

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)

Alexander Cook

2104599

December 2025

Table of Contents

1	Introduction	4
2	Capabilities and Limitations of CAE	5
2.1	Capabilities of CAE in Engineering Design	5
2.2	Known Limitations of CAE and FEA Processes	5
2.3	Validation and Verification in CAE	6
3	Validation Studies	7
3.1	Validation Case 1: Bimetallic strip (Thermal)	7
3.2	Validation Case 2: Cantilever with Axial Load	10
4	Methodology for Piston Case Study	16
4.1	Modelling scope and assumptions	16
4.2	Material models	16
4.3	Load Application from Kinematic Study	19
4.4	Boundary conditions	20
4.5	Mesh Strategy	21
5	Structural and Thermomechanical Results	24
5.1	Material Comparison Results	24
5.2	Thermal influence	27
5.3	Interpretation	27

6	Topology and Mass Reduction Study	29
7	Critical Discussion: CAE as a Design Tool	31
8	Conclusions	32
9	References.....	33
10	Appendix	35
10.1	Validation.....	35
10.2	Piston case study	36

Equation 1 - Maximum deflection - bimetallic strip	8
Equation 2 - Equation for axial deformation	11
Equation 3 - Equation for transversal deformation.....	11
Figure 1 - Bimetallic strip, 0.25mm Hex Dominant Mesh	7
Figure 2 - Bimetallic strip geometry and loading.....	8
Figure 3 - Cantilever beam maximum transverse deformation.....	10
Figure 4 - Cantilever beam geometry and loading.....	10
Figure 5 - Cantilever beam boundary conditions.....	11
Figure 6 - Visualisation of joint probe	20
Figure 7 - Loads applied to the piston (top) Figure 8 - Loads applied to the piston (bottom)	20
Figure 9 - Piston (top view) 4mm mesh Figure 10 - Piston (top view) refined mesh	23
Figure 11 - Piston (bottom view) 4mm mesh Figure 12 - Piston (bottom view) refined mesh	23
Figure 13 - Maximum deformation comparison between materials.....	25
Figure 14 - Maximum stress comparison between materials.....	26
Figure 15 - Visualisation of the deformation of the piston in 4032 T6.....	26
Figure 16 - Visualisation of the stress analysis of the piston in 4032 T6	27
Figure 17 - Material reduction examples of the piston head using topology.....	30

Graph 1 - Bimetallic strip deformation under 100 °C temperature increase	9
Graph 2 - Cantilever beam, maximum axial deformation	12
Graph 3 - Cantilever beam maximum transversal displacement comparison.....	13
Graph 4 - Von Mises stress, axial	14
Graph 5 - Von Mises stress, transversal.....	14
Graph 6 - Isotropic elasticity of all materials	19
Table 1 - Mesh and deformation comparison, bimetallic strip.....	9
Table 2 - Cantilever beam, axial displacement - comparison between mesh types	12
Table 3 - Cantilever beam, maximum transversal displacement - mesh type comparison.....	13
Table 4 - 6061 T6 Young's Modulus, temperature dependent values and Poisson's Ratio.....	17
Table 5 - 4032 T6 Young's Modulus, temperature dependent values and Poisson's ratio	18
Table 6 - 2618 T61 Modulus, temperature dependent values and Poisson's ratio	18
Table 7 - Thermal boundary conditions	21
Table 8 - 4mm mesh, result comparison	22
Table 9 - 2mm mesh, result comparison	22
Table 10 - Refined mesh, result comparison	22
Table 11 - Thermal results comparison between all materials.....	24
Table 12 - Total deformation comparison for all materials	24

Table 13 - Von Mises Stress comparison between materials	25
Table 14 - Results for the topology study	29

1 Introduction

Computer Aided Engineering (CAE) is widely used in modern mechanical design to evaluate structural and thermal behaviour prior to physical testing. Finite Element Analysis (FEA) enables complex components to be analysed under realistic loading and boundary conditions that are difficult to assess using analytical methods alone.

The aim of this report is to demonstrate an understanding of the CAE process through a structured workflow that includes validation, application, and critical interpretation of results. Two benchmark problems are first used to validate modelling approaches against published reference solutions. A piston case study is then analysed to compare material behaviour under combined mechanical and thermal loading, followed by an illustrative topology optimisation study to demonstrate the broader capabilities of CAE tools.

Throughout the report, emphasis is placed on understanding the limitations of numerical modelling and the role of engineering judgement when interpreting simulation results.

2 Capabilities and Limitations of CAE

2.1 Capabilities of CAE in Engineering Design

CAE tools are widely used in modern mechanical design to assess structural, thermal, and dynamic behaviour prior to physical prototyping. FEA allows engineers to analyse problems involving complex three-dimensional geometry, varying material behaviour, and combined loading cases that cannot be solved accurately using traditional hand calculations(Sifakis & Barbič, 2015).

By dividing a continuous component into a finite number of smaller elements, FEA enables stresses, deformations, and temperature distributions to be calculated across complex components in much greater detail than is possible with analytical methods. This makes CAE particularly useful during early design stages, where multiple design variants can be assessed efficiently without the cost and time associated with manufacturing and testing physical prototypes (Szabó & Babuška, 2021).

When applied correctly, CAE supports engineering decision-making by highlighting critical load paths, areas of high stress, and sensitivity to changes in geometry or material properties. This allows targeted design improvements to be made before manufacture. However, the reliability of CAE results depends strongly on the quality of the numerical model and the assumptions applied during simulation setup, meaning that results must always be interpreted with appropriate engineering caution.(Gu & Yang, 2004)

2.2 Known Limitations of CAE and FEA Processes

While CAE provides valuable insight during the design process, its outputs are subject to several well-known technical limitations that arise from numerical modelling and simplifications. In FEA, physical systems are represented through idealised geometry, discretised domains, and assumed loading conditions, meaning that results are sensitive to how the model is constructed rather than representing exact real-world behaviour.

One of the primary limitations of FEA is its dependence on spatial discretisation. The continuous geometry of a component is divided into a finite number of elements, meaning stresses and deformations are sensitive to element size, shape, and formulation. Poor mesh quality or insufficient refinement can lead to inaccurate local stress predictions, while excessive refinement increases

computational cost without necessarily improving result reliability(Belytschko et al., 2014)(Amor-Martin & Garcia-Castillo, 2021).

Boundary condition idealisation represents another major source of uncertainty. Real components are rarely subjected to perfectly fixed supports or uniformly applied loads, yet such simplifications are often required in CAE models. Small changes in constraint definition or load application can significantly affect predicted stresses and deformations, particularly in statically indeterminate systems(Schwer, n.d.).

Material modelling assumptions further limit predictive accuracy. Engineering simulations frequently assume linear elastic, isotropic material behaviour, despite real components exhibiting temperature dependence, nonlinearity, and manufacturing variability. While these simplifications are acceptable for comparative design studies, they reduce confidence in absolute performance predictions (Budynas & Keith Nisbett, n.d.).

These limitations highlight the need for validation and sensitivity studies before CAE results are applied to real engineering components.

2.3 Validation and Verification in CAE

Given the known limitations associated with numerical modelling, validation and verification form a critical part of any CAE workflow. Verification focuses on ensuring that the computational model has been implemented correctly, while validation assesses how well simulation results represent physical behaviour (Schwer, n.d.). In practice, this is commonly achieved through comparison with analytical solutions, benchmark problems, or experimental data. Studies comparing CAE predictions with measured component behaviour show that simulations are most reliable when used to capture trends rather than exact values (Dubensky et al., 1996). These principles underpin the validation studies presented in this report.

3 Validation Studies

Following the validation principles outlined in Section 2, two benchmark problems were modelled using published verification examples from Dlubal. These cases provide reference solutions that allow direct numerical comparison with CAE results.

3.1 Validation Case 1: Bimetallic strip (Thermal)

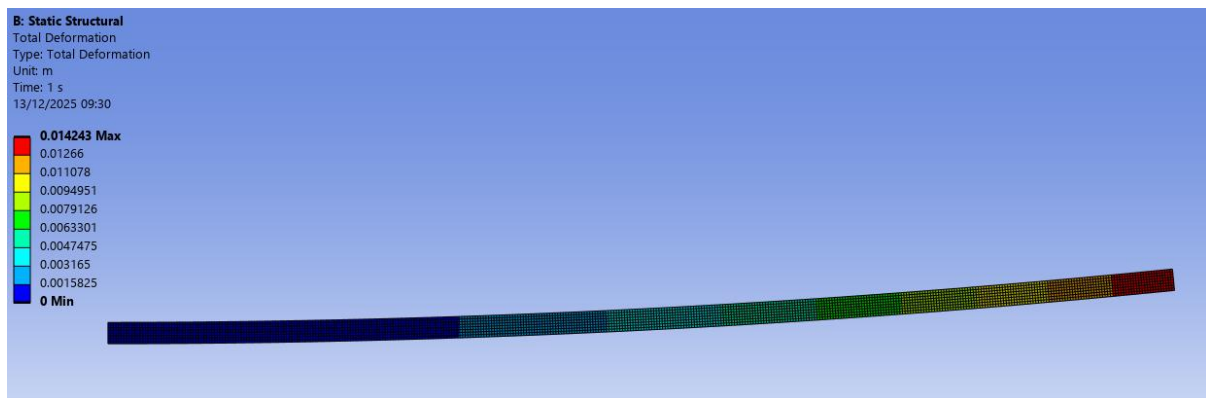


Figure 1 - Bimetallic strip, 0.25mm Hex Dominant Mesh

The first validation case considers a bimetallic strip composed of invar and copper subjected to a uniform temperature increase. One end of the strip is fully constrained, while the opposite end remains free to deform. This configuration follows a published verification example, for which a reference deformation value is available for direct comparison (Dlubal, 2018), as shown in Figure 1.

Material	Invar	Modulus of Elasticity	E_i	137000.000	MPa
		Poisson's Ratio	ν_i	0.280	—
		Coefficient of Thermal Expansion	α_i	1.200×10^{-6}	$^{\circ}\text{C}^{-1}$
	Copper	Modulus of Elasticity	E_c	130000.000	MPa
		Poisson's Ratio	ν_c	0.354	—
		Coefficient of Thermal Expansion	α_c	2.000×10^{-5}	$^{\circ}\text{C}^{-1}$
Geometry	Cross-section Width		w	5.000	mm
	Layer Thickness		$t_i = t_c = t$	1.000	mm
	Length		L	100.000	mm
Load		Thermal Loading	T_c	100.000	$^{\circ}\text{C}$

Figure 2 - Bimetallic strip geometry and loading

$$U_{z,max} = \frac{6(\alpha_i - \alpha_c)(t_i - t_c)T_c L^2}{t_s^2 K_1} \frac{1}{2} \approx -7.049\text{mm}$$

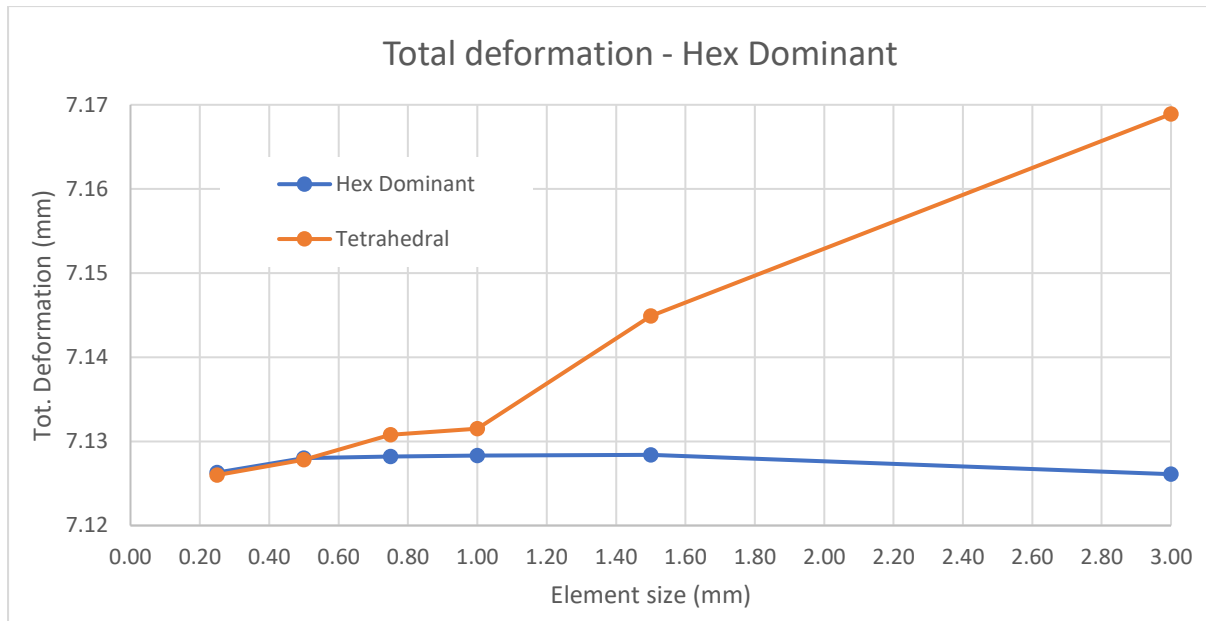
Equation 1 - Maximum deflection - bimetallic strip

Absolute deformation values were used to allow direct comparison with the reference outputs.

$$U_{z,max} \approx 7.049\text{mm}$$

A static structural analysis was performed with a temperature increase from 22 $^{\circ}\text{C}$ to 122 $^{\circ}\text{C}$.

Multiple mesh densities were tested using hex-dominant and tetrahedral elements. Maximum free-end deformation was used as the validation metric, as it can be directly compared to the reference solution.



Graph 1 - Bimetallic strip deformation under 100 °C temperature increase

The results, summarised in Table 1 and illustrated in Graph 1, show that both mesh density and element size influence the predicted deformation. At an element size of 3.0 mm, the tetrahedral mesh shows a higher percentage difference (~1.6%) than the hex-dominant mesh (~1.0%). With mesh refinement, both formulations converge to within approximately 1.01% of the reference solution.

Table 1 - Mesh and deformation comparison, bimetallic strip

	Hex Dominant		Tetrahedral	
	Deformation (mm)		Deformation (mm)	
Element size (mm)	Maximum (mm)	% difference - Dlubal	Maximum (mm)	% difference - Dlubal
3.00	7.1211	1.012%	7.1639	1.604%
1.50	7.1234	1.044%	7.1399	1.273%
1.00	7.1233	1.043%	7.1265	1.087%
0.75	7.1232	1.042%	7.1258	1.078%
0.50	7.1230	1.039%	7.1228	1.036%
0.25	7.1213	1.015%	7.1210	1.011%

For this simple geometry, hex-dominant elements achieve comparable accuracy at larger element sizes, with percentage differences remaining below 1.05% even at larger element sizes. At finer

mesh densities (below approximately 1mm), the difference between element size becomes negligible, indicating numerical convergence. This case demonstrates that convergence should be assessed based on the stabilisation of results relative to a reference solution, rather than element size reduction alone.

3.2 Validation Case 2: Cantilever with Axial Load

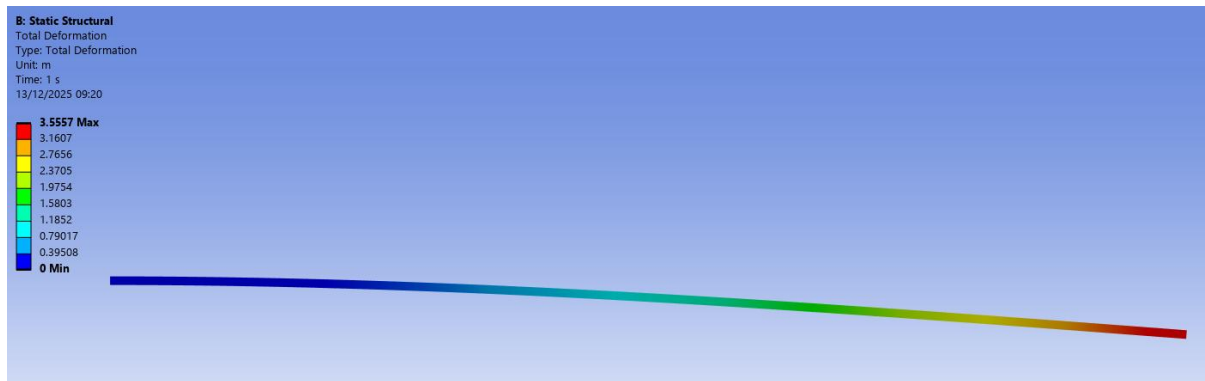


Figure 3 - Cantilever beam maximum transverse deformation

The second validation case considers a cantilever beam subjected to combined axial and transverse loading. Unlike the previous thermomechanical case, this example focuses on linear elastic structural behaviour and allows validation of both axial and bending deformation modes within a single model. The configuration follows a published verification example with analytical reference solutions for maximum axial and transverse displacements (Dlupal, 2015).

Material	Steel	Modulus of Elasticity	E	210000.000	MPa
		Poisson's Ratio	ν	0.300	—
Geometry	Cantilever	Length	L	10.000	m
		Cross-section width	w	0.080	m
Load		Transversal Force	F_z	7.650	kN
		Axial Force	F_x	1.600	kN

Figure 4 - Cantilever beam geometry and loading

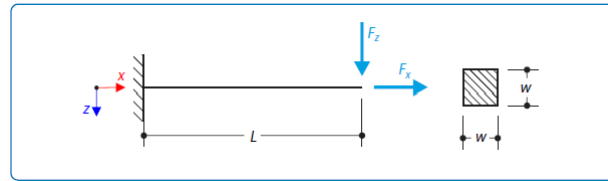


Figure 5 - Cantilever beam boundary conditions

Linear analysis:

$$U_{x,max} = \frac{F_x L}{EA} = 0.012mm$$

Equation 2 - Equation for axial deformation

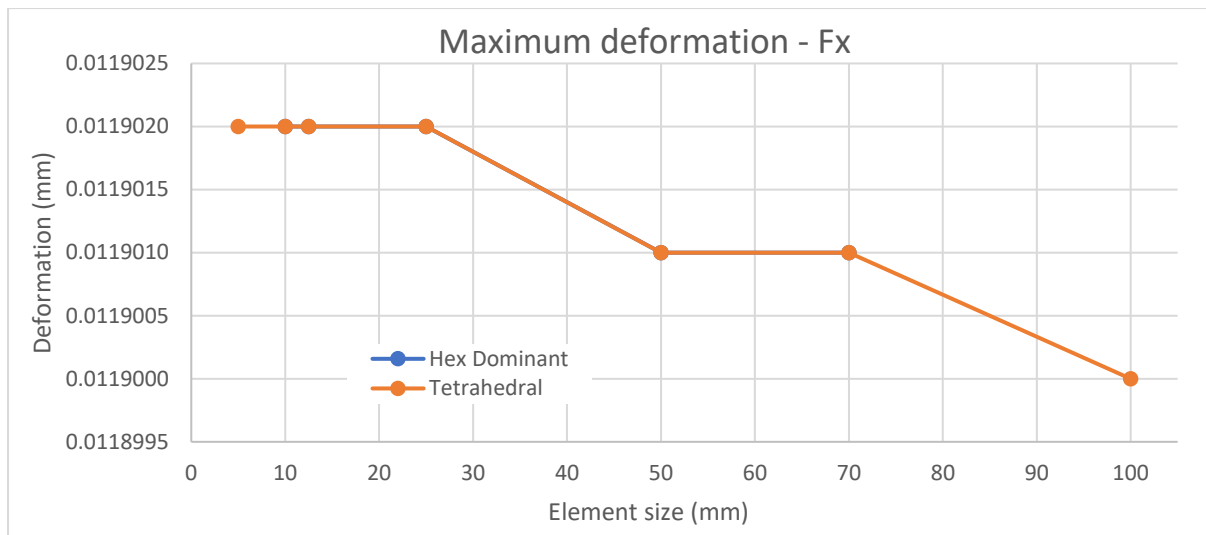
$$U_{z,max} = \frac{F_z L^3}{3EI_y} = 3557.478mm$$

Equation 3 - Equation for transversal deformation

(Dlubal, 2015)

A is the cross-section area and I_y is the quadratic moment of the cross-section to the y-axis.

The beam is fully constrained at one end and loaded at the free end with an axial force in the x-direction and a transverse force in the z-direction. The model was analysed using a linear static structural formulation. Validation was performed by comparing the maximum axial displacement and maximum transverse displacement against the analytical reference values provided in the Dlubal documentation

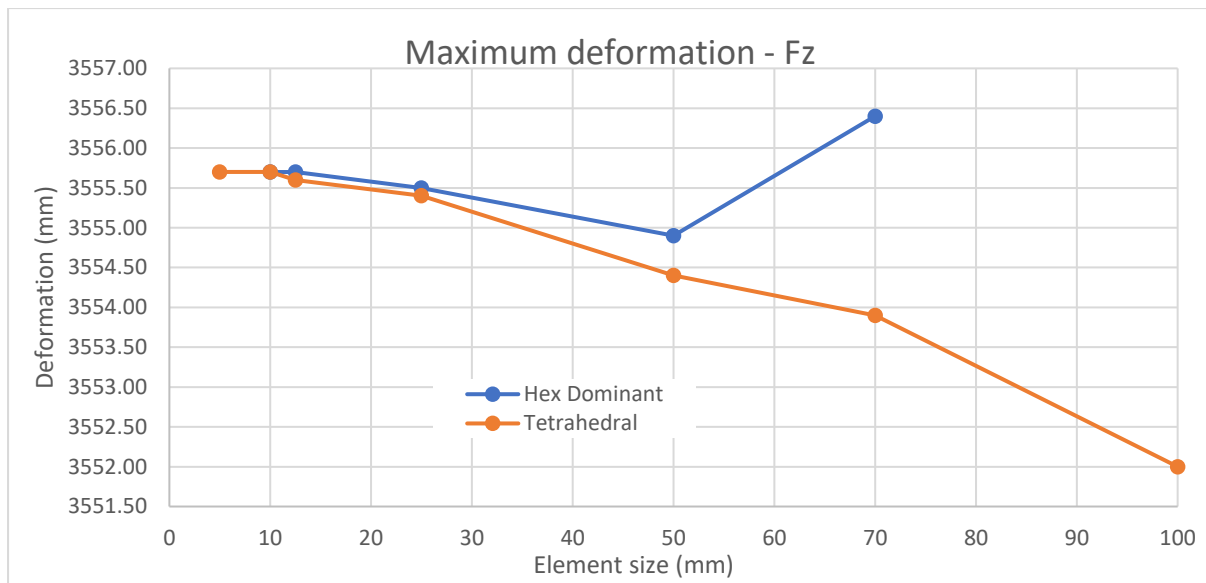


Graph 2 - Cantilever beam, maximum axial deformation

Table 2 - Cantilever beam, axial displacement - comparison between mesh types

Fx - axial deformation				
Hex Dominant			Tetrahedral	
Element size (mm)	Maximum (mm)	% difference	Maximum (mm)	% difference
100			0.0119	0.840%
70	0.0119	0.832%	0.0119	0.832%
50	0.0119	0.832%	0.0119	0.832%
25	0.0119	0.823%	0.0119	0.823%
12.5	0.0119	0.823%	0.0119	0.823%
10	0.0119	0.823%	0.0119	0.823%
5			0.0119	0.823%

Axial deformation is largely insensitive to mesh density. Across all mesh sizes, the axial displacement remains approximately $u_x \approx 0.0119\text{mm}$, with percentage differences below 0.9% relative to the reference solution. This reflects the uniform nature of axial stress and strain along the beam length.



Graph 3 - Cantilever beam maximum transversal displacement comparison

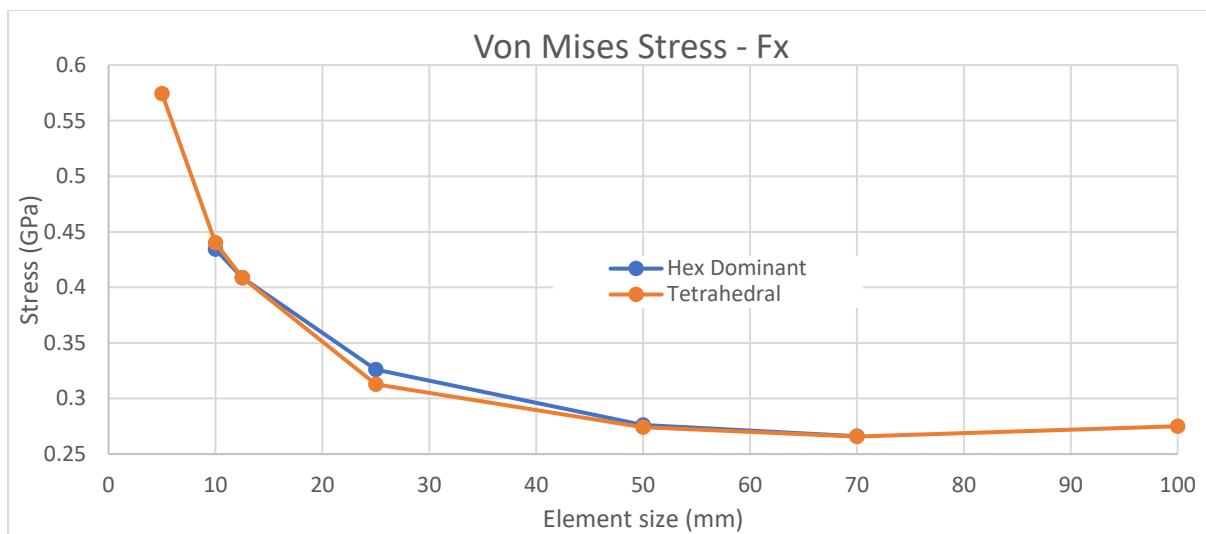
Table 3 - Cantilever beam, maximum transversal displacement - mesh type comparison

Fz - transverse displacement				
Element size (mm)	Hex Dominant		Tetrahedral	
	Maximum (mm)	% difference	Maximum (mm)	% difference
100			3552.00	0.154%
70	3556.40	0.030%	3553.90	0.101%
50	3554.9	0.073%	3554.40	0.087%
25	3555.5	0.056%	3555.40	0.058%
12.5	3555.7	0.050%	3555.60	0.053%
10	3555.7	0.050%	3555.70	0.050%
5			3555.70	0.050%

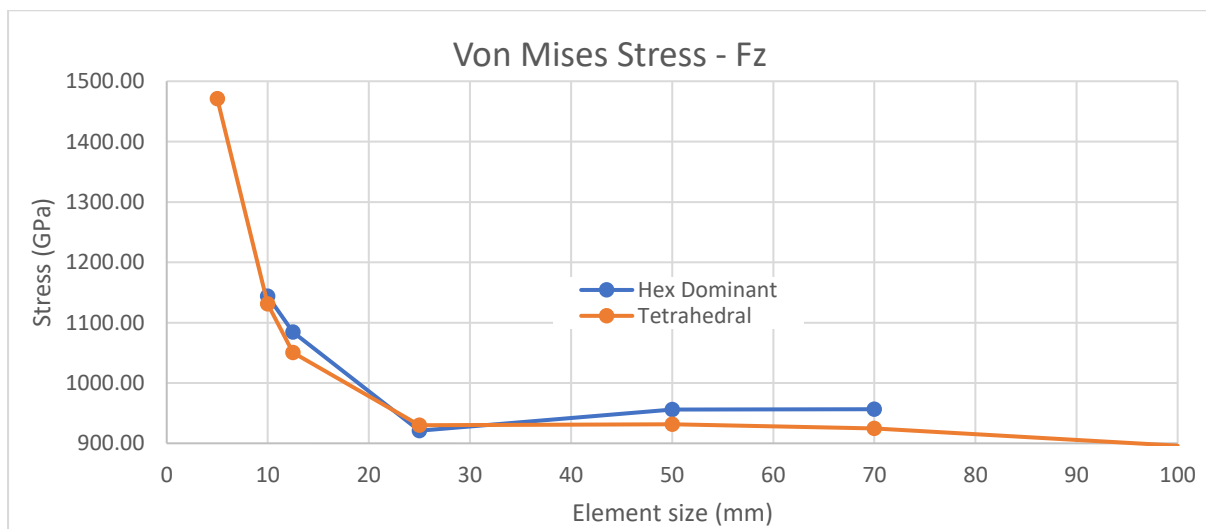
In contrast, transverse displacement shows a higher sensitivity to mesh density. This is particularly clear for the tetrahedral mesh at coarser element sizes. At an element size of 100 mm, the tetrahedral mesh predicts a maximum displacement of 3552.0 mm, corresponding to a percentage difference of 0.154% relative to the reference solution. As the mesh is refined, the percentage

difference reduces steadily, falling below 0.10% at an element size of 50 mm and converging to approximately 0.05% for element sizes between 10 mm and 12.5 mm.

The hex-dominant mesh provides improved accuracy at coarser resolutions. At an element size of 50 mm, the percentage difference is 0.073%, reducing to approximately 0.05% at 12.5 mm. At finer mesh densities, both element formulations converge to similar displacement values of approximately 3555.7 mm. This trend is consistent with bending-dominated deformation, where curvature near the fixed support is more sensitive to element resolution than axial deformation.



Graph 4 - Von Mises stress, axial



Graph 5 - Von Mises stress, transversal

Von Mises stress was examined as a supporting indicator of numerical behaviour. Peak stress values increased with mesh refinement due to improved resolution of stress gradients near the fixed support. As no analytical reference solution exists, stress results are not used for validation.

4 Methodology for Piston Case Study

The methodology used for the piston case study defines how loads, boundary conditions, material behaviour, and mesh strategy were applied within the CAE workflow.(X. F. Liu et al., 2017)

4.1 Modelling scope and assumptions

A three-dimensional structural and thermomechanical analysis of the piston was performed to investigate the influence of material selection and thermal loading on structural behaviour. In addition, a separate topology study was conducted to demonstrate potential mass reduction using CAE tools.

To balance model fidelity with computational efficiency, the following assumptions were applied:

- Linear elastic, isotropic material behaviour was assumed for all materials.
- Temperature-dependent Young's modulus was defined to capture stiffness degradation at elevated temperature.
- Poisson's ratio was held constant for each material.
- Manufacturing effects, residual stresses, surface roughness, and piston pin compliance were not included.

These assumptions allow meaningful comparison between materials and design variants while maintaining a controlled and repeatable CAE workflow.

4.2 Material models

The piston was analysed using three aluminium alloys commonly used in piston applications: 6061-T6, 4032-T6, and 2618-T61. All materials were modelled as linear elastic and isotropic, with temperature-dependent Young's modulus defined to capture stiffness reduction at elevated temperature. Poisson's ratio was held constant for each material.

Young's modulus was defined as a function of temperature using discrete data points between 22 °C and 500 °C, as summarised in the tables below. This approach allows the influence of thermal softening to be included without introducing additional material non-linearities.

Material properties for 6061-T6 were taken from data supplied during lectures, while properties for 4032-T6 and 2618-T61 were obtained from published aluminium alloy property data. The same material modelling approach was applied consistently across all cases to allow direct comparison between materials.

Table 4 - 6061 T6 Young's Modulus, temperature dependent values and Poisson's Ratio

6061 - T6		
Temperature (°C)	Young's modulus (Gpa)	Poisson's Ratio
22	67	0.33
100	70	0.33
150	68	0.33
200	64	0.33
250	56	0.33
300	48	0.33
350	41	0.33
400	32	0.33
450	21	0.33
500	12	0.33

Table 5 - 4032 T6 Young's Modulus, temperature dependent values and Poisson's ratio

4032 T6		
Temperature (°C)	Young's modulus (Gpa)	Poisson's Ratio
22	79	0.34
100	77	0.34
150	75	0.34
200	73	0.34
250	71	0.34
300	69	0.34
350	65	0.34
400	62	0.34
450	59	0.34
500	55	0.34

(Aluminum 4032-T6, n.d.)

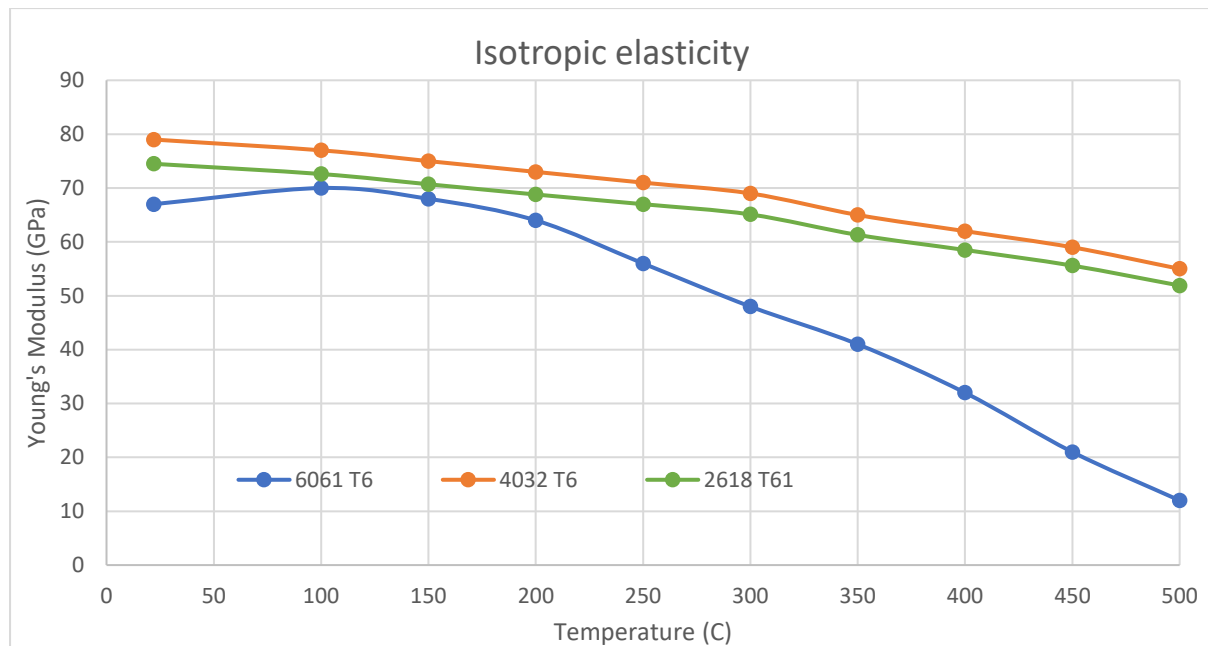
(Aluminium 4032 | Aluminium Alloy 4032, n.d.)

Table 6 - 2618 T61 Modulus, temperature dependent values and Poisson's ratio

2618 T61		
Temperature (°C)	Young's modulus (Gpa)	Poisson's Ratio
22	74.5	0.33
100	72.6	0.33
150	70.7	0.33
200	68.8	0.33
250	67	0.33
300	65.1	0.33
350	61.3	0.33
400	58.5	0.33
450	55.6	0.33
500	51.9	0.33

(Aluminum 2618-T61, n.d.)

(2618A Aluminium | DTD 5014A, n.d.)



Graph 6 - Isotropic elasticity of all materials

The temperature dependence for Young's modulus for each material is shown in the graph above.

4.3 Load Application from Kinematic Study

Loads applied in the structural analysis were derived from a prior kinematic simulation of the crank–connecting rod–piston assembly. Reaction forces and moments were extracted at the gudgeon pin–piston interface using a joint probe, representing the load transfer from the connecting rod to the piston during operation, as illustrated in Figure 6.

These reaction forces and moments were applied to the structural piston model as remote loads at the piston pin boss to represent both translational and moment effects without introducing localised load singularities. In addition, combustion loading was represented as a uniform pressure of 1.05×10^7 Pa applied to the piston crown, corresponding to peak operating conditions Figures 7 & 8.

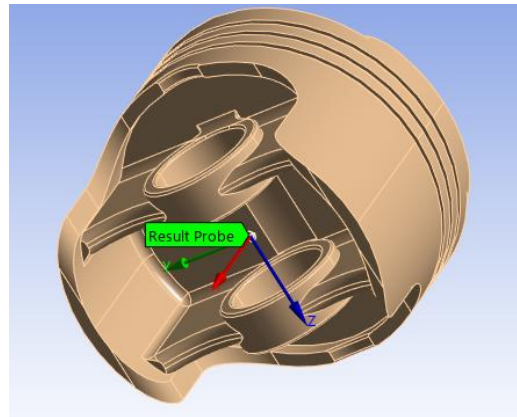


Figure 6 - Visualisation of joint probe

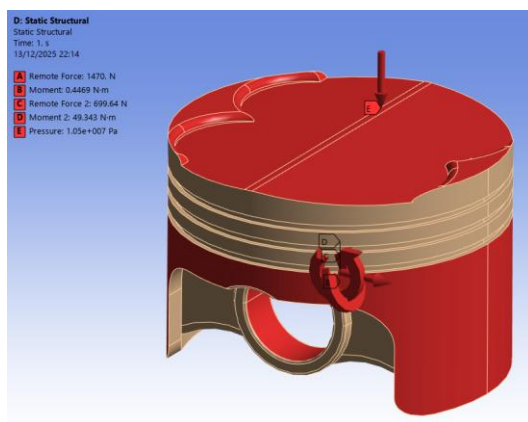


Figure 7 - Loads applied to the piston (top)

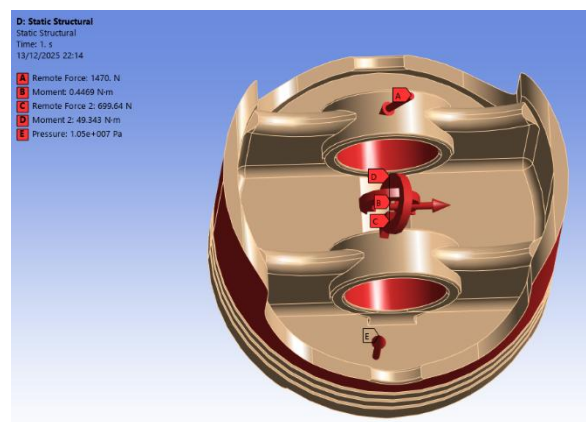


Figure 8 - Loads applied to the piston (bottom)

4.4 Boundary conditions

Mechanical boundary conditions were applied to represent piston guidance within the cylinder bore. Translational degrees of freedom were constrained to allow motion only along the cylinder axis, while rotational degrees of freedom were restricted to prevent rigid body motion. These constraints provide a stable structural solution while maintaining realistic piston kinematics.

Table 7 - Thermal boundary conditions

Thermal Study Boundary Conditions		
	Heat transfer coefficient	Temperature (°C)
Crown	497.5	928.5
Crown side	447.5	425
Top Groove	437.5	190
Middle Groove	437.5	190
Oil Groove	338.75	163.75
Underside	675	91.25
Skirt	462.5	107.5

Thermal boundary conditions were defined using a steady-state convection model. Different convection temperatures and heat transfer coefficients were applied to the piston crown, ring grooves, underside, and skirt to represent the non-uniform thermal environment experienced during operation. The applied thermal boundary conditions are summarised in Table 7. The resulting temperature field was mapped into the structural analysis to account for thermally induced stiffness reduction and thermal expansion effects.(B. Liu et al., 2024)

4.5 Mesh Strategy

Following the mesh refinement study performed on the baseline 6061-T6 configuration, a fixed mesh strategy was selected and retained for all subsequent material and topology studies. This approach ensures that observed differences in response are attributable to changes in material behaviour or geometry rather than numerical variation.

A global element size of 4 mm was used, with local refinement to 2 mm applied at the piston crown and pin holes to better capture stress gradients in these regions. Overall mesh quality was checked to ensure stable and reliable results, and refinement beyond this level did not result in meaningful changes in global response. The selected mesh therefore represents an appropriate balance between numerical accuracy and computational efficiency.

Table 8 - 4mm mesh, result comparison

6061 T6 - 4mm mesh		
Total deformation (m)		
Minimum	Maximum	Average
8.88E-07	3.43E-04	1.38E-04
Stress (Von Mises) (Pa)		
Minimum	Maximum	Average
3.89E+06	2.16E+09	2.62E+08
Strain (Von Mises) (m/m)		
Minimum	Maximum	Average
1.03E-04	3.87E-02	4.96E-03

Table 9 - 2mm mesh, result comparison

6061 T6 - 2mm mesh		
Total deformation (m)		
Minimum	Maximum	Average
5.52E-07	3.43E-04	1.43E-04
Stress (Von Mises) (Pa)		
Minimum	Maximum	Average
3.57E+06	1.98E+09	2.74E+08
Strain (Von Mises) (m/m)		
Minimum	Maximum	Average
6.69E-05	3.53E-02	5.07E-03

Table 10 - Refined mesh, result comparison

6061 T6 - 4 mm - crown + pin holes 2mm		
Total deformation (m)		
Minimum	Maximum	Average
6.97E-07	3.42E-04	1.54E-04
Stress (Von Mises) (Pa)		
Minimum	Maximum	Average
3.02E+06	2.13E+09	2.52E+08
Strain (Von Mises) (m/m)		
Minimum	Maximum	Average
7.18E-05	3.83E-02	4.78E-03

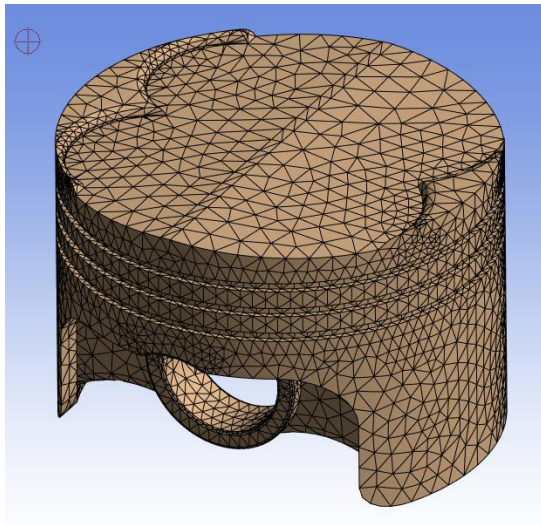


Figure 9 - Piston (top view) 4mm mesh

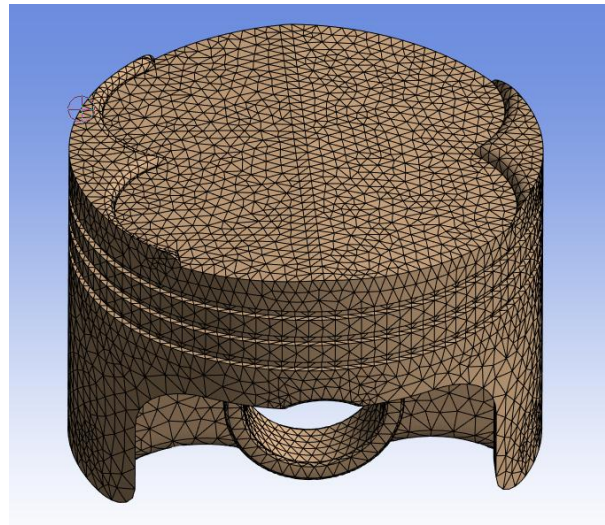


Figure 10 - Piston (top view) refined mesh

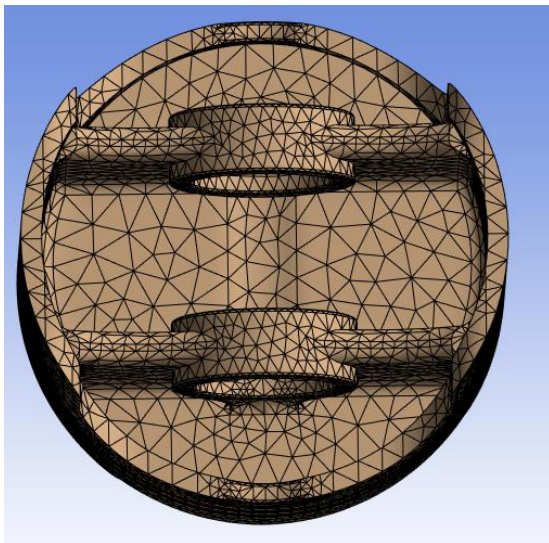


Figure 11 - Piston (bottom view) 4mm mesh

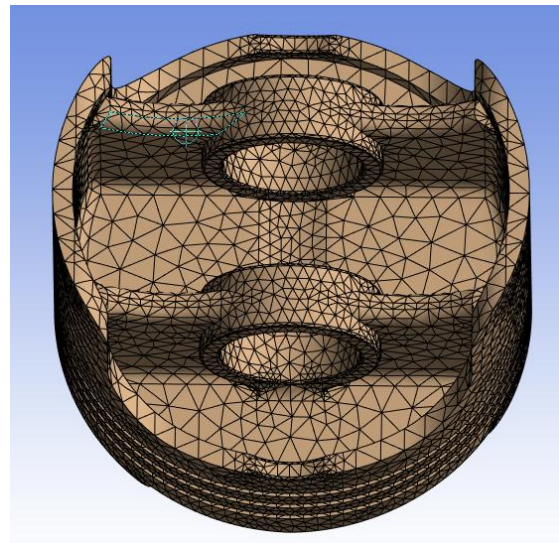


Figure 12 - Piston (bottom view) refined mesh

5 Structural and Thermomechanical Results

5.1 Material Comparison Results

Table 11 - Thermal results comparison between all materials

Thermal comparisons (°C)		
6061 T6		
Minimum	Maximum	Average
161.26	304.59	246.54
4032 T6		
Minimum	Maximum	Average
152.65	315.36	260.32
2618 T61		
Minimum	Maximum	Average
155.2	312.12	259.43

Table 11 summarises the thermal results obtained for the three piston materials under steady-state thermal loading. The maximum piston temperature varies only slightly between materials, ranging from 304.6 °C for 6061-T6 to 315.4 °C for 4032-T6, with similar average temperatures between 246 °C and 260 °C. This indicates that material selection has a limited influence on the overall thermal field under the applied boundary conditions.

Table 12 - Total deformation comparison for all materials

Materials	Total deformation (m)		
	Minimum	Maximum	Average
6061 T6	6.97E-07	3.42E-04	1.54E-04
4032 T6	9.44E-07	2.88E-04	1.29E-04
2618 T61	9.37E-07	3.23E-04	1.45E-04

Structural response shows clearer differences between materials. As illustrated in Figure 13, the 6061-T6 piston exhibits the largest maximum deformation of 3.42×10^{-4} m, compared to 2.88×10^{-4} m for 4032-T6 and 3.23×10^{-4} m for 2618-T61. This behaviour is consistent with the greater reduction in stiffness experienced by 6061-T6 at elevated temperature.

Table 13 - Von Mises Stress comparison between materials

Materials	Stress (Von Mises) (Pa)		
	Minimum	Maximum	Average
6061 T6	3.02E+06	2.13E+09	2.52E+08
4032 T6	3.03E+06	2.31E+09	2.71E+08
2618 T61	3.35E+06	2.41E+09	2.86E+08

Maximum von Mises stress trends are shown in Figure 14 and summarised in Table 13. The highest peak stress is observed for 2618-T61 at 2.41×10^9 Pa, followed by 4032-T6 at 2.31×10^9 Pa and 6061-T6 at 2.13×10^9 Pa. Materials retaining higher stiffness at operating temperature therefore experience lower deformation but higher peak stress under identical loading.

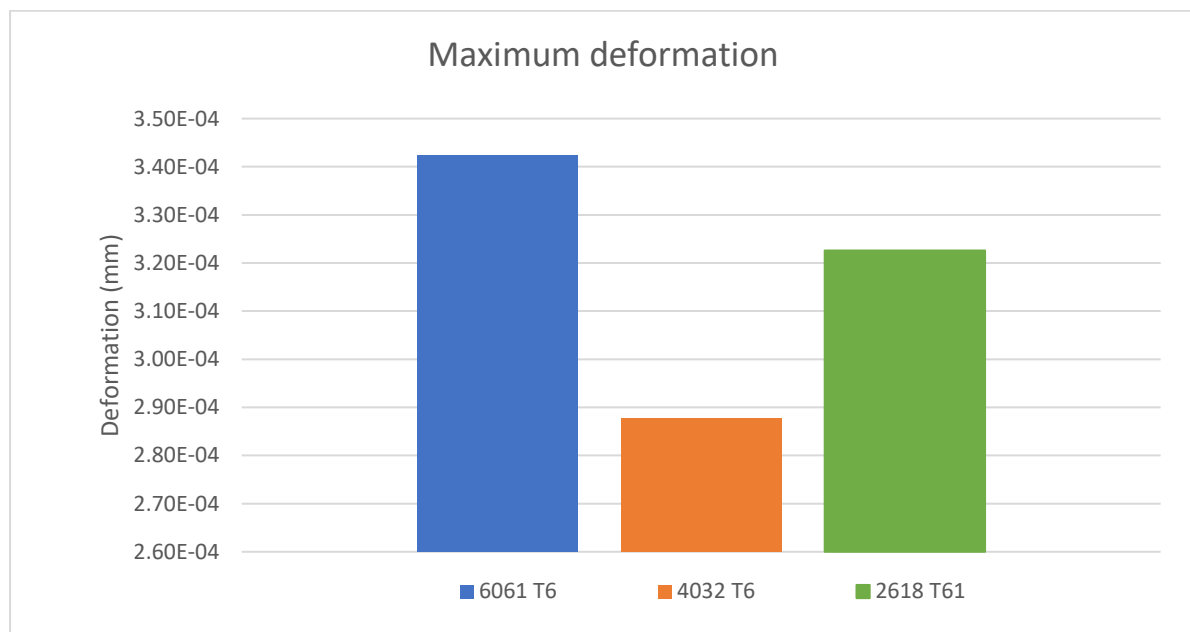


Figure 13 - Maximum deformation comparison between materials

Figure 13 highlights the higher maximum deformation observed for the 6061-T6 piston relative to the other materials.

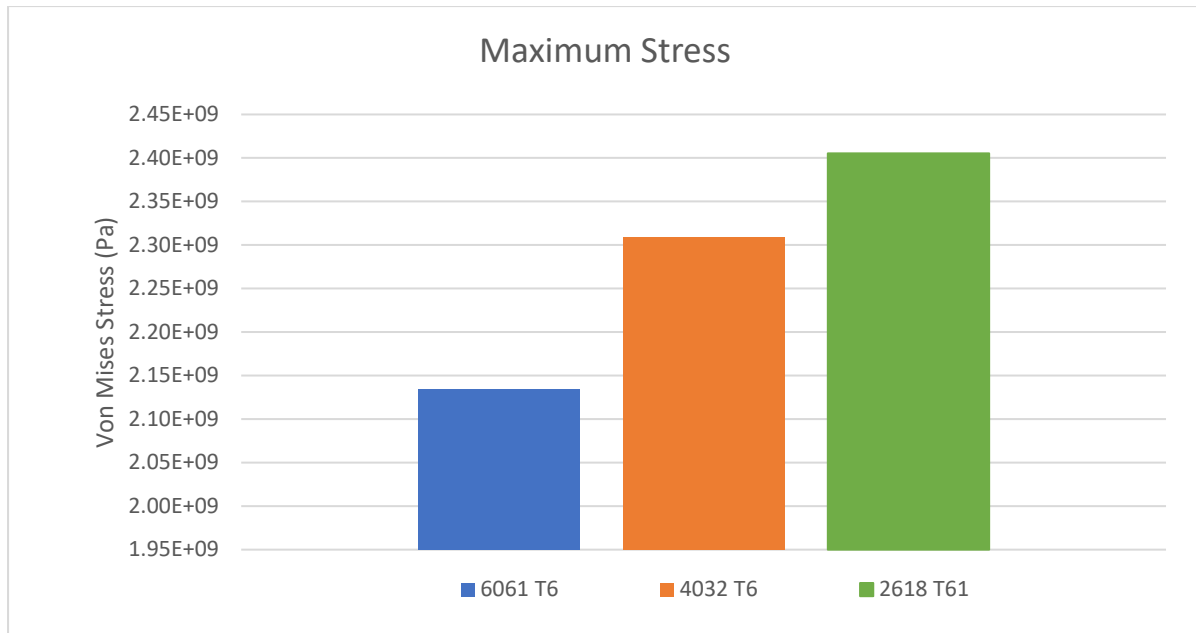


Figure 14 - Maximum stress comparison between materials

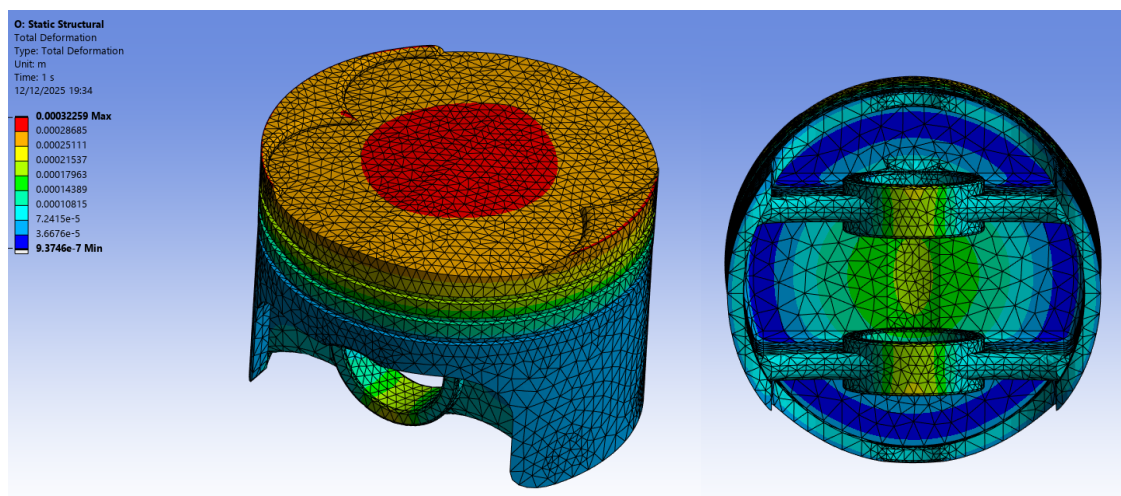


Figure 15 - Visualisation of the deformation of the piston in 4032 T6

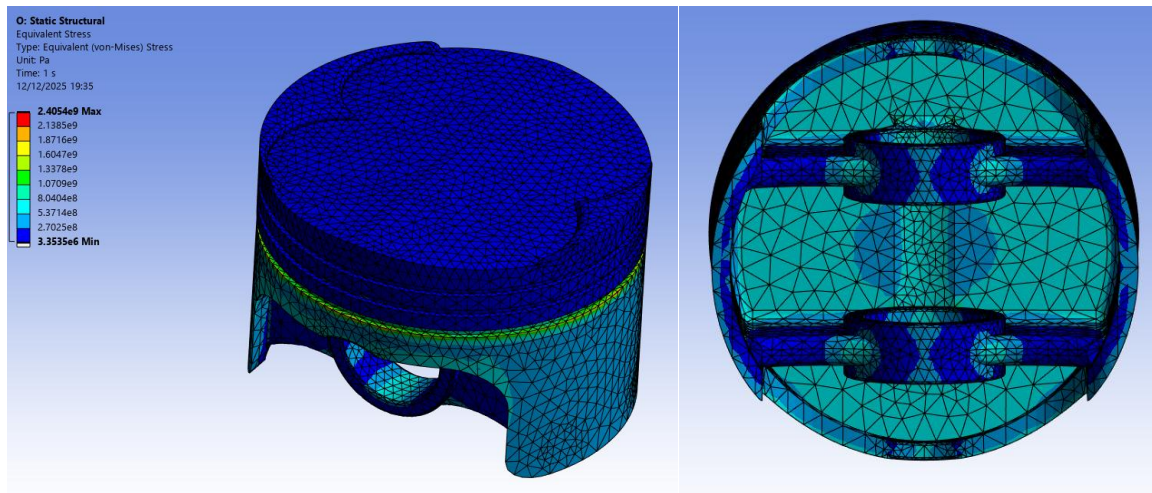


Figure 16 - Visualisation of the stress analysis of the piston in 4032 T6

Peak deformation and stress occur at the piston crown and pin boss regions, as illustrated in Figures 15 and 16.

5.2 Thermal influence

Thermal loading has a clear influence on the structural response of the piston across all materials. Elevated temperatures reduce effective material stiffness, leading to increased deformation and strain compared to a purely mechanical analysis. This effect is most pronounced for 6061-T6, which experiences the largest reduction in Young's modulus at operating temperature, as shown by the material property data.

Although the thermal field itself is similar for all materials, differences in high-temperature stiffness retention result in distinct structural behaviour. Materials such as 4032-T6 and 2618-T61 retain greater stiffness at elevated temperature and therefore exhibit lower deformation, but higher peak Von Mises stress due to reduced compliance under the same loading. This demonstrates that thermal effects must be included when comparing piston materials, as neglecting temperature would underestimate deformation and misrepresent relative stress levels.

5.3 Interpretation

The observed trends in deformation, stress, and strain are consistent with the defined material models and applied boundary conditions. Higher stiffness at operating temperature reduces overall

deformation but increases local stress concentration, while more compliant materials distribute load at the expense of increased displacement. These results therefore provide a meaningful basis for comparative assessment rather than absolute performance prediction.

The analysis is limited by the use of steady-state thermal loading, linear elastic material behaviour, and idealised boundary conditions. Transient thermal gradients, piston pin compliance, and non-uniform combustion pressure were not modelled and may influence peak stress values in practice. However, by maintaining consistent modelling assumptions across all cases, the study demonstrates how CAE can be used to identify relative material behaviour and highlight key trade-offs in piston design.

6 Topology and Mass Reduction Study

Table 14 - Results for the topology study

4032 T6 - Topology study				
Targeted reduction	Mass (kg)	% of original mass	Volume (m ³)	retain threshold
100%	0.25388	100%	9.47E-05	0.5
90%	0.23336	91.92%	8.71E-05	0.5
80%	0.21639	85.23%	8.07E-05	0.5
70%	0.19907	78.41%	7.43E-05	0.5

A topology optimisation study was carried out to demonstrate the capability of CAE tools to identify potential mass reduction opportunities within the piston geometry. The study was conducted using the same loading and boundary conditions as the structural analysis, with target mass reductions of 10%, 20%, and 30%. The objective was not to generate a validated redesign, but to explore feasible material removal regions and illustrate how CAE can inform early-stage design decisions.

The results, summarised in Table 16, show a progressive reduction in mass from 0.254 kg for the baseline geometry to 0.199 kg for the 30% target case, corresponding to a retained mass of approximately 78% of the original. As the target reduction increased, the topology solution consistently removed material from low-load regions within the piston skirt and internal structure, while retaining material around the crown, pin bosses, and major load paths.

The resulting topology distributions, shown in Figure 17, indicate that significant mass reduction is theoretically achievable without compromising the primary load-carrying regions of the piston. However, no structural response analysis was performed on the optimised geometries, and the results should therefore be interpreted qualitatively rather than as evidence of structural feasibility. Manufacturing constraints, fatigue performance, and thermal effects were also not considered.

Despite these limitations, the topology study demonstrates the value of CAE in guiding design exploration by highlighting where material contributes least to structural performance. Used appropriately, these tools can support informed engineering judgement and reduce reliance on trial-and-error approaches during early design development.

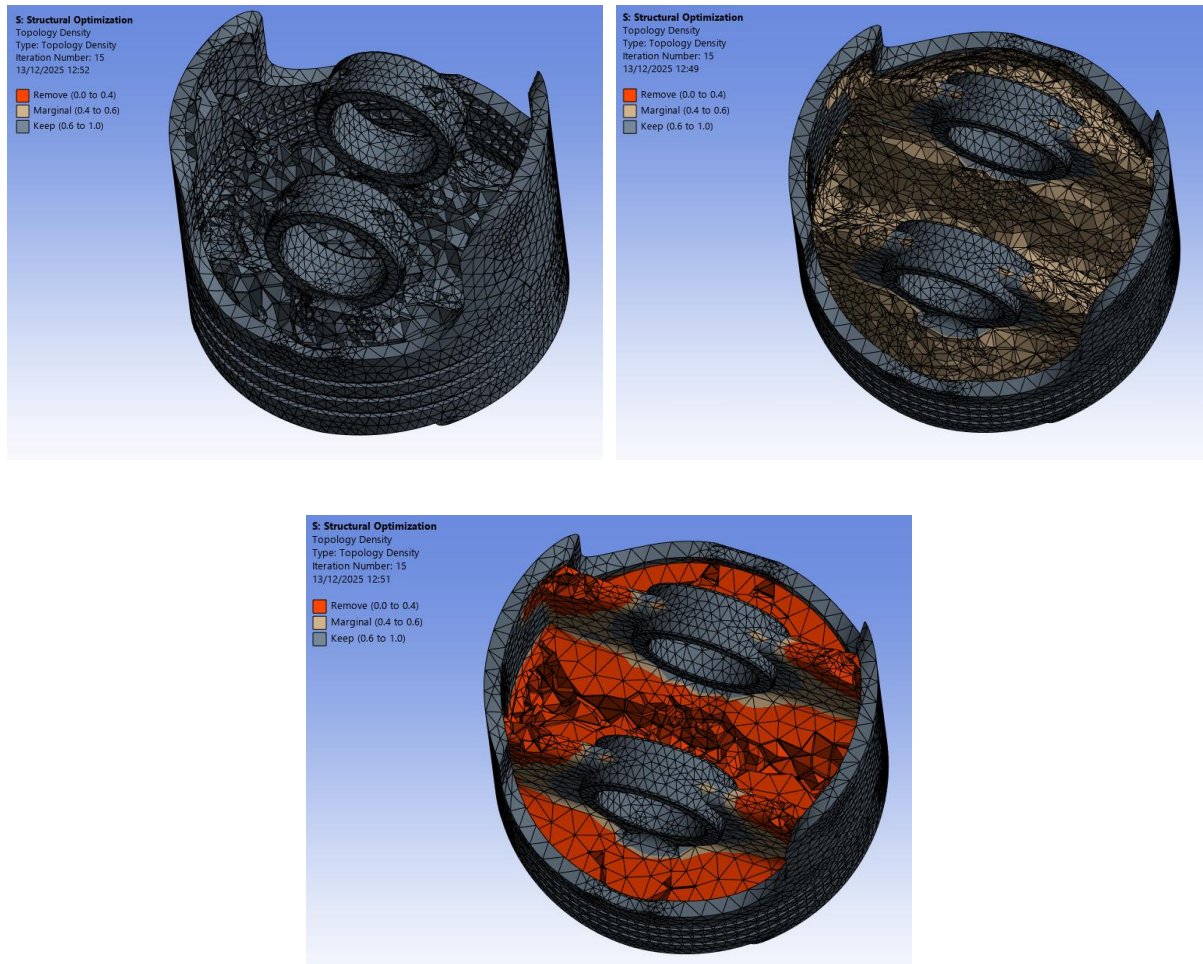


Figure 17 - Material reduction examples of the piston head using topology

7 Critical Discussion: CAE as a Design Tool

CAE proved effective in identifying structural trends, comparing material behaviour, and highlighting regions of high stress and deformation across a range of case studies. Validation against benchmark problems demonstrated that, when appropriately meshed and constrained, numerical models can reproduce reference solutions with a high level of accuracy. This provides confidence in the use of CAE for comparative assessment and early-stage design decision making.

However, the results also highlight how CAE can be misleading if model assumptions are not fully understood. Idealised boundary conditions, simplified material models, and steady state loading all influence predicted responses and can lead to overconfidence in absolute values. Local stress magnitudes are sensitive to mesh refinement and constraint definition and should not be interpreted as direct indicators of failure without further validation. Fatigue behaviour was not explicitly analysed in this study. However, the predicted von Mises stress distributions identify the piston crown and pin boss regions as likely fatigue-critical locations under cyclic loading, highlighting where further life assessment would be required in a full design validation process.(B. Liu et al., 2024)

Validation improves trust in the modelling approach but does not establish numerical results as physical truth. Even well-validated models remain approximations of real systems. Engineering judgement is therefore essential when interpreting CAE outputs, particularly when results are used to inform design changes. The studies presented demonstrate that CAE is most effective when applied as a decision-support tool, guiding understanding and comparison rather than replacing physical testing or analytical reasoning.

8 Conclusions

This report has demonstrated a structured application of CAE processes through validation, analysis, and critical interpretation. Benchmark validation studies confirmed the importance of mesh refinement, element selection, and appropriate boundary conditions when assessing numerical accuracy. These principles were applied to a piston case study, where material behaviour under combined mechanical and thermal loading was compared in a controlled and consistent manner.

The results showed that temperature-dependent material stiffness has a significant influence on deformation and stress response, reinforcing the need to include thermal effects in structural assessment. An illustrative topology optimisation study further demonstrated how CAE can be used to explore mass reduction potential and guide early design decisions.

Overall, the work highlights that CAE is a powerful engineering tool when used with validation and judgement, but that its outputs must be interpreted within the context of modelling assumptions and known limitations.

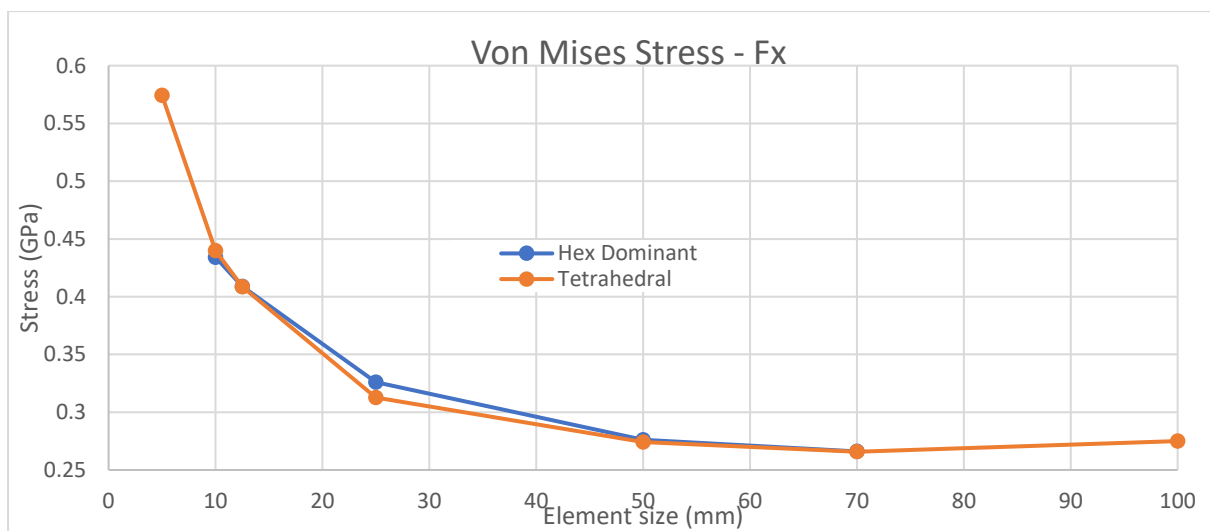
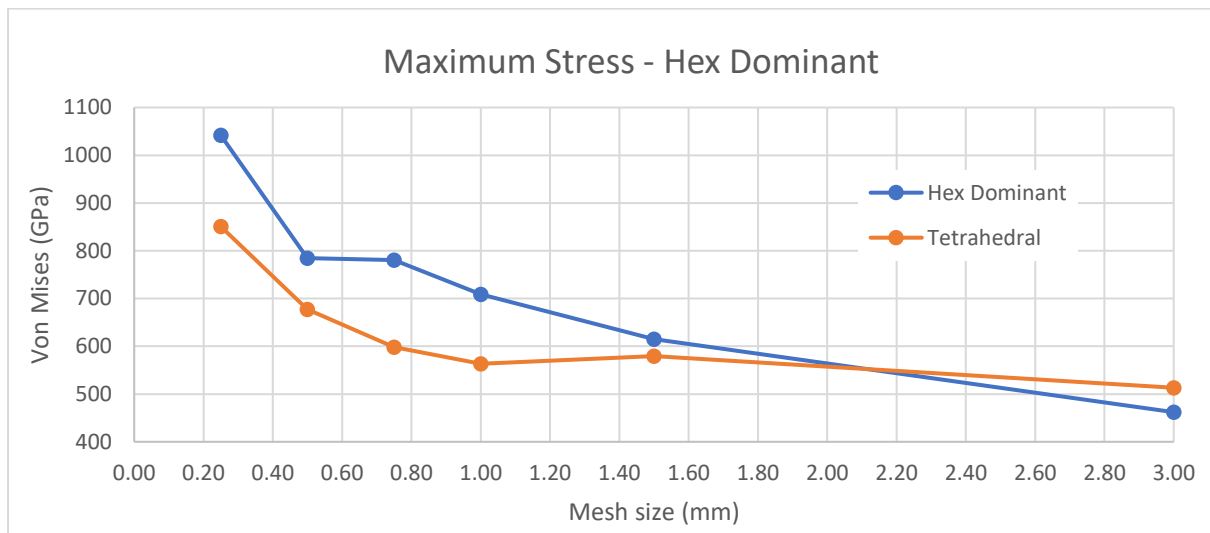
9 References

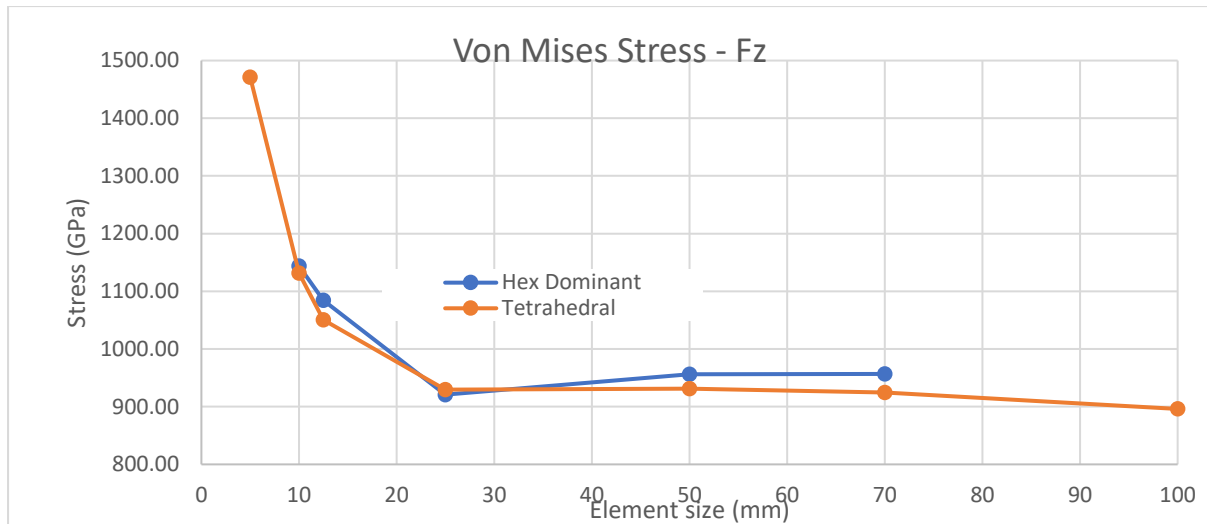
- 2618A Aluminium | DTD 5014A*. (n.d.). Retrieved December 13, 2025, from <https://www.smithshp.com/2618a-aluminium.htm>
- Aluminium 4032 | Aluminium Alloy 4032*. (n.d.). Retrieved December 13, 2025, from <https://www.smithshp.com/4032-aluminium.htm>
- Aluminum 2618-T61*. (n.d.). Retrieved December 14, 2025, from <https://www.matweb.com/search/DataSheet.aspx?MatGUID=f6d0bebbfc7248838243b7fa141431ba>
- Aluminum 4032-T6*. (n.d.). Retrieved December 14, 2025, from <https://www.matweb.com/search/datasheet.aspx?matguid=35707660584d4f7caef591324e592396&ckck=1>
- Amor-Martin, A., & Garcia-Castillo, L. E. (2021). Adaptive semi-structured mesh refinement techniques for the finite element method. *Applied Sciences (Switzerland)*, 11(8). <https://doi.org/10.3390/app11083683>
- Belytschko, Ted., Liu, W. K. ., Moran, B. ., & Elkhodary, K. I. . (2014). *Nonlinear finite elements for continua and structures*. Hoboken, New Jersey : Wiley. https://books.google.com/books/about/Nonlinear_Finite_Elements_for_Continua_a.html?id=BQpfAQAAQBAJ
- Budynas, R. G., & Keith Nisbett, J. (n.d.). *Shigley's Mechanical Engineering Design* (Eighth).
- Dlubal. (2015). *Verification Example 0042-Bending Cantilever with Axial Force*.
- Dubensky, R. G., Salansky, R. K., Isaac, M. G., King, R., & Mohr, G. (1996). Dimensional Validation of Production Parts - Comparison to CAE Design. *SAE Technical Papers*. <https://doi.org/10.4271/960552>
- Gu, L., & Yang, R. J. (2004). CAE Model Validation in Vehicle Safety Design. *SAE Technical Papers*. <https://doi.org/10.4271/2004-01-0455>

- Liu, B., Aggarwal, S. K., Zhang, siyu, Tchakam, H. U., & Luo, H. (2024). Thermo-mechanical coupling strength analysis of a diesel engine piston based on finite element method. *International Journal of Green Energy*, 21(4), 732–744. <https://doi.org/10.1080/15435075.2023.2211139>
- Liu, X. F., Wang, Y., & Liu, W. H. (2017). Finite element analysis of thermo-mechanical conditions inside the piston of a diesel engine. *Applied Thermal Engineering*, 119, 312–318. <https://doi.org/10.1016/j.applthermaleng.2017.03.063>
- Schwer, L. E. (n.d.). *An overview of the PTC 60/V&V 10: guide for verification and validation in computational solid mechanics*. <https://doi.org/10.1007/s00366-007-0072-z>
- Sifakis, E., & Barbič, J. (2015). Finite Element Method Simulation of 3D Deformable Solids. In *Synthesis Lectures on Visual Computing* (Vol. 7, Issue 3). Springer Science and Business Media LLC. <https://doi.org/10.2200/S00674ED1V01Y201509CGR021>
- Szabó, B., & Babuška, I. (2021). Finite element analysis: method, verification, and validation. In B. Szabó & I. Babuška (Eds.), *Wiley series in computational mechanics* (2nd ed.). John Wiley & Sons, Inc. https://books.google.com/books/about/Finite_Element_Analysis.html?id=V_UqEAAAQBAJ

10 Appendix

10.1 Validation





10.2 Piston case study

