

University of Wales Trinity Saint David
School of Engineering

Application of Computer Aided Engineering Processes
Structural and Fluid Analysis ACMA6010

Nathan Skyrme

2111290

10th November 2023

Contents

1	Introduction	4
2	Validation	5
2.1	Aim of Validation	5
2.2	Bimetallic Strip Specification	5
2.3	Analytical Solution	5
2.4	Bimetallic Strip Validation.....	6
2.4.1	Mesh Validation - 0.5mm	7
2.4.2	Mesh Validation – 1mm.....	9
2.4.3	Mesh Validation – 0.25mm.....	10
2.4.4	Mesh Validation – 0.1mm.....	11
2.5	Mesh Validation - Failure Point	12
2.6	Validation Conclusion	13
3	Engine Analysis	14
3.1	Engine System Specification	14
3.2	Rigid Dynamics – Maximum Joint Loading	16
4	Single Component Analysis.....	17
4.1	Piston	17
4.2	Thermal Conditions.....	17
4.3	Static Structural	18
4.4	Structural Analysis Conclusion.....	23
5	Continuous Improvement.....	24
5.1	Stress and Strain	24
5.2	Strain Difference	24
6	Conclusion.....	27
	References	28

Figure 1 - Static Structural Solver in ANSYS	6
Figure 2 - Bimetallic Strip - 0.5mm Mesh	7
Figure 3 - Bimetallic Strip - Deformation Profile - 0.5mm Mesh	8
Figure 4 - Bimetallic Strip - 1mm Mesh	9
Figure 5 - Bimetallic Strip - Deformation Profile - 1mm Mesh	9
Figure 6 - Bimetallic Strip - 0.25mm Mesh	10
Figure 7 - Bimetallic Strip - Deformation Profile - 0.25mm Mesh	10
Figure 8 - Bimetallic Strip - 0.1mm Mesh	11
Figure 9 - Bimetallic Strip - Deformation Profile - 0.1mm Mesh	11
Figure 10 - Close-up Bimetallic Strip - Deformation Profile - 0.1mm Mesh	12
Figure 11 - Error Message	12
Figure 12 - Engine Assembly	14
Figure 13 - Piston to Gudgeon Pin Joint	15
Figure 14- Crankshaft - Body-to-Ground Joint.....	15
Figure 15 - Piston - Body-to-Ground Joint	16
Figure 16 - Joint Probe - Loading and Results.....	16
Figure 17 - Piston Mesh	17
Figure 18 - Piston - Convection Points.....	18
Figure 19 - Piston - Temperature Profile	18
Figure 20 - Piston - Motion Loads.....	19
Figure 21 - Piston Crown - Pressure.....	19
Figure 22 - Piston - Deformation Profile.....	20
Figure 23 - Piston Crown - Deformation Profile	20
Figure 24 - Piston Underside - Deformation Profile	21
Figure 25 - Piston Pin Bore - Deformation Profile	21
Figure 26 - Piston - Stress Profile	22
Figure 27 - Piston Crown - Stress Profile	22
Figure 28 - Piston Underside - Stress Profile	23
Figure 29 - Piston Bore Hole - Stress Profile	23
Figure 30 - Piston - Temperature Profile - Carbon Steel	24
Figure 31 - Piston - Temperature Profile - Carbon Steel	25
Figure 32 - Piston - Stress Profile - Carbon Steel	25
Figure 33 - Piston - Stress Profile - Carbon Steel	25
 Table 1. Bimetallic Strip Specification [3]	 5
Table 2 - Copper Materials Properties.....	7
Table 3 - Invar Materials Properties	7
Table 4 - Total Deformation - 1mm Mesh	8
Table 5 - Total Deformation - 1mm Mesh	9
Table 6 - Total Deformation - 0.25mm Mesh	10
Table 7 - Total Deformation - 0.1mm Mesh	12

1 Introduction

Computer-aided engineering (CAE) plays a fundamental role in the design and validation processes in engineering. It enables the simulation and analysis of diverse physical phenomena, including structural integrity, fluid dynamics, heat transfer, and electromagnetic fields. CAE is particularly useful when dealing with intricate geometries that are difficult or unfeasible to solve manually [1]. By using advanced computer software and numerical methods, CAE enables engineers to virtually test and optimise designs before physical prototyping, saving time and resources [2]. Additionally, CAE allows for the exploration of different design alternatives and the identification of potential issues or weaknesses, leading to improved product performance and reliability [5] [6].

In order to examine and assess the capabilities and constraints of CAE software and methodologies, a comprehensive analysis employing finite element analysis (FEA) has been conducted utilising ANSYS. This study focuses on evaluating the structural, thermal, and kinematic characteristics of an engine system model to ascertain its performance and identify potential areas for enhancement within the model. This paper aims to explore the potential applications of these technologies throughout the wider engineering sector.

2 Validation

2.1 Aim of Validation

To evaluate the suitability of ANSYS for conducting analysis, it is crucial to perform validation testing to ascertain the software's capability in accurately calculating output results. In order to accomplish this, a basic test case was conducted using ANSYS software to analyse the overall deflection at the free end of a bimetallic strip. The obtained findings were compared with the analytical solution of the test case in order to evaluate the accuracy of the results and analyse the influence of mesh sizing on the outcomes, aiming to identify a point of mesh independence.

2.2 Bimetallic Strip Specification

The bimetallic strip is comprised of two materials, invar and copper. The left end of the bimetallic strip is affixed, while the right end remains free, it is subjected to a thermal gradient denoted as T_c [8].

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	$^{\circ}\text{C}$

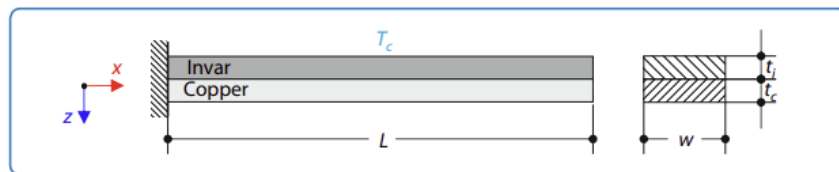


Table 1. Bimetallic Strip Specification [3]

2.3 Analytical Solution

The analytical answer is derived using the methodology presented in [7]. The initial equation provides the dimensionless coefficient K_1 , which is utilised in calculating the equivalent stiffness.

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

The following formula then calculates the maximum deflection of the bimetallic strip.

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

2.4 Bimetallic Strip Validation

Based on the validation example in reference [7], a basic three-dimensional model of a bimetallic strip was generated using Solidworks software. The process of generating the strip involved constructing a rectangle with the specified dimensions outlined in the validation example, namely 1mm by 5mm. Subsequently, this shape was extended (extruded) in the third dimension to achieve a length of 100mm. Then finally the component was incorporated into an assembly, wherein two instances of the component were joined (mated) together.

So as to evaluate the total deflection, a static structural solver was established within the ANSYS software.

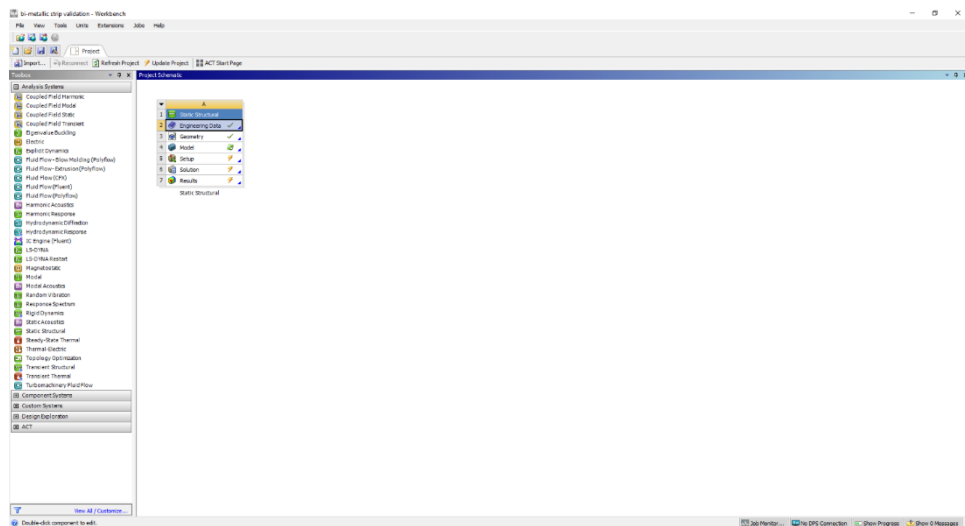


Figure 1 - Static Structural Solver in ANSYS

The data obtained from the validation example was then inputted into the Engineering Data section and used to generate the material profiles for invar and copper.

Properties of Outline Row 3: Copper				
	A	B	C	D E
1	Property	Value	Unit	
2	Material Field Variables	Table		
3	Isotropic Instantaneous Coefficient of Thermal Expansion	2E-05	C ⁻¹	
4	Isotropic Elasticity			
5	Derive from	Young's ...		
6	Young's Modulus	1.3E+05	MPa	
7	Poisson's Ratio	0.354		
8	Bulk Modulus	1.484E+11	Pa	
9	Shear Modulus	4.8006E+10	Pa	

Table 2 - Copper Materials Properties

Properties of Outline Row 4: Invar				
	A	B	C	D E
1	Property	Value	Unit	
2	Material Field Variables	Table		
3	Isotropic Instantaneous Coefficient of Thermal Expansion	1.2E-06	C ⁻¹	
4	Isotropic Elasticity			
5	Derive from	Young's ...		
6	Young's Modulus	1.37E+05	MPa	
7	Poisson's Ratio	0.28		
8	Bulk Modulus	1.0379E+11	Pa	
9	Shear Modulus	5.3516E+10	Pa	

Table 3 - Invar Materials Properties

2.4.1 Mesh Validation - 0.5mm

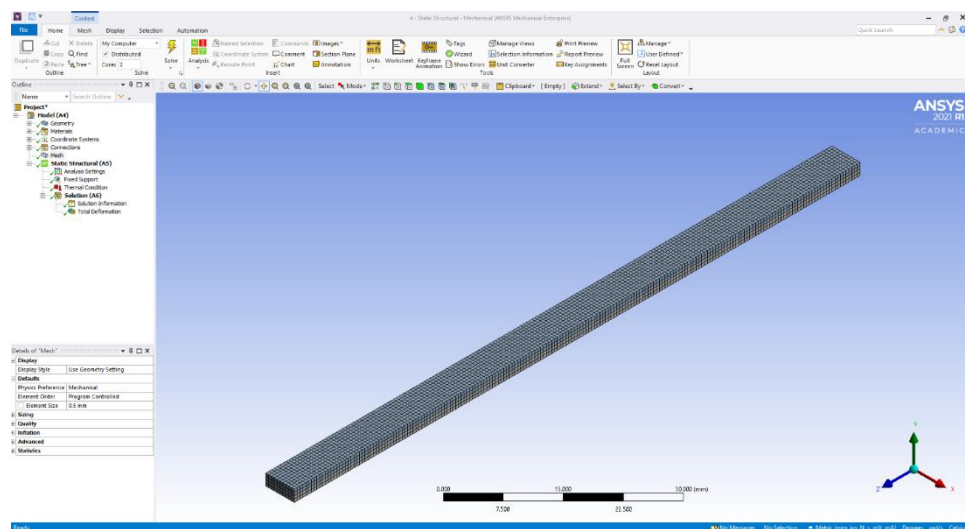


Figure 2 - Bimetallic Strip - 0.5mm Mesh

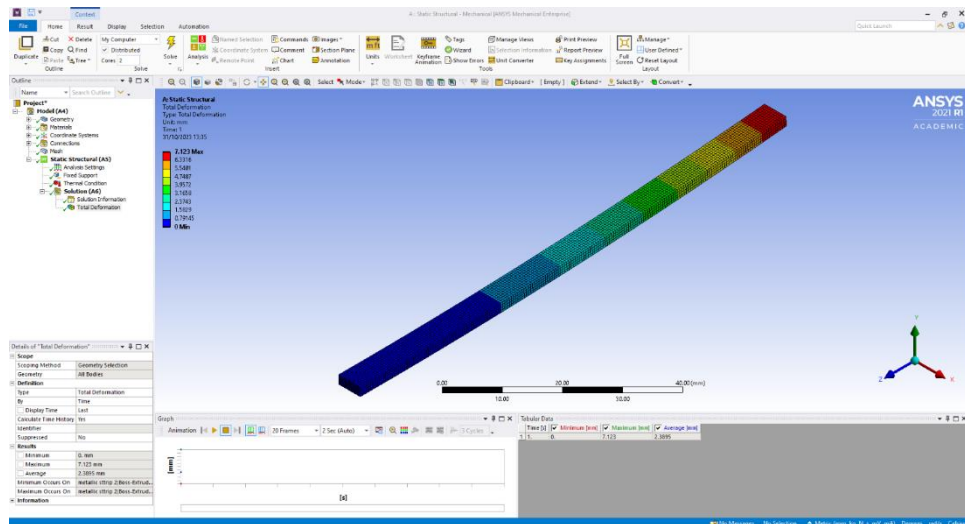


Figure 3 - Bimetallic Strip - Deformation Profile - 0.5mm Mesh

Tabular Data				
	Time [s]	☒ Minimum [mm]	☒ Maximum [mm]	☒ Average [mm]
1	1.	0.	7.123	2.3895

Table 4 - Total Deformation - 1mm Mesh

$$\text{Percentage Error} = \left(\frac{(7.123 - 7.049)}{7.049} \right) \times 100 \approx 1.0498\%$$

As indicated in section 2.3, the deflection computed by analytical methods amounts to 7.049mm. When utilising a 0.5mm mesh, ANSYS yields a deflection measurement of 7.123mm, which is comparable to the expected outcome but exhibits a little deviation from accuracy. When considering a single component in isolation, the observed margin of error can be considered acceptable. However, in a more intricate simulation including numerous components, even little discrepancies can accumulate and result in a significantly greater percentage error. This may exceed a permissible tolerance level.

2.4.2 Mesh Validation – 1mm

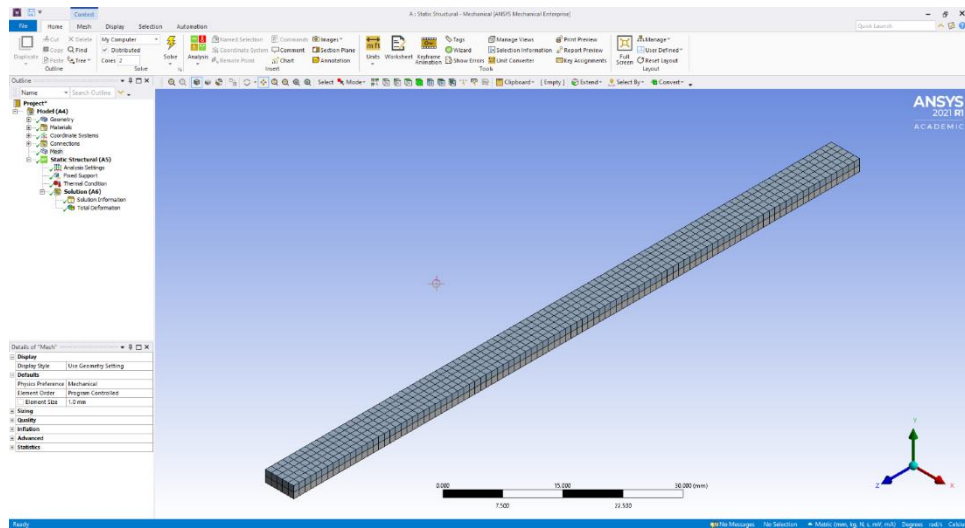


Figure 4 - Bimetallic Strip - 1mm Mesh

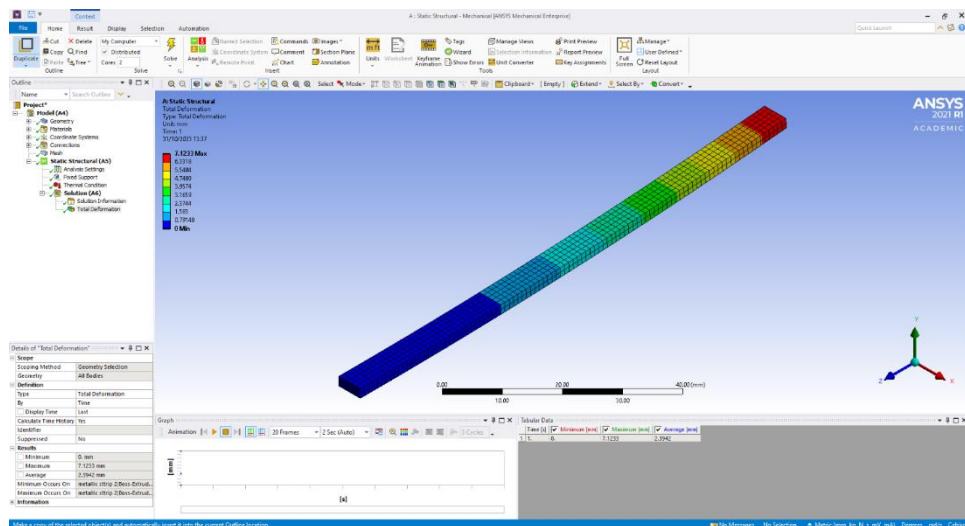


Figure 5 - Bimetallic Strip - Deformation Profile - 1mm Mesh

Tabular Data				
	Time [s]	☒ Minimum [mm]	☒ Maximum [mm]	☒ Average [mm]
1	1.	0.	7.1233	2.3942

Table 5 - Total Deformation - 1mm Mesh

$$\text{Percentage Error} = \left(\frac{(7.1233 - 7.049)}{7.049} \right) \times 100 \approx 1.0541\%$$

When utilising a mesh size of 1mm, which is twice as large as the previous mesh size, the obtained findings exhibit only a minor disparity. Specifically, the deflection values change by a mere 0.0003mm, and the level of inaccuracy increases by only approximately 0.0043%. The

presence of these minor discrepancies may suggest the existence of underlying factors contributing to the inaccuracies, such as intrinsic limitations in the CAE and ANSYS software, inaccuracies in the modelling process, deficiencies in the mathematical models employed, inadequate computational resources, and so forth.

2.4.3 Mesh Validation – 0.25mm

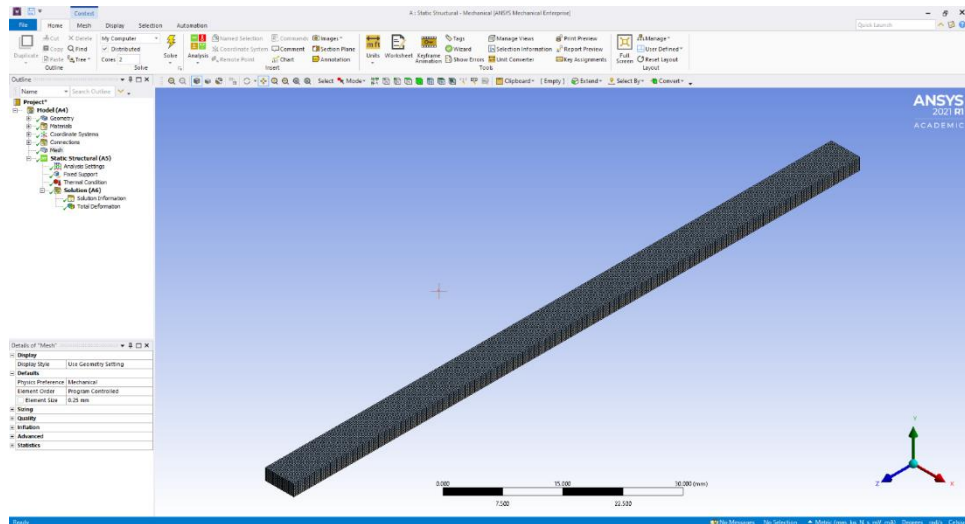


Figure 6 - Bimetallic Strip - 0.25mm Mesh

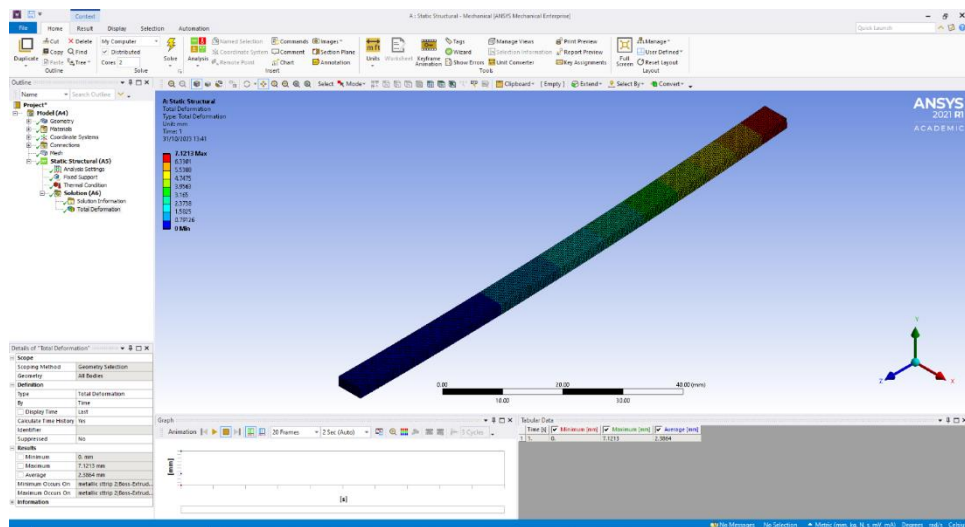


Figure 7 - Bimetallic Strip - Deformation Profile - 0.25mm Mesh

Tabular Data				
	Time [s]	☒ Minimum [mm]	☒ Maximum [mm]	☒ Average [mm]
1	1.	0.	7.1213	2.3864

Table 6 - Total Deformation - 0.25mm Mesh

$$\text{Percentage Error} = \left(\frac{(7.1213 - 7.049)}{7.049} \right) \times 100 \approx 1.0257\%$$

The utilisation of a reduced mesh size (0.25mm) has resulted in an anticipated outcome of enhanced accuracy, leading to a decrease in the percentage error by approximately 0.0241% in comparison to the findings obtained using a 0.5mm mesh. The observed phenomenon indicates that reducing the mesh size by half does not result in a proportional reduction in error. This finding suggests that there is no direct correlation between mesh size and accuracy, as supported by the evidence presented in reference [4].

2.4.4 Mesh Validation – 0.1mm

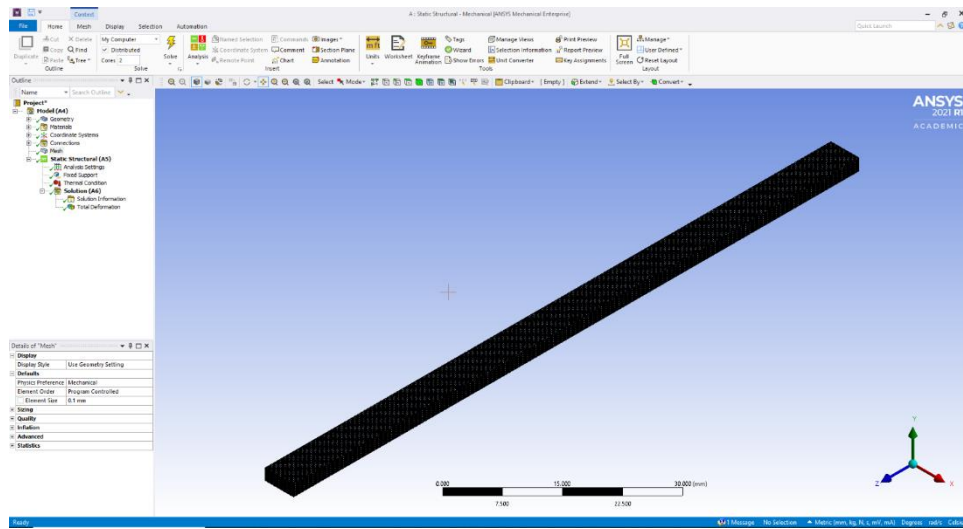


Figure 8 - Bimetallic Strip - 0.1mm Mesh

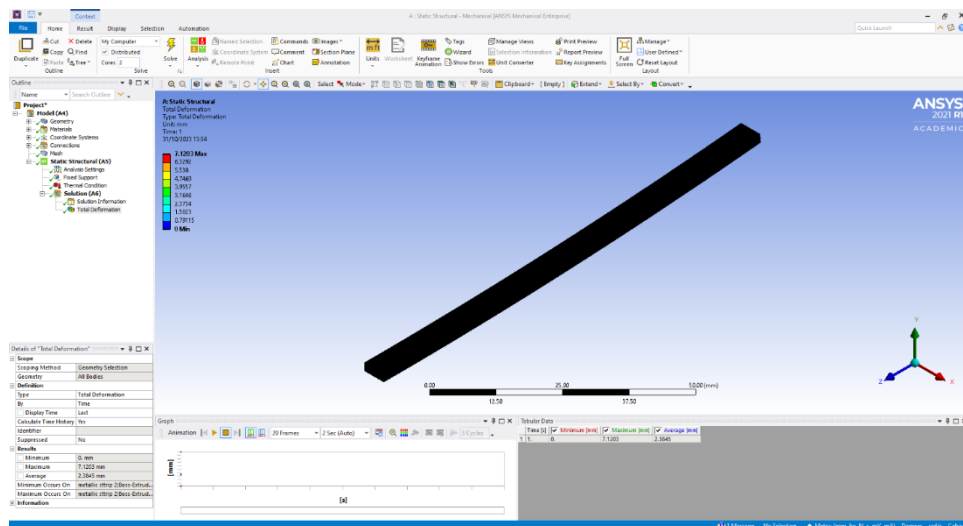


Figure 9 - Bimetallic Strip - Deformation Profile - 0.1mm Mesh

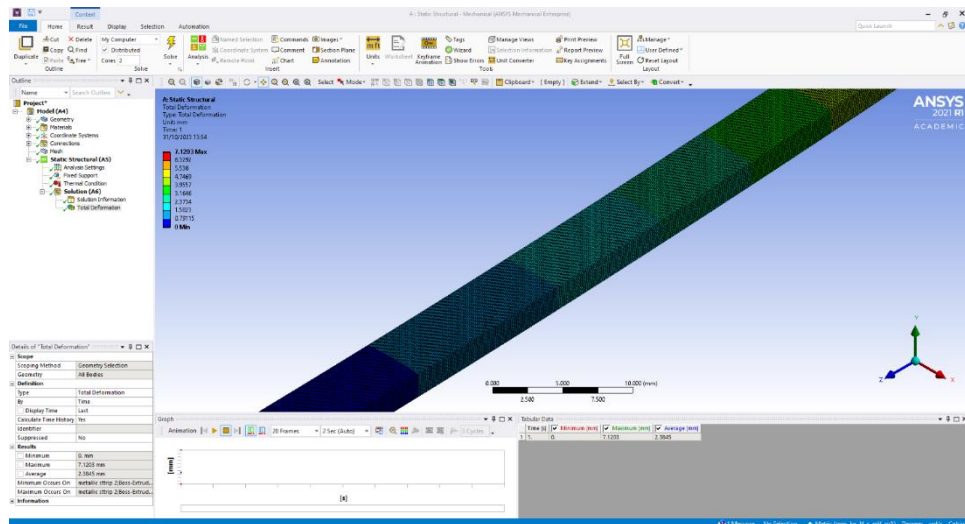


Figure 10 - Close-up Bimetallic Strip - Deformation Profile - 0.1mm Mesh

Tabular Data				
	Time [s]	☒ Minimum [mm]	☒ Maximum [mm]	☒ Average [mm]
1	1.	0.	7.1203	2.3845

Table 7 - Total Deformation - 0.1mm Mesh

$$\text{Percentage Error} = \left(\frac{(7.1203 - 7.049)}{7.049} \right) \times 100 \approx 1.0115\%$$

The precision of the outcomes progressively improves with the reduction in mesh size, suggesting the potential for identifying the threshold of mesh independence. The determination of mesh independence was achieved through a systematic process of trial and error, as documented in study [2]. The study revealed that larger mesh sizes tend to yield less accurate results, while smaller meshes may lead to the propagation of local damages from an initiation point, resulting in atypical phenomena such as uneven distribution of damage or buckling. The implementation of a mesh with a smaller size of 0.1mm results in a marginal improvement of 0.0426% in accuracy when compared to the first mesh with an aperture size of 0.5mm.

As the mesh size decreases, there are evident computational restrictions in terms of graphical clarity. Specifically, as the mesh becomes smaller, the mesh grid and the overall deformation profile become less distinct. This effect is also present but less pronounced when the mesh size is set at 0.25mm. The observed phenomenon can be attributed to limitations in processing capacity and software resolution, which interact with one other.

2.5 Mesh Validation - Failure Point

Messages			
Error	Text	Association	Timestamp
	Insufficient memory was available for the solver engine to obtain a solution. Try simplifying the model.	Project>Model>Static Structural>Solution	Tuesday, October 31, 2023 2:17:57 PM

Figure 11 - Error Message

When attempting to calculate the deformation using an element size of 0.05mm, an error message was displayed indicating that there is not enough memory available for the solver engine to produce a solution. The error message suggests simplifying the model in order to reduce the issue size. This observation highlights a distinct fundamental constraint in the computational capability of the system. If there had been an adequate amount of processing capacity, it may be contended that it might have been feasible to determine the point at which mesh independence is achieved. It could also be argued that the mesh was already approaching this point, as seen by the minimum percentage change observed. Given the simplicity of the model's geometry, it seems probable that the observed percentage error stems from fundamental software limitations rather than resolvable human mistake.

2.6 Validation Conclusion

Based on the findings obtained from the validation and mesh independence analysis, it can be inferred that ANSYS software is a suitable tool for conducting a deformation study. Moreover, its use can be extended to encompass both larger systems and individual components within such systems. The findings indicate that ANSYS demonstrates the ability to provide outcomes that closely align with the analytical solution, within an acceptable margin of error.

3 Engine Analysis

3.1 Engine System Specification

The Computer Aided Design (CAD) model, is of a typical engine crankshaft with one connecting rod, gudgeon pin and piston. The additional connecting rods, gudgeon pins, and pistons, have been removed in order to simplify the model, and assess the effects to just one of the pistons. This allows for a more detailed analysis of the impact of the single piston on the engine's performance. The piston is composed of aluminium, and the crankshaft, connecting rod and gudgeon pin, for simplicity, structural steel. The model is generalised rather than engine specific, in order to provide a comprehensive understanding of the average engine's performance.

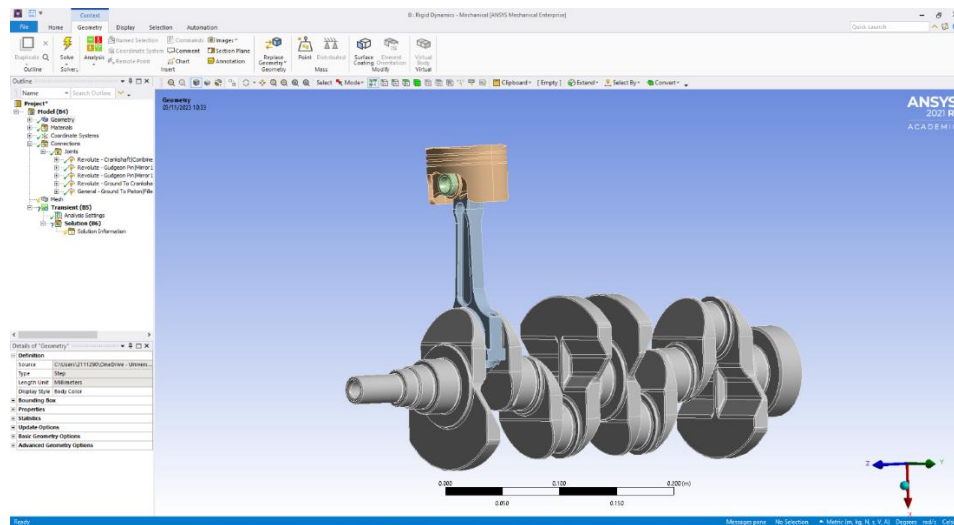


Figure 12 - Engine Assembly

In order to simulate the physical behaviour of the model, joints were first established between the various components. This enables the locking, rotation, and translation of components in a realistic manner. A body-to-ground connection between the crankshaft and the 'ground' was established as a way to secure the assembly in position. Moreover, so as to enable the rotational movement of the piston around the crankshaft body throughout the simulation, a subsequent body-to-ground joint was then also created for the piston, allowing translational motion along the Z-axis while being constrained along the X-axis and Y-axis.

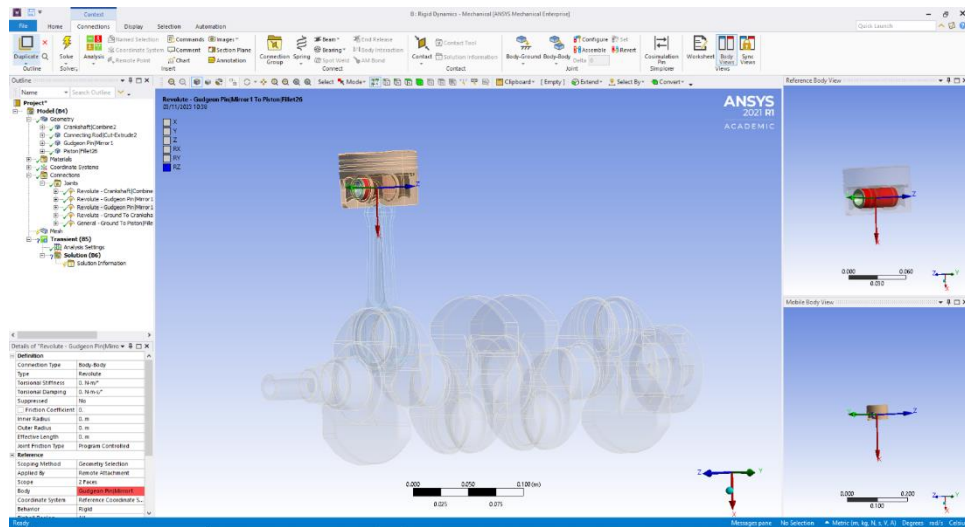


Figure 13 - Piston to Gudgeon Pin Joint

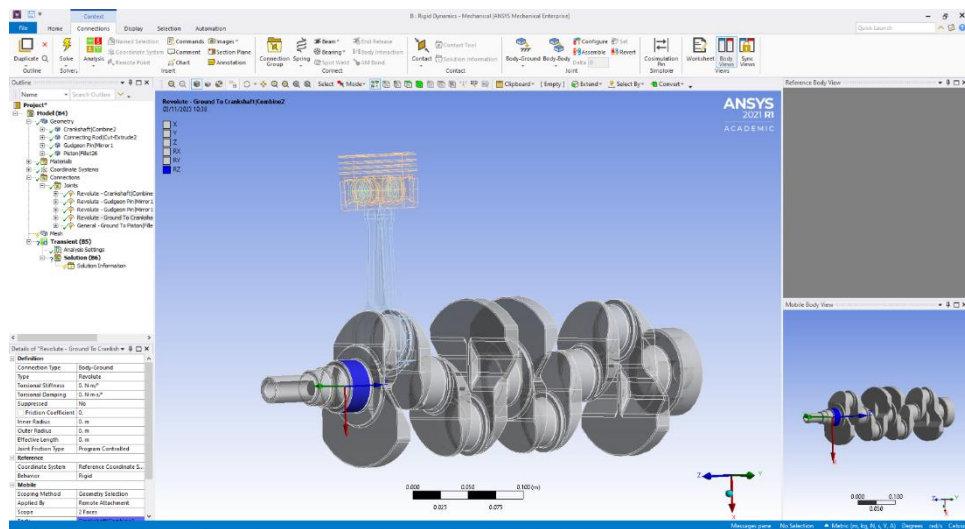


Figure 14- Crankshaft - Body-to-Ground Joint

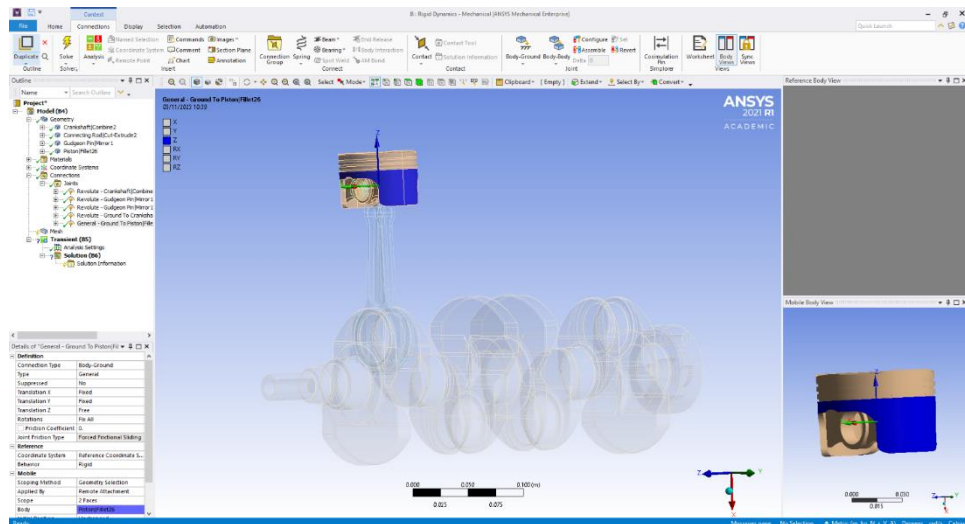


Figure 15 - Piston - Body-to-Ground Joint

3.2 Rigid Dynamics – Maximum Joint Loading

An evaluation of the system's kinematics was initially conducted utilizing a rigid dynamics solver, with the purpose of ascertaining the loadings produced by inertia. The calculation performed analysed the impact of force loadings on the joint connecting the piston and the gudgeon pin, at a rotational velocity of 4000 revolutions per minute (rpm). The findings indicated that the maximum level of joint loading observed in the system was 2391.9N and that the minimum was 0. For analysing the piston this maximum outcome will be used as this allows assessment of the most detrimental stress and strain scenario that the system and its components could potentially be subjected to.

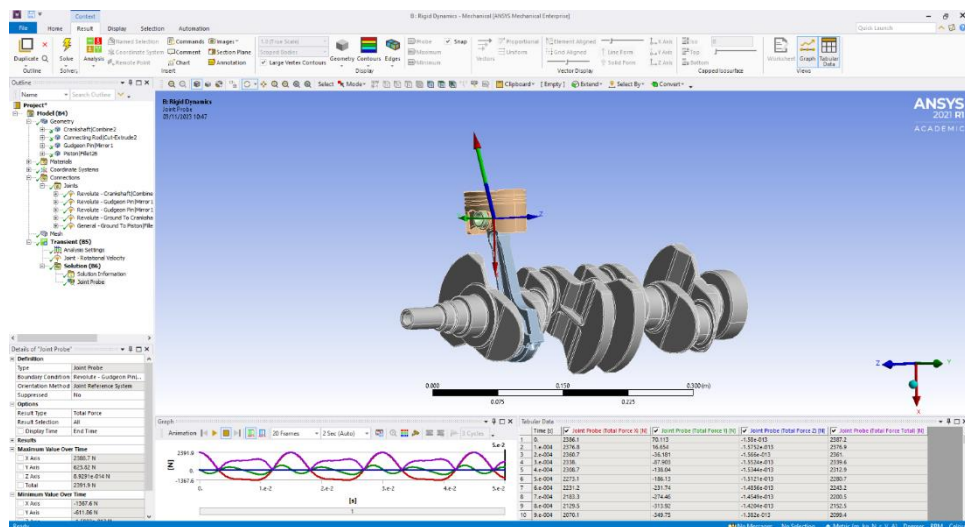


Figure 16 - Joint Probe - Loading and Results

4 Single Component Analysis

4.1 Piston

To be able to evaluate the potential influence of an individual component within the system, a static structural analysis was conducted specifically on the isolated piston. The evaluation of an individual element within the system provides valuable understanding regarding the impacts of various stressors on the system. The evaluation of the piston has been conducted in order to identify areas for improvement, as outlined in section 4.5.

The initial step in the assessment was utilising the outcomes of the rigid dynamics investigation to evaluate the overall deformation and equivalent stress experienced by the piston. The initial determination of the joint loading on the system allows for the identification of realistic loading conditions that would otherwise be uncertain. This information may then be utilised as boundary conditions to enhance the accuracy of subsequent simulations.

4.2 Thermal Conditions

To determine the temperature profile of the piston, a mesh was initially created with a size of 4mm. As a way to account for the varying temperatures experienced by different regions of a piston, convection points were established. These convection points incorporate the heat transfer coefficient and average temperature values for the various surfaces of the piston.

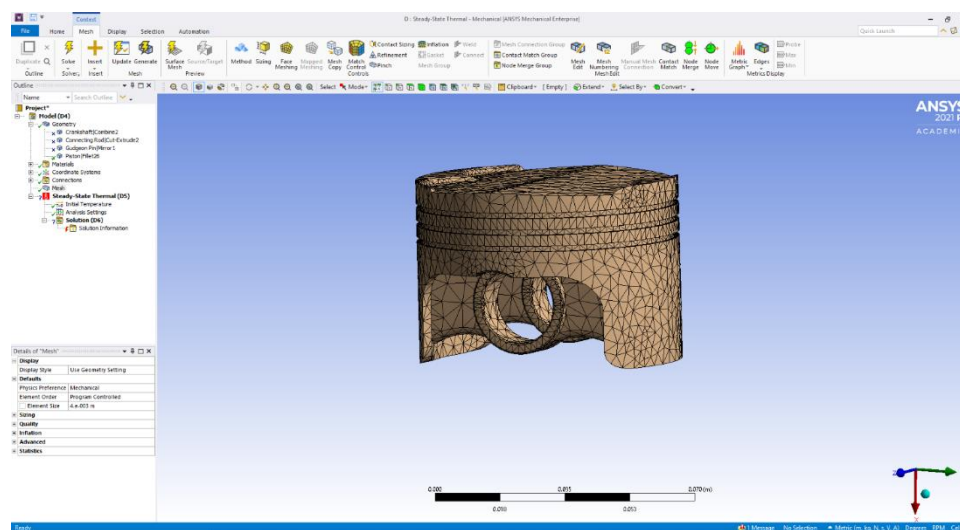


Figure 17 - Piston Mesh

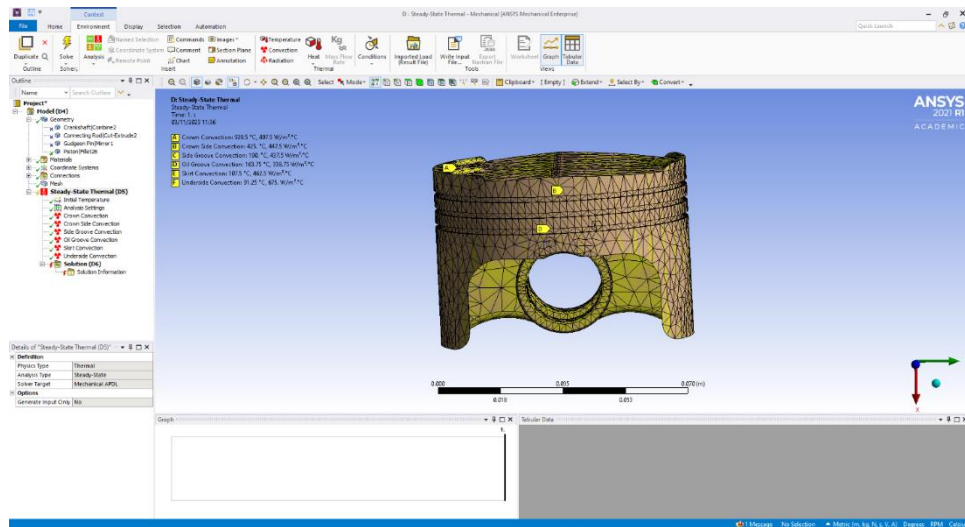


Figure 18 - Piston - Convection Points

From the temperature profile generated it can be seen that the piston experiences a maximum temperature of 302.64°C, peaking at the crown, and a minimum temperature of 161°C around the bottom of the skirt and the bottom of the underside wall.

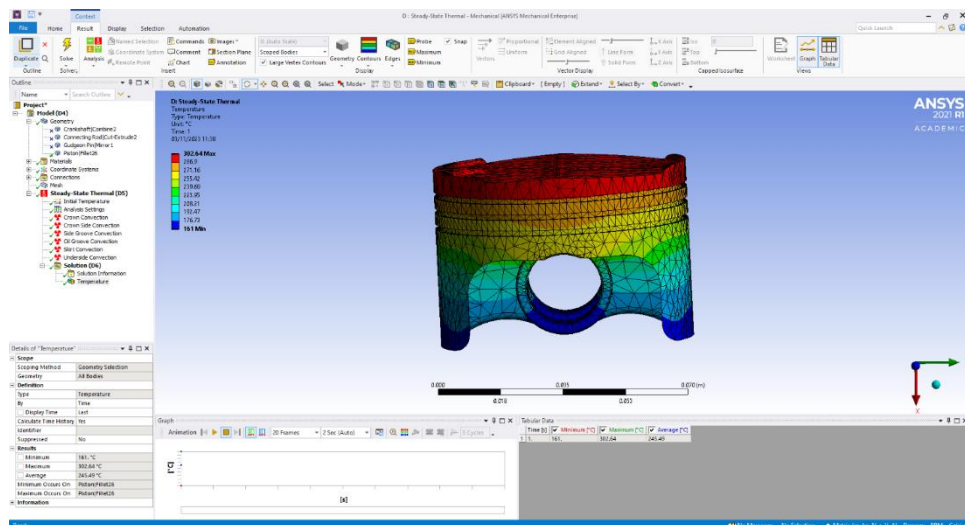
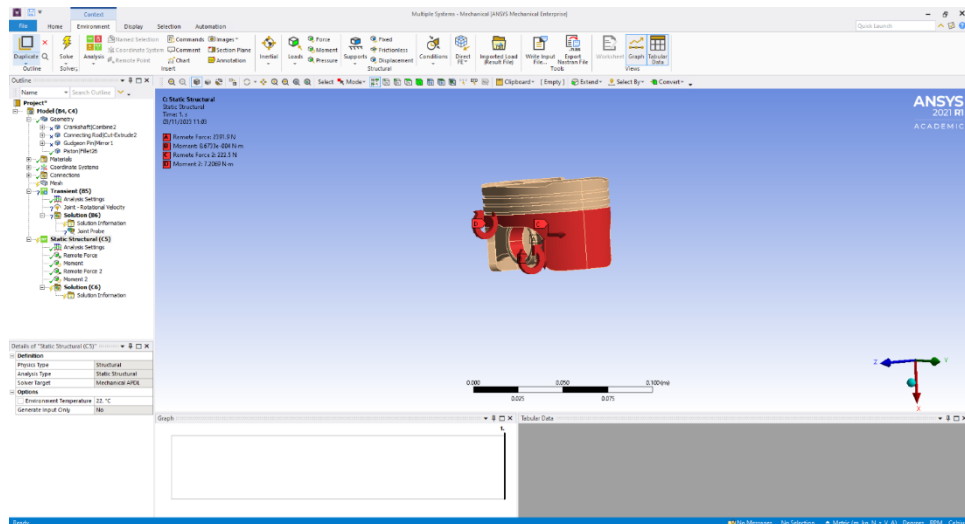


Figure 19 - Piston - Temperature Profile

4.3 Static Structural

To begin static structural study, the solver was connected to the rigid dynamics and steady-state thermal solver to import the previous results and boundary conditions. To apply the loading conditions to the structural study, the motion loads of the maximum joint loading were exported and applied to the face of the piston.



To be able to perform a static structural analysis, it is essential to incorporate a fixed support for the component. However, the inclusion of such support may obstruct certain regions of the piston and consequently result in an incomplete representation of stress distribution. To address this issue, the ANSYS software offers a feature known as "weak springs." The component will be encased by a support system consisting of springs on its external surfaces. Although the springs exhibit minimal stiffness, their presence introduces a certain degree of discrepancy in the outcomes, so augmenting the percentage error and causing the output to deviate further from its true value.

To increase the realism of the simulations, a boundary condition was imposed by incorporating a maximum pressure of 105.37 bar on the crown of the piston, as it would typically experience combustion pressure.

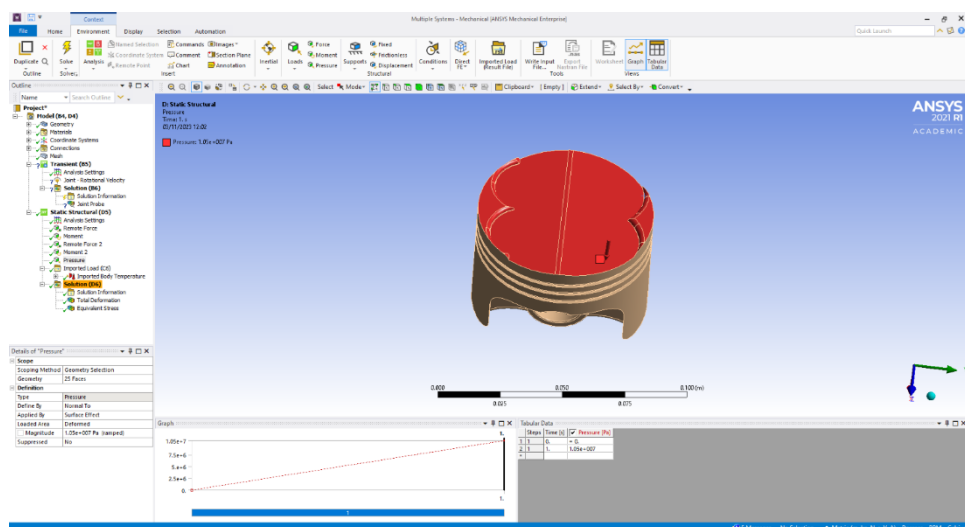


Figure 21 - Piston Crown - Pressure

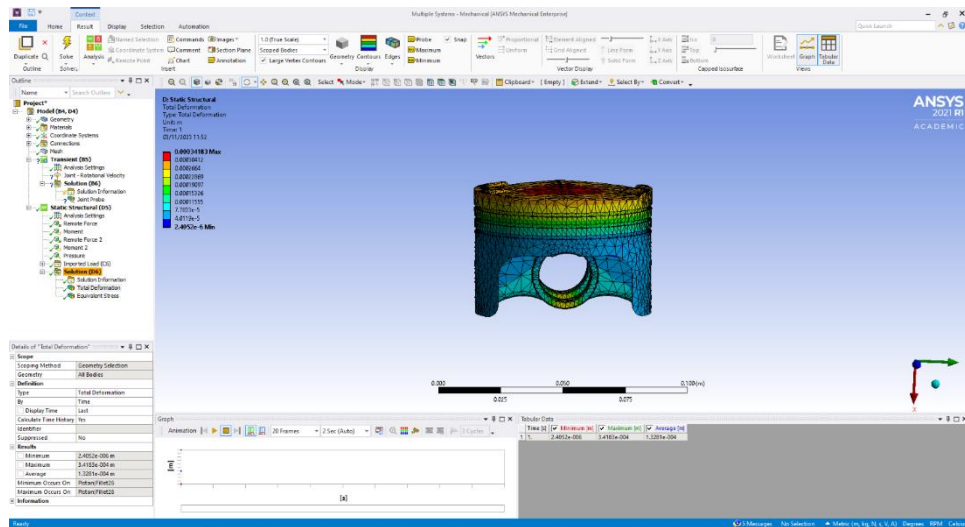


Figure 22 - Piston - Deformation Profile

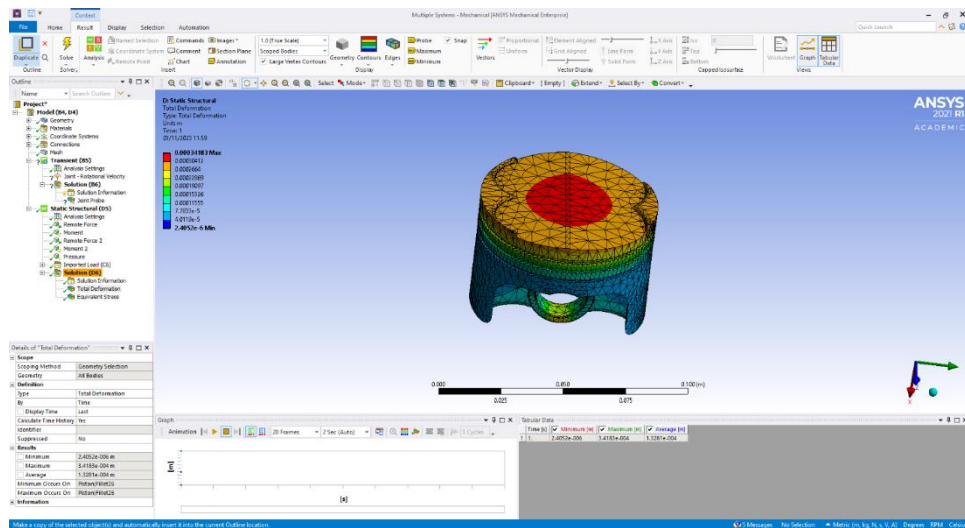


Figure 23 - Piston Crown - Deformation Profile

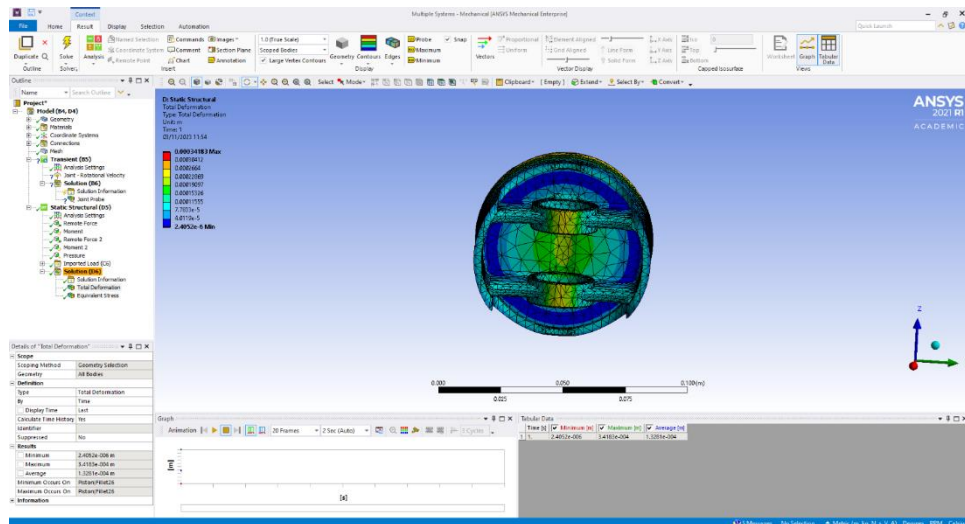


Figure 24 - Piston Underside - Deformation Profile

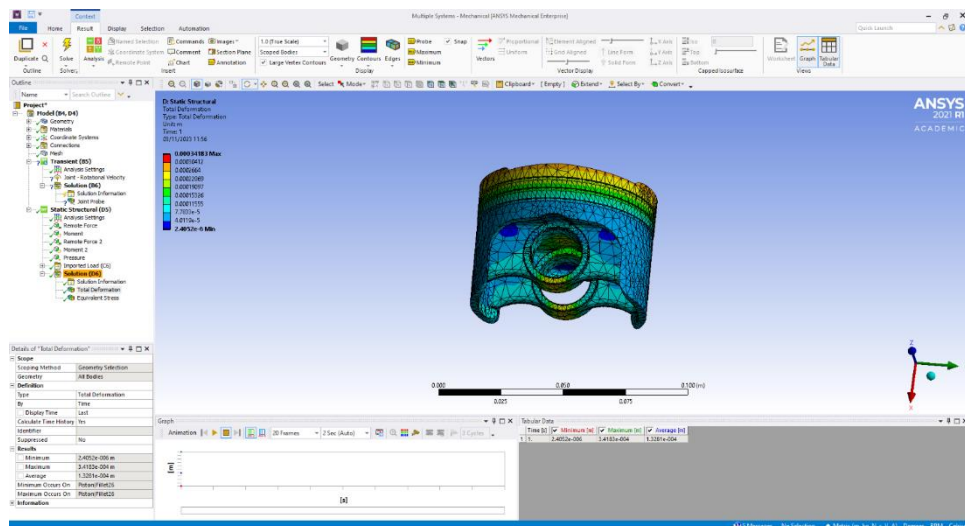


Figure 25 - Piston Pin Bore - Deformation Profile

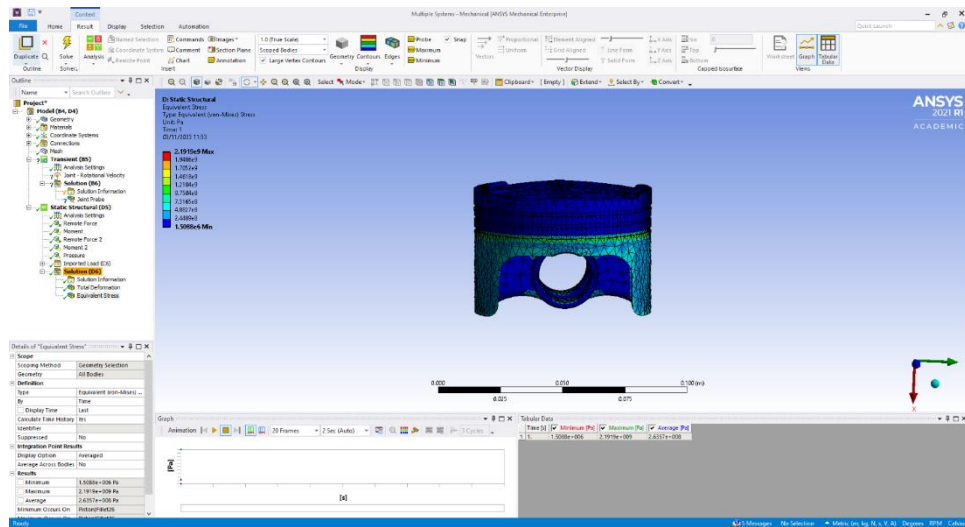


Figure 26 - Piston - Stress Profile

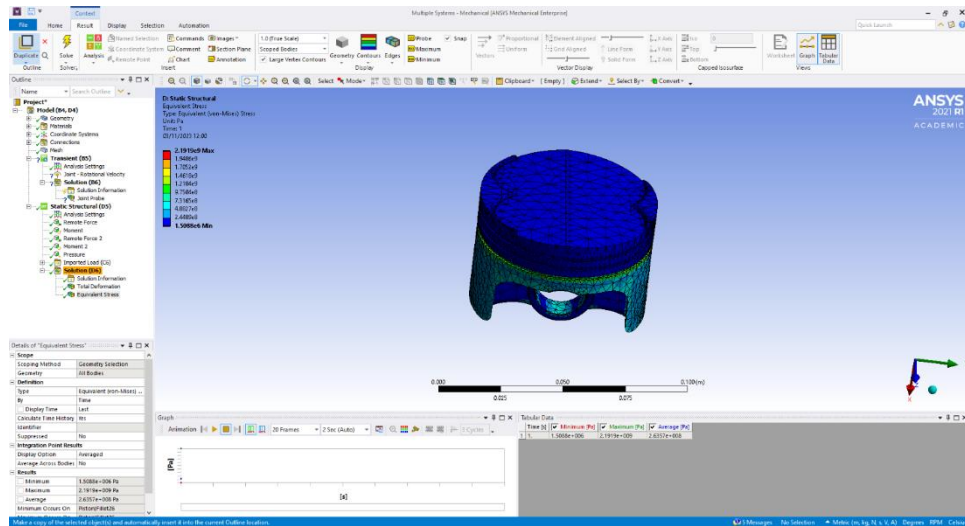


Figure 27 - Piston Crown - Stress Profile

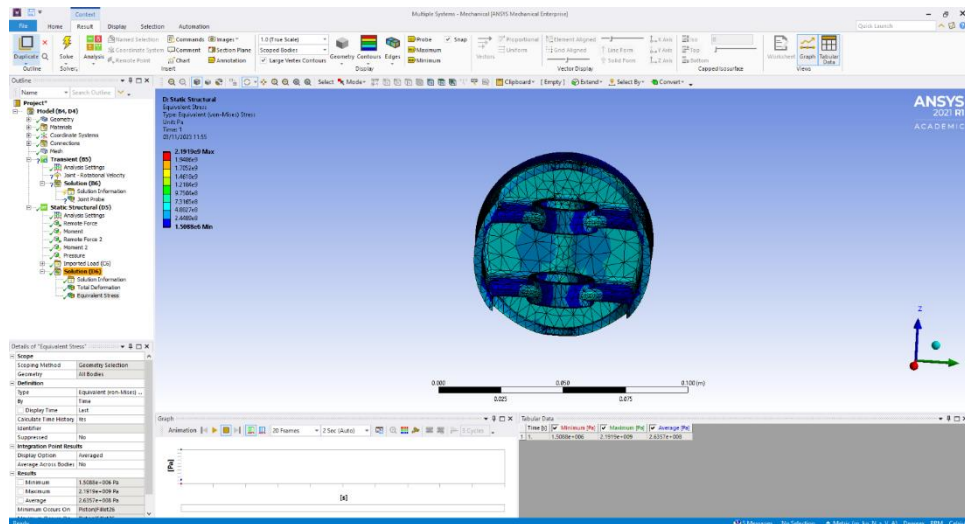


Figure 28 - Piston Underside - Stress Profile

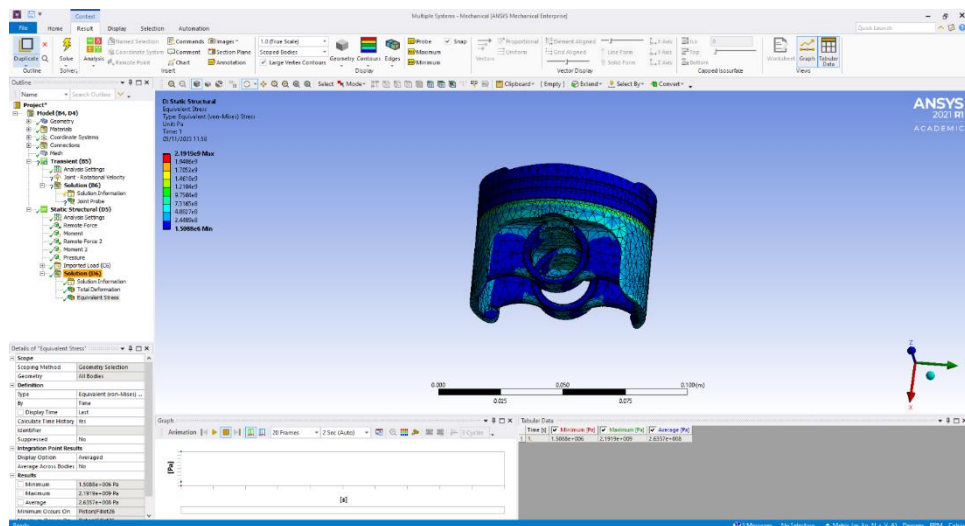


Figure 29 - Piston Bore Hole - Stress Profile

4.4 Structural Analysis Conclusion

From the results calculated, it is known that the piston exhibits a maximum deflection of $3.42 \times 10^{-4} \text{ m}$ and a minimum deflection of $2.41 \times 10^{-6} \text{ m}$. In addition, the greatest stress experienced by the piston is $2.19 \times 10^9 \text{ Pa}$, while the minimum stress is $1.51 \times 10^6 \text{ Pa}$.

Within the deformation and stress profiles there will be some level of inaccuracy due to the isolation of the component. As seen in figs. 24, 25, 28 and 29, across the underside and through the pin bore hole there is deformation and stress occurring in areas that would normally be obscured by the connecting rod and/or gudgeon pin. It may be that the maximum or minimum level of stress or deformation that ANSYS is calculating occurs in these regions, skewing the results. Further, the profiles themselves are not completely representative because of this modelling inaccuracy.

5 Continuous Improvement

The performance and characteristics of core components can significantly impact the overall capabilities of a system. Hence, it is imperative to pursue continuous improvements in order to safeguard the entire system's capabilities and performance from being significantly compromised by minor design imperfections [9].

5.1 Stress and Strain

Enhancing the longevity of components has paramount significance as it can substantially mitigate expenses and augment the dependability of systems. The material properties of a system may be a significant contributing factor to the stress and strain on the system. The mass of components is influenced by the density of the material and the volume of the geometry. Structural steel was used for the crankshaft, connecting rod and gudgeon pin, this is not necessarily the most representative material as the typically these components are made of alternate materials, a crankshaft for example, is usually composed of forged steel or cast iron [10] [11].

As a means to evaluate the consequences of inadequate materials selection, the piston material was changed to a low carbon steel, to evaluate the effect to the equivalent stress when using a more realistic material [12]. The aim of this will be to reduce the stress in the piston and thereby the overall system.

5.2 Strain Difference

The same process was followed for the piston, only this time using the material carbon steel AISI 1010 [13]. The temperature and stress profiles of the testing are shown below.

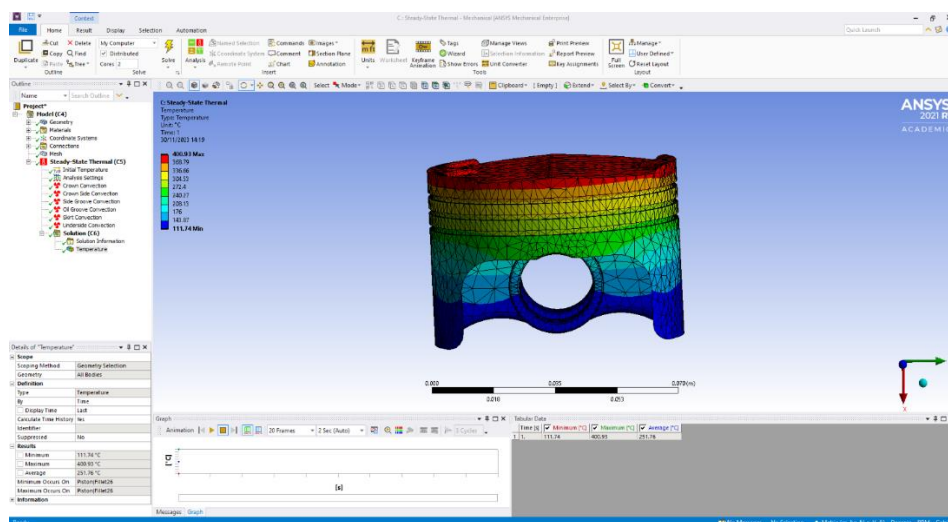


Figure 30 - Piston - Temperature Profile - Carbon Steel

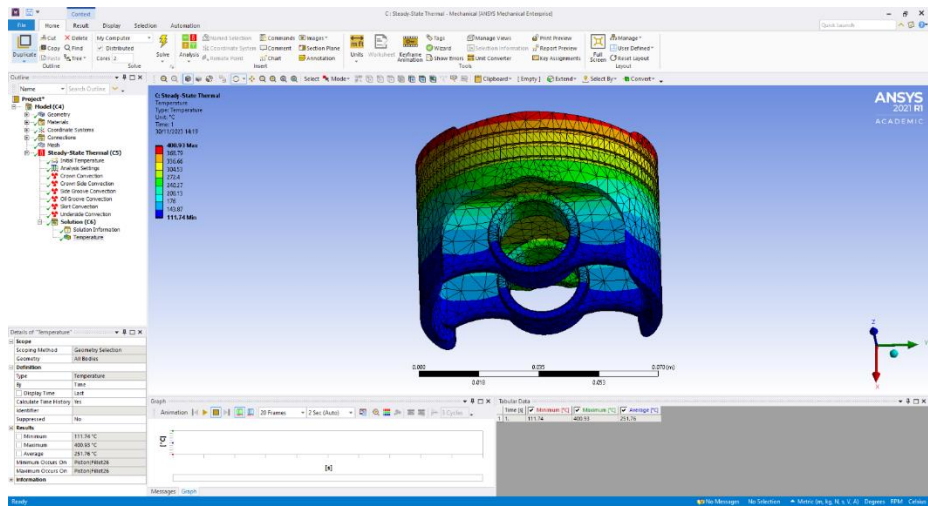


Figure 31 - Piston - Temperature Profile - Carbon Steel

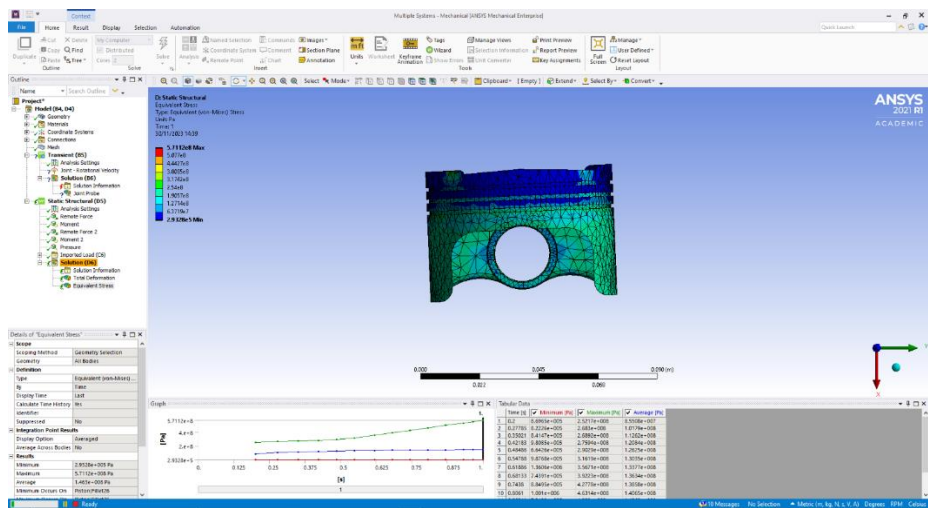


Figure 32 - Piston - Stress Profile - Carbon Steel

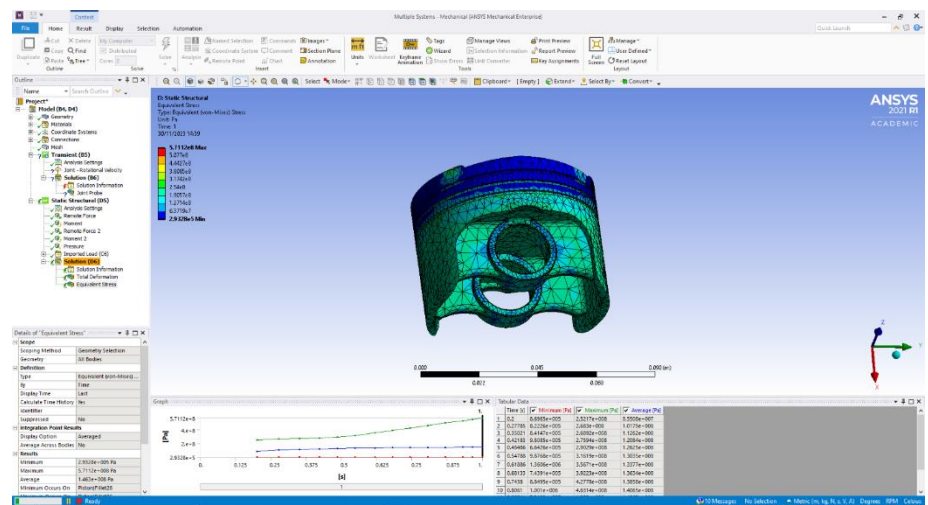


Figure 33 - Piston - Stress Profile - Carbon Steel

From the results of the stress analysis it can be seen that the piston has a maximum stress of 5.71×10^8 Pa and a minimum stress of 2.93×10^5 . The alteration of the material selection has led to a significant reduction in the stress experienced by the piston. The maximum stress has had a reduction of 720.91 Pa, while the minimum stress has undergone a decrease of 173.97 Pa.

6 Conclusion

The effectiveness of Ansys as a software package for assessing stress and strain in a system and its components has been demonstrated, with the software successfully generated an accurate outcome with a tolerable margin of error. The study demonstrated that the outcomes are notably influenced by the mesh size. However, the presentation of this relationship was hindered by the computational constraints imposed by the limited computer resources. Subsequently, the software successfully computed the stress and strain values pertaining to both the system and the piston system component. Moreover, the relationship between outcomes and design parameters was demonstrated by the alteration of the piston material, resulting in an enhanced design characterised by reduced component stress.

Several areas of concern were identified in the software, which could result in computational errors, such as the requirement to utilise weak springs. Additional concerns have been identified regarding the graphics and processing capabilities at hand, however, it would be more appropriate to ascribe these flaws not to Ansys, but rather to the hardware used for the computer system.

Additional research is warranted to conduct a comprehensive analysis of the fatigue life cycle of the piston. This entails exploring alternative methods to enhance the component's performance, going beyond the scope of material modification alone [14]. Enhancing the lifecycle is critical for a real system, improving vastly the reliability and reducing costs.

Furthermore, the development of an improved simulation would be necessary in order to effectively construct this system. This research provides a simplified examination of these systems, acknowledging that a comprehensive study would encompass additional facts and considerations. The objective of this study was to demonstrate the essential principles and the manner in which data can be derived from them. Although that is not to undermine that the simulation results remain realistic to the operating conditions which are applied.

References

- [1] Z. Bi, "Overview of Finite Element Analysis," in *Finite Element Analysis Applications, A Systematic and Practical Approach*, Indiana: Academic Press, 2018. [E-book]
- [2] R. I. Stephens, "Fatigue Design Criteria," *Encyclopedia of Materials: Science and Technology*, 2nd ed. Iowa City: Pergamon, 2001.
- [3] Dlubal Software Inc., *Verification Example - 0078*, Philadelphia: Dlubal Software, 2018, p.1, Fig.1.
- [4] M. Elchalakani, B. Yang, K. Mao, and T.-H. Pham, "Numerical methods," in *Geopolymer Concrete Structures with Steel and FRP Reinforcements*, Cambridge: Woodhead Publishing, 2023. [E-book]
- [5] A. S. Bahman and F. Iannuzzo, "Computer-aided engineering simulations," in *Wide Bandgap Power Semiconductor Packaging*, Cambridge: Woodhead Publishing, 2018. [E-book]
- [6] TWI Ltd., "What is Computer Aided Engineering (CAE)?", TWI, 2023. [Online]. Available: <https://www.twi-global.com/technical-knowledge/faqs/what-is-computer-aided-engineering> [Accessed Nov. 9 2023].
- [7] W. Young, R. Budynas, and A. Sadegh. *Roark's Formulas for Stress and Strain*, 8th ed. New York: McGraw-Hill Education, 2011.
- [8] Dlubal Software Inc., "Verification Example – 0078," *Dlubal Software*, 2018. [Online] Available: <https://www.dlubal.com/-/media/B437DA0115FB496C9C7BA58F08960A5E.ashx> . [Accessed Nov. 4, 2018].
- [9] J.P.T. Mo, C. Bil, A. Sinha, "Continuous Improvement" in *Engineering Systems Acquisition and Support*, Cambridge: Woodhead Publishing, 2015, ch.12, pp.205-212. [E-book]
- [10] M. Stewart, "Reciprocating Pumps," in *Surface Production Operations Volume IV – Pump and Compressor Systems: Mechanical Design and Specification*, Houston: Gulf Professional Publishing, 2019, ch.4, pp.311-414.
- [11] T. Allen, "An In-Depth Look at Crankshafts", *ENGINEBUILDER*, Feb. 1, 2018. [Online] Available: <https://www.enginebuildermag.com/2018/02/depth-look-crankshafts/#:~:text=Factory%20forged%20cranks%20are%20made,makes%20the%20forging%20less%20brittle> [Accessed Nov. 14, 2023].
- [12] J. Skowronska, A. Kosucki, L. Stawinski, "Overview of Materials Used for the Basic Elements of Hydraulic Actuators and Sealing Systems and Their Surfaces Modification Methods", National Library of Medicine, Bethesda, 2021. [Online] Available: <https://www.ncbi.nlm.nih.gov/pmc/articles/PMC7999906/> [Accessed Nov. 15, 2023].
- [13] MatWeb, "AISI 1010 Steel, cold drawn", *MatWeb, LLC*, 2023. [Online] Available: <https://www.matweb.com/search/DataSheet.aspx?MatGUID=025d4a04c2c640c9b0eaaef28318d761> [Accessed Nov. 30, 2023].
- [14] A.M. Alshoaibi, Y.A. Fageehi, "Numerical Analysis on Fatigue Crack Growth at Negative and Positive Stress Ratios" in *Materials - Metals Processing, Finite Element Analysis and Fatigue Design*, Jazan: Jazan University, 2023.

[Main Body Word Count: 2981]