



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

BEng Motorsport Engineering

Application of Computer Aided Engineering Processes

Module Name	Structural Design & Fluid Analysis
Module Number	ACHA6010
Student Name	Sean Doyle
Student Number	2305948
Date of Submission	17/12/2023

University of Wales Trinity Saint David
School of Engineering
2023-2024

Contents

Table of Figures	3
Table of Tables	4
1 Introduction.....	5
2 Literature Review.....	6
2.1 CAE Studies.....	6
2.1.1 Kinematics Study	6
2.1.2 Thermal Study.....	6
2.1.3 Structural Study.....	6
2.2 CAE Validation.....	7
2.2.1 Thermal Validation	10
2.3 Comparing Meshing Results	12
2.3.1 Cantilever Beam Mesh Comparison	12
2.3.2 Bimetallic Strip Mesh Comparison.....	14
2.4 Piston Forces.....	16
3 Methodology	17
3.1 Base Model	18
3.1.1 Kinematics Methodology	18
3.1.2 Steady-State Thermal.....	19
3.1.3 Static Structural.....	20
3.2 New Material Testing.....	22
3.2.1 Kinematics Study	22
3.2.2 Thermal Study.....	23
3.2.3 Static Structural Study	24
3.3 New Piston Design.....	26
3.3.1 Kinematics Study	26
3.3.2 Thermal Study.....	27
3.3.3 Static Structural Study	28
4 Analysis.....	30

4.1	Limitations of CAE.....	31
5	Conclusion	32
6	Reference List	33

Table of Figures

Figure 1: Cantilever Beam (Dub, 2021).....	8
Figure 2: SolidWorks Cantilever Validation Results.....	9
Figure 3: Ansys Cantilever Results Mesh 0.2m.....	9
Figure 4: Heat Transfer Rate Diagram.....	10
Figure 5: Bimetallic Strip Deformation	11
Figure 6: Ansys Cantilever Beam 0.1m Meshing.....	12
Figure 7: Ansys Cantilever Beam 0.3m Meshing.....	13
Figure 8: Ansys Cantilever Beam 0.4m Meshing	13
Figure 9: Bimetallic Strip 0.1mm Mesh.....	14
Figure 10: Bimetallic Strip 0.5mm Mesh.....	15
Figure 11: Bimetallic Strip 1mm Mesh.....	15
Figure 12: Piston Cyclic Loading Forces (MAHLE GmbH, 2013).....	16
Figure 13: Piston and Crankshaft Assembly.....	17
Figure 14: Ansys Kinematic Study	17
Figure 15: Rigid Dynamics Study.....	18
Figure 16: Piston Temperature 1.....	19
Figure 17: Piston Temperature 2.....	19
Figure 18: Static Structural Deformation 1.....	20
Figure 19: Static Structural Deformation 2.....	20
Figure 20: Static Structural Stress Results 1	21
Figure 21: Static Structural Stress Results 2	21
Figure 22: New Material Kinematic Study	22
Figure 23: New Material Piston Temperature 1.....	23
Figure 24: New Material Piston Temperature 2.....	23
Figure 25: New Material Deformation 1.....	24
Figure 26: New Material Deformation 2.....	24
Figure 27: New Material Stress Results 1.....	25

Figure 28: New Material Stress Results 2.....	25
Figure 29: Piston Skirt New Design	26
Figure 30: New Design Kinematic Study	26
Figure 31: New Design Piston Temperature 1.....	27
Figure 32: New Design Piston Temperature 2.....	27
Figure 33: New Design Deformation Results 1	28
Figure 34: New Design Deformation Results 2	28
Figure 35: New Design Stress Results 1	29
Figure 36: New Design Stress Results 2.....	29

Table of Tables

Table 1: Elastic Bending Properties (Dub, 2021)	7
Table 2: Bimetallic Strip Properties (Dub, 2017)	10
Table 3: Comparing Results.....	30

1 Introduction

Computer Aided Engineering (CAE) is a powerful tool for modern engineers. It enables semi-accurate results to be simulated for real world design problems. As results are only assumptions, validation is required to verify the accuracy of a solution. This can be determined and compared with analytical methods. Representing a geometry with a suitable mesh is difficult to achieve correctly. This document will detail the varying percentage accuracy of results, focusing on the impact of changing mesh sizing. A kinematics, thermal and structural study is used to develop a piston within an assembly by using an iterative design technique to reduce the mass. These results will be analysed with CAE limitations considered.

2 Literature Review

CAE is the use of computers and software to simulate product performance. It assists in design improvements and engineering problems, “this includes simulation, validation and optimization of products processes and manufacturing tools” (Siemens, 2023). CAE can be used within numerous finite element analysis (FEA) studies: thermal, structural, kinematics and more.

The aim of redesigning most components is to either make it lighter or improve the properties (performance, durability, etc.). The aim for the piston was to minimise stress by making it lighter to reduce piston friction, engine weight and vibration (U.S. Office, n.d.).

2.1 CAE Studies

2.1.1 Kinematics Study

Kinematics are the study of forces in motion for an assembly or component. In FEA this can be combined with thermal and structural studies to gain additional data (Cybellium Ltd, 2023).

2.1.2 Thermal Study

As temperature across a component is unknown, thermal studies are used to give a thermal profile. Heat flow cannot be static, only “steady state” (heat flow does not change) or transient (changes with time) as heat flow never stops (Kurowski, 2019).

2.1.3 Structural Study

A static structural study can be either linear or non-linear. It gives engineers an understanding of the physical conditions of a component regarding stress, strain, deformation and more (SimScale, 2023).

2.2 CAE Validation

Validation is required as results are assumptions. The number of assumptions depend upon the software and/or users ability. However, no result is truly accurate as not every condition (known or unknown) can be accounted for (Tickoo, 2021). To demonstrate the need for validation a couple of examples will be demonstrated. The first, ‘Elastic Bending – Continuous Load’ with analytical validation and testing within SolidWorks and Ansys with varying mesh sizes (Dub, 2021). The dimensions are specified in table 1 with figure 1 representing the model.

Table 1: Elastic Bending Properties (Dub, 2021)

Properties	Material Properties
Modulus of Elasticity E	210000 MPa
Poisson’s Ratio	0.0
Shear Modulus (G)	105000 MPa
Length (L)	1 m
Width (w)	0.05 m
Layer Thickness (t)	0.005 m
Uniform Pressure (P)	2750 Pa

To validate the model analytically, the maximum displacement (u_{max}) can be calculated using the following formulas (Dub, 2021):

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

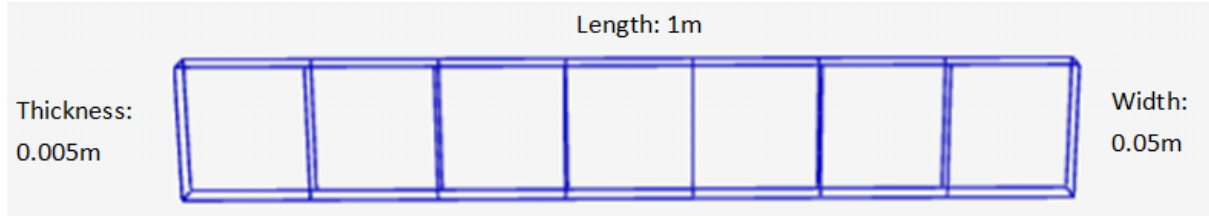


Figure 1: Cantilever Beam (Dub, 2021).

$$q = pt$$

$$I = \frac{tw^3}{12}$$

$$\therefore u_{max} = \frac{ptL^4}{8E \frac{tw^3}{12}}$$

$$u_{max} = \frac{(2750 \times 0.005 \times 1^4)}{\left(8 \times (2.1 \times 10^{11}) \times \frac{0.005 \times 0.05^3}{12}\right)}$$

$$u_{max} = 0.15714 \times 10^{-3} \text{ m}$$

$$u_{max} = 0.15714 \text{ mm}$$

Completing this validation setup within SolidWorks (figure 2) and Ansys (figure 3) gave a maximum displacement of 0.1575 mm at the end of the cantilever beam furthest from the fixed face; whilst analytical methods above gave a value of 0.1571 mm. When comparing the two values, difference between the two is 0.0004 millimetres. Using equations below, percentage error is minuscule at 0.254% (Meyer, 2012).

$$\text{Percentage Error} = \frac{\text{Measured Value} - \text{Actual Value}}{\text{Actual Value}} \times 100$$

$$\text{Percentage Error} = \frac{0.1571 - 0.1575}{0.1575} \times 100$$

$$\text{Percentage Error} = 0.254 \%$$

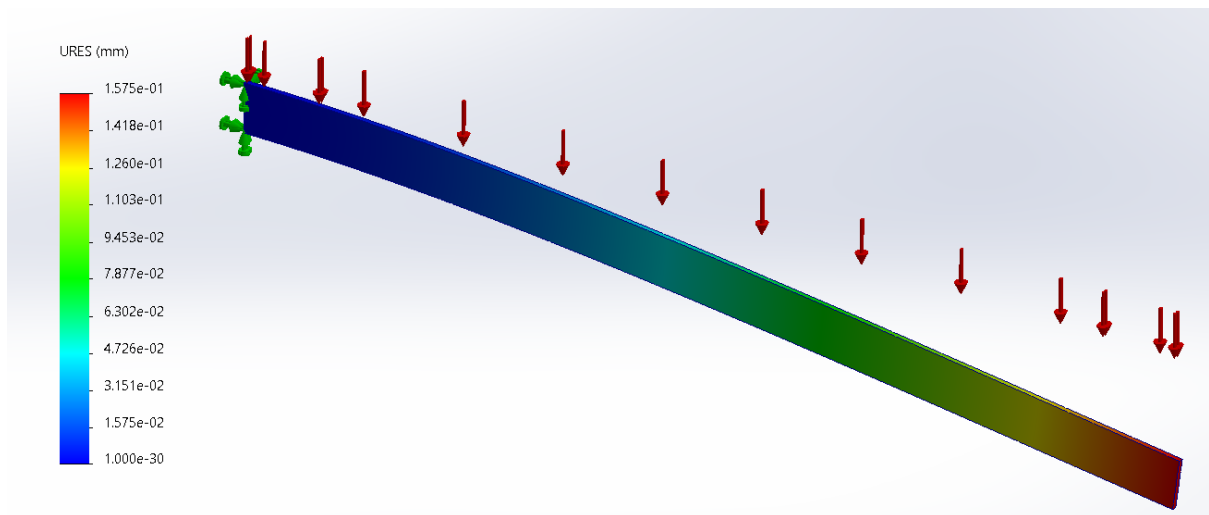


Figure 2: SolidWorks Cantilever Validation Results

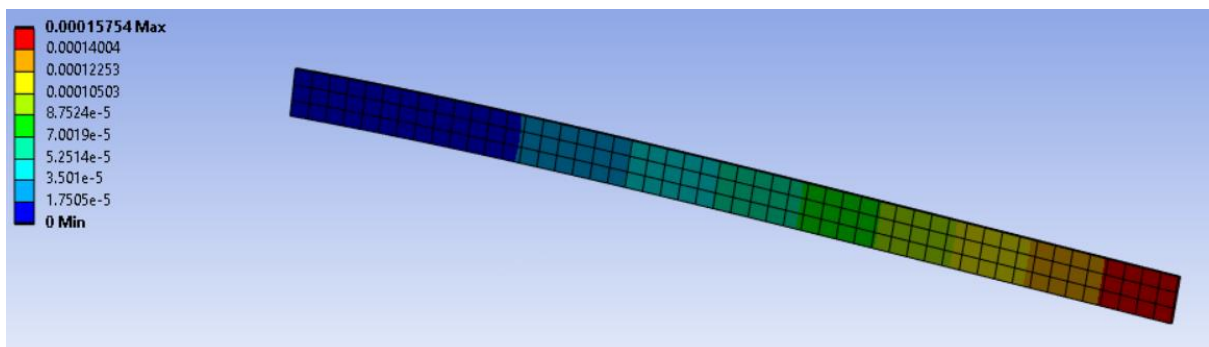


Figure 3: Ansys Cantilever Results Mesh 0.2m

2.2.1 Thermal Validation

The bimetallic strip example can be used for thermal validation due to the effect of heat transfer on the cantilever beam. When heat is applied to an object, the internal energy within, changes (Naterer, 2002). Using the equations below, the model can be validated with basic thermodynamic principles in accordance with the materials thermal conductivity for heat transfer rate. The following equation calculates heat transfer rate ($\dot{Q}$) by utilizing figure 4 where ΔT is change in temperature and one of the main factors affecting transfer rate. Material thickness (x), thermal conductivity (k) and area (A) also contribute to heat transfer rate (Twigg, 2012).

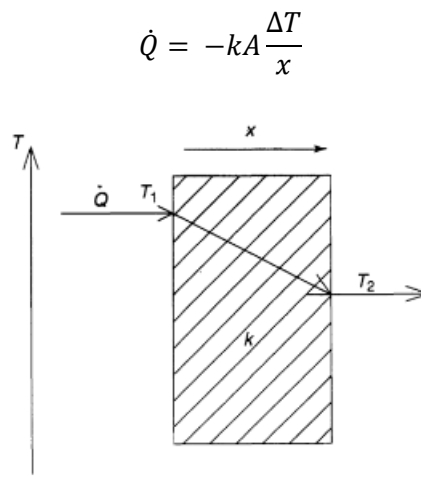


Figure 4: Heat Transfer Rate Diagram

Table 2: Bimetallic Strip Properties (Dub, 2017)

Properties	Invar (i)	Copper (c)	Units
Modulus of Elasticity (E)	137000	130000	MPa
Poisson's Ratio (v)	0.28	0.354	-
Coefficient of Thermal Expansion (α)	1.2×10^{-6}	2.0×10^{-5}	$^{\circ}\text{C}^{-1}$
Cross-section Width (w)	5.0		mm
Layer Thickness (t)	1.0		mm
Length (L)	100.0		Mm
Thermal Loading (T_c)	100.0		$^{\circ}\text{C}$

To validate results from Ansys, data from table 2 and the following validation method and equations were used. The analytical method results are 7.049mm of displacement (upwards). However, this considers two different material's heat transfer rate, copper and invar. Equivalent stiffness (k_1) is calculated for use in calculating maximum displacement $u_{z,max}$ (Dub, 2017):

$$k_1 = 4 + 6 \frac{t_c}{t_i} + 4 \left(\frac{t_c}{t_i} \right)^2 + \frac{E_c}{E_i} \left(\frac{t_c}{t_i} \right)^3 + \frac{E_i t_i}{E_c t_c}$$

$$u_{z,max} = \frac{6(\alpha_i - \alpha_c)(t_i + t_c)T_c}{t_s^2 k_1} \frac{L^2}{2} \cong -7.049 \text{ mm}$$

An Ansys static structural study was conducted in accordance with the setup description using a mesh sizing of 0.25mm (Dub, 2017). Giving the results in figure 5 where maximum deformation was slightly underestimated at 7.1569mm when compared to the real value of 7.049mm. In the equations below the percentage error within this calculation is shown to be 1.531% (Meyer, 2012):

$$\text{Percentage Error} = \frac{7.1569 - 7.049}{7.049} \times 100$$

$$\text{Percentage Error} = 1.531 \%$$

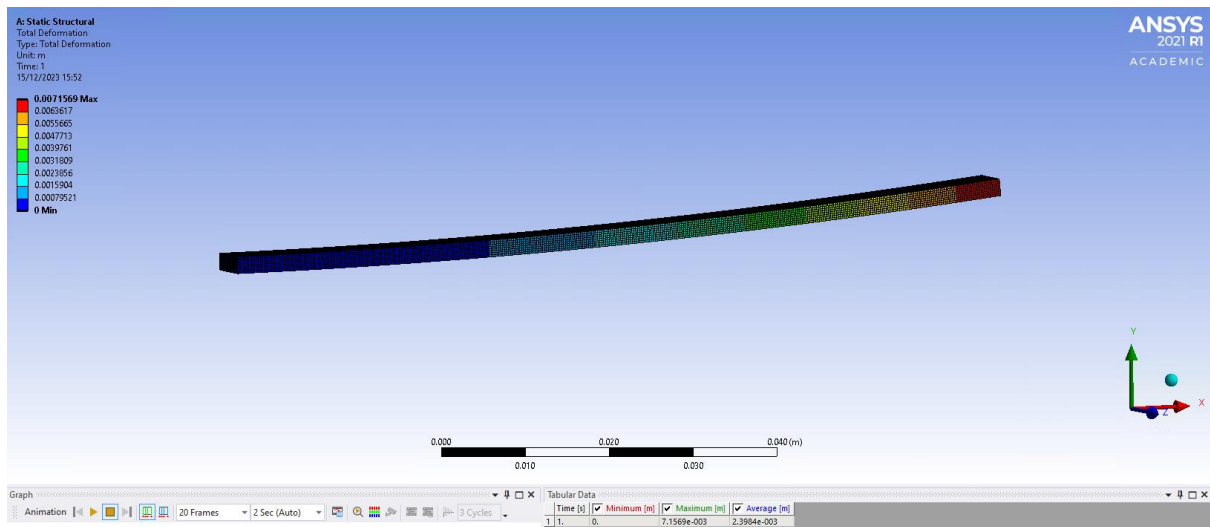


Figure 5: Bimetallic Strip Deformation

2.3 Comparing Meshing Results

To use CAE methods correctly, appropriate mesh sizes must be defined. Complex geometry items will require more refinement and attention to detail around prominent geometry changes. The first study will determine if settings are correct. The results determine if mesh settings are too fine or coarse. If large changes between adjacent nodes occur this could be an indication of over sizing (Champion & Ensminger, 1988). If specific areas are suspected to contain higher stress or strain levels mesh refinement should be considered.

2.3.1 Cantilever Beam Mesh Comparison

Figures 6, 7 and 8 show the effects of changing mesh sizing on the cantilever beam. Minimal differences are noticeable where slight changes occur in mid-range of each test but maximum deflection does not.

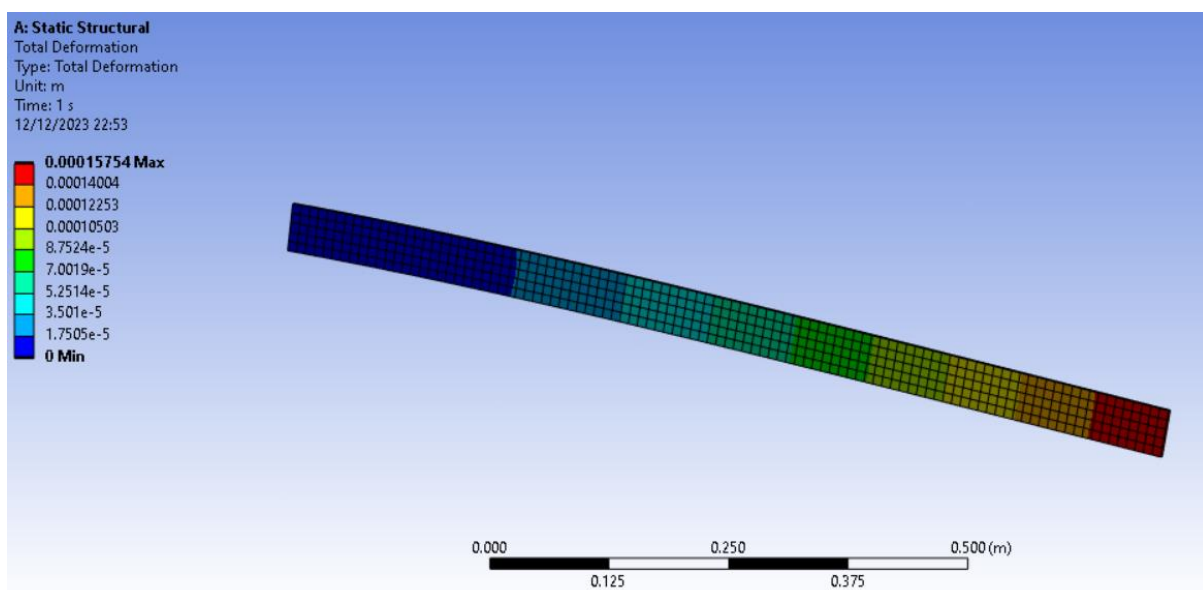


Figure 6: Ansys Cantilever Beam 0.1m Meshing.

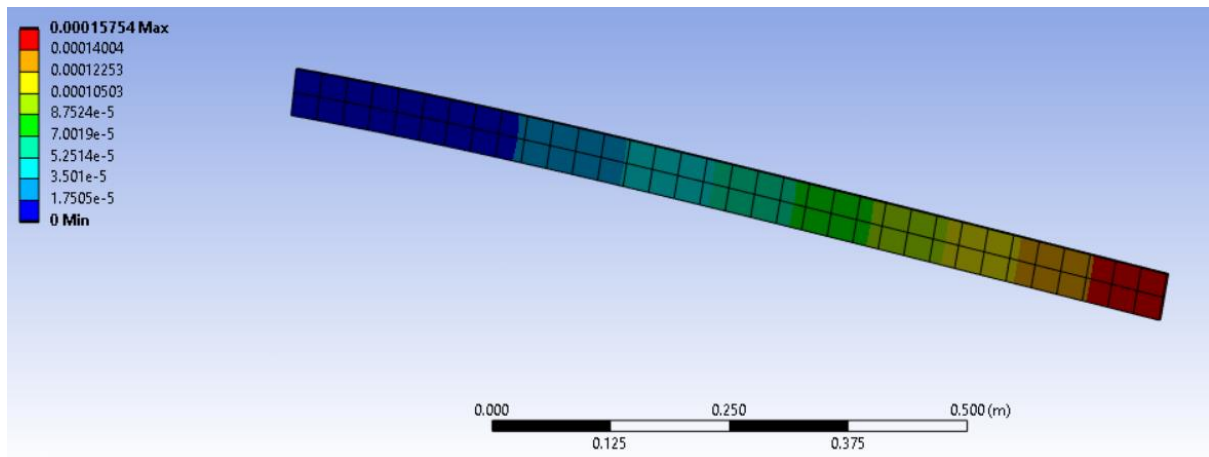


Figure 7: Ansys Cantilever Beam 0.3m Meshing.

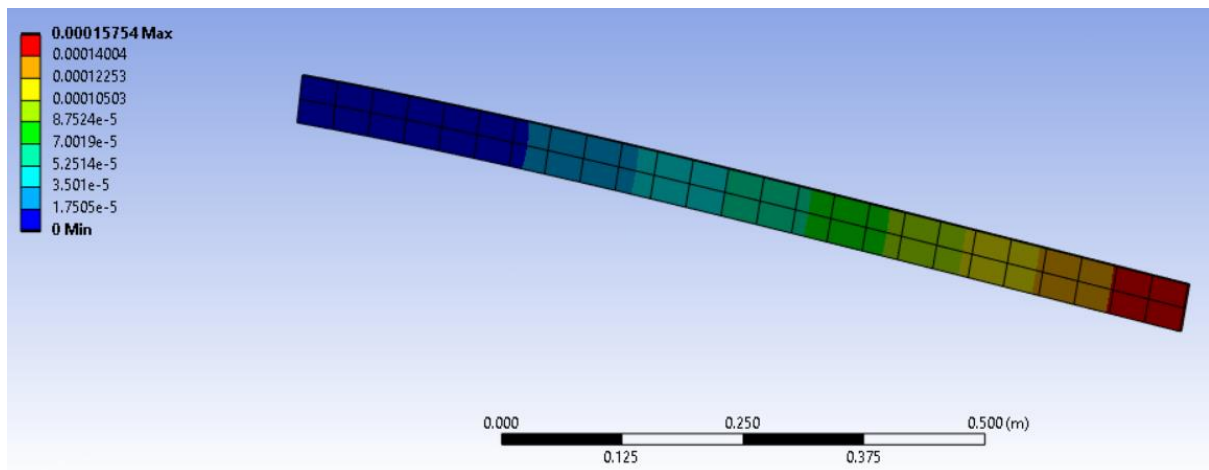


Figure 8: Ansys Cantilever Beam 0.4m Meshing

2.3.2 Bimetallic Strip Mesh Comparison

Adjusting the meshing for the bimetallic strip shows changes in maximum deflection. Referring to the actual value maximum deflection should be 7.049mm and recommended mesh setting values deflection at 7.1569mm (figure 5). By decreasing the mesh element sizing to 0.1mm from 0.25mm, maximum deflection was decreased to 7.1559mm (figure 9). This is more accurate but takes longer. Increasing element size to 0.5mm, the deflection is increased to 7.1587mm (figure 10) which is less accurate. Mesh element sizing is further increased to 1mm in figure 11 where accuracy decreases again to maximum deflection 7.1589mm. From all percentage error calculations shown below, it is apparent there is a negative correlation, as mesh element sizing increases, accuracy decreases and vice versa.

Percentage Error for Figure 9:

$$\text{Percentage Error} = \frac{7.1559 - 7.049}{7.049} \times 100$$

$$\text{Percentage Error} = 1.517 \%$$

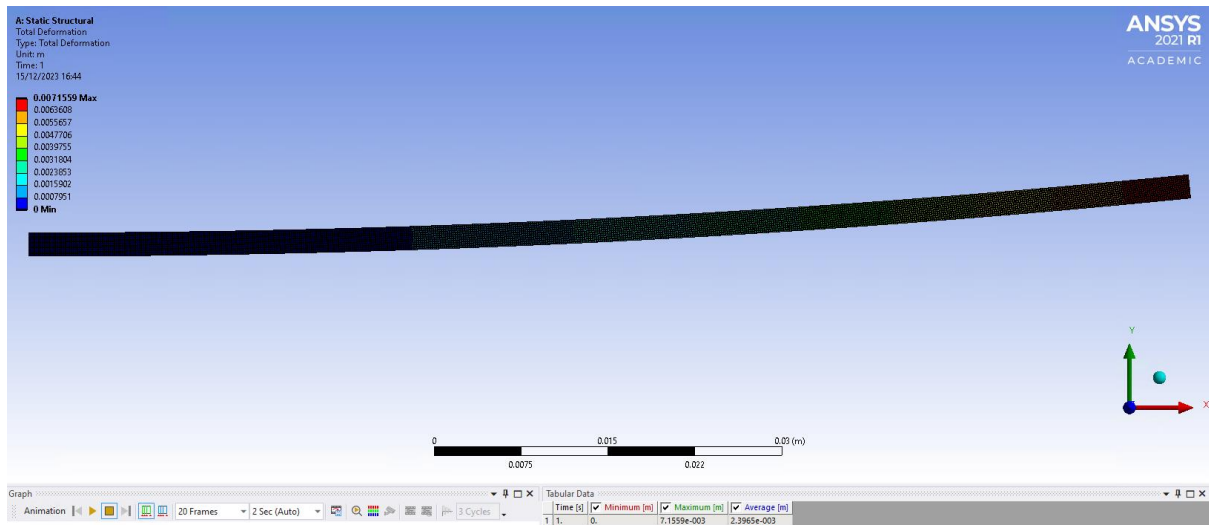


Figure 9: Bimetallic Strip 0.1mm Mesh

Percentage Error for Figure 10:

$$\text{Percentage Error} = \frac{7.1587 - 7.049}{7.049} \times 100$$

$$\text{Percentage Error} = 1.556 \%$$

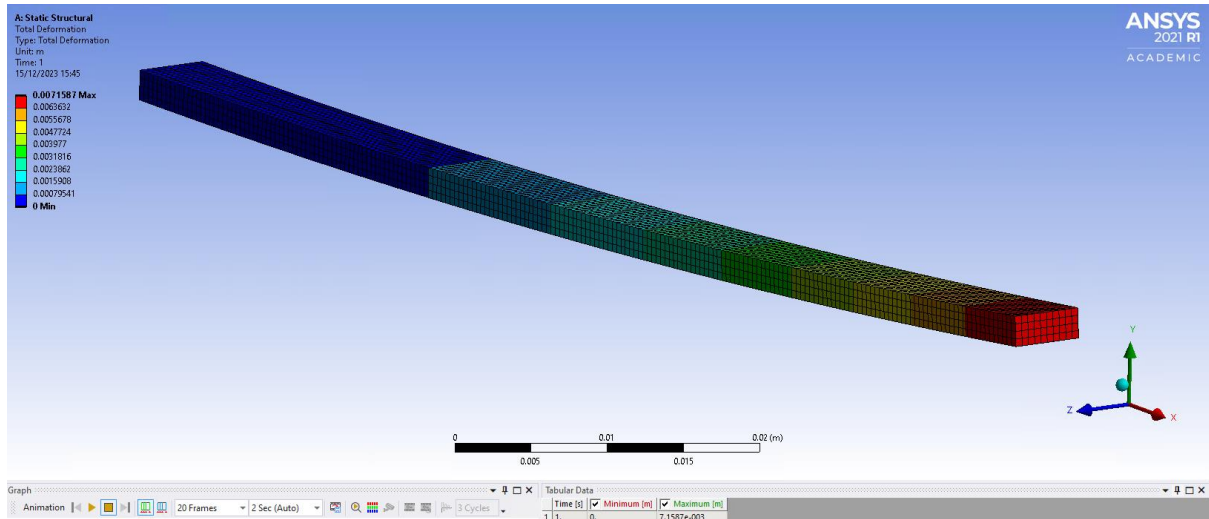


Figure 10: Bimetallic Strip 0.5mm Mesh

Percentage Error for Figure 11:

$$\text{Percentage Error} = \frac{7.1589 - 7.049}{7.049} \times 100$$

$$\text{Percentage Error} = 1.559 \%$$

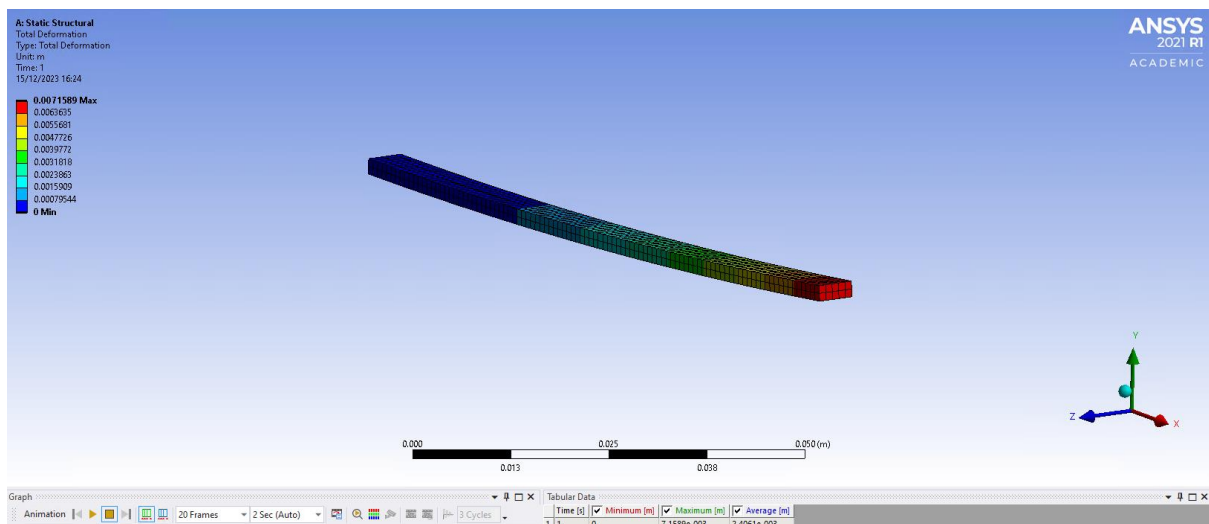


Figure 11: Bimetallic Strip 1mm Mesh

2.4 Piston Forces

When an engine is operating the piston is subject to extreme pressures, temperatures and forces. Pistons undergo cyclic loading which is application of repeated or fluctuating (nonproportional) loading of stress and strain (Fukumoto & Lee, 1992). The highest forces exerted on a piston in motion are when direction changes occur at top dead centre (TDC) and bottom dead centre (BDC), figure 12. However, the addition of combustion pressure at TDC exerts a larger force than anywhere else (MAHLE GmbH, 2013).

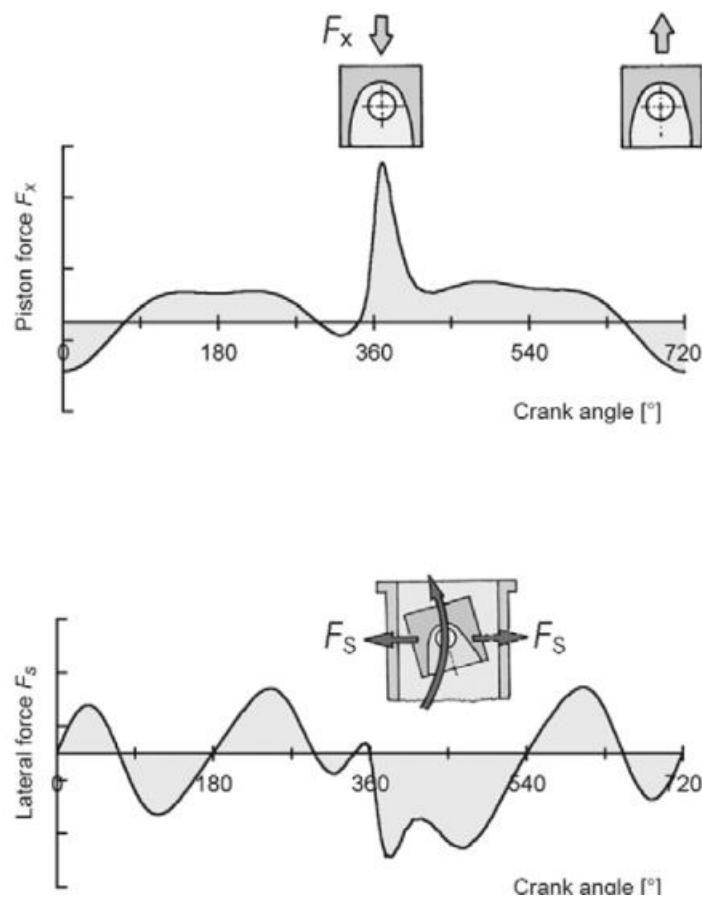


Figure 12: Piston Cyclic Loading Forces (MAHLE GmbH, 2013)

3 Methodology

Testing and development used the piston and crankshaft assembly (figure 13). The component to analyse and improve is the piston. By completing a kinematic study for thermal and structural conditions this can be analysed for loadings and thermal properties. Imported into Ansys Workbench, the study starting with rigid dynamics, continues to steady-state thermal and finishes with static structural (figure 14). Materials assigned were Aluminium alloy 7020 T6 for pistons and “Carbon Steel, AISI 1141, oil quenched and tempered at 650°C” for all remaining components (Ansys, 2023). To optimise design, these results can be used to reduce mass. Although useful, CAE software also has limitations and concerns for solution accuracy (Champion & Ensminger, 1988).

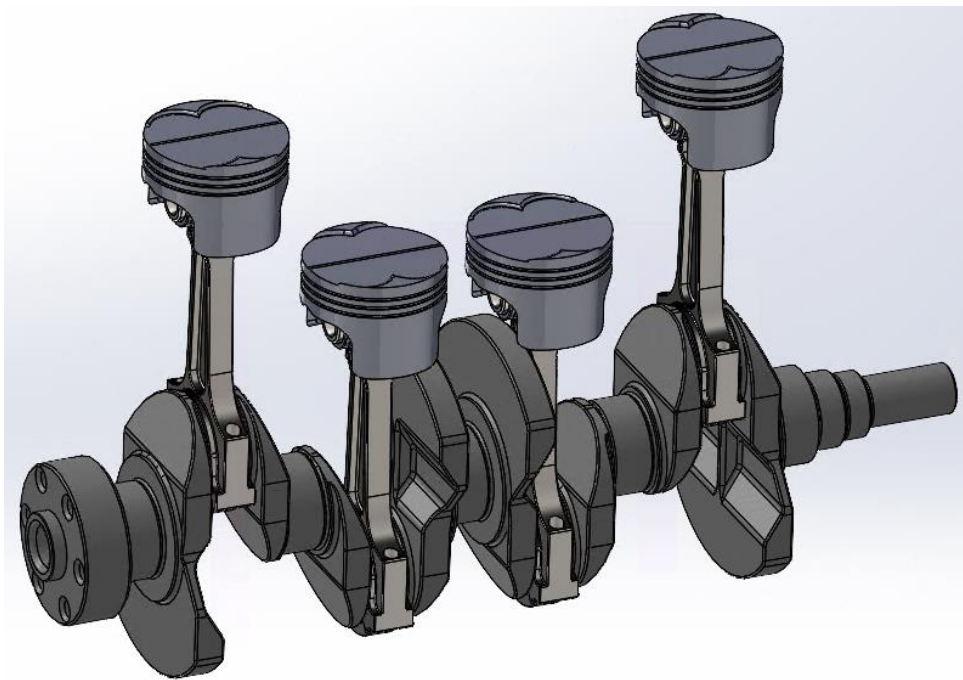


Figure 13: Piston and Crankshaft Assembly

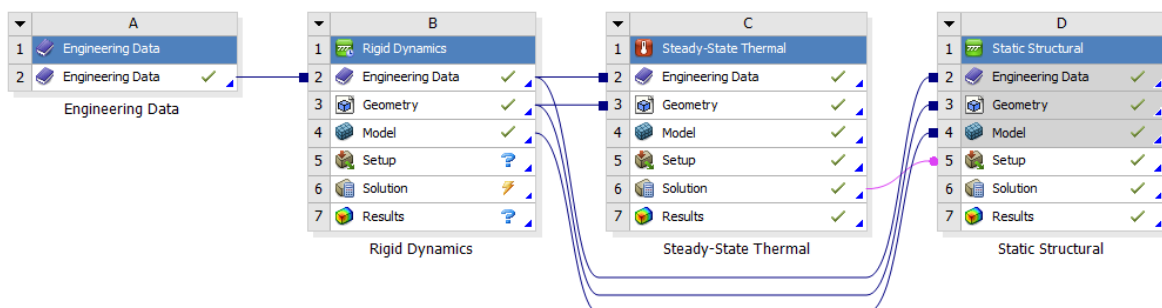


Figure 14: Ansys Kinematic Study

3.1 Base Model

3.1.1 Kinematics Methodology

The rigid dynamics study (figure 15) is used to gain the highest force within the assembly, used later within the thermal and static structural study. A revolution speed is applied to the crankshaft to simulate engine rotating at 4000 RPM. Maximum loads are found when the piston reaches TDC and changes direction where force is approximately 2438 Newtons.

The “probe point” with this highest force is then exported as a motion load to be imported for the “steady-state thermal” analysis.

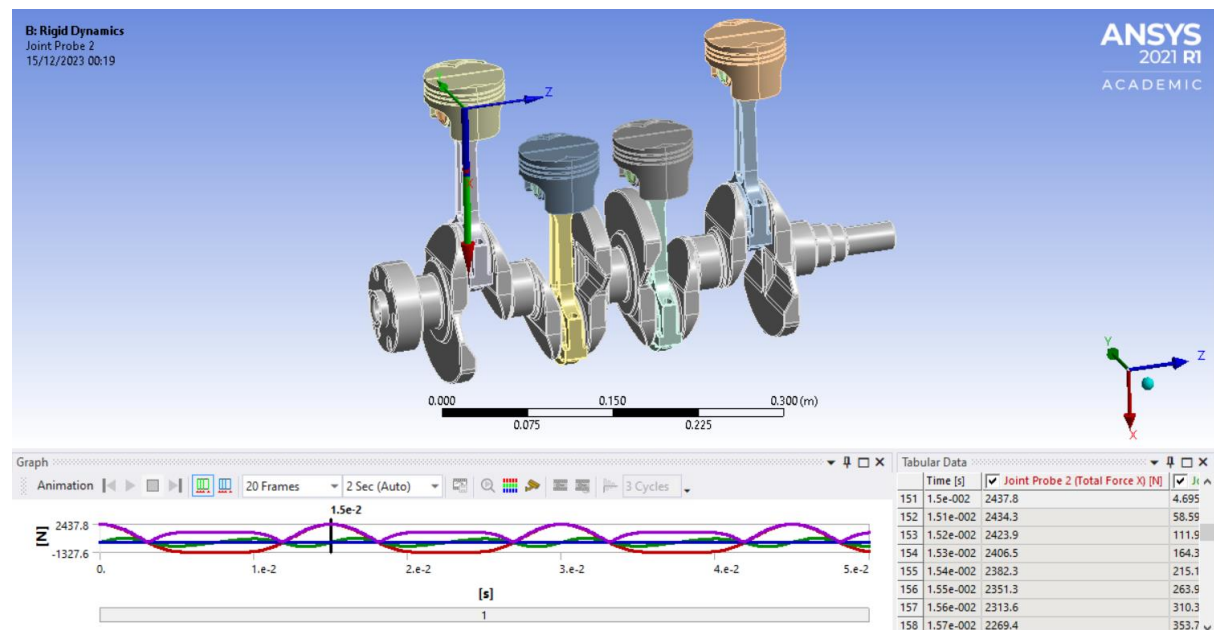


Figure 15: Rigid Dynamics Study

3.1.2 Steady-State Thermal

The steady state thermal analysis uses temperature inputs determining heat transfer across the component due to conduction and temperature gradients between different surfaces (Kurowski, 2019). By adding realistic convection temperature values over the component for combustion, oil temperature, ring grooves and bore/skirt this establishes an accurate temperature profile for the component. Figure 16 and 17, shows the temperature gradient over the piston where temperature is highest at the crown, approximately 316.03°C. The piston skirt is coolest where heat is transferred from the piston to oil, with an estimated temperature of 159.69°C.

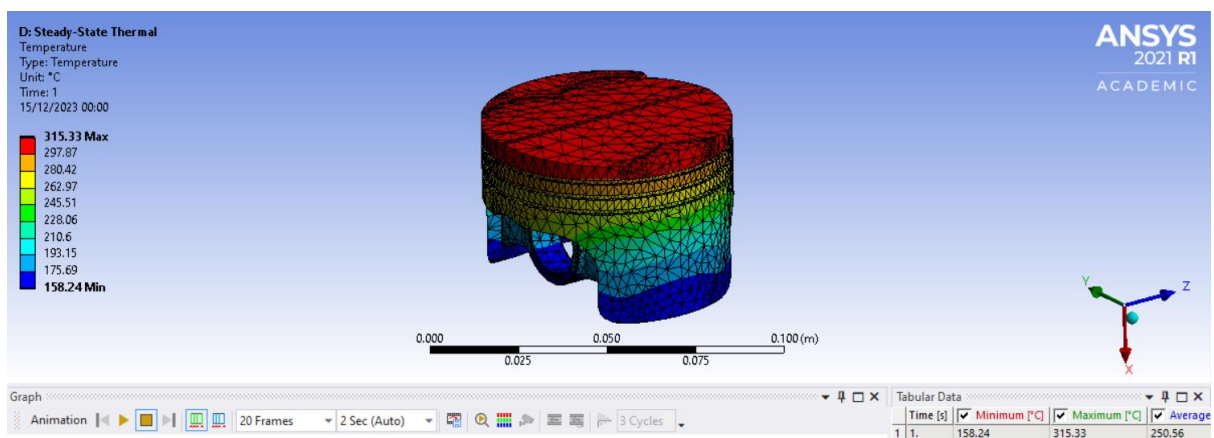


Figure 16: Piston Temperature 1

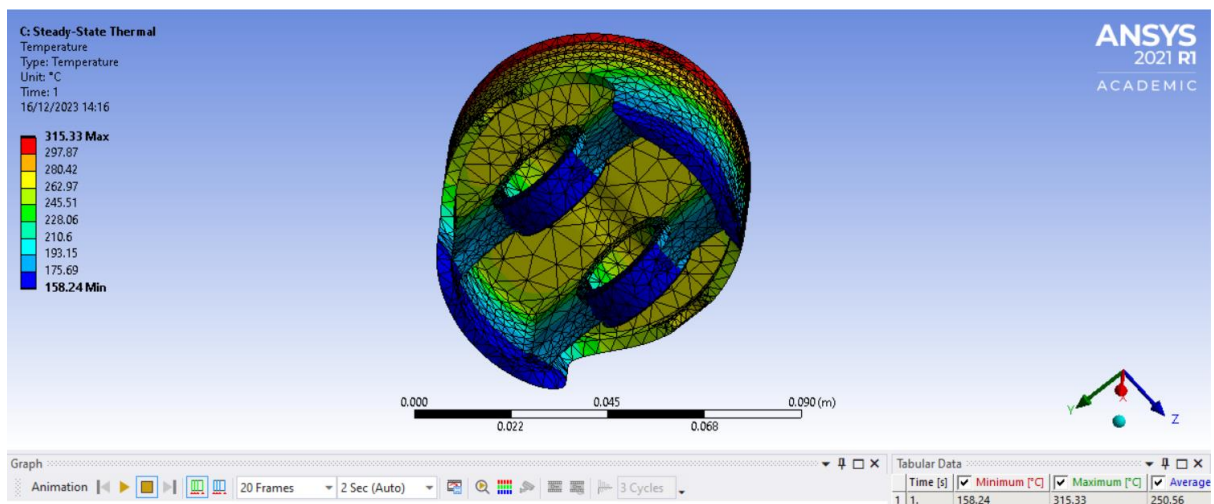


Figure 17: Piston Temperature 2

3.1.3 Static Structural

The static structure test uses all results from previous tests calculating stress and deformation. Temperature is critical as components have been modelled to change modulus of elasticity accordingly. Results are shown in figures 18, 19, 20 and 21 where deformation is concentrated around the pin boss as the cyclic loading contains sudden change in force direction (MAHLE GmbH, 2013). When conducting the simulations, weak springs would not support the model, therefore for each test a “Compression Only Support” was used to allow for skirt displacement (Ansys Learning, 2022). Deformation occurs most around the crown of the piston, which is to be expected, with temperatures and pressures highest at this area (figure 18 and 19).

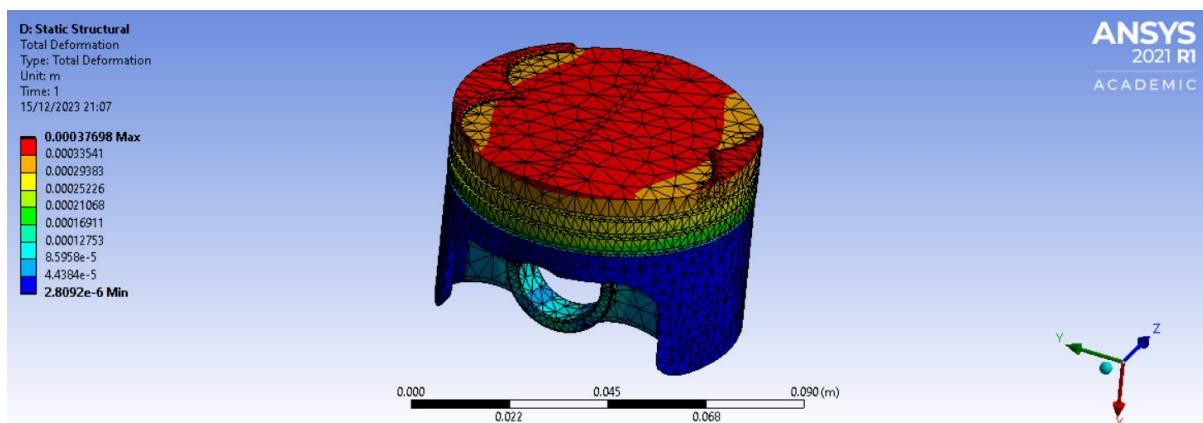


Figure 18: Static Structural Deformation 1

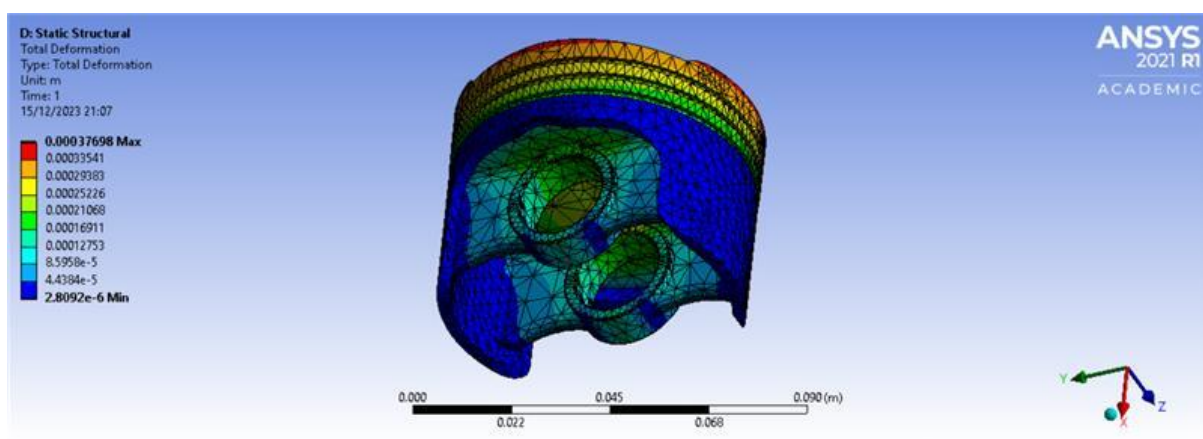


Figure 19: Static Structural Deformation 2

Figures 20 and 21 demonstrates highest stress is at the skirt of the piston and around the gudgeon pin slots. These results are expected around the gudgeon pin slots due to the sudden cyclic loading as the piston changes direction. Referring to the lateral forces in Figure 12, caused by the normal forces from the changing angle of the conrod, stress can be expected on the piston skirt where only vertical travel is permitted overcoming the lateral forces.

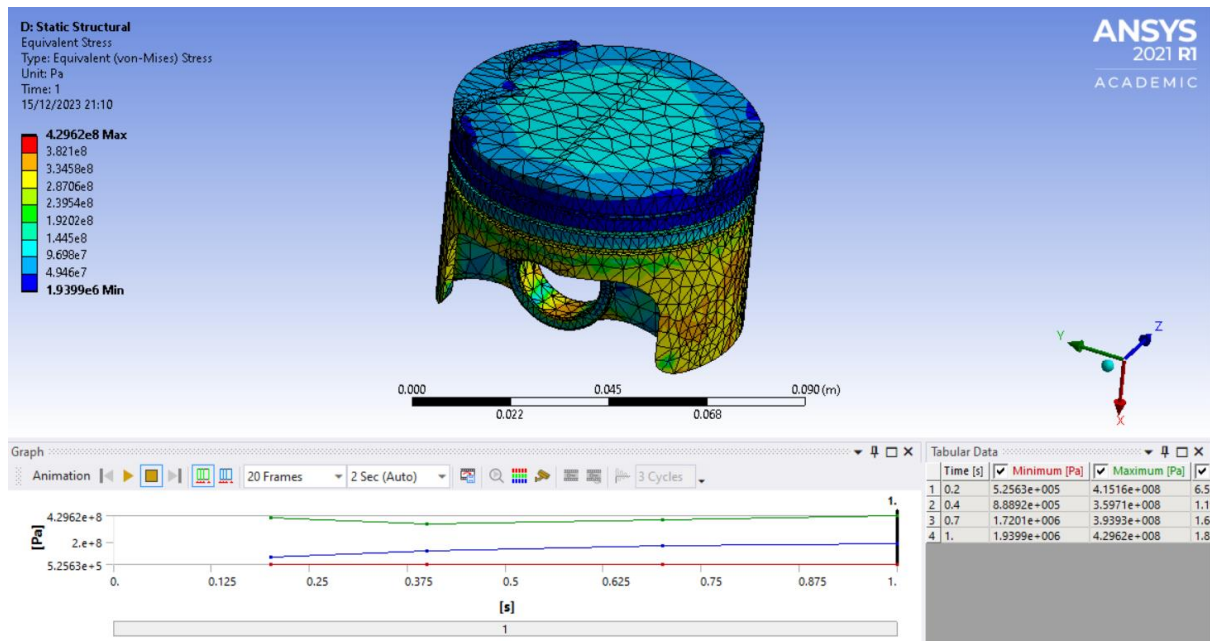


Figure 20: Static Structural Stress Results 1

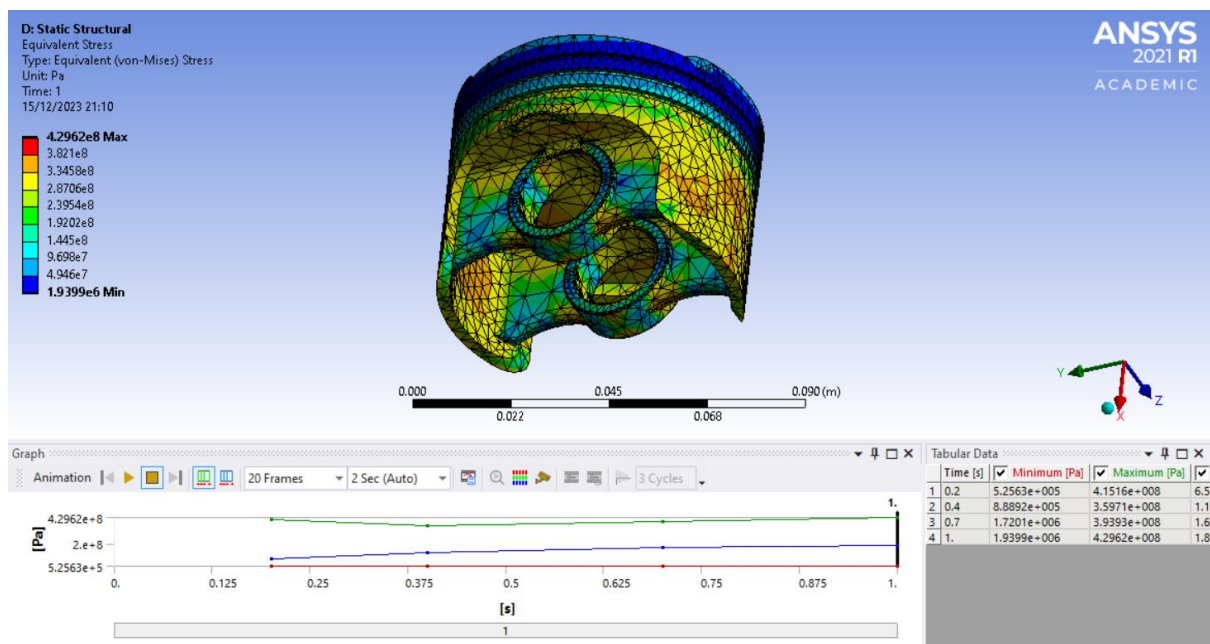


Figure 21: Static Structural Stress Results 2

3.2 New Material Testing

To improve the assembly, the aim was to reduce the mass. By changing the material from Aluminium 7020 T6 to Aluminium 6061 T451 mass was reduced from 0.26237 Kg to 0.25605 kg, giving a weight saving of 0.00632 Kg, approximately 2.41%. This is expected to reduce the forces within the kinematics study, because of Newton's laws with reduced inertia, as a result of a lower force equal to mass multiplied by acceleration (Hall, 2023).

3.2.1 Kinematics Study

The same studies and criteria from the base model were used to ensure control parameters and comparable results (figure 22).

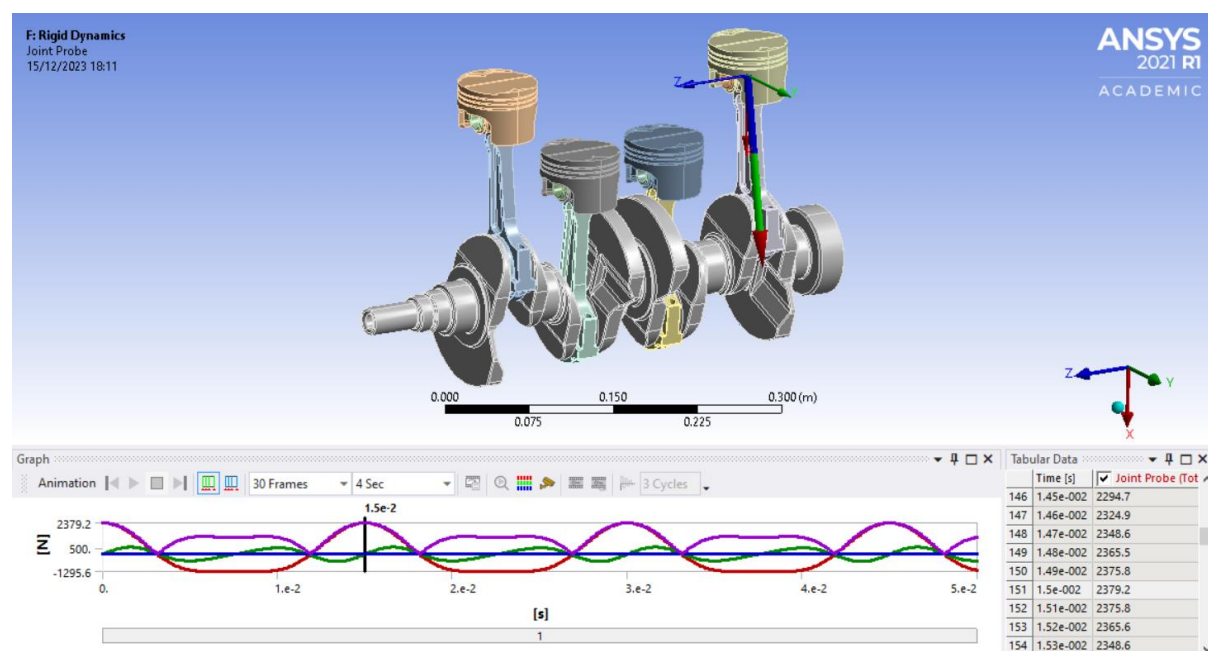


Figure 22: New Material Kinematic Study

3.2.2 Thermal Study

The piston temperature environment was setup identically to the previous study with results shown figures 23 and 24.

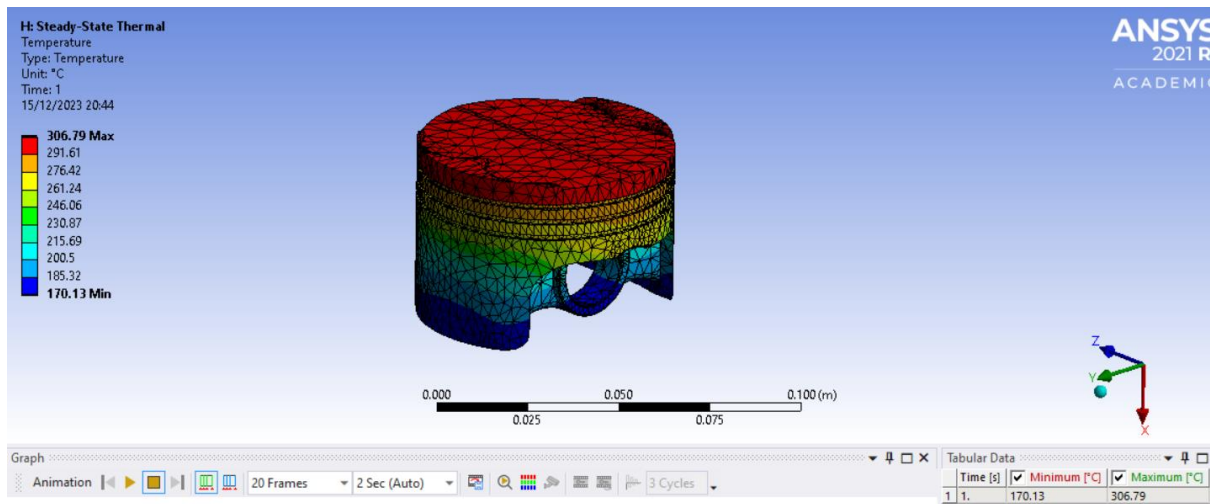


Figure 23: New Material Piston Temperature 1

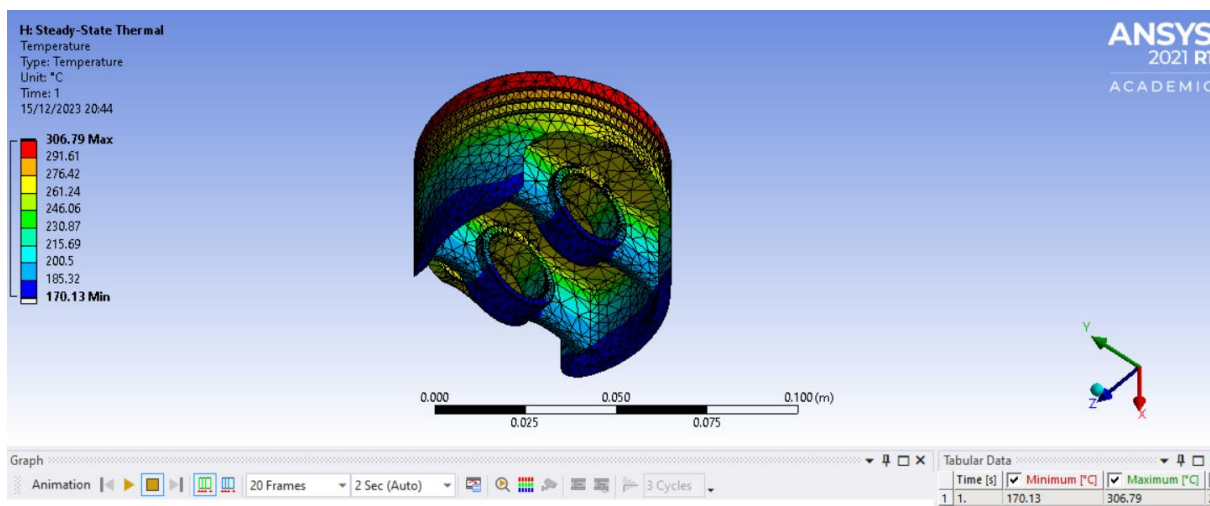


Figure 24: New Material Piston Temperature 2

3.2.3 Static Structural Study

Figures 25 and 26 show the deformation on the piston with the Aluminium 6061.

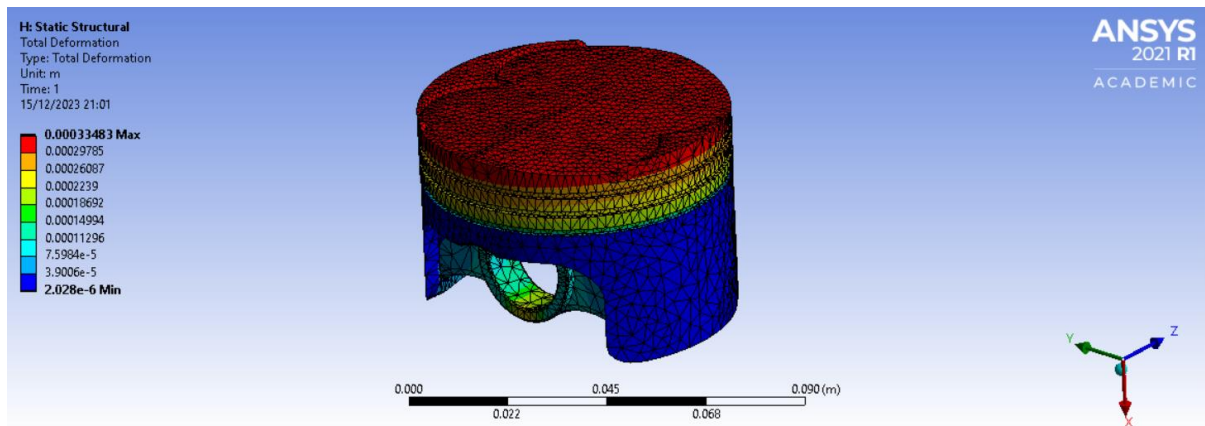


Figure 25: New Material Deformation 1

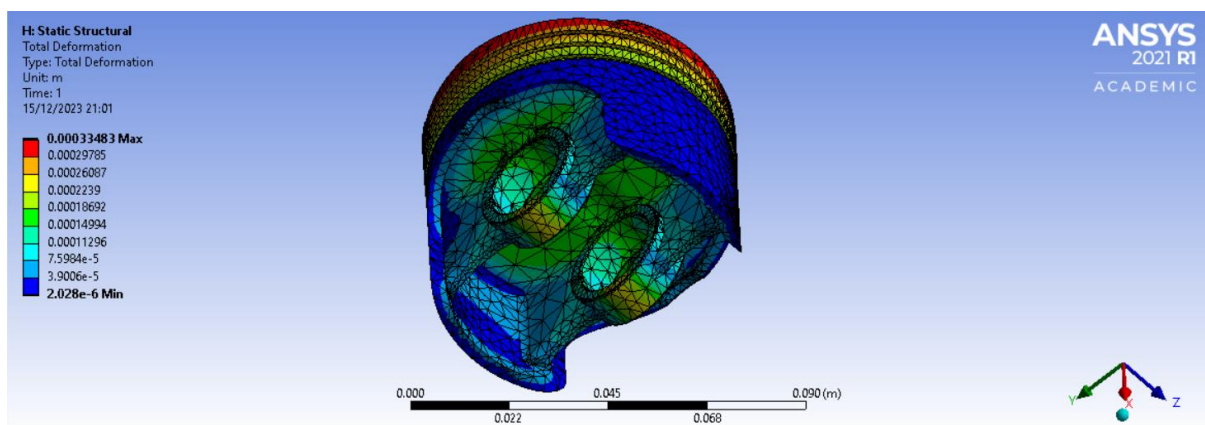


Figure 26: New Material Deformation 2

Figures 27 and 28 show piston stress with Aluminium 6061.

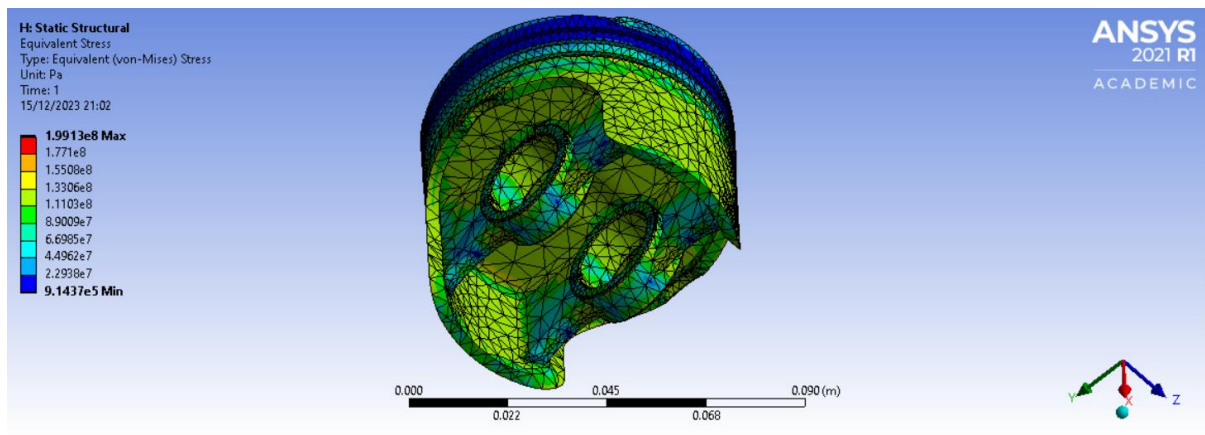


Figure 27: New Material Stress Results 1

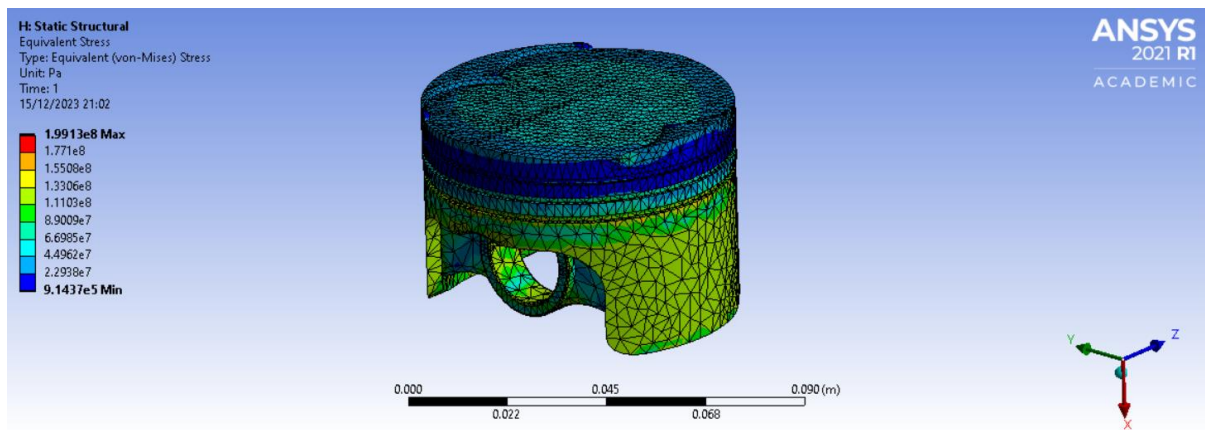


Figure 28: New Material Stress Results 2

3.3 New Piston Design

When analysing stress and deformation results from previous simulations and research, it was determined that weight could also be reduced by removing material. The piston skirt experiences less deformation and lower temperatures therefore allowing a reduction in thickness by 0.75mm (figure 29), saving 0.00474 Kg when using Aluminium 7020.

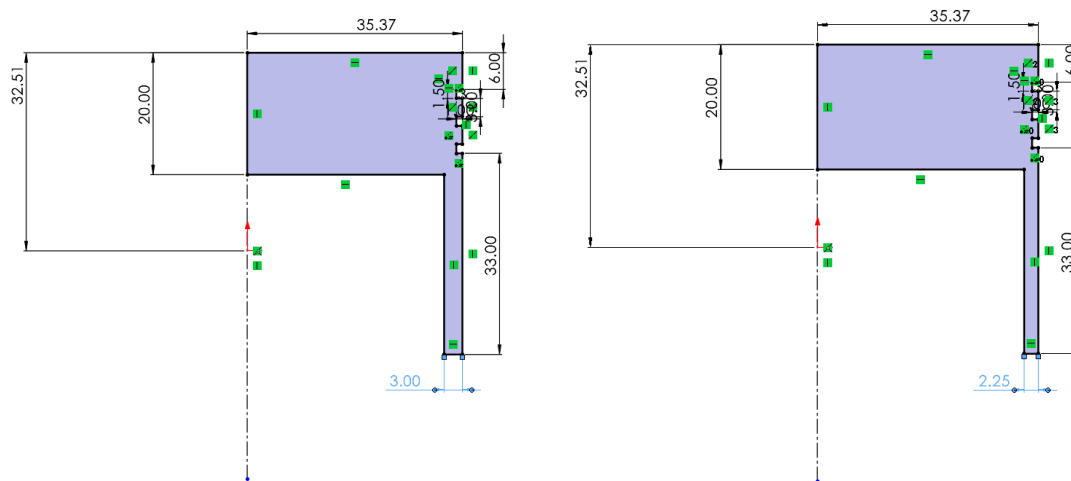


Figure 29: Piston Skirt New Design

3.3.1 Kinematics Study

New kinematics study shown figure 30 demonstrates the maximum force exerted is 2393.8 Newtons.

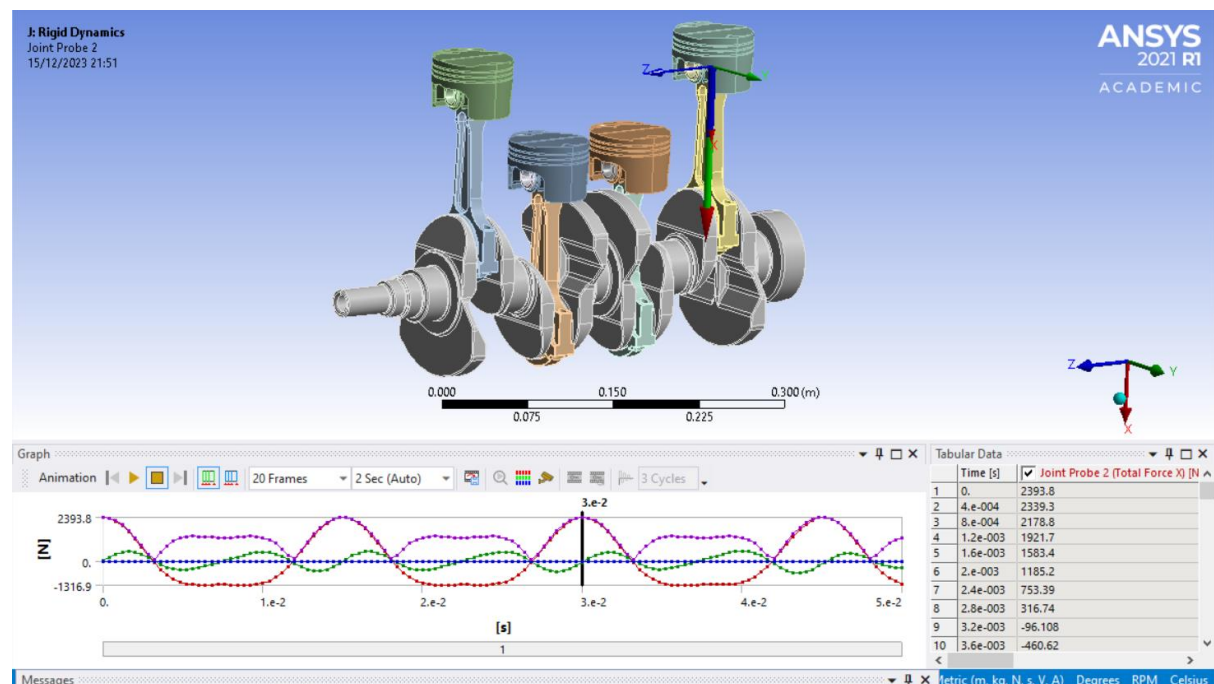


Figure 30: New Design Kinematic Study

3.3.2 Thermal Study

Figures 31 and 32 show thermal study results for the new design.

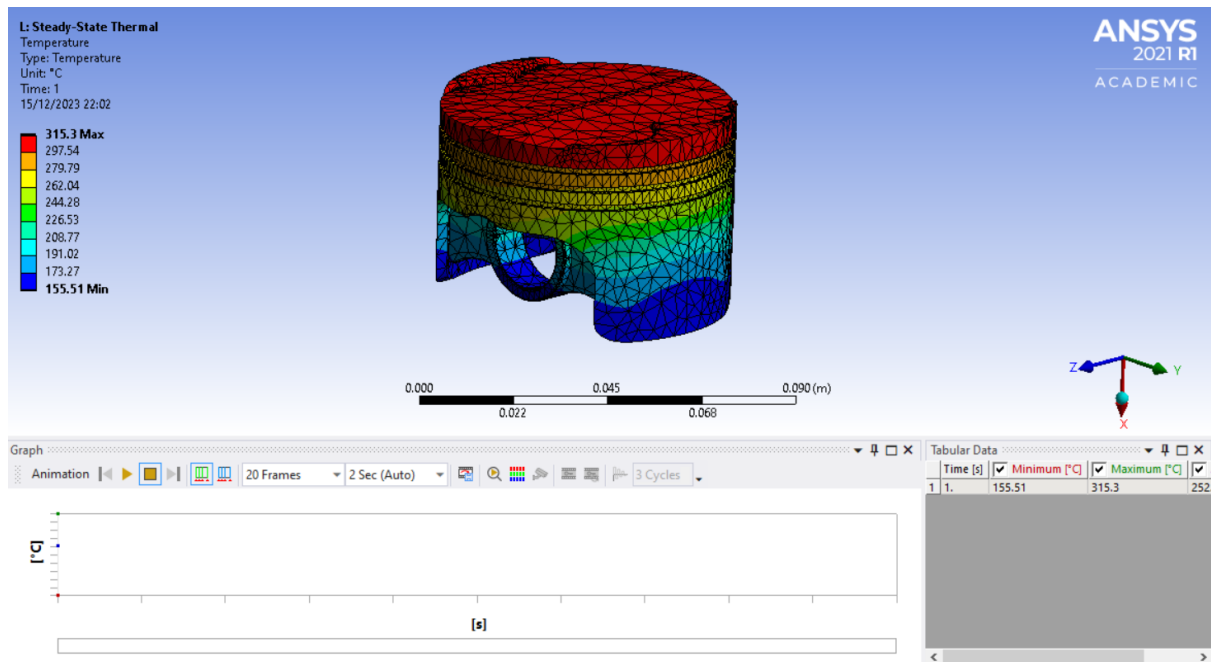


Figure 31: New Design Piston Temperature 1

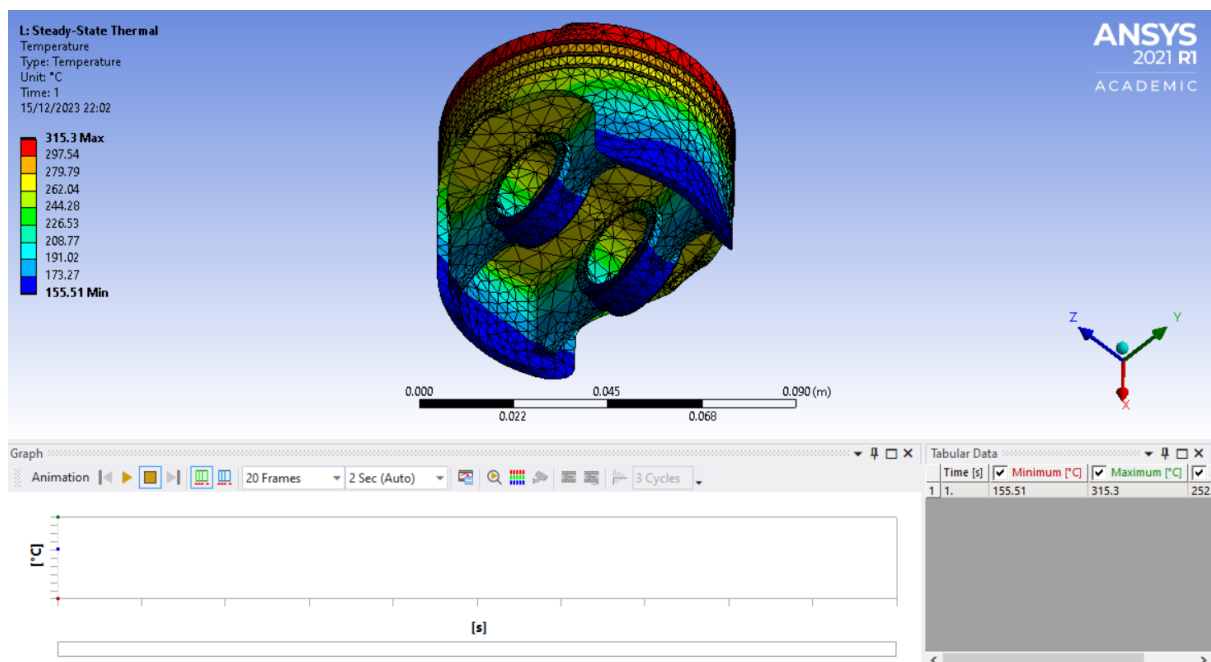


Figure 32: New Design Piston Temperature 2

3.3.3 Static Structural Study

Piston deformation is shown figure 33 and 34.

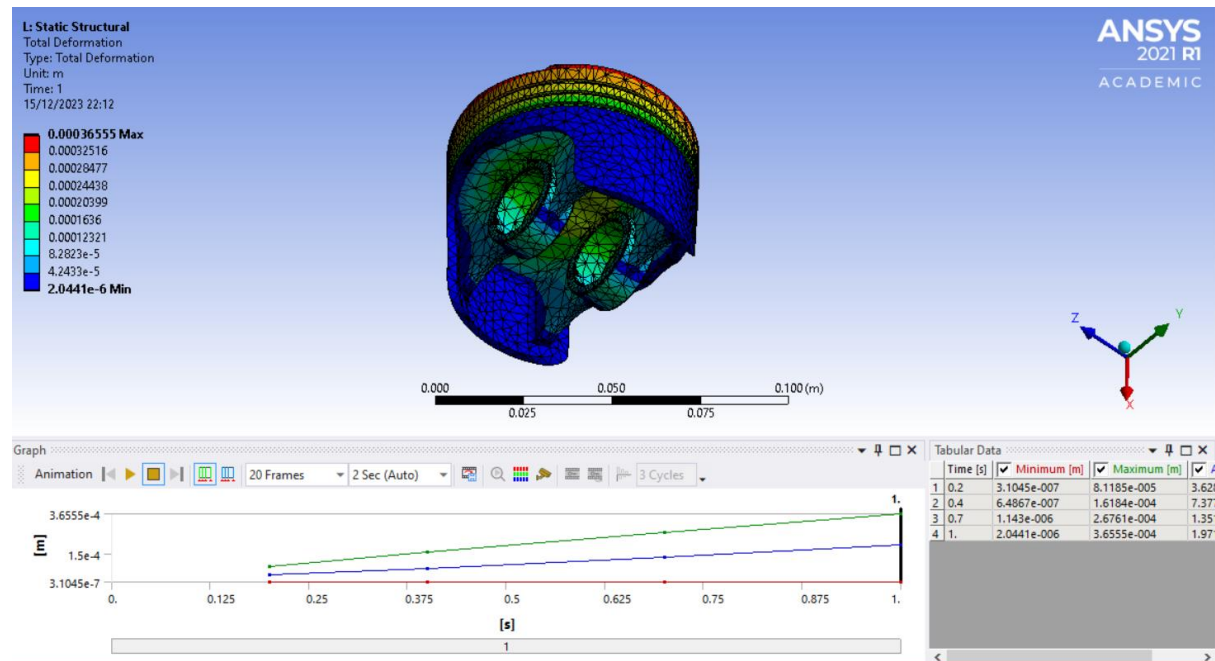


Figure 33: New Design Deformation Results 1

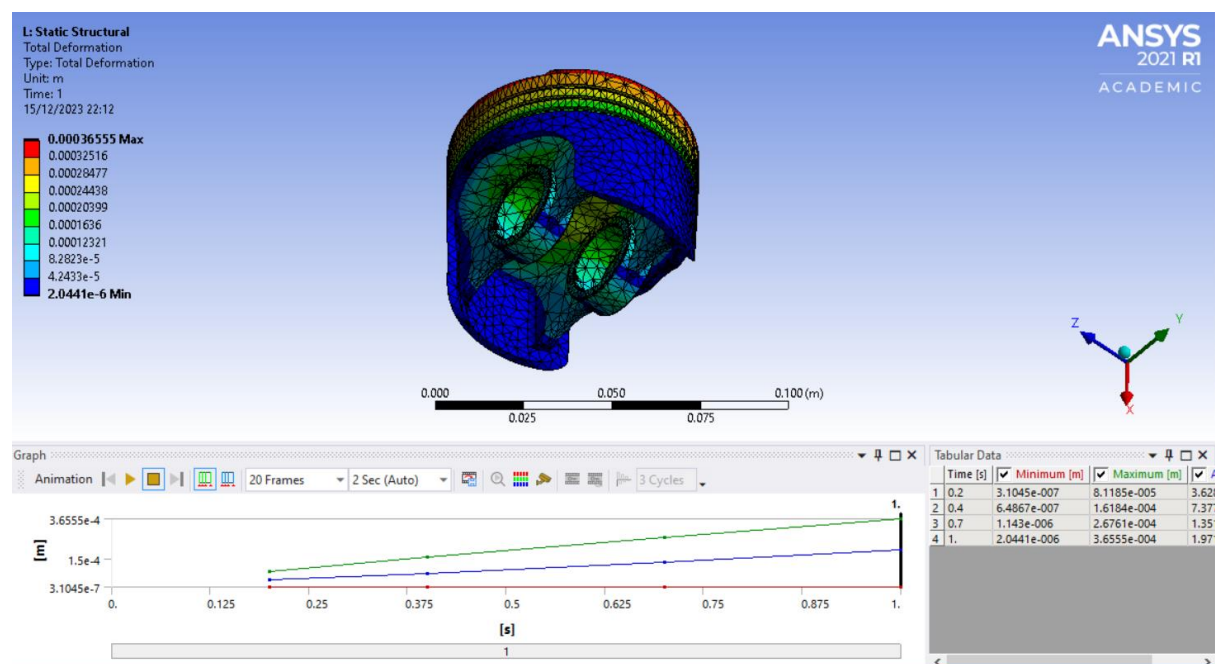


Figure 34: New Design Deformation Results 2

Piston stress results are shown figure 35 and 36.

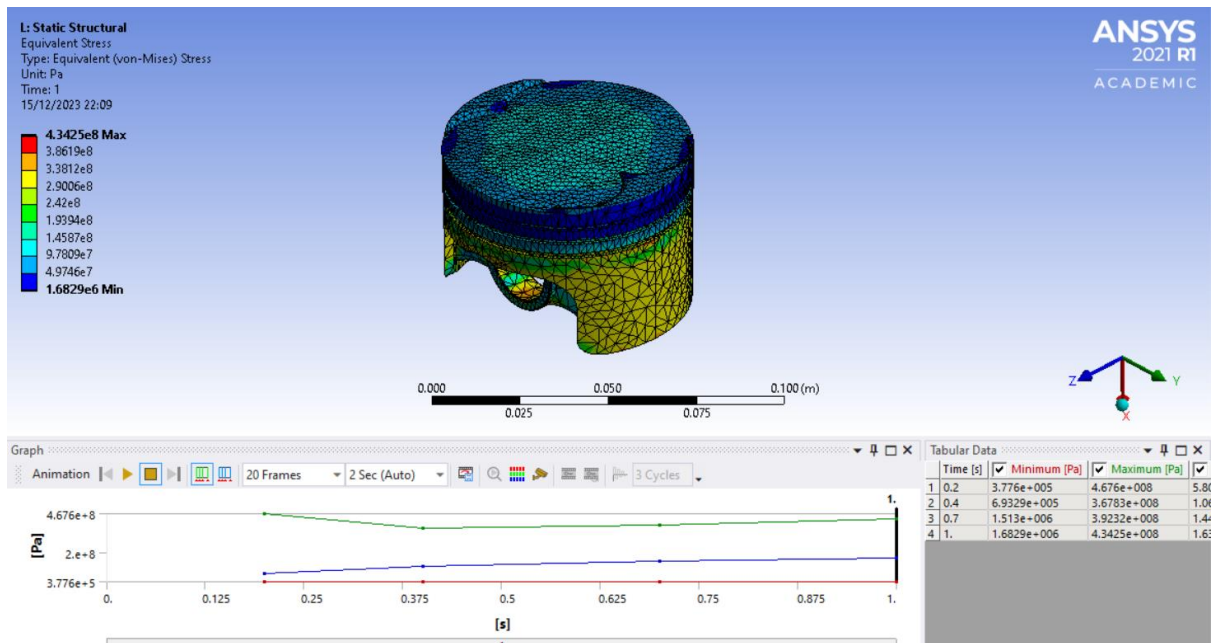


Figure 35: New Design Stress Results 1

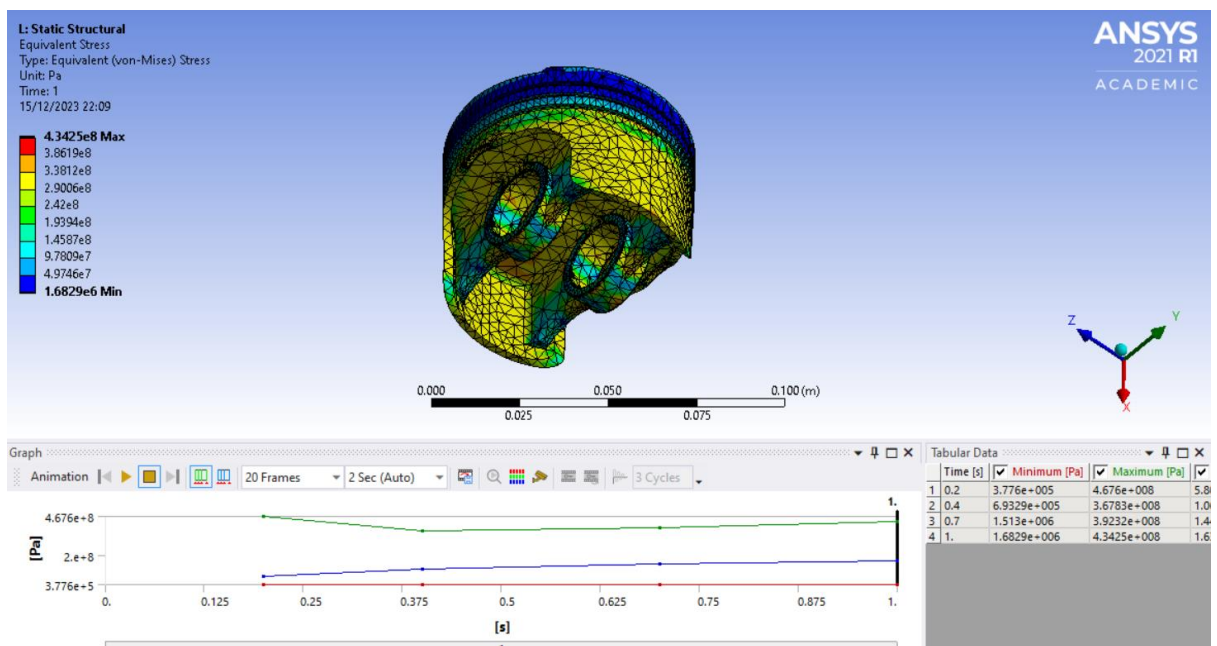


Figure 36: New Design Stress Results 2

4 Analysis

Using the data in table 3, shows all results. The original mass of 0.26237 kg is reduced to 0.25605 Kg for the new material design and 0.25763mm for the new piston design. By changing to Aluminium 6061, this met the aims and objectives specified, reducing the mass as mentioned as well as force, temperature, deformation and most significantly stress. Making a longer lasting and more durable component. The outcome of these results can be verified by “mass reduction reduces friction mean effective pressure” (U.S. Office, n.d.), supporting the theory reduced heat energy is wasted/transferred to the piston. By reducing the maximum occurring displacement and stress it supports the validity of results.

By ensuring a fair test throughout, enabled comparable end results to select the most effective design. It can also be estimated from the results, by combining the aluminium 6061 T451 with the new piston design, mass could further be reduced.

Table 3: Comparing Results

Model Variant	Mass (Kg)	Maximum Force (N)	Maximum Temperature (°C)	Max Displacement (mm)	Maximum Stress (MPa)
Base Model (Aluminium 7020 T6)	0.26237	2437.8	315.33	0.37698	429.6
New Material (Aluminium 6061 T451)	0.25605	2379.2	306.79	0.33483	199.1
New Design (Aluminium 7020 T6)	0.25763	2393.8	315.3	0.36555	434.3

Similarities between temperature gradients compared with the original model shows peak temperatures at the crown, slightly lower on the new material with a drop in temperature down to the piston skirt. The skirt temperature is higher on Aluminium 6061 T451 by 12°C indicating Aluminium 6061 T451 is a more suitable material with a higher thermal conductivity. Ability to transfer heat faster and easier means the piston can be cooled more efficiently when oil splashes onto the skirt (Li & Ren, 2011). Temperature is only slightly reduced with the new design as the reduction in force, thermal conductivity remains the same as material does not change.

Overall validity of simulation results follow the predicted and scientific studies, although the true values might vary slightly. Further refinement to model meshes and additional setup parameters would improve the numerical accuracy (Champion & Ensminger, 1988). Using validation examples outlined in the literature review, estimates show a maximum percentage error of 1.5%. Percentage error here is expected to be higher, due to the larger numbers found and setup conditions used affecting the validity of static structural results.

4.1 Limitations of CAE

Limitations of CAE and FEA are important to consider when testing and designing components. With only approximate solutions supplied, results should not be taken as factual. FEA software is also expensive and should only be used in industry by trained and qualified personnel (Tickoo, 2021).

Understanding the limitations and potential problems helps users to adjust simulations and analyse results accordingly. All software is different and how it is operated in accordance to its limitations are critical to simulation accuracy. Human error can also cause discrepancies by inputting incorrect or inaccurate data, discrediting results (Nkuleon, 2021).

Meshing is a key factor in simulation accuracy. The testing section 2.3 shows the impact of changing mesh sizing on a simple cantilever beam on the result accuracy. With more complex geometry the impact of incorrect meshing could cause a higher error in calculations. Meshing is also limited by computer hardware, as it becomes finer, simulation time increases and may not work on less powerful devices (Rebelo, 2021).

Material properties must be precise and sometimes cannot be defined accurately enough to represent certain characteristics. Lack of material knowledge or choices within software can cause problems for engineers, simulating unsuitable results (Nkuleon, 2021). Limited knowledge of new materials (without a full technical sheet) cannot be utilised within CAE until detailed testing has been conducted on the material itself. This therefore cannot be used to predict a designs characteristic with a developing material.

5 Conclusion

In conclusion, it is clear in results that changing material and designs in accordance with simulation data can improve a components structure. Accuracy of simulations can be questionable and it is important to validate these results with proven actual values and compare this with mesh sizing. A simulations numerical answers may not be accurate but determining this is vital to overcome engineering problems safely and successfully. Trending in the validation section, percentage error raised with an increase in mesh element size at the expense of time. By using trends and reliable patterns within the simulations such as stress points, design improvements can be focused around specific areas. There can be many limitations and inaccuracies within studies caused by human error, hardware or lack of input data. This is important to consider when using results for final products.

6 Reference List

Ansys Learning, 2022. *Youtube: Determining Which Support to Use in Ansys Mechanical — Lesson 1*. [Online]

Available at: <https://www.youtube.com/watch?v=H5TnGWFS05U>

[Accessed 15 December 2023].

Ansys, 2023. *Ansys Granta EduPack*. [Online]

Available at: <https://www.ansys.com/en-gb/products/materials/granta-edupack>

[Accessed 14 December 2023].

Champion, E. & Ensminger, M., 1988. *Finite Element Analysis with Personal Computers*. Egypt: Taylor & Francis.

Cybellium Ltd, 2023. *Mastering Kinematics*. n.p: Cybellium Ltd.

Dub, M., 2017. *Dlubal Verification Example: Bimetallic Strip*. [Online]

Available at: <https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/000078>

[Accessed 15 December 2023].

Dub, M., 2021. *Verification Example: Geometric Linear Analysis*. [Online]

Available at: <https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/009016>

[Accessed 12 December 2023].

Fukumoto, Y. & Lee, G., 1992. *Stability and Ductility of Steel Structures Under Cyclic Loading*. United Kingdom: CRC-Press.

Hall, N., 2023. *NASA: Newton's Laws of Motion*. [Online]

Available at: <https://www1.grc.nasa.gov/beginners-guide-to-aeronautics/newtons-laws-of-motion/>

[Accessed 14 December 2023].

Kurowski, P., 2019. *Thermal Analysis with SOLIDWORKS Simulation 2019 and Flow Simulation 2019*. United States: SDC Publications.

Li, Y. & Ren, S., 2011. Basic Properties of Building Decorative Materials. In: *Building Decorative Materials*. United Kingdom: Elsevier Science, pp. 10-24.

MAHLE GmbH, 2013. *Pistons and Engine Testing*. 1st ed. Germany: Springer.

Meyer, W., 2012. Evaluation of Mathematical Modeling. In: *Concepts of Mathematical Modeling*. United Kingdom: Dover Publications, pp. 143-145.

Naterer, G., 2002. *Heat Transfer in Single and Multiphase Systems*. Ukraine: CRC Press.

Nkuleon, 2021. *FEA limitations in Mechanical Engineering*. [Online]

Available at: <https://nkuleon.com/FEA-LIMITATIONS/>

[Accessed 16 December 2023].

Rebelo, F., 2021. *Advances in Ergonomics in Design: Proceedings of the AHFE 2021 Virtual Conference on Ergonomics in Design, July 25-29, 2021, USA*. Germany: Springer International Publishing.

Siemens, 2023. *Computer Aided Engineering (CAE)*. [Online]

Available at: <https://www.plm.automation.siemens.com/global/en/our-story/glossary/computer-aided-engineering-cae/13112>

[Accessed 12 December 2023].

SimScale, 2023. *What Is FEA | Finite Element Analysis?*. [Online]

Available at: <https://www.simscale.com/docs/simwiki/fea-finite-element-analysis/what-is-fea-finite-element-analysis/#>

[Accessed 15 December 2023].

Tickoo, S., 2021. *SOLIDWORKS 2022: A Tutorial Approach*. 6th ed. n.p.: CAD/CIM Technologies.

Twigg, P., 2012. *Science for Motor Vehicle Engineers*. United Kingdom: Elsevier Science.

U.S. Office, n.d.. *Advanced automotive technology : visions of a super-efficient family car*. United States: Diane Publishing Co.