



Wales Institute of Science and Art

Application of CAE Processes

Structural & Fluid Analysis (ACHA6010)

Brody Crockford

2300783

14/12/2025

Contents

Table of Figures	III
Table of Abbreviations	IV
1 Introduction	1
1.1 Problem context & motivation	1
1.2 Objectives & success criteria	1
2 Literature Review	2
3 Assembly & Specifications	3
3.1 Material Specifications	4
4 Analysis Setup & Testing	6
4.1 Rigid Dynamics	6
4.2 Steady-State Thermal	7
4.3 Static Structural	8
5 Analysis & Optimisation	10
5.1 Iteration 1	10
5.2 Iteration 2	11
5.3 Iteration 3	13
5.4 Iteration 4	15
5.5 Overview & Future Iterations	18
5.6 Fatigue Analysis	18
6 Validation & Boundary Conditions	19
6.1 Analytical Case	19
6.2 ANSYS Replication	19
6.2.1 Cantilever	19
6.2.2 Bimetallic Strip	20
6.3 Constraint Location Study	22
6.3.1 Gudgeon Pin	22
6.3.2 Piston Skirt	23
6.3.3 Crown Underside	24
6.3.4 Oil Ring	25
6.4 Mesh Independence Study	26
7 Discussion & Conclusions	30
7.1 Benefits & Limitations of CAE	30
7.2 Conclusions	30
8 References	32
9 Appendix	33
9.1 Appendix A: Piston Temperature & Pressure Data	33

9.2	Appendix B: Validation Analytical Cases.....	34
9.2.1	Structural Validation Case: Cantilever with Axial Load	34
9.2.2	Thermal Validation Case: Bimetallic Strip	36

Table of Figures

Figure 1: The starting engine assembly in SolidWorks.	3
Figure 2: The assembly within ANSYS Mechanical.	3
Figure 3: Material properties against temperature for Aluminium 6061-T6	5
Figure 4: ANSYS Workbench workflow for a complete analysis.	6
Figure 5: Resulting force results from the motion simulation.	7
Figure 6: Temperature study result, with heat propagation through the piston.....	8
Figure 7: ANSYS Mechanical simulation tree for the Static Structural solver.....	8
Figure 8: Results from the structural test on the base design.	9
Figure 9: Factor of safety result of the base design, with key points annotated.	10
Figure 10: Base design (left) compared to Iteration 1 (right).....	10
Figure 11: Results of Iteration 1.	11
Figure 12: Iteration 2 CAD model.....	12
Figure 13: Results of Iteration 2.	13
Figure 14: Iteration 3 CAD Model.....	14
Figure 15: Factor of safety result for Iteration 3.	15
Figure 16: Iteration 4 CAD model.....	16
Figure 17: Thermal result for Iteration 4.	16
Figure 18: Factor of safety result for Iteration 4.	17
Figure 19: Simulation tree for the uniform load validation.....	19
Figure 20: The cantilever model in ANSYS Mechanical, with loadings applied.	20
Figure 21: Simulation tree for the bimetallic strip validation.	21
Figure 22: The bimetallic strip model in ANSYS Mechanical, with thermal loads applied.	21
Figure 23: Results of the bimetallic strip simulation.	22
Figure 24: Results using the gudgeon pin as the fixed location	23
Figure 25: Results using the skirt as the fixed location.	24
Figure 26: Results using the crown underside as the fixed location.....	25
Figure 27: Results using the oil ring as the fixed location.	26
Figure 28: Von Mises stress change for uniform mesh size.....	27
Figure 29: Von Mises stress using a uniform 5mm mesh.	28
Figure 30: Von Mises stress results using the user-defined variable mesh.....	28
Figure 31: Mesh quality results for the user-defined mesh.	29

Table of Abbreviations

Definition	Abbreviation
Computer Aided Engineering	CAE
Finite Element Analysis	FEA
Computational Fluid Dynamics	CFD
Degrees of Freedom	DoF
Factor of Safety	FoS

1 Introduction

1.1 Problem context & motivation

Computer Aided Engineering (CAE) is now a common practice in engineering, with simulation tools like Finite Element Analysis (FEA) and Computational Fluid Dynamics (CFD) being standard practice during the development of a design, along with kinematic simulations for moving assemblies. By using CAE, iteration time is reduced, testing infrastructure is almost redundant and overall design efficiency is increased. (Vishwas, 2013)

This report focuses on the use of CAE tools to optimise the design of a piston, with the target to reduce mass – a common task in the motorsport world. By using dynamic, thermal and structural simulation, validated results allow for iterative design improvements to take place.

1.2 Objectives & success criteria

The author aims to complete a full simulation and validation of a piston and its relative assembly, looking into boundary conditions and the solver's accuracy. Using these simulations, iterative improvements to the piston's design can take place, reducing mass while still operating within the yield strength of the chosen material.

2 Literature Review

CAE enables rapid iteration and reduced prototype dependence, but its effectiveness relies heavily on solver accuracy and appropriate modelling assumptions (Vishwas, 2013). For piston studies, accurate temperature-dependent material data is essential, as aluminium alloys show significant variation in strength and stiffness under thermal loading (Shetty et al., 2014). Broader reviews confirm that alloy composition and manufacturing method further influence mechanical behaviour, making material definition a major contributor to simulation fidelity.

(Liu and Glass, 2013) demonstrate that mesh density is another critical factor, with coarse meshes distorting stress fields while excessively fine meshes provide diminishing returns, reinforcing the need for mesh-independence checks rather than default settings. Equally, (Malak and Paredis, 2004) emphasise that CAE models must be validated within the context of their intended use, as predictions are only reliable when their assumptions and boundary conditions align with reality.

Collectively, the literature highlights that CAE is highly effective for optimisation but depends on accurate material modelling, suitable mesh strategies, and structured validation—principles central to the methodology in this study.

3 Assembly & Specifications

Figure 1 shows the full assembly used in this study, as modelled in SolidWorks by (Noble, 2025a).

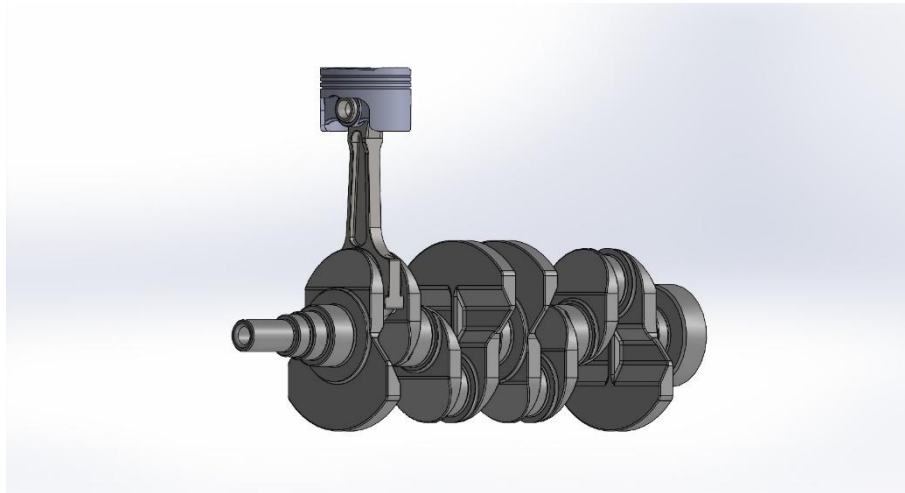


Figure 1: The starting engine assembly in SolidWorks.

This model is imported to ANSYS Workbench for the simulation. While the solvers automatically detect mating faces, these contacts must be overwritten by manual inputs, declaring the degrees of freedom (DoF) each component relationship can have.

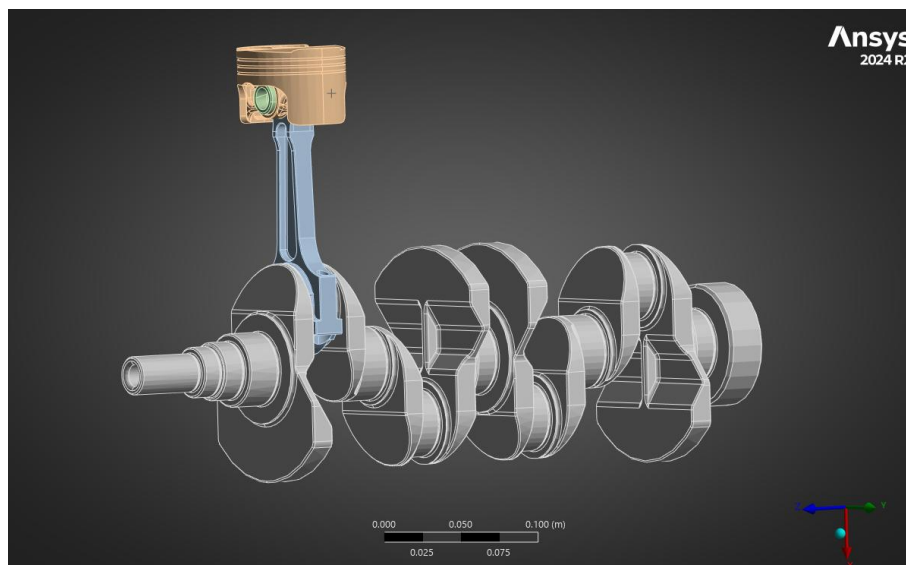


Figure 2: The assembly within ANSYS Mechanical.

Table 1: Part contacts: type & degree of freedom axis.

Part	Related Part	Movement Type	Axis (Global)
Crankshaft	Ground	Revolute	Z
Crankshaft	Connecting Rod	Revolute	Z
Connecting Rod	Gudgeon Pin	Revolute	Z
Gudgeon Pin	Piston	Revolute	Z
Piston	Ground	Translation	X

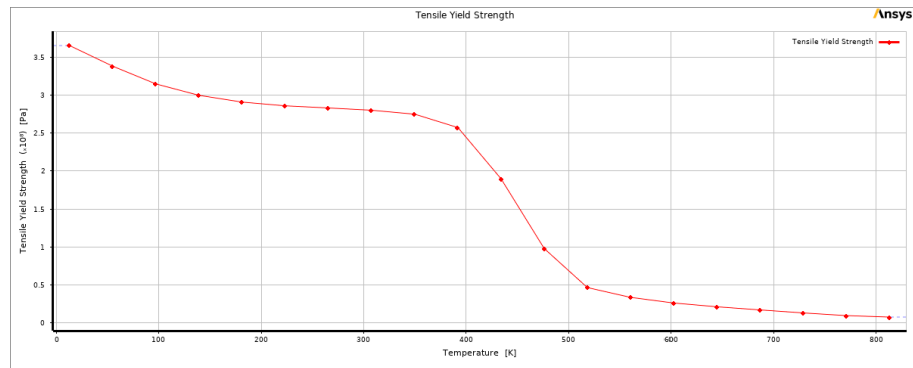
A limitation of defining relationships in this way is that connections are treated as frictionless, whereas the real joints have friction and lubrication. This idealised assumption removes rotational drag and the consequent temperature effects from the simulation. Although ANSYS can model these behaviours, doing so has computational cost and requires input data that is not available for this study.

3.1 Material Specifications

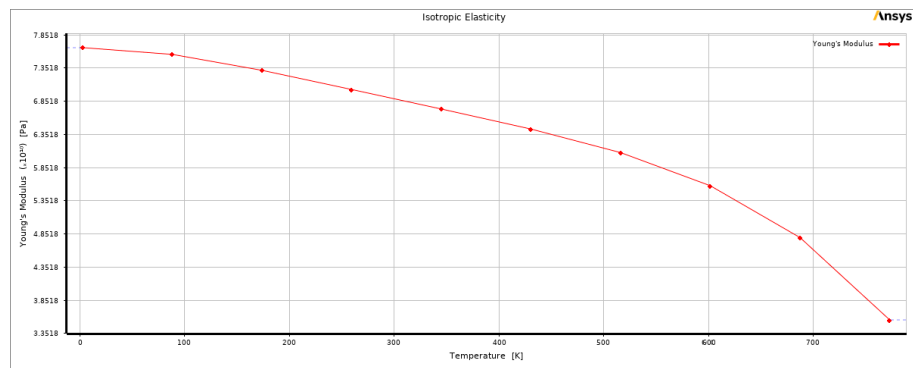
Table 2: Material specified for each component.

Part	Material	Source
Crankshaft	Structural Steel	ANSYS Workbench
Connecting Rod	Structural Steel	ANSYS Workbench
Gudgeon Pin	Structural Steel	ANSYS Workbench
Piston	Aluminium 6061-T6	ANSYS Granta EduPack

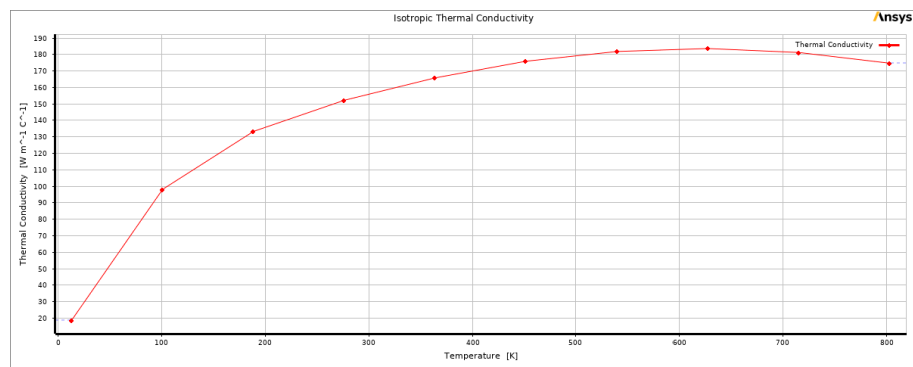
Aluminium 6061-T6 is selected as the material for the analysed component due to its common use for pistons according to (Shetty *et al.*, 2014). Using ANSYS Granta EduPack, a preset material specification can be loaded to Ansys Workbench for the study. Using this allows for higher quality results as the material specification is known for a range of temperatures (as shown in Figure 3) which is important for this use case.



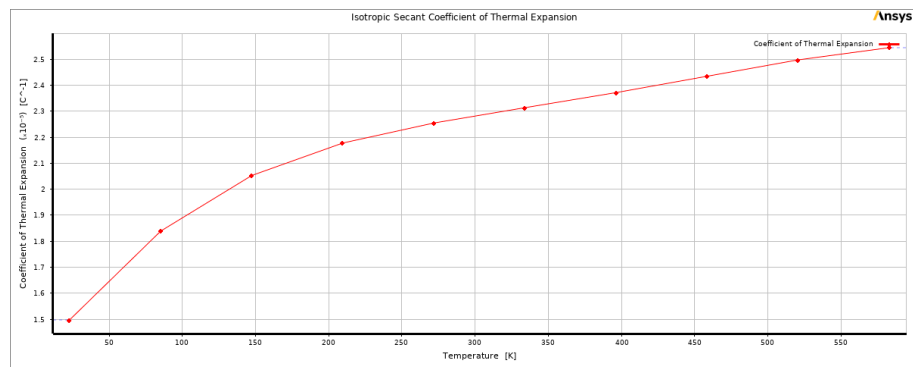
(a) Tensile yield strength



(b) Young's modulus



(c) Thermal conductivity



(d) Thermal expansion

Figure 3: Material properties against temperature for Aluminium 6061-T6

4 Analysis Setup & Testing

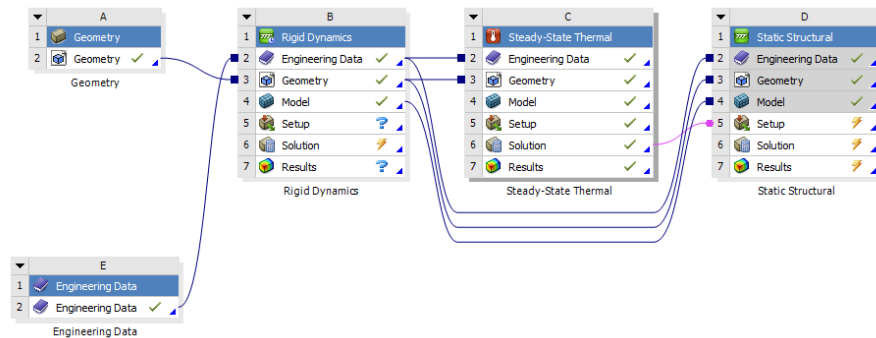


Figure 4: ANSYS Workbench workflow for a complete analysis.

4.1 Rigid Dynamics

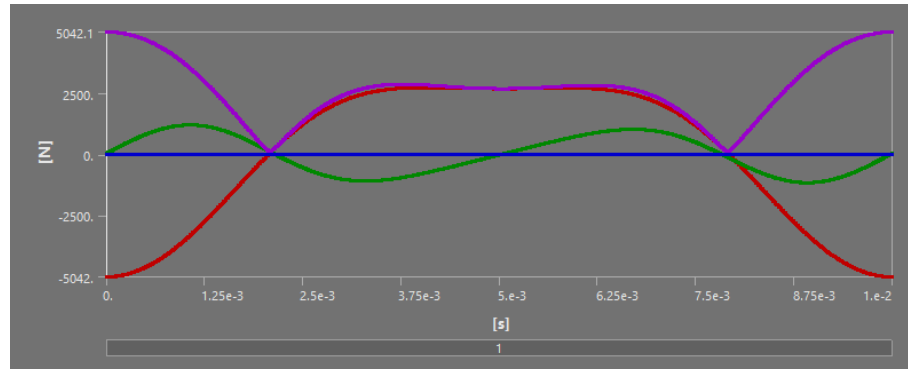
To derive the forces acting on the piston, a kinematic analysis must be done on the assembly to calculate forces at a joint over a movement cycle. Once a model is loaded and the DoF are set, a rotational velocity input is defined, using the crankshaft to ground contact. For this study, 6000RPM is used, as this is a generic high RPM value for automotive applications and will give significant forces for the simulation.

User-defined simulation parameters also include end time and step time, to control how long the simulation is run for. In this study, the following parameters are used.

Table 3: Rigid Dynamics simulation parameters.

Rigid Dynamics - Step Controls	
Step End Time	1.00E-02
Auto Time Stepping	Off
Time Step	1.00E-03

The output of the simulation is defined as the force acting on a joint. For this study, the joint between the piston and gudgeon pin is used, to establish forces on the piston.



(a) Graph for one cycle (complete study)

	Time [s]	Joint Probe (Total Force X) [N]	Joint Probe (Total Force Y) [N]	Joint Probe (Total Force Z) [N]	Joint Probe (Total Force Total) [N]
1	0.	-5042.	40.42	3.5519e-013	5042.1
2	1.e-005	-5041.7	57.801	3.5517e-013	5042.
3	2.e-005	-5041.1	75.175	3.5513e-013	5041.7
4	3.e-005	-5040.2	92.539	3.5507e-013	5041.1
5	4.e-005	-5039.	109.89	3.5498e-013	5040.2
6	5.e-005	-5037.4	127.22	3.5487e-013	5039.
7	6.e-005	-5035.5	144.53	3.5474e-013	5037.6
8	7.e-005	-5033.3	161.82	3.5458e-013	5035.9
9	8.e-005	-5030.8	179.08	3.544e-013	5033.9
10	9.e-005	-5027.9	196.31	3.542e-013	5031.7
11	1.e-004	-5024.7	213.5	3.5398e-013	5029.2
12	1.1e-004	-5021.2	230.65	3.5373e-013	5026.5
13	1.2e-004	-5017.4	247.76	3.5346e-013	5023.5
14	1.3e-004	-5013.2	264.82	3.5317e-013	5020.2
15	1.4e-004	-5008.7	281.83	3.5285e-013	5016.7
16	1.5e-004	-5003.9	298.78	3.5251e-013	5012.9
17	1.6e-004	-4998.8	315.68	3.5215e-013	5008.8
18	1.7e-004	-4993.4	332.52	3.5177e-013	5004.4
19	1.8e-004	-4987.6	349.3	3.5136e-013	4999.8

(b) Table with numerical results

Figure 5: Resulting force results from the motion simulation.

As shown in Figure 5, the forces happen through multiple axis. To export the data, the highest total load is identified and saved as the motion loads, as this will include forces along all axes. This file allows parts to be suppressed for later simulations without losing results.

4.2 Steady-State Thermal

From this stage onwards, all parts are suppressed apart from the piston, as this separates the part of interest.

Using data from (Noble, 2025b), detailed in Appendix A, a steady-state thermal simulation was run to calculate the material temperature of the piston at different points. This process means inputting data as convections onto surfaces with different temperatures and transfer rates, allowing the solver to calculate the temperature dissipation within the material.

A thermal simulation is crucial for this analysis due to the way materials alter their properties with temperature, as previously shown in 3.1. As a piston is subject to significant heat for prolonged periods of time, the structural simulation must use the material properties accurate to the conditions the part will work in.

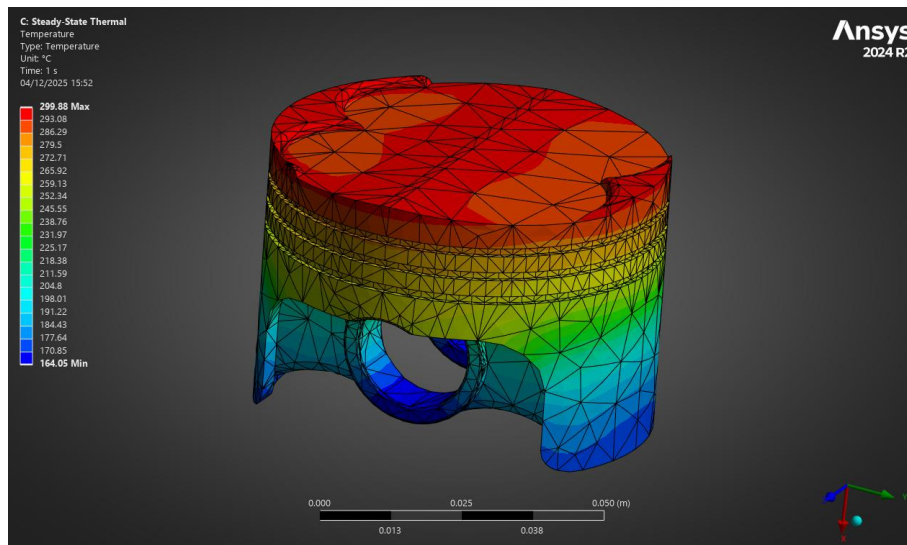


Figure 6: Temperature study result, with heat propagation through the piston.

4.3 Static Structural

The thermal result is fed into the structural solver, with the motion loads being imported as the assembly is suppressed. Along with this, a pressure is applied to the piston crown, detailed in Appendix A, representing combustion and contributing to input loads. Constraints are also set at this point, using a fixed location as discussed in 6.3. The resulting simulation tree is shown in Figure 7.

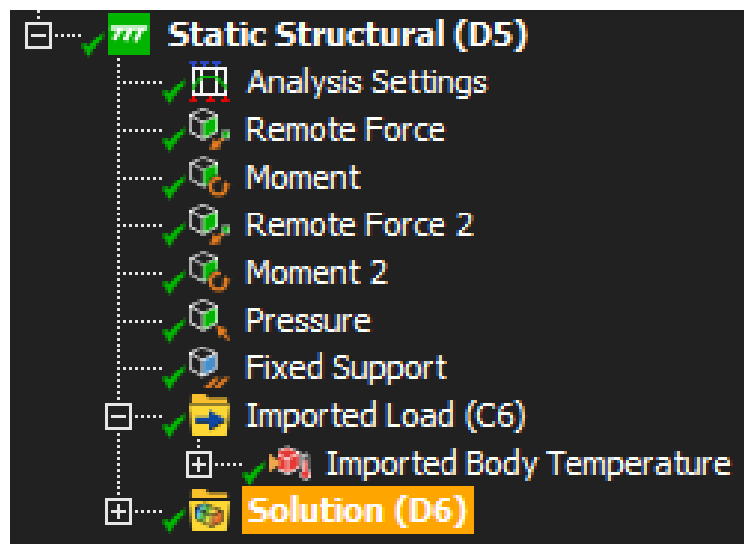
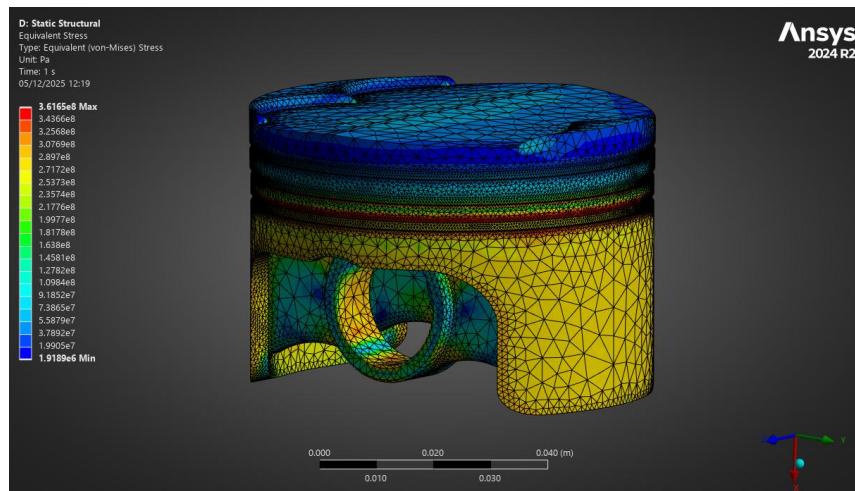
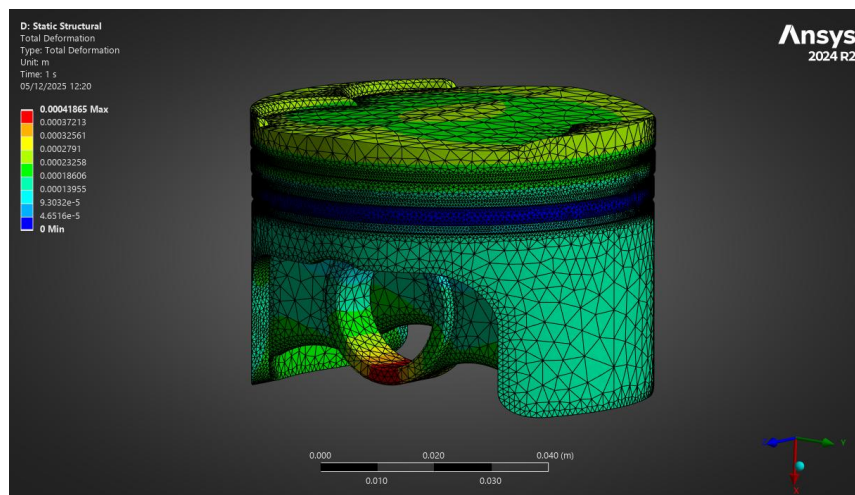


Figure 7: ANSYS Mechanical simulation tree for the Static Structural solver.

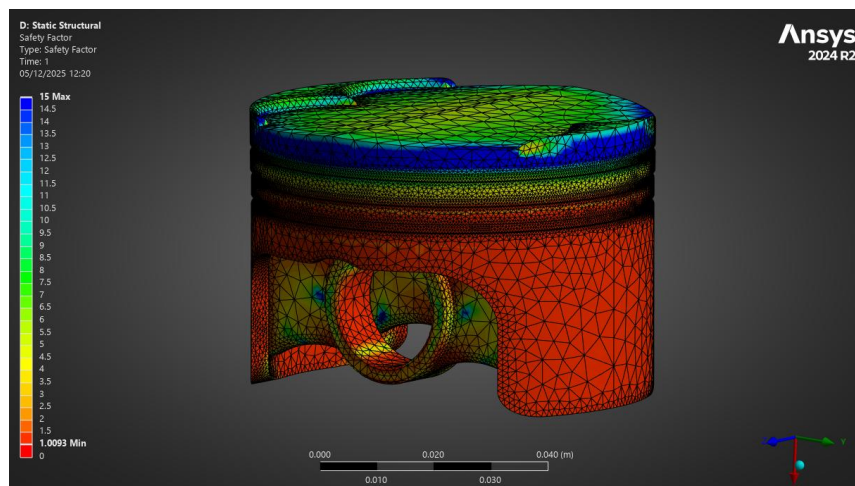
Once set up with the appropriate loads, boundary conditions and mesh, solutions can be defined, and the simulation can be run. Results for the base model can be seen in Figure 8.



(a) Von-Mises (equivalent) stress.



(b) Total deformation.



(c) Factor of Safety, in relation to the Von-Mises stress.

Figure 8: Results from the structural test on the base design.

5 Analysis & Optimisation

From the results in Figure 8, several areas clearly offer potential for material removal, with the FoS result providing an immediate indication of strong and weak geometry.

For each iteration, the rigid dynamics and steady-state thermal studies are rerun to regenerate the input conditions, as changes in mass or surface area would otherwise skew these results.

5.1 Iteration 1

Initial observations on the base model show that the gudgeon pin supports have areas of high safety, between 3 and 15 across the structure, as shown in Figure 9.

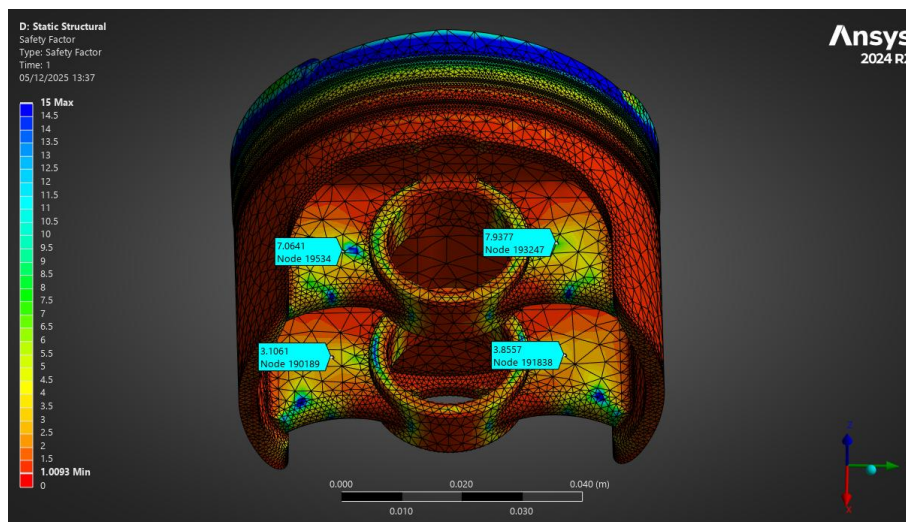


Figure 9: Factor of safety result of the base design, with key points annotated.

This meant that material could be removed from these structures, with cutouts being made as shown in Figure 10. This design reduces mass by 8.89 grams.

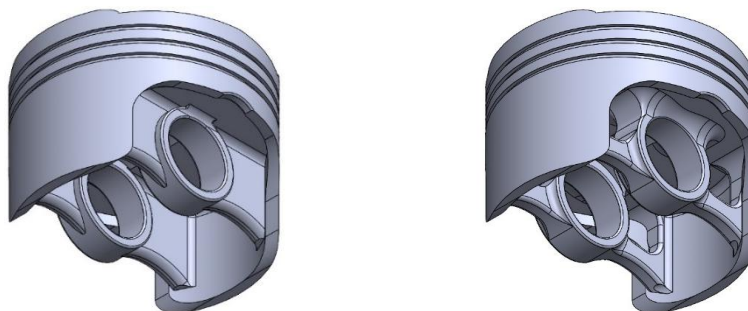
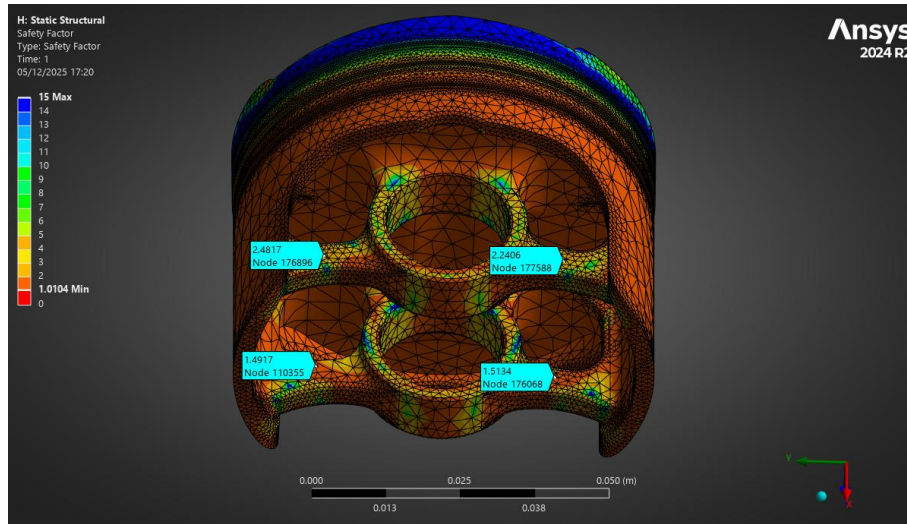
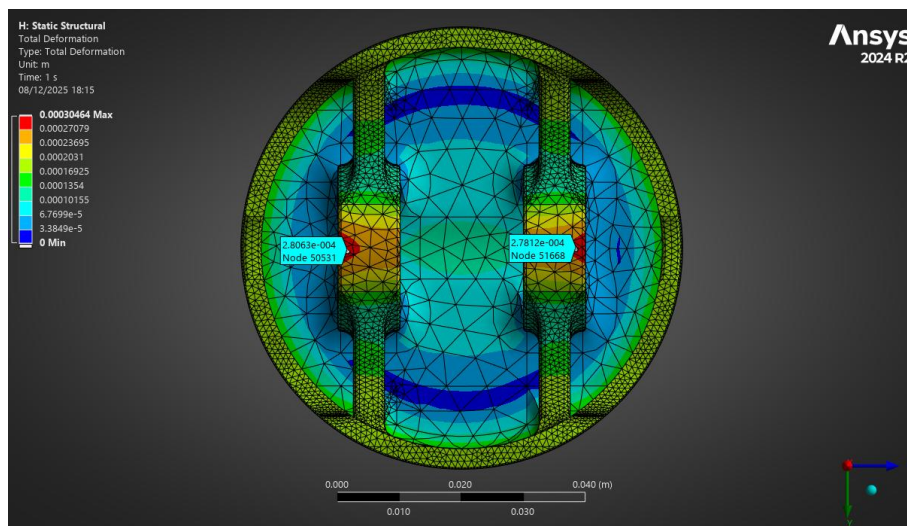


Figure 10: Base design (left) compared to Iteration 1 (right).



(a) Factor of safety, with key points annotated.



(b) Total Deformation, with key points annotated.

Figure 11: Results of Iteration 1.

As shown in Figure 11(a), the FoS along the braces has become much more consistent, varying from 1.5 to 2.5 along most of the surface. This is due to the stress being more evenly distributed through the new design. In Figure 11(b), it is shown that peak deformation happens at the lower gudgeon pin locations due to the downwards force, meaning additional bracing could be beneficial for this area.

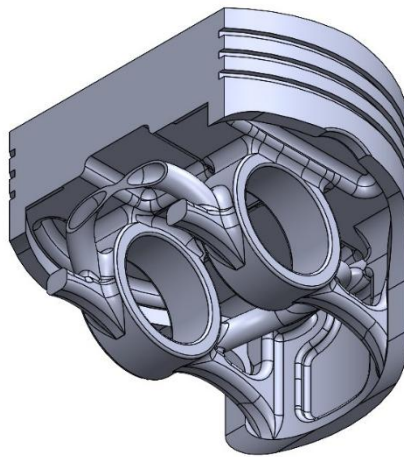
5.2 Iteration 2

Building on Iteration 1, Iteration 2 adds 5 mm bracing tubes from each gudgeon pin boss to the piston underside to cut deformation in that region while enabling further material removal from the main brace. The tubes use a 0.5 mm wall to save mass, allowing the main brace to be reduced by 1 mm. Fillet radii were trimmed where carried over, and pockets were added to the underside and skirt to remove mass and raise the second moment of area. Overall, these

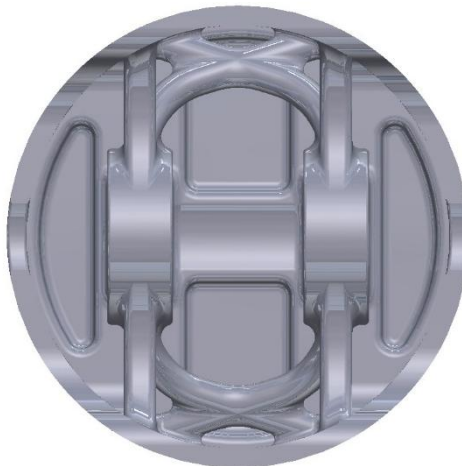
changes remove a further 14.41 g, bringing the reduction to 23.3 g compared with the base design.

$$I_{NA} = \frac{bd^3}{12}$$

Equation 1: Second Moment of Area

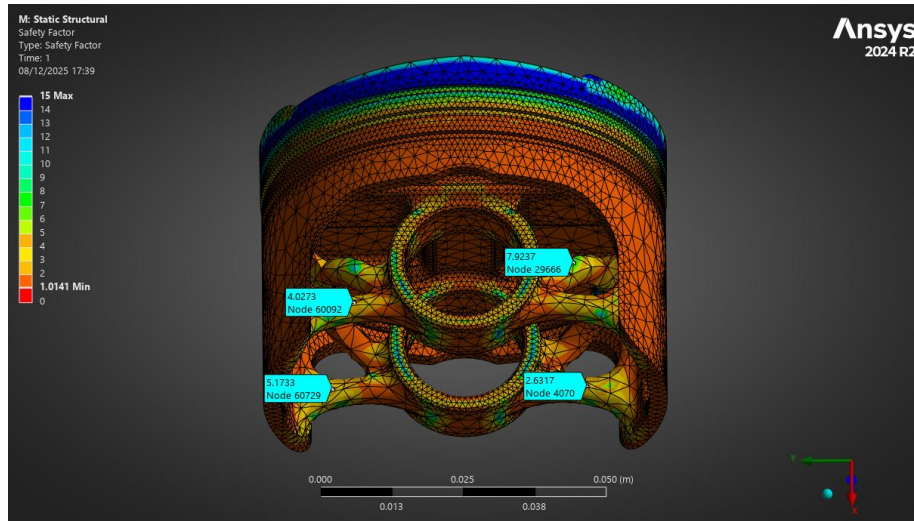


(a) Section view.

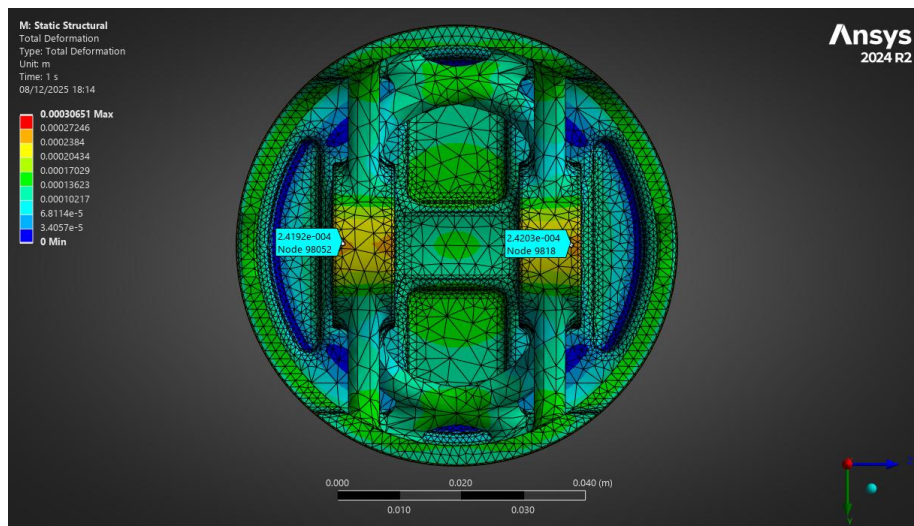


(b) Underside.

Figure 12: Iteration 2 CAD model.



(a) Factor of Safety, in relation to Von-Mises stress, with key points annotated.



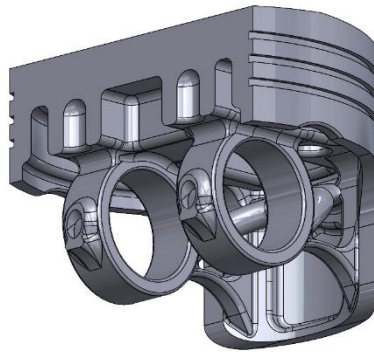
(b) Total Deformation, with key points annotated.

Figure 13: Results of Iteration 2.

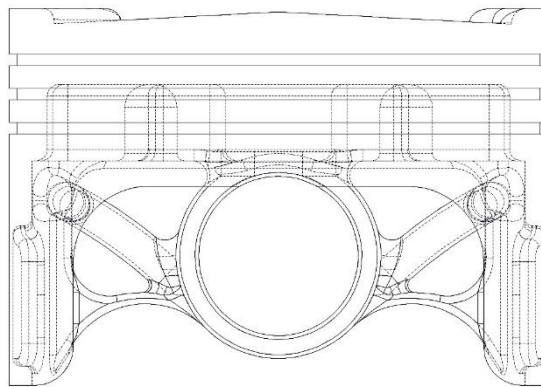
Iteration 2 results show that this design increases both strength and rigidity over Iteration 1, with factor of safety in the main supports varying from 2.6 upwards. Deformation has also been reduced on the underside, as peak deformation no longer occurs at the gudgeon pin but on the crown. The pocketing has caused more inconsistent deformation, although this is still marginal.

5.3 Iteration 3

Since Iteration 2 still met its structural requirements, the next iteration pushed the same principles further. The main gudgeon pin braces were thinned to 2 mm at the mid-point, pocket depths were increased, and additional pockets were added to remove more material. These changes achieved a 27.81 g mass reduction.



(a) Section view.



(b) Transparent view.

Figure 14: Iteration 3 CAD Model

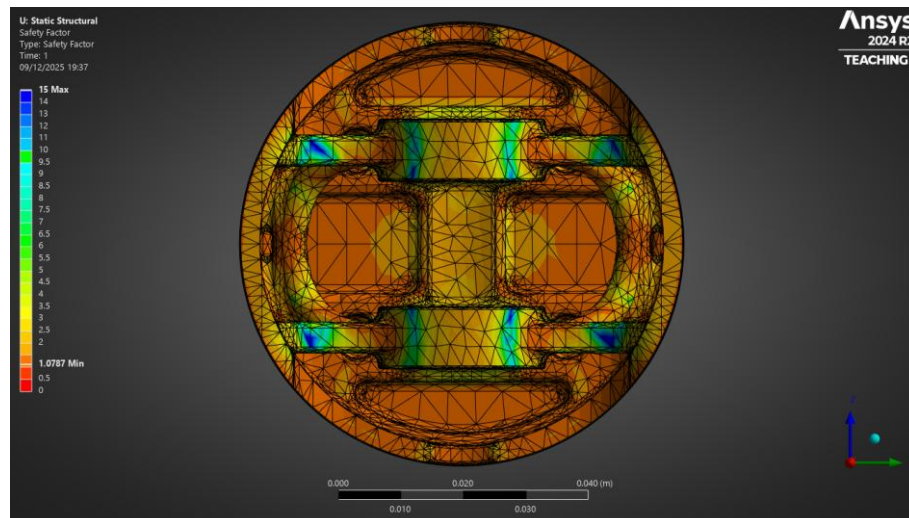
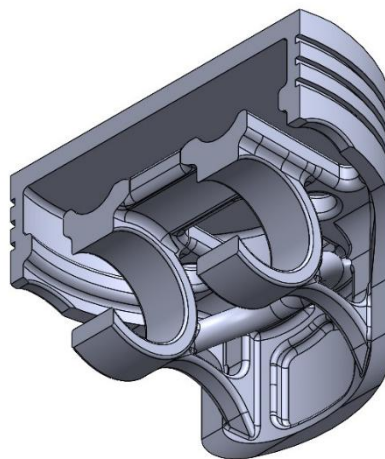


Figure 15: Factor of safety result for Iteration 3.

This iteration increases the minimum FoS, reducing stress in the main supporting structures while maintaining crown-underside strength. The improvement comes from the higher second moment of area, with the added ribbing providing greater stiffness than a flat surface.

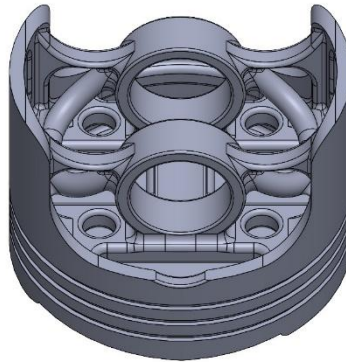
5.4 Iteration 4

As Iterations 2 and 3 showed improved strength, Iteration 4 introduced a hollow section beneath the crown, leaving the bracing “floating” and only tied into the skirt. This approach delivers a large mass reduction and is intended to push the design to failure to understand the limiting geometry.



(a) Section view.

(b)



(b) Underside

Figure 16: Iteration 4 CAD model.

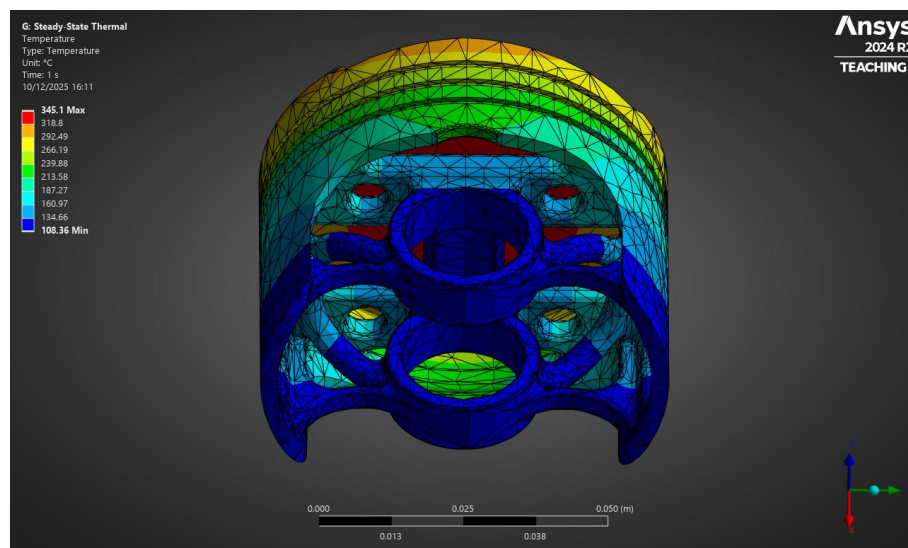
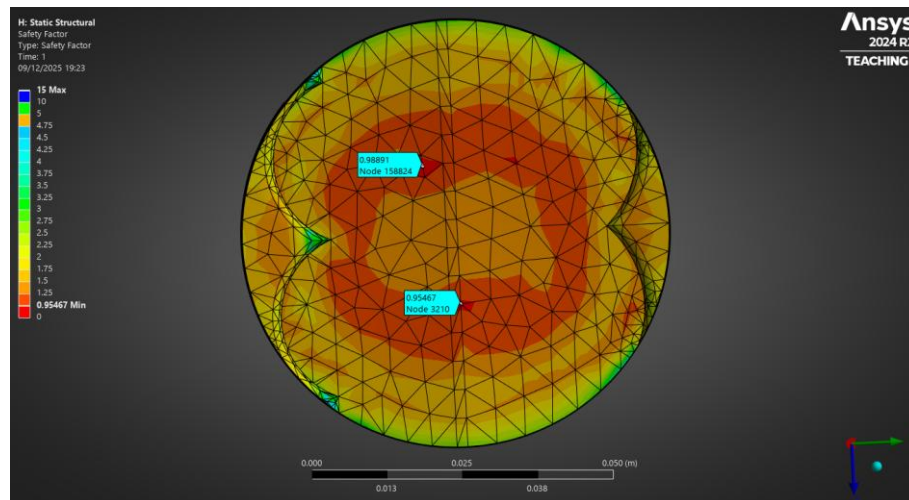
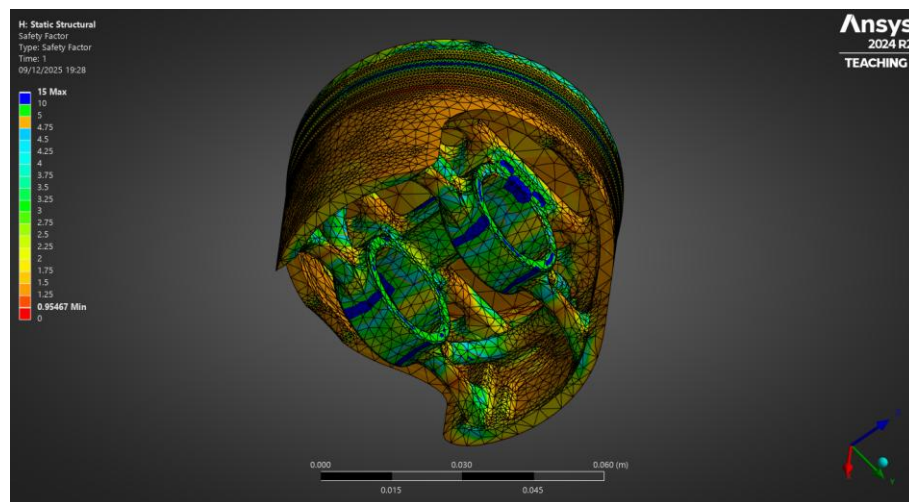


Figure 17: Thermal result for Iteration 4.

The temperature results in Figure 17 highlight a major factor behind the strength gains in earlier iterations. Increasing surface area, here through a large open section, allows more heat from combustion to dissipate, lowering temperatures around the gudgeon pin. As shown in the material data in 3.1, this temperature drop directly improves material strength.



(a) Crown, top-down view, with key points annotated.



(b) Underside

Figure 18: Factor of safety result for Iteration 4.

This part fails at the crown due to the combustion pressure, as shown in Figure 18, due to the thickness of the part being <5mm at those points. This failure could be easily repaired by increasing thickness or adding a brace along the high stress 'ring' on the piston crown. The other overwhelming issue with this new design is that the lack of vertical support causing large deformation, which in turn creates stress within the supporting features of the gudgeon pin. While these areas do not exceed a FoS of one, the design is using the materials elasticity to deform and relieve stress, which is problematic for a piston.

5.5 Overview & Future Iterations

Table 4: Overview of analysed designs.

Iteration Overview				
Iteration #	Mass [g]	Mass [%]	Maximum Deflection [mm]	Minimum FoS
Base	255.77	100.0	0.41865	1.0093
Iteration 1	246.88	96.5	0.30464	1.0104
Iteration 2	232.47	90.9	0.30651	1.0141
Iteration 3	204.66	80.0	0.2797	1.0787
Iteration 4	130.27	50.9	1.3847	0.9547

Building on this study, future iterations could refine Iteration 4 by adding bracing around the weak crown region or locally increasing thickness to improve stress distribution across the top surface. These changes could also provide extra vertical support to reduce gudgeon pin deformation. Increasing stiffness will likely shift stress elsewhere, so low-FoS regions may need reinforcing to accommodate the redistribution.

5.6 Fatigue Analysis

Using the initial and final designs, a preliminary zero-based fatigue analysis was carried out to estimate cycle life before failure. No mean-stress correction was applied, highlighting how the study could be refined with further research. Early results indicate that the final, lighter design achieves a longer life, likely due to its reduced operating temperatures and increased structural strength.

Table 5: Fatigue study results and comparison.

Iteration #	Minimum # of Cycles	Life Expectancy (%)
Base	19432	100.0
Iteration 3	55538	285.8

6 Validation & Boundary Conditions

Before starting the main methodology, a thorough validation and boundary-condition review is carried out to ensure the study's conclusions could be trusted. This stage is essential in CAE, confirming that the solver produces reliable, repeatable results without inconsistencies.

6.1 Analytical Case

Simple analytical cases from (Dlubal Software, 2023) were chosen to validate both the structural and thermal solvers. Details of these cases can be found in Appendix B.

6.2 ANSYS Replication

Using the parameters from the analytical case, replica tests are created in ANSYS, using the static structural solver.

6.2.1 Cantilever

This test validates the structural solver against the analytical case and another solver, using the same boundary conditions, mesh parameters and loads. This is to prove that the structural results are comparable to traditional methods. A study is made in ANSYS Mechanical as shown in Figure 19 and 20.

The material used for this study is user-defined using the values found in the analytical case.

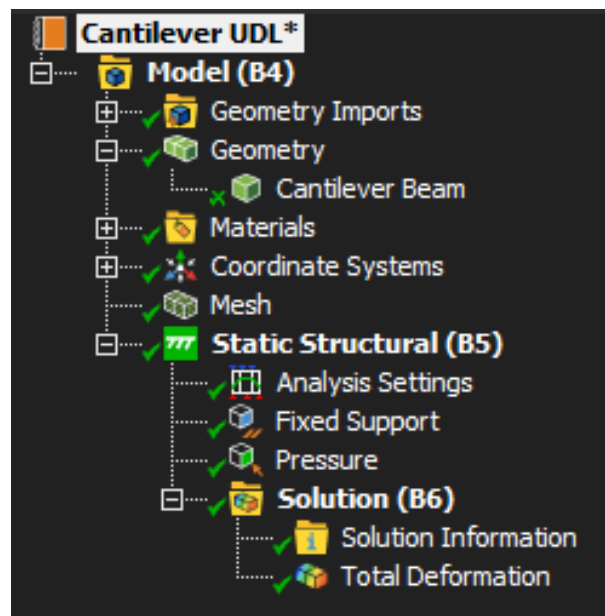


Figure 19: Simulation tree for the uniform load validation.

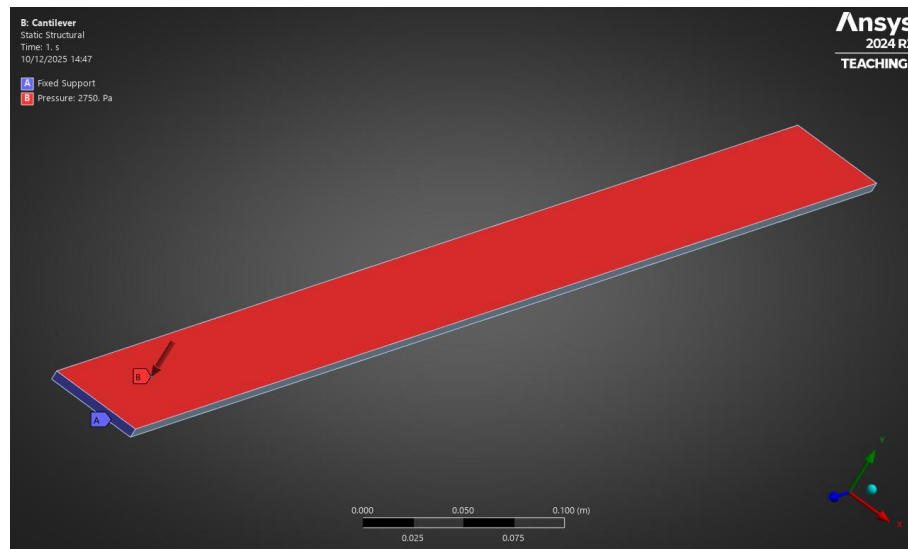


Figure 20: The cantilever model in ANSYS Mechanical, with loadings applied.

Table 6: Results comparison for the cantilever validation.

Cantilever with Uniform Load		
Source	Deformation	% Diff
Dlubal - Calculation	157.14	N/A
Dlubal - RFEM 5, Plate	157.13	99.99%
ANSYS	157.15	100.00%

The results in Table 6 show the ANSYS solver to be almost identical to both the analytical solution and RFEM 5 solver, allowing structural results to be trusted as true.

6.2.2 Bimetallic Strip

This test is to validate ANSYS's method for processing temperature expansion, by giving two different materials a temperature loading and measuring the bend due to varying expansion. This shows the thermal effects are contributing to the structural solver and yielding comparable results.

The material used for this study is user-defined using the values found in the analytical case.

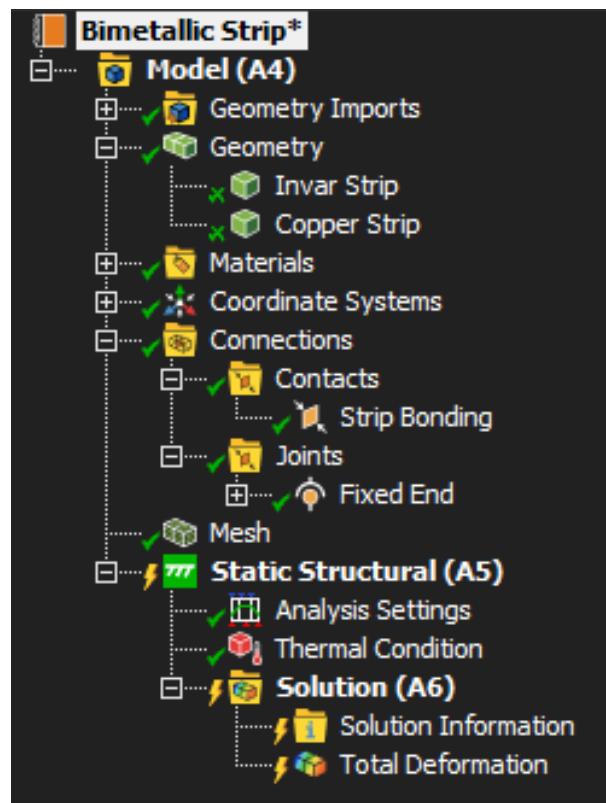


Figure 21: Simulation tree for the bimetallic strip validation.

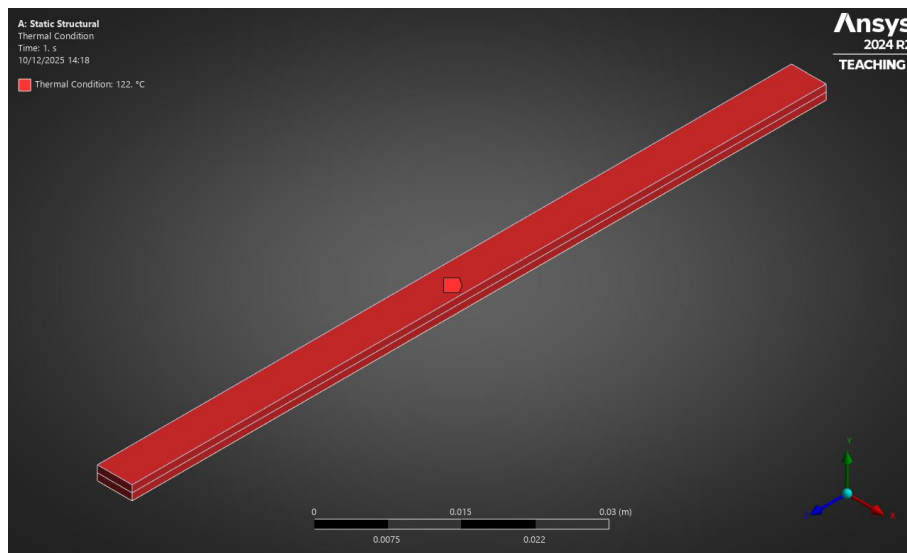


Figure 22: The bimetallic strip model in ANSYS Mechanical, with thermal loads applied.

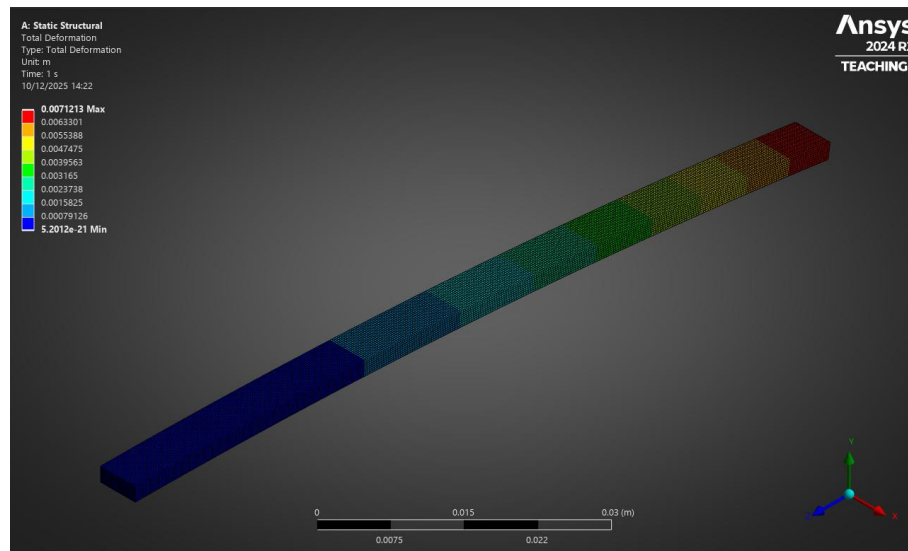


Figure 23: Results of the bimetallic strip simulation.

Table 7: Results Comparison for the bimetallic strip validation.

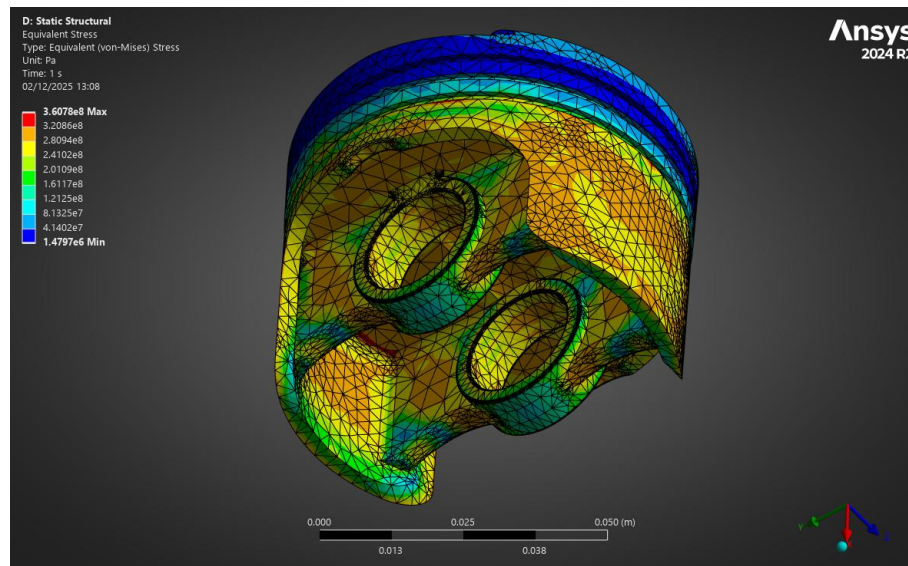
Bimetallic Strip		
Source	Deformation	% Diff
Dlubal - Calculation	7.049	N/A
Dlubal - RFEM 5, Plate	7.047	99.97%
Dlubal - RFEM 5, Solid	7.064	100.21%
ANSYS	7.121	101.02%

6.3 Constraint Location Study

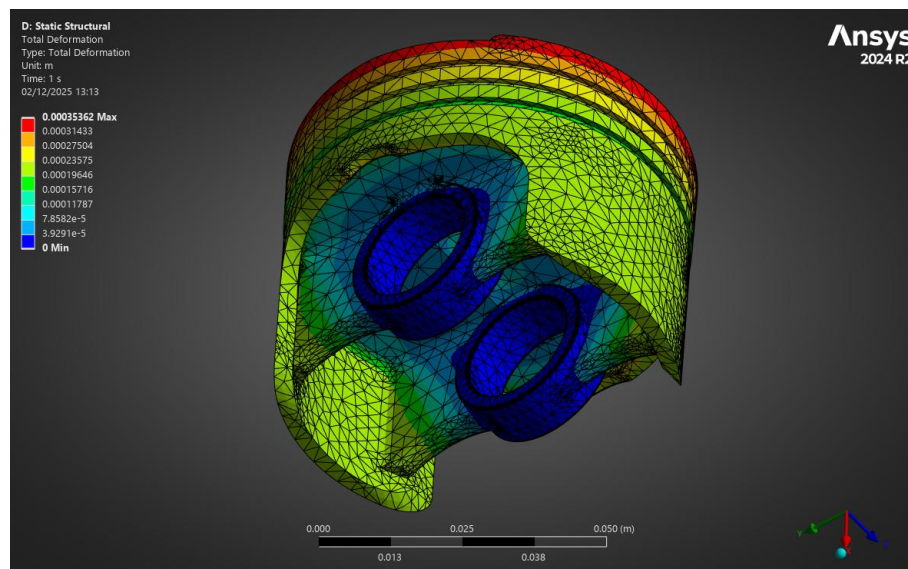
When working with less rigid materials, weak springs - typically used to stabilise a model - are overwhelmed by motion loads, leading to excessive deformation. A fixed constraint is therefore needed to anchor the model realistically. Several constraint locations were tested to determine the most suitable approach, using the starting piston model and mesh settings.

6.3.1 Gudgeon Pin

The gudgeon pin bore is fully fixed to represent a rigid connection to the conrod.



(a) Von-Mises stress.



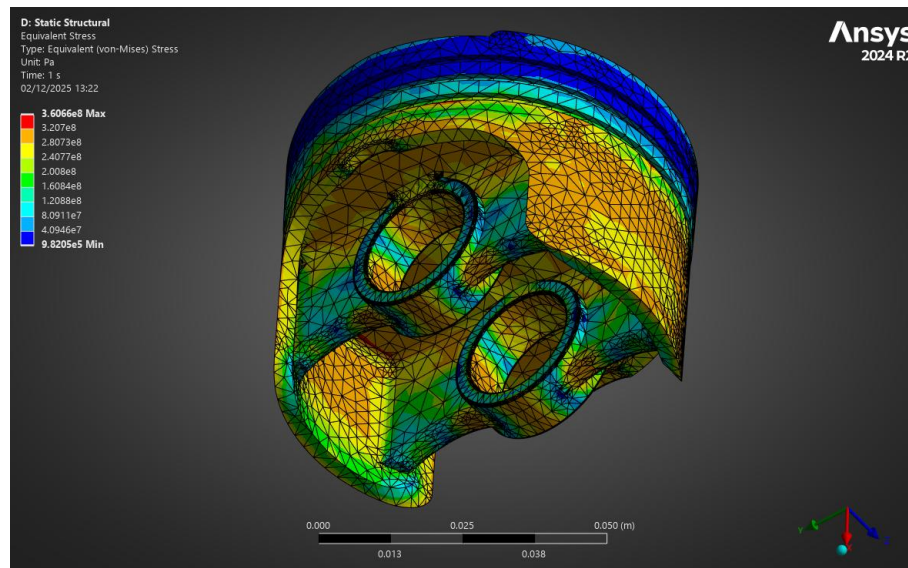
(b) Total deformation.

Figure 24: Results using the gudgeon pin as the fixed location

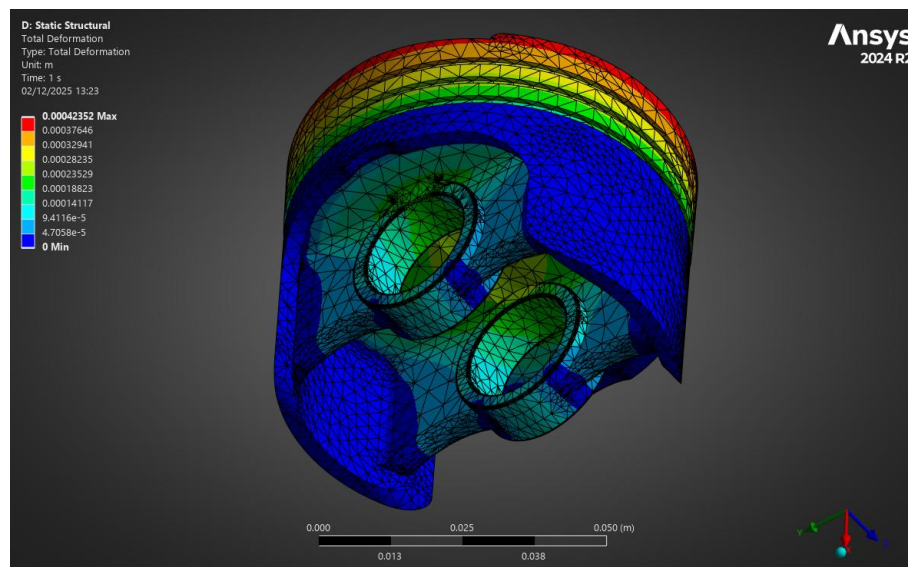
Since the loads on this part are through the small end gudgeon pin, constraining the part in this location will cause inaccurate results. An example of this is the lack of stress or deformation at the bottom of the holes, where the imported loads should cause a slight elongation and bring in an induced stress from the downwards force.

6.3.2 Piston Skirt

The piston skirt is constrained to represent its location inside the cylinder.



(a) Von-Mises stress.



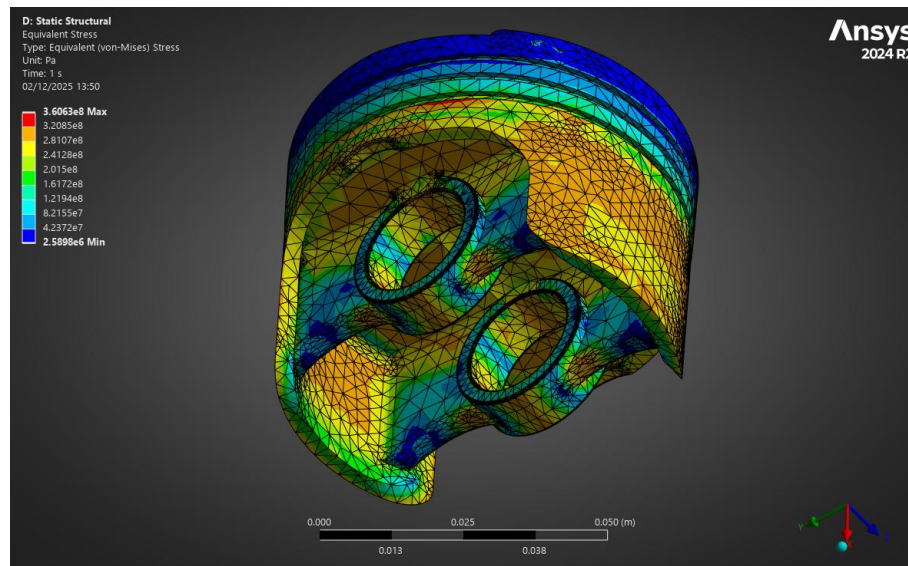
(b) Total deformation.

Figure 25: Results using the skirt as the fixed location.

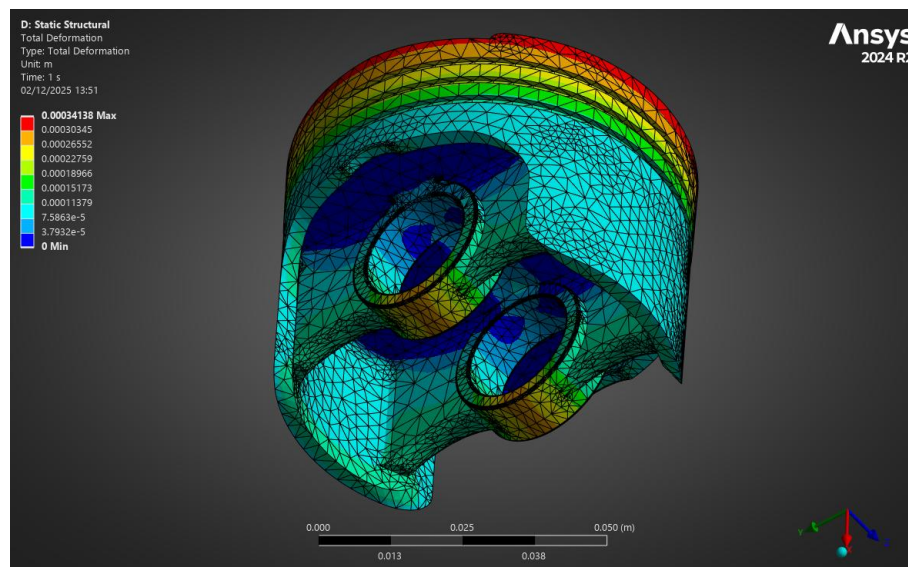
Although these results allow for more clarity in the small end results, the lack of forgiveness brought in by the constraint means that a higher stress is introduced on the inside of the skirt, where it meets the crown underside.

6.3.3 Crown Underside

The underside of the crown is constrained to allow the small end bearing and skirt to move, as these have the most direct forces acting upon them



(a) Von-Mises stress.



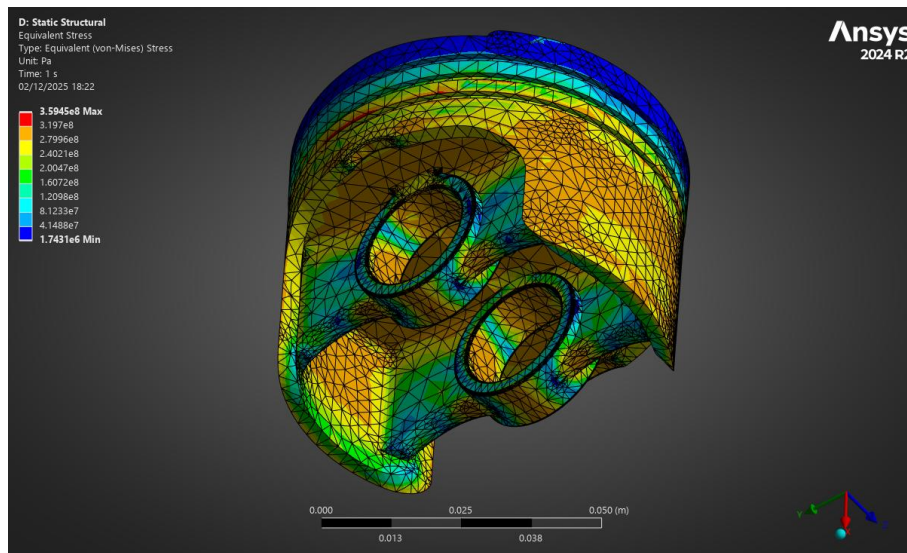
(b) Total deformation.

Figure 26: Results using the crown underside as the fixed location.

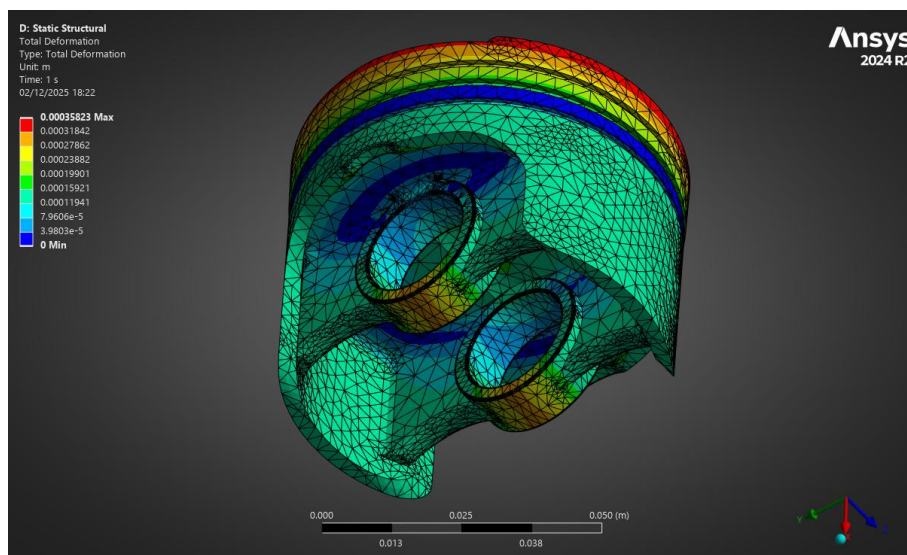
Constraining the part in this way allowed for stress to be realistically measured on the small end bearing and skirt, however the pressure acting on top of the piston crown has caused less deformation than previous runs, as it has been fixed from the underside.

6.3.4 Oil Ring

The oil ring is constrained as a neutral point between the forces on top of the piston and on the small end bearing, following on from the crown underside test.



(a) Von-Mises stress.



(b) Total deformation.

Figure 27: Results using the oil ring as the fixed location.

The oil ring results are giving the most realistic results, with stress and deformation showing on all the key surfaces. The downside of this method is that stress risers on the edge of the ring grooves are increased, as the deformation pulls against the fixed surface. For this reason, results around this area were interpreted with caution, although no geometry was change in these places, so the results are still comparable.

6.4 Mesh Independence Study

To identify optimal mesh parameters, an independence study is conducted on the base piston model. To do this, the global mesh size was changes in 0.25 intervals up to the computational limit. For each test, the maximum, minimum and average Von-Mises stress was logged, until the results became consistent.

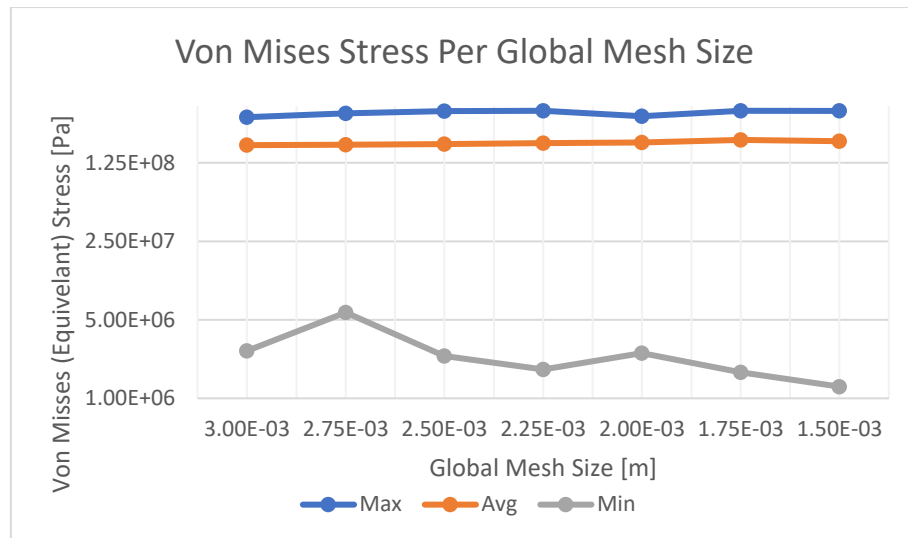


Figure 28: Von Mises stress change for uniform mesh size.

As the computational limit for this geometry was found beyond 1.5mm mesh size, a further study was then done into using variable mesh sizing to provide more consistent results. As shown in Figure 29 and 30, stress risers on the oil ring edge move from being sporadic to constant, showing that the mesh density along that edge directly effects the clarity of the FEA results. The cause of the inconsistent edge results in the uniform mesh is the fact that there is only one mesh element running along the piston ring grooves, and so in the user-defined mesh the sizing was altered to allow two mesh elements to spread across this distance (Liu and Glass, 2013).

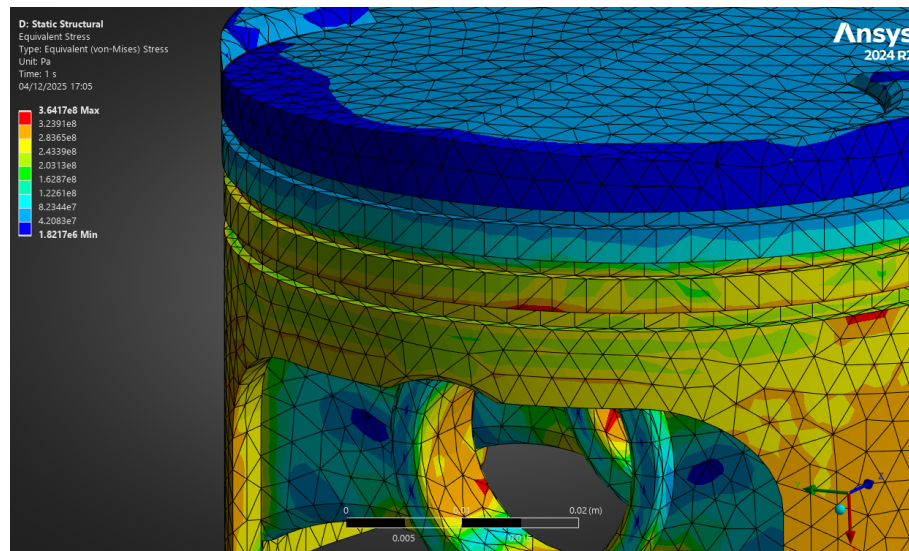


Figure 29: Von Mises stress using a uniform 5mm mesh.

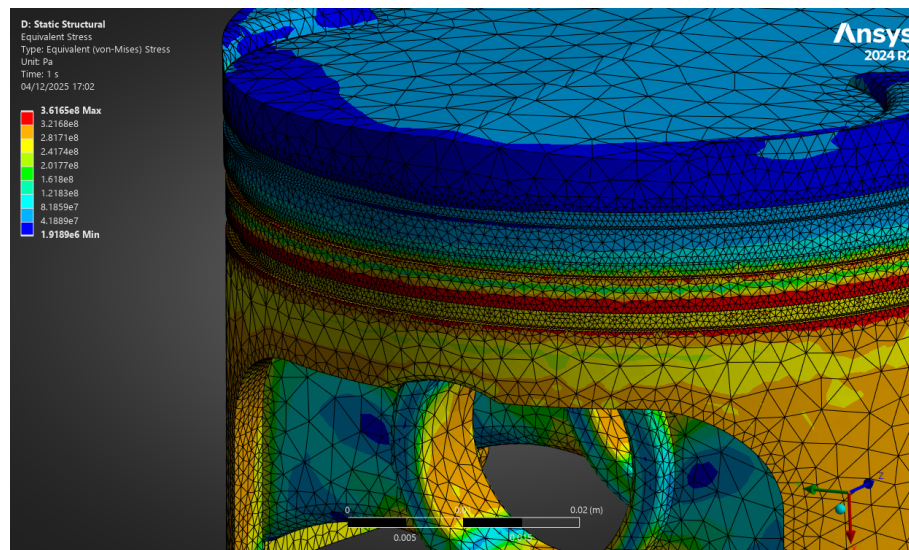


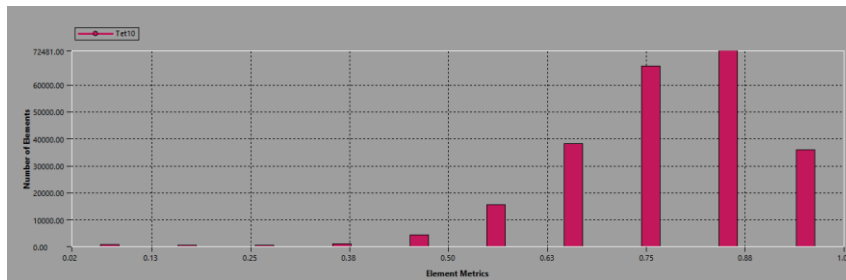
Figure 30: Von Mises stress results using the user-defined variable mesh.

Following on from this study, the mesh parameters found in Table 8 were used for the design analysis. The parameters are attempted to be maintained throughout each design iteration, although may vary due to part geometry or computational power.

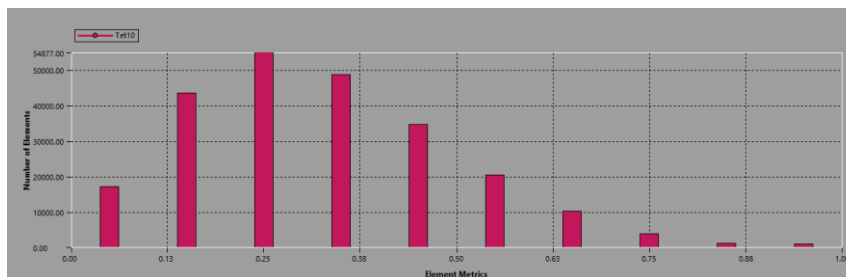
Table 8: User-defined mesh parameters used for the simulation.

Global Parameters			Underside Edges	
Element Size	5.00E-03	m	Type	Element Size
Use Adaptive Sizing	No		Element Size	7.50E-04 m
Growth Rate	Default (1.85)		Behaviour	Soft
Max Size	Default (1.00E-2)	m	Growth Rate	Default (1.85)
Piston Rings			Gudgeon Pin Location	
Type	Element Size		Type	Element Size
Element Size	5.00E-04	m	Element Size	1.50E-03 m
Behaviour	Soft		Behaviour	Soft
Growth Rate	Default (1.85)		Growth Rate	Default (1.85)
Crown Features			Piston Skirt Edges	
Type	Element Size		Type	Element Size
Element Size	1.25E-03	m	Element Size	1.00E-03 m
Behaviour	Soft		Behaviour	Soft
Growth Rate	Default (1.85)		Growth Rate	Default (1.85)

Further analysis could improve the quality of this mesh, creating more accurate and consistent results. This is shown in Figure 31 where the mesh quality plots reveal areas of poor quality, allowing these areas to be made more defined in the mesh parameters. The error due to mesh is deemed too small for this report's scope, so the parameters in Table 8 will continue to be used.



(a) Minimum element quality histogram, with 1 being a perfect mesh element.



(b) Maximum skewness histogram, with 0 being a perfect mesh element.

Figure 31: Mesh quality results for the user-defined mesh.

7 Discussion & Conclusions

7.1 Benefits & Limitations of CAE

In this report, CAE methods allow for a component to be tested and improved upon without the need for physical testing. This is massively beneficial for the engineering industry due to the rapid iterations which can be done, as well as the money saved on prototype production, physical testing, and the time taken to do this. Especially for a component like a piston, physical testing cannot replicate real world conditions exactly and therefore must be done in a running engine. This causes great expense, and part failure has enormous repercussions when testing in this way. By using CAE methods, the risk of this is diminished significantly as well as having an increase in development speed – a major factor in monetary efficiency and success in environments such as motorsport. (MAHLE GmbH, 2012) (Vishwas, 2013)

Limitations in CAE come from the accuracy of the solver and the inputs into it. This is why validation is so important to the process, as results can vary significantly depending on mesh sizing, input loads and environmental conditions. This makes CAE in disciplines such as motorsport or aerospace crucial, as a lower factor of safety puts more influence on the inaccuracy of the solver used, increasing the need for validation and physical testing to also be completed on a prototype prior to part use. Constant validation of the solver reassures that results are accurate, and validating with physical tests (such as controlled cyclic loading and strain sensor testing) means that boundary conditions can be adapted in the solver to improve accuracy and therefore reduce the need for this type of testing. (Malak and Paredis, 2004)

7.2 Conclusions

This study set out to create a validated workflow capable of analysing the stresses acting on a piston using generated motion loads and thermal conditions, allowing the design to be optimised. These objectives were met, demonstrating both the capability and the limitations of CAE within the component design process.

Across the four revised designs, Iteration 3 provided the most effective solution compared to the base model, achieving a 20% mass reduction while retaining a factor of safety above failure and reducing deformation. The fatigue study further indicated that these refinements contributed to a longer working life and improved cyclic loading behaviour. Iteration 4, however, highlighted the consequences of excessive or misapplied mass-reduction strategies, where increased deformation led to structural failure. While this reinforces the importance of understanding load paths during optimisation, the design still forms a strong foundation for future development.

The validation exercises, mesh independence study, and boundary-condition investigation demonstrated the solver's consistency but also its sensitivity to input parameters. These processes underline the need for well-informed modelling assumptions within CAE workflows and reinforce the importance of understanding the factors that influence solver behaviour and result accuracy.

8 References

- Dlupal Software (2023) *Verification Examples*. [online] Available at: <https://www.dlupal.com/en/downloads-and-information/examples/verification-examples> [Accessed 9 December 2025].
- Liu, Y. and Glass, G. (2013) 'Effects of mesh density on finite element analysis', in *SAE Technical Papers*. SAE International. Available at: <https://doi.org/10.4271/2013-01-1375>.
- MAHLE GmbH (2012) *Pistons and engine testing*. 1st ed. Edited by MAHLE GmbH. Stuttgart: Vieweg+Teubner Verlag.
- Malak, R.J. and Paredis, C.J.J. (2004) 'ON CHARACTERIZING AND ASSESSING THE VALIDITY OF BEHAVIORAL MODELS AND THEIR PREDICTIONS', in *ASME 2004 Design Engineering Technical Conferences and Computers and Information in Engineering Conference*. Salt Lake City.
- Noble, A. (2025a). *3D CAD model of engine internal assembly*. Unpublished teaching material, ACHA6010, University of Wales Trinity Saint David.
- Noble, A. (2025b). *Piston temperature and pressure data*. Unpublished teaching material, ACHA6010, University of Wales Trinity Saint David.
- Shetty, A.K. et al. (2014) 'Design and Analysis of Piston using Aluminium Alloys', *International Journal of Innovative Research in Advanced Engineering* [Preprint]. Available at: www.ijirae.com.
- Vishwas, M. (2013) 'Role of Computer Aided Design and Engineering in Product Development', in *International Conference on Advanced Materials, Manufacturing, Management, Thermal science, At Siddaganga Institute of Technology, Tumkur*. Tumkur.

9 Appendix

9.1 Appendix A: Piston Temperature & Pressure Data

Table 9: Piston temperature and pressure data.

Area	Heat Transfer Coefficient [$\text{W/m}^2\cdot^\circ\text{C}$]	Surface Temperature [$^\circ\text{C}$]
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
Maximum Pressure [Pa]		
10537060		

9.2 Appendix B: Validation Analytical Cases

9.2.1 Structural Validation Case: Cantilever with Uniform Load

Verification Example

Program: RFEM 5

Category: Geometrically Linear Analysis, Isotropic Linear Elasticity, Plate

Verification Example: 0016 – Elastic Bending - Continuous Load

0016 – Elastic Bending - Continuous Load

Description

A thin plate is fully fixed on the left end ($x = 0$) and is subjected to a uniform pressure p according to the Figure 1. The problem is described by the following set of parameters.

Material	Elastic-Plastic	Modulus of Elasticity	E	210000.000	MPa
		Poisson's Ratio	ν	0.000	
		Shear Modulus	G	105000.000	MPa
Geometry	Plate	Length	L	1.000	m
		Width	w	0.050	m
		Thickness	t	0.005	m
Load	Plate	Pressure	p	2750.000	Pa

Small deformations are considered and the self-weight is neglected in this example. Determine the maximum deflection $u_{z,\max}$. The aim of this example is to show that a surface of the surface stiffness type *Without Tension* behaves linearly under bending.

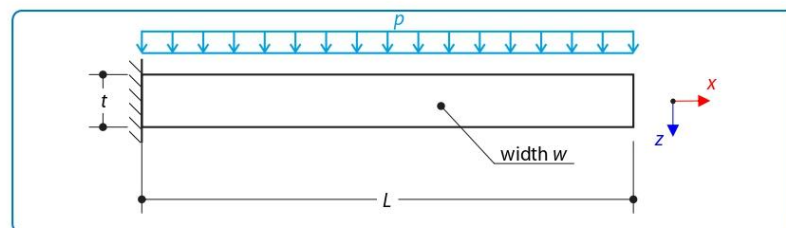


Figure 1: Problem sketch

Analytical Solution

The bending moment M for the plate under continuous loading $q = pw$ is defined by

$$M = -\frac{q(L-x)^2}{2} \quad (16-1)$$

Verification Example: 0016 – Elastic Bending - Continuous Load

Recalling the elementary equation for the beam deflection using the Euler-Bernoulli theory

$$E I_y \frac{d^2 u_z}{dx^2} = M \quad (16 - 2)$$

where I_y is the quadratic moment of a cross-section with respect to the y -axis and it is given by

$$I_y = \frac{wt^3}{12} \quad (16 - 3)$$

We integrate the equation (16 – 2) from 0 to L twice. Upon applying the boundary conditions for cantilever beam $u_z(0) = u_z'(0) = 0$, the solution takes the following form

$$u_z(x) = \frac{qx^2(6L^2 - 4Lx + x^2)}{24E I_y} \quad (16 - 4)$$

The maximum displacement occurs at the point $x = L$:

$$u_{z,\max} = \frac{qL^4}{8E I_y} = \frac{3pL^4}{2Ew^2t} = 157.143 \text{ mm} \quad (16 - 5)$$

RFEM 5 Settings

- Modeled in RFEM 5.04.0024
- The element size is $l_{FE} = 0.020 \text{ m}$
- Geometrically linear analysis is considered
- The Mindlin plate theory is used
- Isotropic linear elastic material model is used

Results

Structure File	Surface Stiffness Type		
0016.01	Standard		
0016.02	Without Tension		

Surface Stiffness Type	Analytical Solution	RFEM 5	
	$u_{z,\max}$ [mm]	$u_{z,\max}$ [mm]	Ratio [–]
Standard	157.143	157.130	1.000
Without Tension		157.150	1.000

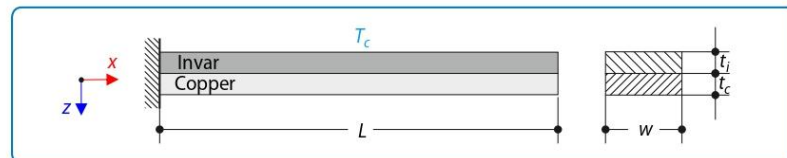
9.2.2 Thermal Validation Case: Bimetallic Strip

**Verification
Example**

Program: RFEM 5
Category: Geometrically Linear Analysis, Isotropic Linear Elasticity, Temperature Dependency, Plate, Solid
Verification Example: 0078 – Bimetallic Strip
0078 – Bimetallic Strip
Description

A bimetallic strip is composed of the Invar¹ and copper. The left end of the bimetallic strip is fixed, and the right end is free, loaded by temperature difference T_c according to **Figure 1**. While neglecting self-weight, determine the deflection $u_{z,max}$ of the bimetallic strip (free end). The problem is described by the following set of parameters.

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 1: Problem Sketch

¹ Invar is an alloy of iron and nickel with very low coefficient of thermal expansion.

Verification Example: 0078 – Bimetallic Strip**Analytical Solution**

The bimetallic strip is made of two metals with different coefficients of thermal expansion. They are frequently used in instruments to sense or control temperatures. When the ambient temperature is changing, the bimetallic strip is bending. That is caused by the different elongations of the used metals.

The analytical solution is based on the approach introduced in [1]. The following formula gives the dimensionless coefficient K_1 which can be used for the calculation of the equivalent stiffness

$$K_1 = 4 + 6 \frac{t_c}{t_i} + 4 \left(\frac{t_c}{t_i} \right)^2 + \frac{E_c}{E_i} \left(\frac{t_c}{t_i} \right)^3 + \frac{E_i t_i}{E_c t_c}. \quad (78 - 1)$$

The maximum deflection of the bimetallic strip is then defined by the following formula

$$u_{z,\max} = \frac{6(\alpha_i - \alpha_c)(t_i + t_c)T_c L^2}{t_s^2 K_1} \approx -7.049 \text{ mm}. \quad (78 - 2)$$

RFEM 5 Settings

- Modeled in RFEM 5.16.01
- The minimum element size is $l_{FE} = 0.250 \text{ mm}$
- Isotropic linear elastic model is used

Results

Structure Files	Program	Entity
0078.01	RFEM 5	Plate
0078.02	RFEM 5	Solid

Model	Analytical Solution	RFEM 5	
	$u_{z,\max}$ [mm]	$u_{z,\max}$ [mm]	Ratio [-]
RFEM 5, Plate	-7.049	-7.047	1.000
RFEM 5, Solid		-7.064	1.002

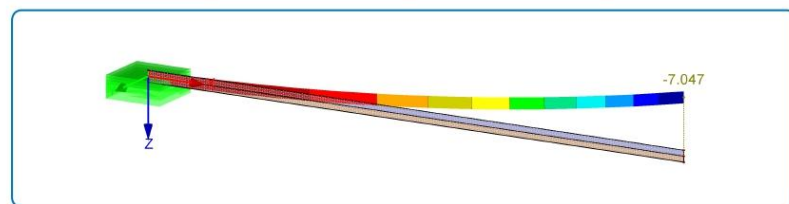


Figure 2: RFEM 5 results – Plate model

References

- [1] YOUNG, W., BUDYNAS, R. and SADEGH, A. *Roark's Formulas for Stress and Strain, 8th Edition*. McGraw-Hill Education, 2011.