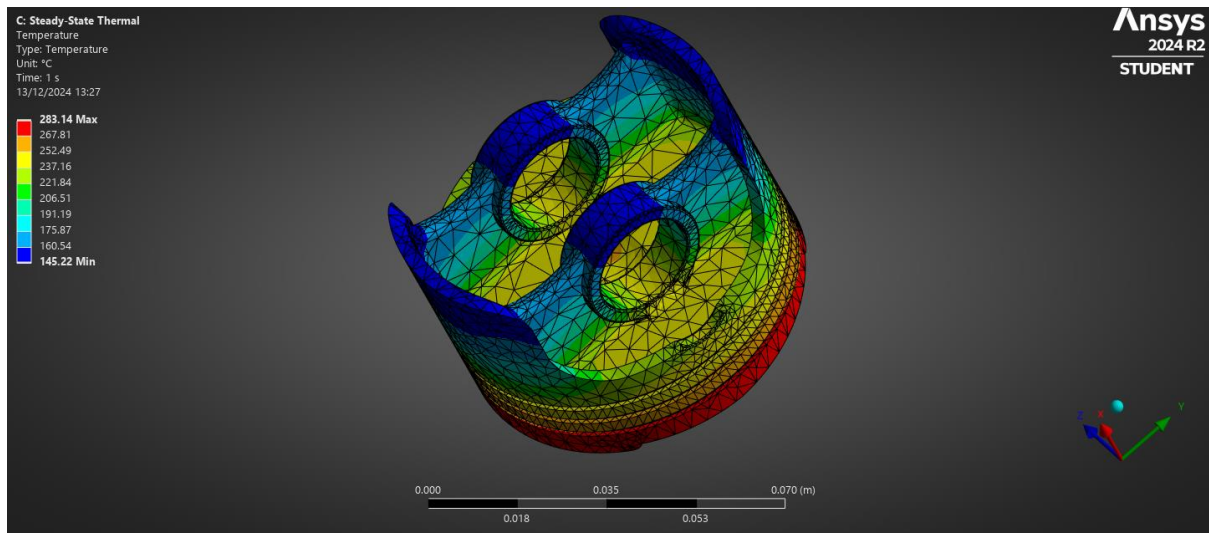


Figure 3 Temperature distribution on the bottom of the piston using aluminium alloy

Results	
Minimum	145.22 °C
Maximum	283.14 °C
Average	227.41 °C

Figure 4 Results of the thermal simulation using aluminium alloy

2.3 Static Structural Analysis

2.3.1 Total Deformation

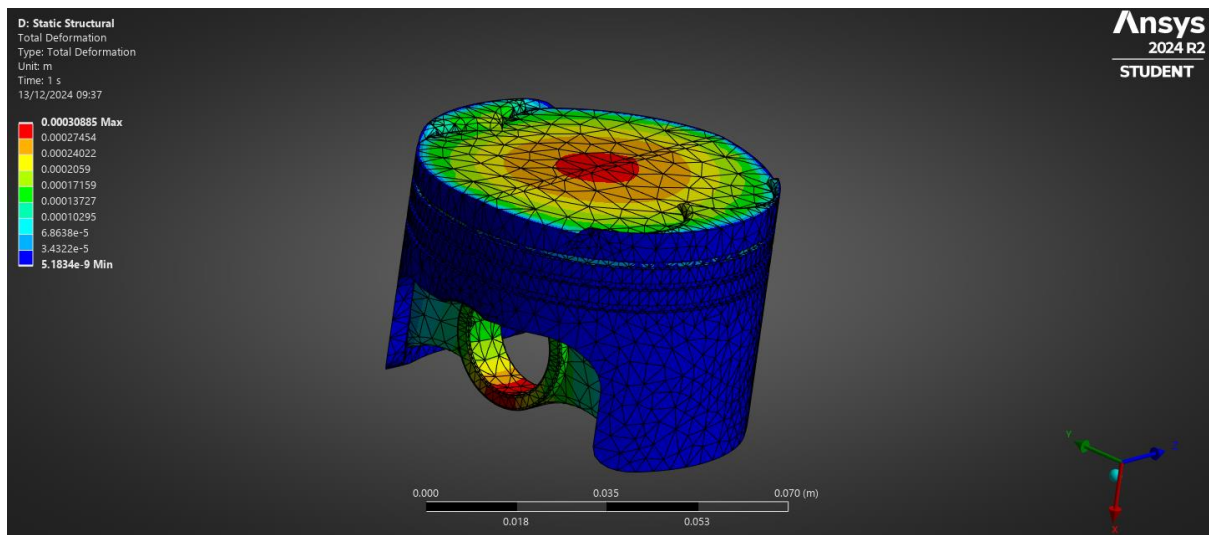


Figure 5 Total deformation of the top of the piston using aluminium alloy

The static structural analysis of the piston was conducted to evaluate its deformation and equivalent stresses under operational loads. The simulation results indicate a maximum deformation of 0.00030885 meters, while the minimum deformation was calculated as 5.1834e-9 meters. The average deformation throughout the piston was 7.0147e-5 meters, signifying minimal distortion under the applied loads. The attached deformation plots show that the highest deformation is concentrated in regions subject to the most significant stress, such as fillet areas near the piston

crown. These results highlight the structural integrity of the piston material and design, confirming its ability to withstand operational forces and ensuring the component's functionality and durability.

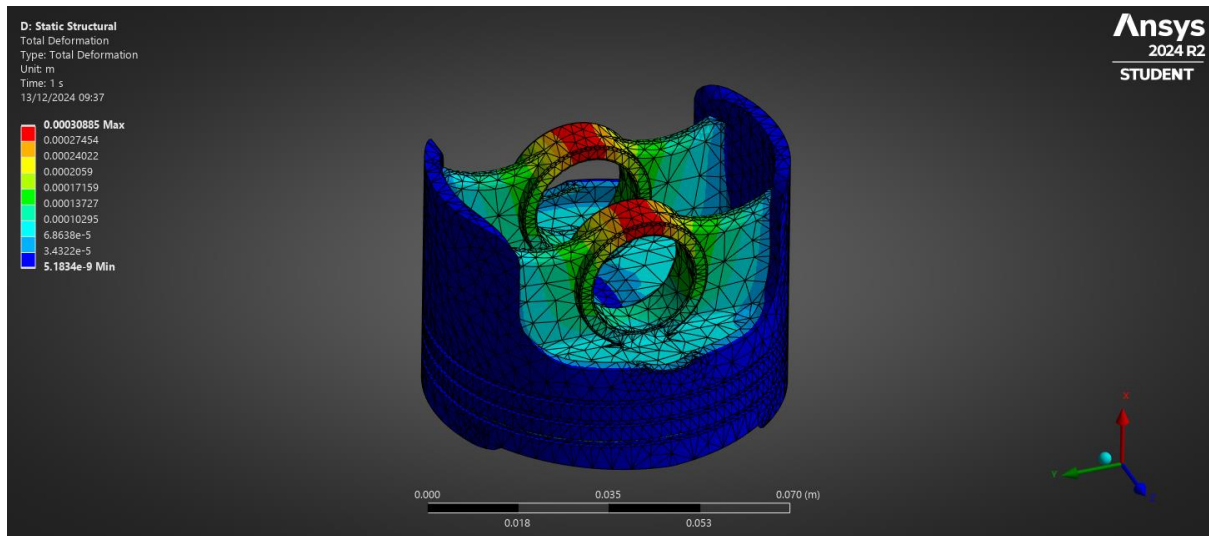


Figure 6 Total deformation of the bottom of the piston using aluminium alloy

Results	
Minimum	5.1834e-009 m
Maximum	3.0885e-004 m
Average	7.0147e-005 m

Figure 7 Total deformation results of the static structural study using aluminium alloy

2.3.2 Equivalent Von-Mises Stress

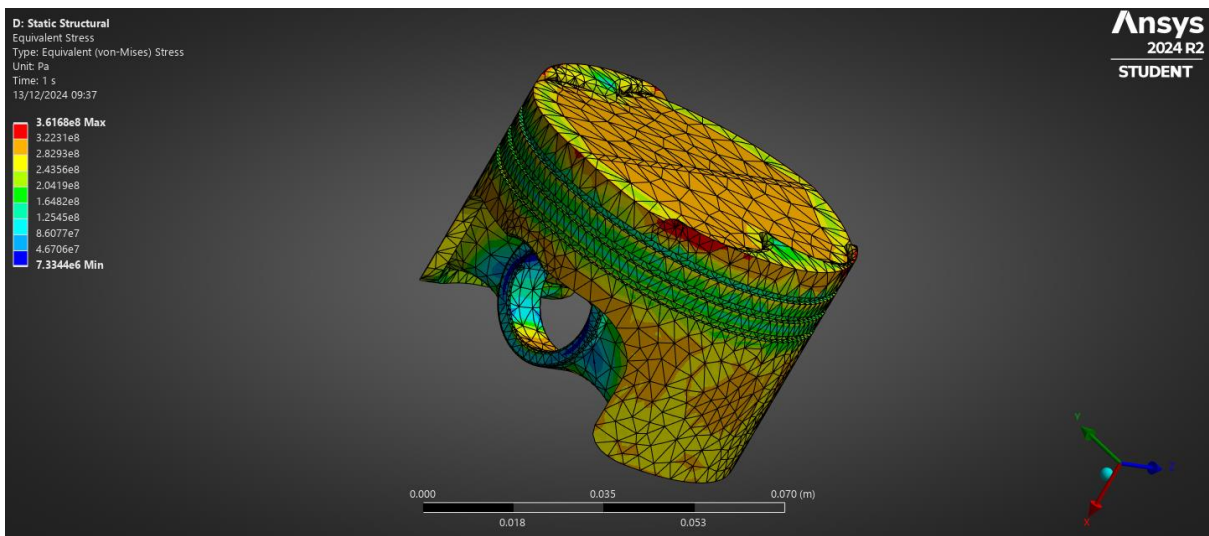


Figure 8 Equivalent Von-Mises Stress of the top of the piston using aluminium alloy

The static structural analysis for the piston's equivalent (von Mises) stress distribution has revealed critical insights into the stress concentration regions. The maximum equivalent stress was 3.6168×10^8 Pa, while the minimum stress was 7.3344×10^6 Pa, with an average stress of 2.2895×10^8 Pa. The maximum stress that occurs indicates regions of high stress due to geometric features and loading conditions. The simulation results, visualized in the stress distribution plots, show that the highest stress regions are concentrated around the piston pin bosses and the crown. Subject to the most

significant operational loads, these areas play a crucial role in stress distribution. These findings are crucial for identifying potential failure points and optimizing the piston design to enhance its durability under high-pressure and high-temperature conditions.

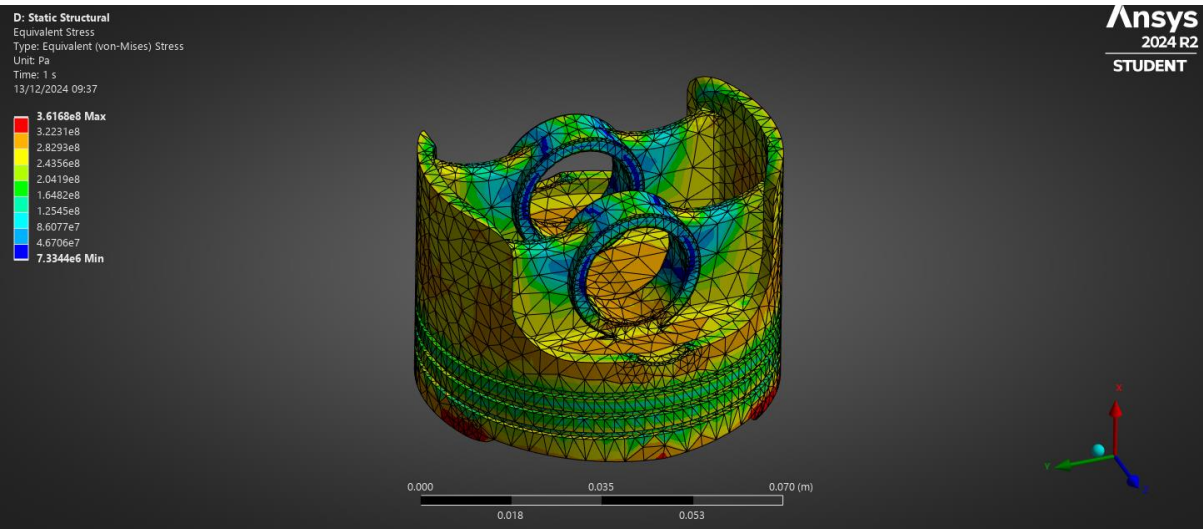


Figure 9 Equivalent Von-Mises Stress of the bottom of the piston using aluminium alloy

Results	
☐ Minimum	7.3344e+006 Pa
☐ Maximum	3.6168e+008 Pa
☐ Average	2.2895e+008 Pa

Figure 10 Equivalent Von-Mises Stress results of the static structural study using aluminium alloy

3.0 Design Improvement

3.1 Material Selection and Optimisation

A design improvement was implemented to enhance the piston's structural performance by replacing the original aluminium alloy with an aluminium-silicon carbide (Al-SiC) composite material. This composite, with its enhanced mechanical properties, is a perfect fit for high-performance and high-stress applications. Its higher stiffness improved thermal stability, and increased resistance to deformation under load make it an ideal choice. A comparative simulation was performed using the Al-SiC composite under identical loading and boundary conditions to evaluate its impact on total deformation. The simulation results unveiled a substantial reduction in deformation with the Al-SiC composite. The maximum deformation, which was 0.00030885 m with the aluminium alloy, was reduced to 0.00019548 m with the composite material, marking a remarkable 36.7% decrease. Similarly, the average deformation reduced from 7.0147×10^{-5} m to 3.754×10^{-5} m, demonstrating the composite's superior stiffness. Notably, the minimum deformation remained negligible at near-zero values, a clear indication of the material's ability to distribute stresses and uniformly minimise strain concentrations. This improvement in deformation characteristics underscores the benefits of using Al-SiC in this application where mechanical precision and durability are critical. The composite material reduces the likelihood of mechanical failure under high loads and enhances the piston's overall operational reliability and lifespan. Furthermore, its superior thermal stability mitigates the risk of deformation due to high-temperature conditions, a common challenge in internal combustion engines. These results not only validate but also strongly endorse the Al-SiC composite as a highly effective material choice, providing a clear path for further optimisation and refinement of the piston design.

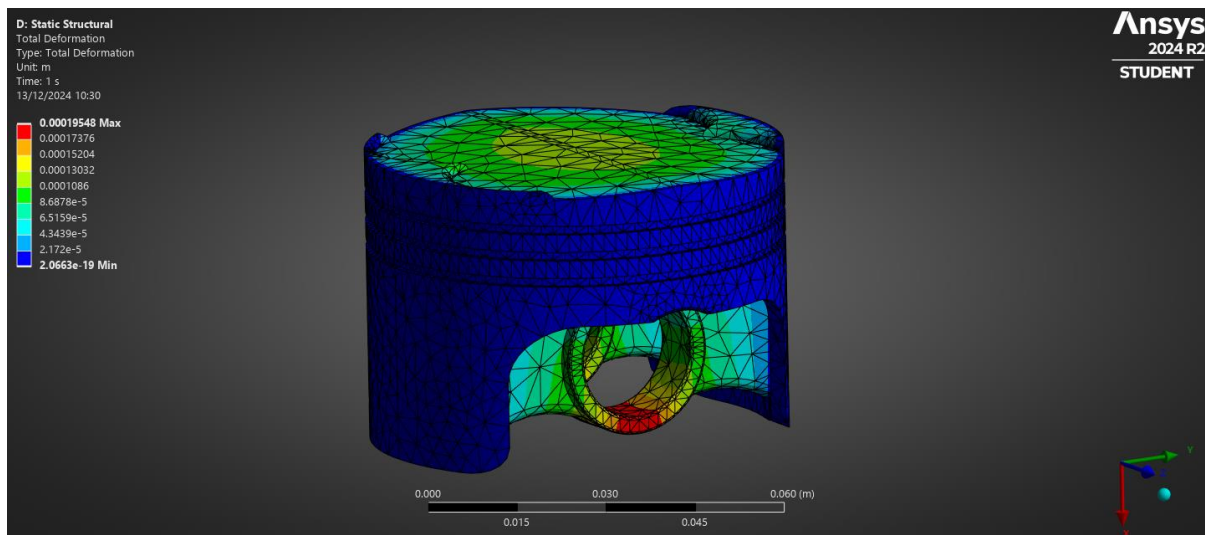


Figure 11 Total deformation of the top of the piston using aluminium-silicon carbide composite material

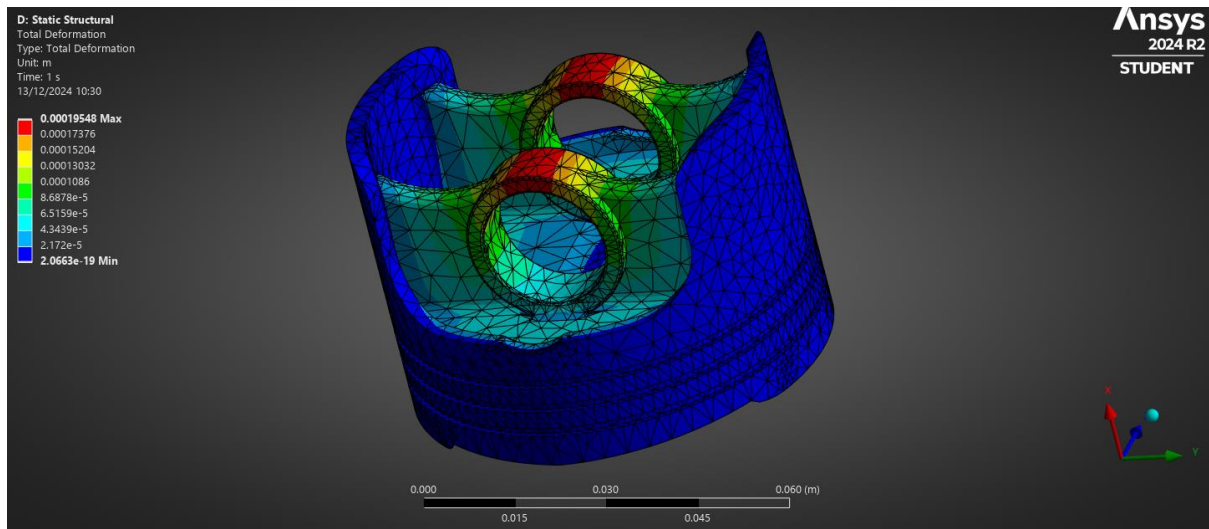


Figure 12 Total deformation of the bottom of the piston using aluminium-silicon carbide composite material

Results	
Minimum	2.0663e-019 m
Maximum	1.9548e-004 m
Average	3.754e-005 m

Figure 13 Total deformation results of the static structural study using aluminium-silicon carbide composite material

An Equivalent Von-Mises Stress simulation was conducted using the Al-SiC composite under the same loading and boundary conditions to compare its performance with the aluminium alloy. The results indicate that the maximum equivalent stress for the Al-SiC composite piston was 3.9061×10^9 Pa, slightly higher than the 3.0486×10^9 Pa recorded for the aluminium alloy. This marginal increase can be attributed to the higher stiffness of the Al-SiC composite, which leads to localized stress concentrations in critical areas such as the gudgeon pin bosses and the crown. However, the higher stress levels are well within the material's increased yield strength, ensuring the component's structural integrity under operational loads.

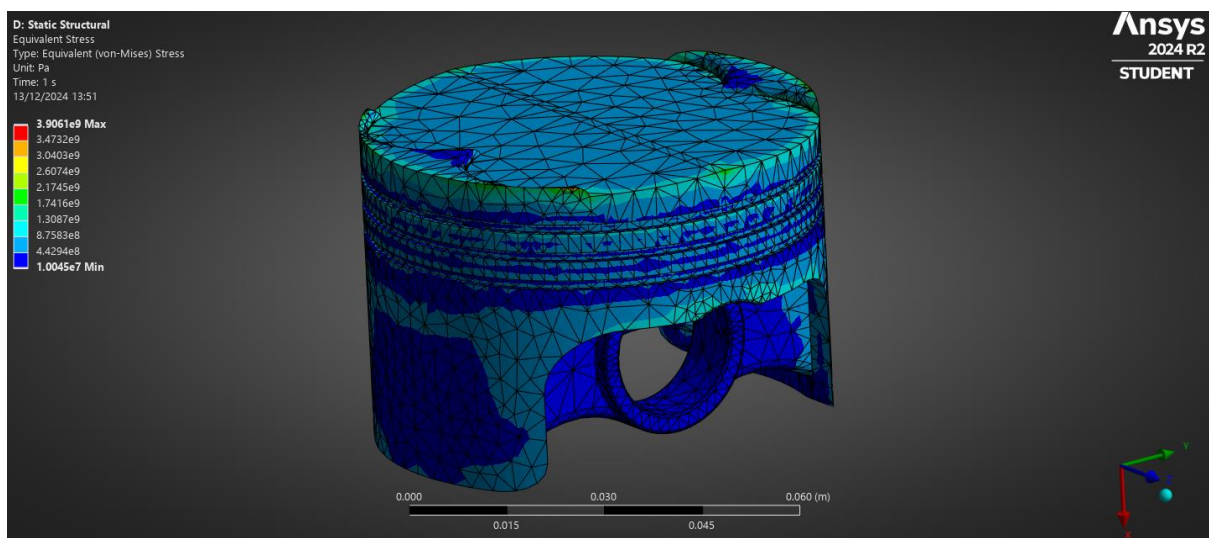


Figure 14 Equivalent Von-Mises Stress of the top of the piston using aluminium-silicon carbide composite material

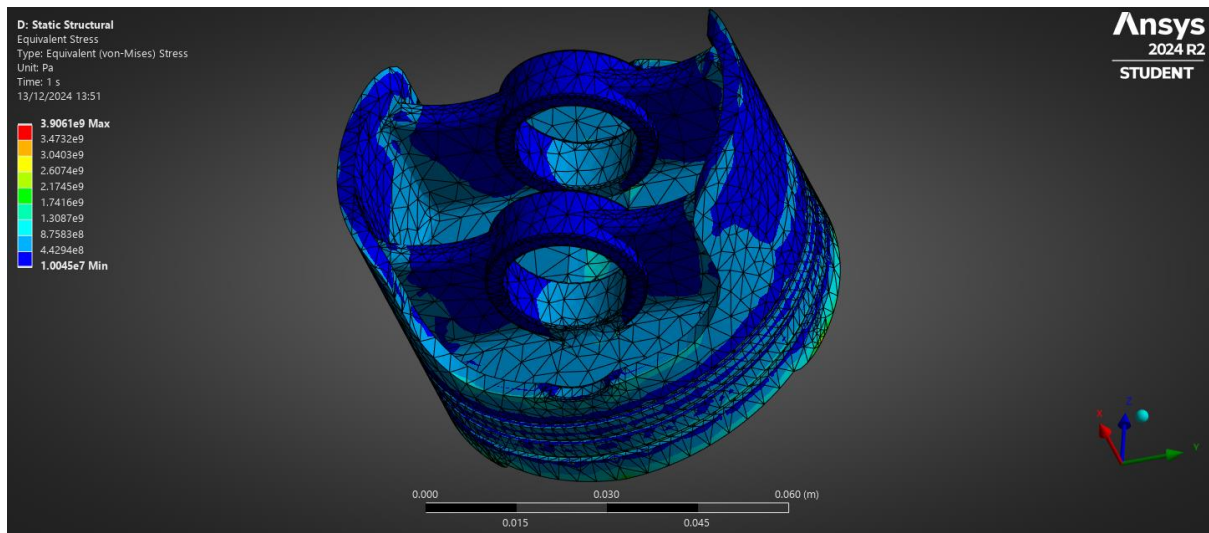


Figure 15 Equivalent Von-Mises Stress of the bottom of the piston using aluminium-silicon carbide composite material

Results	
Minimum	1.0045e+007 Pa
Maximum	3.9061e+009 Pa
Average	4.799e+008 Pa

Figure 16 Equivalent Von-Mises Stress results of the static structural study using aluminium-silicon carbide composite material

3.1 Topology Optimisation

A topology optimization study was performed based on the results of the static structural simulation to identify non-critical areas of the piston where material could be safely removed, however the material used was the aluminium-silicon carbide (Al-SiC) composite material mentioned above. The study's objective was to reduce the overall mass and volume of the piston while retaining its operational integrity and ensuring that it could still function as intended. The study was conducted preserving critical areas such as the top face, side faces, grooves along the side for piston rings and oil, and the regions where the gudgeon pin attaches. These areas are essential for maintaining the piston's structural integrity, sealing capability, and interaction with other engine components. Most of the material removal occurred in the internal regions of the piston, leaving an essentially hollow interior. This resulted in a significant reduction in mass and volume, with the final mass and volume of the piston reduced to 58.172% of their original values. Such a design could potentially revolutionize engine efficiency, offering substantial benefits regarding reduced inertia and improved engine performance. However, the hollow nature of the piston raises concerns about its ability to withstand the high thermal and mechanical stresses encountered during regular engine operation.

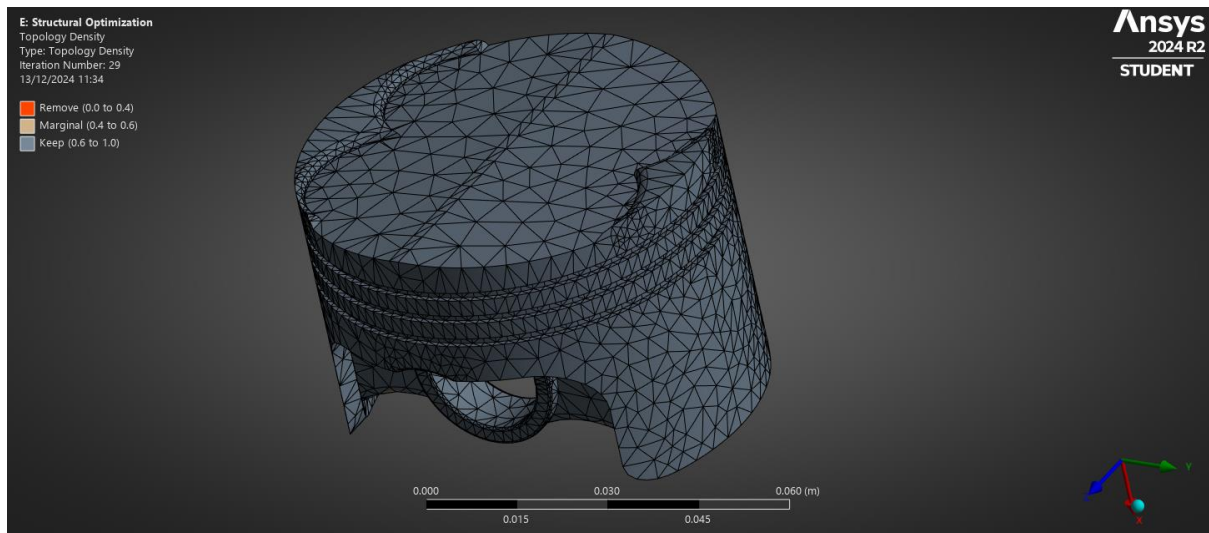


Figure 17 Top of the topology optimised piston

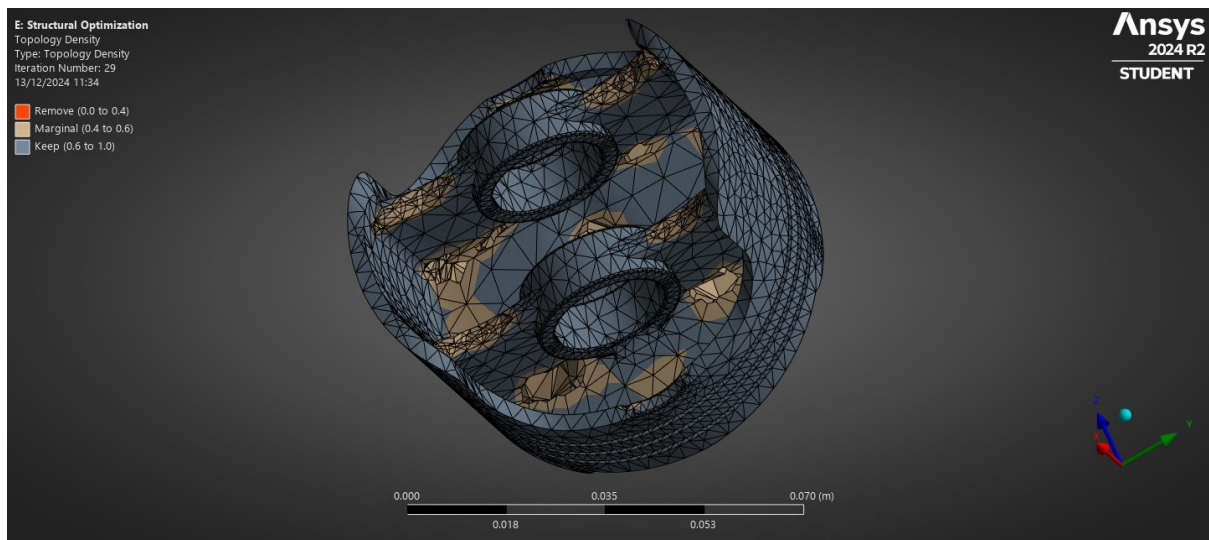


Figure 18 Bottom of the topology optimised piston

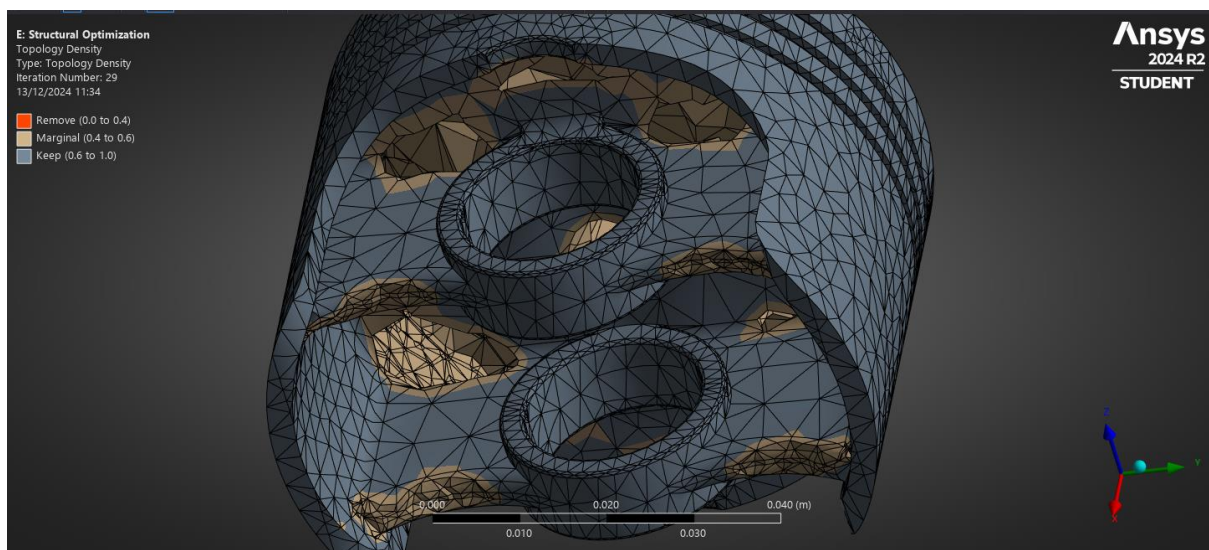


Figure 19 Closer view of the bottom of the topology optimised piston

Original Volume	9.4732e-005 m ³
Final Volume	5.5108e-005 m ³
Percent Volume of Original	58.172
Original Mass	0.27283 kg
Final Mass	0.15871 kg
Percent Mass of Original	58.172

Figure 20 Results of the topology optimisation of the piston

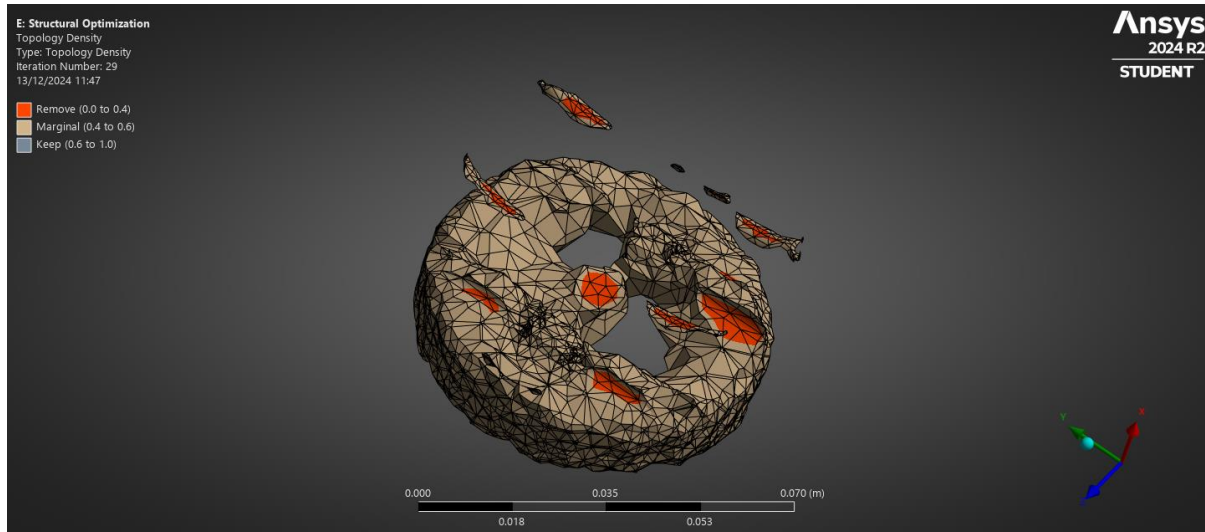


Figure 21 A view of the material removed from the topology optimisation of the piston

While the topology-optimized piston offers promising weight savings, and the results from simulation analysis show that the piston would be structurally stable, its reliability and durability must be thoroughly validated through experimental testing. These tests, which would assess critical performance metrics, such as thermal resistance, structural integrity under cyclic loading, and impact on the overall engine dynamics, are crucial for the study's progression. Overall, the topology optimization study represents a significant step forward in the mass reduction of the piston but underscores the importance of balancing material reduction with operational reliability.

3.2 Simulation of Topology Optimised Piston

Following the topology optimization study, a deformation and stress simulation was conducted on the optimized piston design to evaluate its structural performance under typical operating conditions. The primary objective of this analysis was to assess how the removal of interior material, resulting in an essentially hollow piston, affected its ability to withstand mechanical loads and thermal stresses. By applying identical loading and boundary conditions as in the initial simulations, this study aimed to validate whether the optimized design retained sufficient structural integrity and durability to function reliably within the engine environment.

3.2.1 Total Deformation Analysis

The deformation simulation of the topology-optimized piston revealed that its total deformation levels are comparable to those of the original mechanically inferior aluminium alloy piston despite the significant reduction in material and mass. The maximum deformation observed in the topology-optimized piston was 0.0003634 m, slightly higher than the 0.00030885 m recorded for the original

aluminium alloy design. This similarity demonstrates that the topology optimization process successfully maintained the piston's structural integrity, even after removing a substantial amount of material. The deformation results indicate that the retained structural elements, including the top face, grooves, and gudgeon pin attachment points, are adequately designed to handle operational loads. Overall, the deformation simulation reaffirms the success of the topology optimization process in creating a lightweight yet mechanically robust piston.

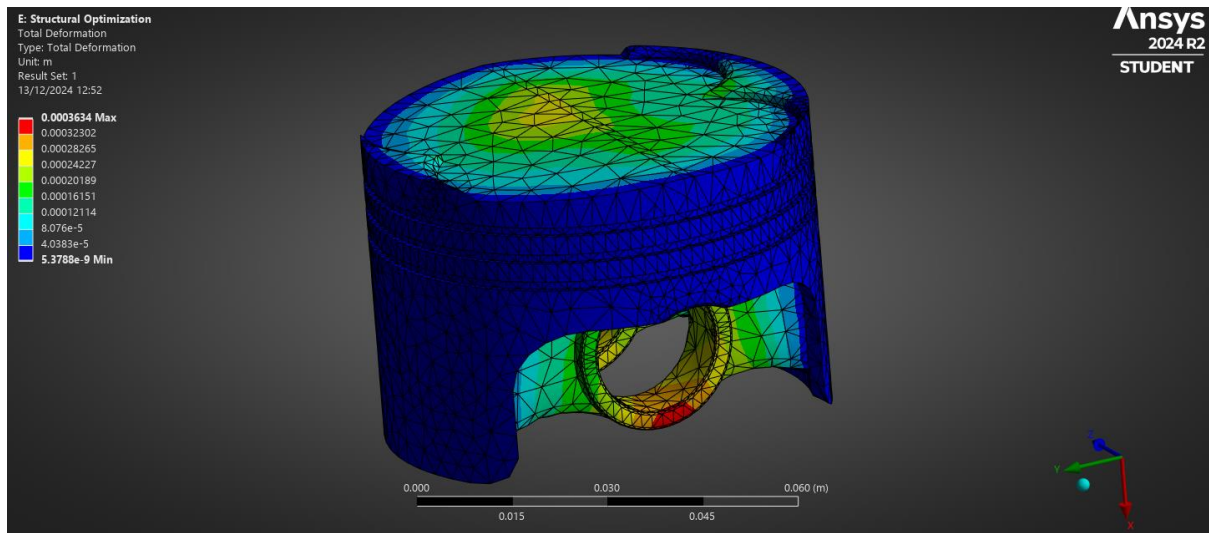


Figure 22 Total deformation of the top of the topology optimised piston

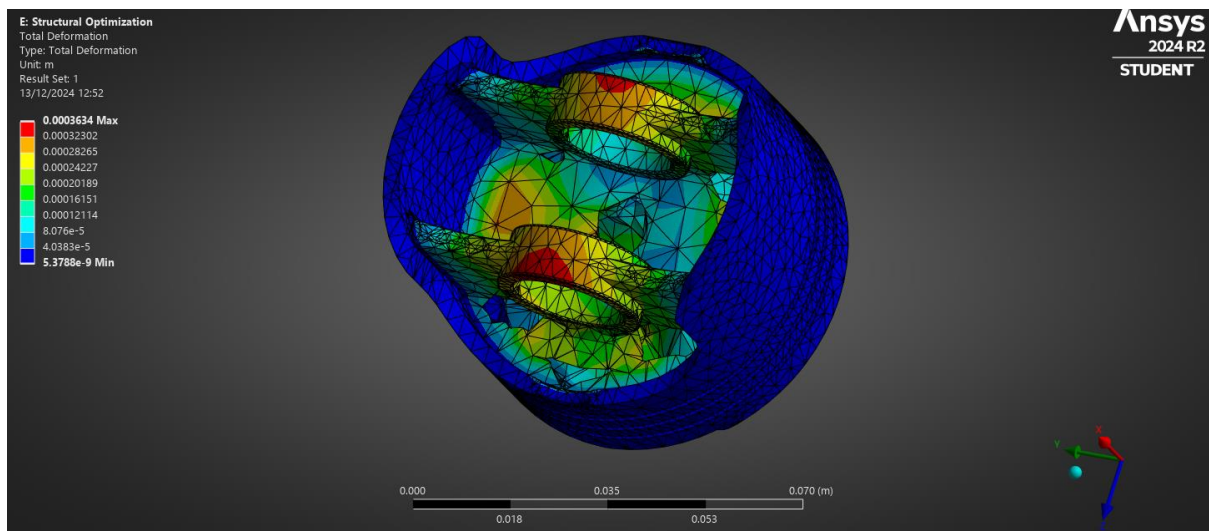


Figure 23 Total deformation of the bottom of the topology optimised piston

Results	
☐ Minimum	5.3788e-009 m
☐ Maximum	3.634e-004 m

Figure 24 Total deformation results of the static structural study using topology optimised piston

3.2.2 Equivalent Stress Analysis

The equivalent stress simulation of the topology-optimized piston revealed that the maximum stress levels are higher than the original aluminium alloy piston. The maximum equivalent stress observed in the topology-optimized piston was 3.0486×10^9 Pa, compared to the 2.684×10^9 Pa recorded for the aluminium alloy design. This increase in stress can be attributed to the removal of significant interior material during the topology optimization process, which resulted in localized stress concentrations in areas such as the gudgeon pin bosses and the crown. Despite this increase in stress levels, the topology-optimized piston demonstrated structural performance sufficient to handle operational loads under the simulated conditions. While the stress levels are higher in the optimized design, the overall mass savings make it a promising lightweight alternative for applications where efficiency and weight reduction are critical. However, the higher stress concentrations in the optimized piston raise concerns about its long-term durability and performance under real-world cyclic and thermal loading. Additional design refinements may be required to address these localized stresses while maintaining the mass reduction achieved. Further validation through fatigue and thermal stress simulations and experimental testing is necessary to ensure that the topology-optimized piston can maintain its structural integrity and reliability over prolonged operation.

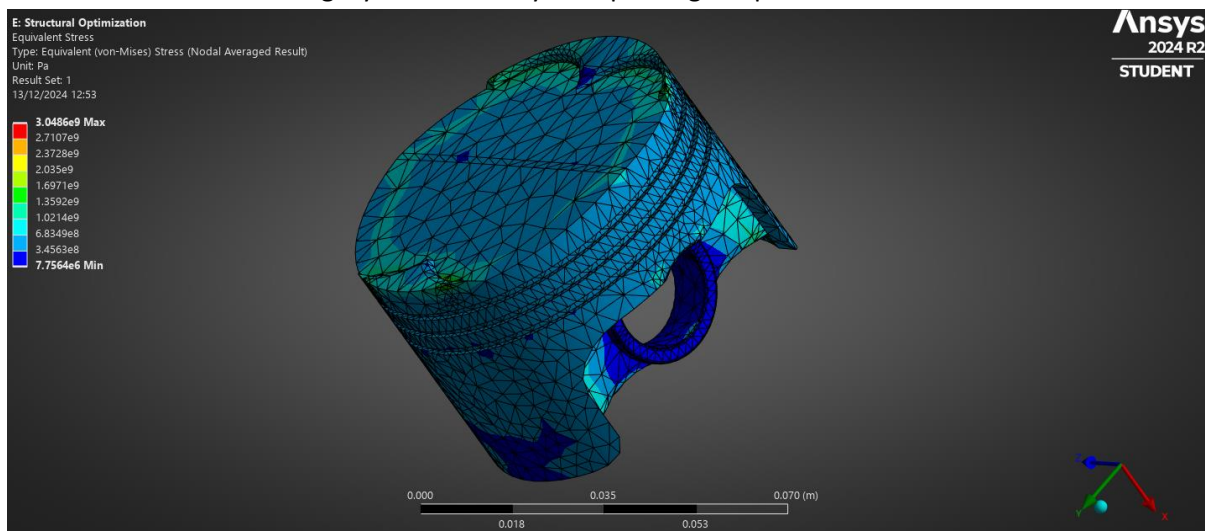


Figure 25 Equivalent Von-Mises Stress of the top of the topology optimised piston

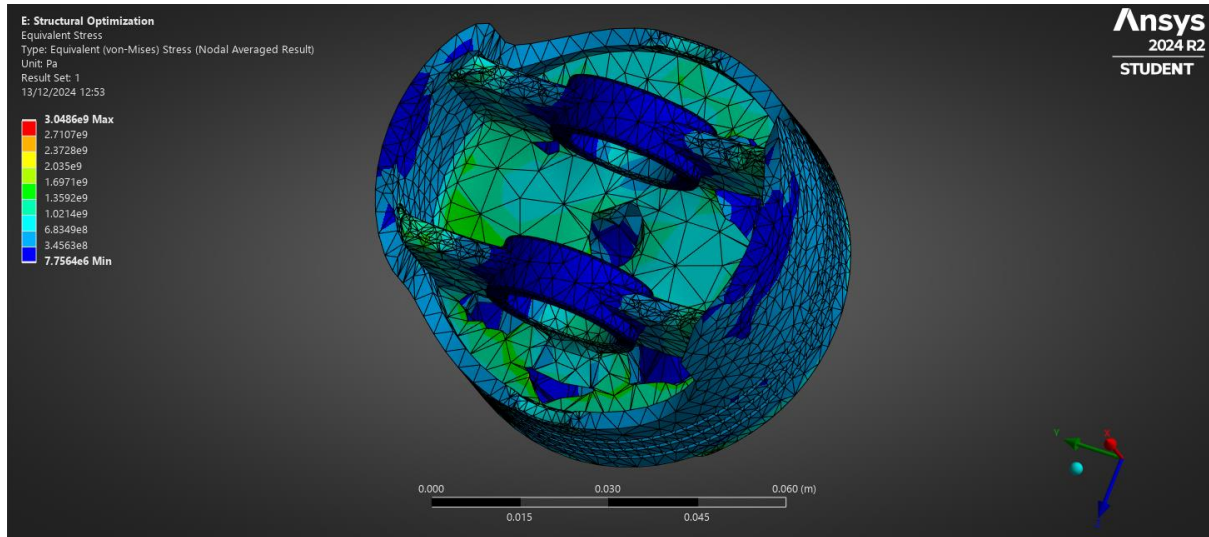


Figure 26 Equivalent Von-Mises Stress of the bottom of the topology optimised piston

Results	
Minimum	7.7564e+006 Pa
Maximum	3.0486e+009 Pa

Figure 27 Equivalent Von-Mises Stress results of the static structural simulation using the topology optimised piston

4.0 Validation Testing

4.1 Elastic Bending - Continuous Load

This validation example aims to verify the accuracy of the Ansys mechanical simulation software in modelling the elastic bending of a thin plate under continuous uniform loading, ensuring it aligns with analytical solutions based on classical beam theory. This example examines the elastic behaviour of a thin plate, fully fixed on one end and subjected to a uniform pressure load. The plate has specific geometric and material parameters: a length of 1 meter, width of 0.05 meters, thickness of 0.005 meters, modulus of elasticity of 210,000 MPa, and Poisson's ratio of 0. The load applied is a uniform pressure of 2,750 Pa, and small deformations are considered while neglecting self-weight.

4.1.1 Analytical Analysis

Bending Moment: The bending moment $M(x)$ at a distance x from the fixed end is given by:

$$M(x) = -\frac{q(L-x)^2}{2}$$

where:

- $q = p \cdot w$ is the distributed load,
- L is the plate length.

Beam Deflection Equation: Using Euler-Bernoulli beam theory, the relationship between bending moment and deflection is:

$$EI_y \frac{d^2 u_z}{dx^2} = M(x)$$

where:

- E is the modulus of elasticity,
- I_y is the moment of inertia of the cross-section.

Moment of Inertia: The moment of inertia I_y for a rectangular cross-section is:

$$I_y = \frac{wt^3}{12}$$

Deflection Function: Integrating the equation for deflection twice and applying boundary conditions for a cantilever beam ($u_z(0) = 0$ and $u'_z(0) = 0$), the deflection function is:

$$u_z(x) = \frac{qx^2(6L^2 - 4Lx + x^2)}{24EI_y}$$

Maximum Deflection: The maximum deflection occurs at the free end ($x = L$) :

$$u_{z,\max} = \frac{qL^4}{8EI_y}$$

Substitution of Parameters: Substitute $q = p \cdot w$ and $I_y = \frac{wt^3}{12}$ into the maximum deflection formula:

$$u_{z,\max} = \frac{pL^4}{2Ewt^3}$$

Numerical Values:

- $E = 210,000\text{MPa}$,
- $L = 1.0 \text{ m}$,
- $w = 0.05 \text{ m}$,
- $t = 0.005 \text{ m}$,
- $p = 2750 \text{ Pa}$.

Substitute these values:

$$u_{z,\max} = \frac{3 \cdot 2750 \cdot 1^4}{2 \cdot 210,000 \cdot 0.05 \cdot 0.005^3}$$

Calculate step-by-step:

$$u_{z,\max} = \frac{3 \cdot 2750}{2 \cdot 210,000 \cdot 0.05 \cdot 0.000125}$$

$$u_{z,\max} = \frac{8250}{2.625}$$

$$u_{z,\max} = 157.143 \text{ mm}$$

4.1.2 Simulation Analysis

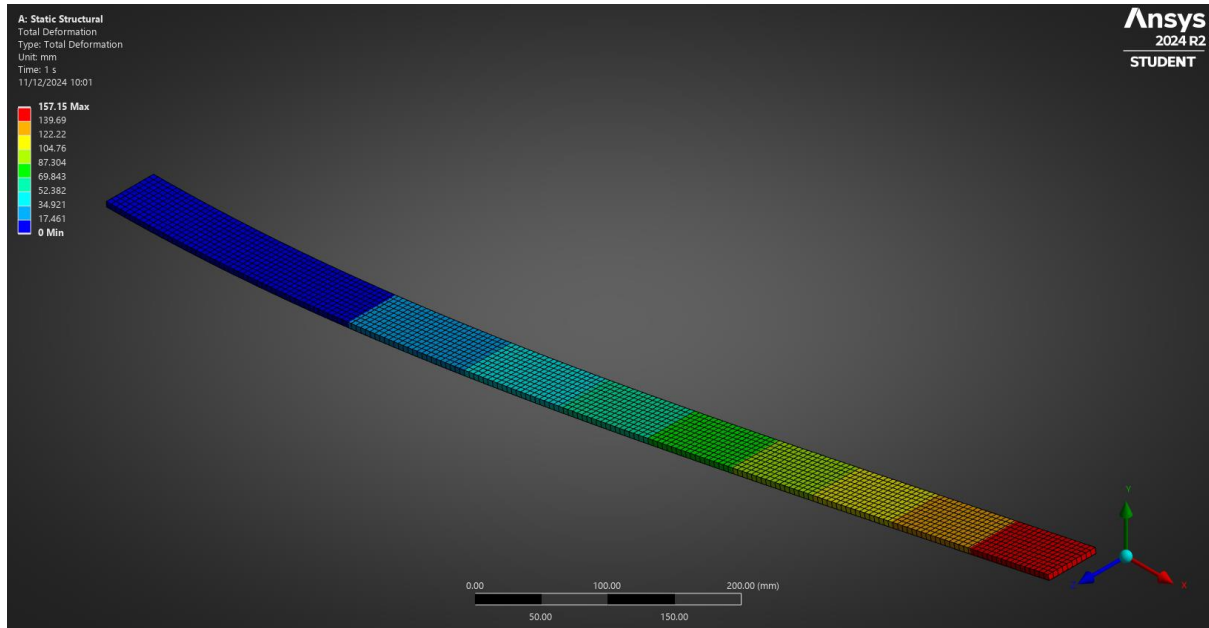


Figure 28 Mechanical simulation of the Elastic Bending - Continuous Load validation example

4.1.3 Comparison

According to Euler-Bernoulli beam theory, the analytical calculation for the maximum deflection of a cantilever beam under uniform load results in a value of 157.143 mm. In contrast, the value obtained from ANSYS is 157.15 mm. The slight difference of 0.007 mm can be attributed to the numerical approximations inherent in simulation software, which employs finite element methods to discretize the structure and solve equations. These methods can introduce minor variations due to element size and numerical rounding. Additionally, the settings of the finite element model, including the type of elements used and the analysis methods applied, can further impact the results. The negligible discrepancy of 0.007 mm highlights the accuracy of both the analytical and numerical approaches, validating their consistency and demonstrating the robustness of the simulation model.

4.1.4 Mesh Independence Study

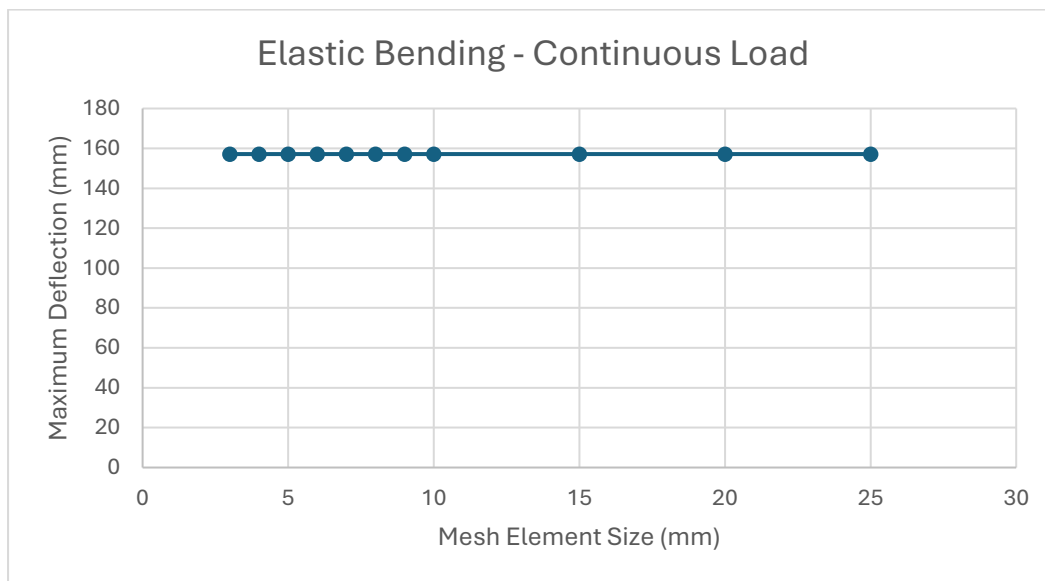


Figure 29 Graph comparing mesh element size against calculated maximum deflection for the Elastic Bending - Continuous Load validation example

Elastic Bending - Continuous Load	
Mesh Element Size (mm)	Maximum Deflection (mm)
25	157.15
20	157.15
15	157.15
10	157.15
9	157.15
8	157.15
7	157.15
6	157.15
5	157.15
4	157.15
3	157.15

The table and graph depict the relationship between mesh element size and maximum deflection obtained from a mechanical simulation in ANSYS under continuous loading conditions. The table shows that the mesh element size ranges from 25 mm to 3 mm, while the maximum deflection remains constant at 157.15 mm for all cases. The graph, which plots mesh element size on the x-axis and maximum deflection on the y-axis, highlights this consistency, with all points aligning horizontally at 157.15 mm. Comparing these results to the analytical solution of 157.143 mm reveals a negligible difference of 0.007 mm, likely due to numerical rounding or software approximations. This indicates that the simulation results are accurate and mesh-independent, demonstrating convergence as further mesh refinement does not alter the results. The close alignment with the analytical value validates the simulation model, confirming that the chosen mesh density is sufficient to accurately capture the system's behaviour.

4.2 Bimetallic Strip

This validation example aims to verify the accuracy of the Ansys mechanical simulation software in modelling and evaluating the deflection of a bimetallic strip consisting of invar and copper subjected to a temperature difference. The analysis evaluates isotropic linear elasticity and geometrically linear analysis principles to validate the results against an analytical solution. One end of the strip is fixed, while the other is free and subjected to a temperature difference of 100 degrees Celsius while neglecting self-weight. The analysis focuses on determining the maximum deflection at the free end. The analytical solution uses material properties such as Young's modulus, Poisson's ratio, and thermal expansion coefficients to derive the deflection through a dimensionless stiffness coefficient formula.

4.2.1 Analytical Analysis

Dimensionless Stiffness Coefficient (K_1) :

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

Substituting $t_i = t_c = t$:

$$K_1 = 4 + 6(1) + 4(1)^2 + \frac{130,000}{137,000} (1)^3 + \frac{137,000 \cdot 1}{130,000 \cdot 1}$$
$$K_1 \approx 4 + 6 + 4 + 0.9496 + 1.0538 \approx 15.0034$$

Maximum Deflection ($u_{z, \max}$) :

$$u_{z, \max} = \frac{6(\alpha_i - \alpha_c)(t_i + t_c)T_c L^2}{t^2 K_1}$$

Material Properties:

Invar:

$$E_i = 137,000 \text{ MPa}, \nu_i = 0.280, \alpha_i = 1.2 \times 10^{-6} \text{ } ^\circ\text{C}^{-1}$$

Copper:

$$E_c = 130,000 \text{ MPa}, \nu_c = 0.354, \alpha_c = 2.0 \times 10^{-5} \text{ } ^\circ\text{C}^{-1}$$

Geometry:

- Width of cross-section: $w = 5 \text{ mm}$
- Thickness of each layer: $t_i = t_c = t = 1 \text{ mm}$
- Length: $L = 100 \text{ mm}$

Thermal Loading:

- Temperature difference: $T_c = 100^\circ\text{C}$

Substituting values:

$$u_{z,\max} = \frac{6(1.2 \times 10^{-6} - 2.0 \times 10^{-5})(1 + 1)(100)(100)^2}{(1)^2(15.0034)}$$

$$u_{z,\max} = \frac{6(-1.88 \times 10^{-5})(2)(10,000)}{15.0034}$$

$$u_{z,\max} = \frac{-2.256 \times 10^{-2}}{15.0034} \approx -7.049 \text{ mm}$$

4.2.2 Simulation Analysis

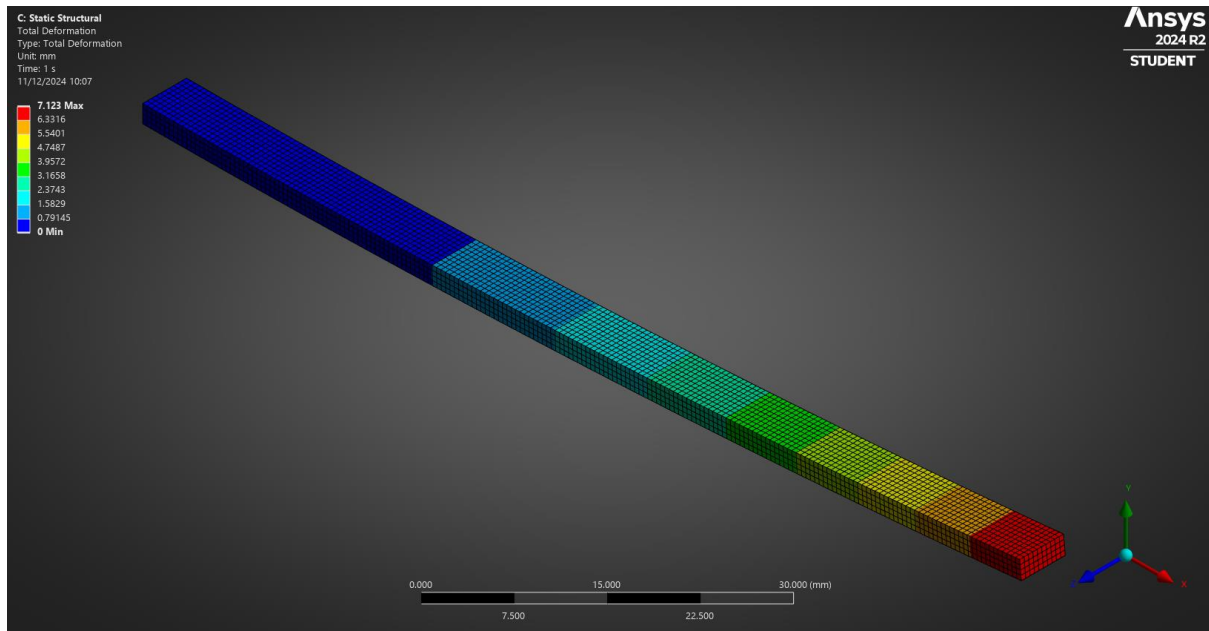


Figure 30 Mechanical simulation of the Bimetallic Strip validation example

4.2.3 Comparison

The analytical deflection value for the bimetallic strip is calculated to be -7.049mm, while the deflection obtained from Ansys Mechanical simulation software is 7.123mm at a mesh size of 0.5mm. This slight discrepancy can be attributed to several factors. Ansys employs numerical methods, such as finite element analysis (FEA), to approximate solutions, which may result in slight differences compared to the exact analytical formula. Additionally, the simulation might interpret or apply constraints differently, leading to variations in the results. The analytical solution assumes uniform properties and idealized geometry, while simulations consider more realistic system complexities. Furthermore, the analytical model involves simplifications, such as assuming linear elasticity and ideal conditions, which may overlook subtle effects like higher-order deformation or material anisotropy. These factors collectively account for the minor differences between the two results.

4.2.4 Mesh Independence Study

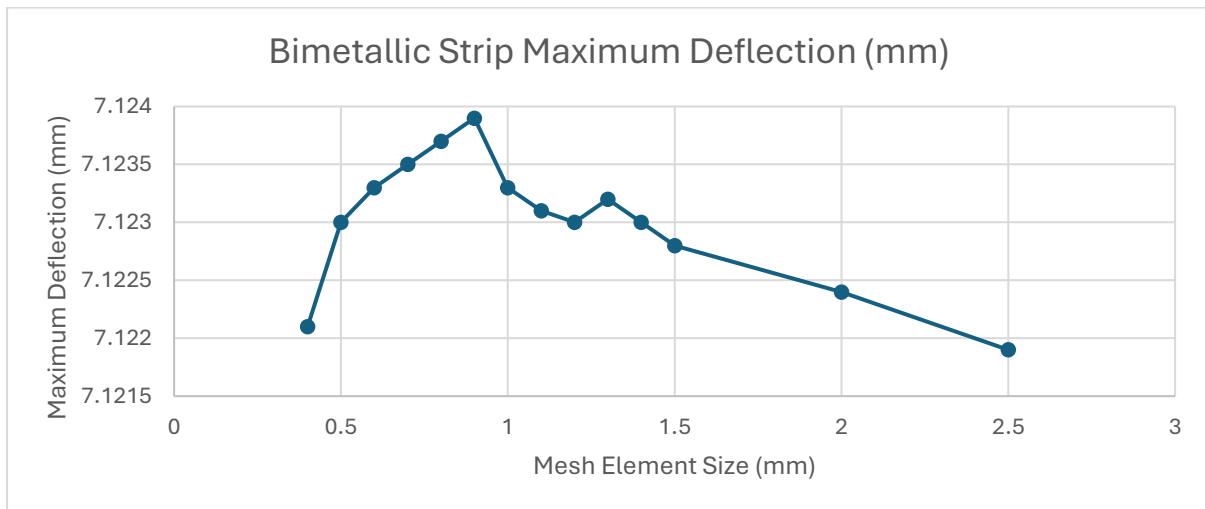


Figure 31 Graph comparing mesh element size against calculated maximum deflection for the Bimetallic Strip validation example

Bimetallic Strip	
Mesh Element Size (mm)	Maximum Deflection (mm)
2.5	7.1219
2	7.1224
1.5	7.1228
1.4	7.123
1.3	7.1232
1.2	7.123
1.1	7.1231
1	7.1233
0.9	7.1239
0.8	7.1237
0.7	7.1235
0.6	7.1233
0.5	7.123
0.4	7.1221

The table shows that as the mesh size decreases, the deflection values converge, with smaller mesh sizes (below 1 mm) exhibiting minimal variation, indicating convergence of the simulation results. The deflection deviates significantly for larger mesh sizes (above 1.5 mm), suggesting less accurate approximations. The corresponding data highlights that the maximum deflection peaks near a mesh size of 1.1 mm (~7.1233 mm) before decreasing slightly for coarser meshes. The deflection values from the simulation (~7.12–7.124 mm) are consistently higher than the analytical value of 7.049 mm, which could be attributed to simplifications in the analytical model, such as neglecting nonlinear effects or real-world complexities, as well as potential inaccuracies in the simulation for coarser meshes. The convergence at finer meshes indicates increasing accuracy, but the consistent

overestimation suggests that the simulation captures additional factors not included in the analytical calculation.

4.3 Boundary Condition Influence

In mechanical simulations, the application of boundary conditions plays a pivotal role in determining the accuracy and reliability of the results. Boundary conditions define how the model interacts with its surroundings, including constraints, loads, and environmental factors. Improperly applied or oversimplified boundary conditions can lead to significant errors, as they may need to represent the simulated real-world scenario adequately. For instance, rigid supports, free boundaries, or assumed symmetry planes might introduce unrealistic stress concentrations or artificially eliminate key deformation modes. Conversely, over-constraining a model can lead to overly conservative results, while under-constraining may cause unrealistic displacements or instability. Therefore, it is prudent to carefully assess the sensitivity of simulation outputs to the choice and implementation of boundary conditions, ensuring they align closely with the physical context. Such assessments can include parametric studies or validations against experimental data to improve confidence in the model's predictions.