



University of Wales Trinity Saint David

School of Engineering

BEng Motorsport Engineering

<i>Title of Report</i>
Application of Computer Aided Engineering Processes
<i>Module Name and Code</i>
Structural Design & Fluid Analysis ACHA6010
<i>Student Name & Number</i>
Iven Maximo 2306047
<i>Submission Date</i>
17/12/2023

University of Wales Trinity Saint David

2023-2024

BEng Motorsport Engineering

Contents Page

Table of Contents

Contents Page.....	2
Table of Figures	3
Introduction.....	4
Literature Review.....	5
Task 1- Ansys Validation Testing.....	6
Task 1.1- Ansys Software Overview.....	6
Task 1.2- Software Results Validation	7
Task 1.2.1- Software Results Validation Overview.....	7
Task 1.2.2- Elastic Bending Continuous Load Validation	7
Task 1.2.3- Bimetallic Strip Validation	9
Task 2- Engine Assembly Simulation	12
Task 2.1- Engine Assembly Simulation Overview	12
Task 2.2- Engine Assembly Simulation Analysis	13
Task 2.2.1- Kinematics Analysis.....	13
Task 2.2.2- Thermal Influence Analysis.....	14
Task 2.2.3- Structural Analysis	16
Task 2.2.4- Engine Assembly Modification and Analysis	17
Conclusion	20
References	21

Table of Figures

Figure 1- Continuous Load Validation Plate Fixture and Load	8
Figure 2- Continuous Load Plate Default Mesh Element Size Results	8
Figure 3- Continuous Load Plate 0.02-0.005m Mesh Element Size Results	9
Figure 4- Bimetallic Strip Validation Fixture and Thermal Loading	11
Figure 5- Bimetallic Strip Validation 0.25mm Mesh Element Size results.....	11
Figure 6- Engine Assembly Kinematics Analysis Simulation Tree	13
Figure 7- Engine Assembly Kinematic Analysis Axial Force Graph.....	14
Figure 8- Engine Assembly Thermal Influence Analysis Mesh Sizing Controls.....	14
Figure 9- Engine Assembly Thermal Influence Analysis Convection Boundary Conditions	15
Figure 10- Engine Assembly Thermal Influence Analysis Results.....	15
Figure 11- Engine Assembly Structural Analysis Pressure Load.....	16
Figure 12- Engine Assembly Structural Analysis Total Deformation Results	17
Figure 13- Engine Assembly Piston Dimension Modification	17
Figure 14- Modified Engine Assembly Kinematic Analysis Axial Force Graph.....	18
Figure 15- Modified Engine Assembly Thermal Influence Analysis Results	18
Figure 16- Modified Engine Assembly Structural Analysis Total Deformation Results	19
Figure 17- Modified Engine Assembly Structural Analysis von-Mises Stress Results	19

Introduction

In this assignment, the principles of CAE are going to be discussed along with the respective advantages and disadvantages. Moreover, the accuracy of simulation results are going to be assessed using an analytical method represented for a model, where the impact of mesh size is also discussed. In addition, an engine assembly is used to conduct a realistic simulation using a variety of solvers which simulate the ruling conditions of an operating cycle. The software will then be used as a design tool, to evaluate the impact of material change, along with feature change on selected component geometry.

Literature Review

Computer aided engineering (CAE) is used to simulate components/ assemblies through the projected loads which the same will experience. This allows the user to select the desired constraints and other respective boundary conditions. This process involves the representation of the desired component/ assembly in a selected simulation package, in which various solvers types are available to the user. Each solver depicts a different types of governing equations used to solve the simulation. As an example, a static structural solver will conclude an analysis using the equations described in the elastic bending continuous load simulation. Software used will simply conduct analytical methods within the solver calculator, therefore, the overall accuracy of these can be checked using a simple model which can be easily solved analytically. Subsequently, this should be conducted within the software package using the right solver (static structural, rigid dynamics, modal analysis, etc). CAE is also primarily used as a design tool, as it grants the user to use the results provided by a simulation to change the ruling design. This can be done iteratively, where small features are modified, in order to analyse the result change between these.

Moreover, the key advantages of CAE are depicted by the equipment required being significantly lower in expense and possibly quantity when compared to physical component testing. Another comparable aspect of CAE is the low need for component/ product prototyping, which is directed to costs. “CAE becomes ideal too when a large number of constructions are evaluated to determine the best solution subjected to given constraints” (Bi, 2018).

Lastly, there are also limitations shown within CAE, such as the computing power required to solve complex simulations may require higher specification hardware. Another leading limitation depicted is the Finite element method (FEM), where the model is represented within a simulation as a group of element which determine how many calculations are required to solve the model. Complex features may affect these, therefore, in some applications, the model is relatively difficult to represent the actual desired component/ assembly.

Task 1- Ansys Validation Testing

Task 1.1- Ansys Software Overview

The Ansys software allows for simulating realistic conditions which a component/ assembly will have to experience throughout the intended application. This software also provides a wide selection of simulation solvers namely, thermal, structural, etc. Simulation trees can also be used in order to connect different solvers, allowing the implementation of various load types, resulting in realistic results, which can be used as the ruling reference in terms of a design tool.

Task 1.2- Software Results Validation

Task 1.2.1- Software Results Validation Overview

In order to validate the results provided by the software, a selection of validation test cases were conducted, assessing the similarity of results achieved between an appropriate analytical method and Ansys simulation. Furthermore, the test cases were completed for a structural and thermal approach.

Task 1.2.2- Elastic Bending Continuous Load Validation

For this validation, a 50x5x1000mm plate is fixed and a uniform pressure of 2.75KPa is applied along the top surface of the plate. The leading aim is to record/ calculate the maximum deflection present. To obtain the maximum deflection caused by the continuous load the following formula is used (Dub, 2021):

$$U_z max = \frac{3pL^4}{2Ew^2t} \quad (1)$$

Where:

$U_z max$ - Maximum Deflection (m)	E- Young's Modulus (MPa)
p- Pressure Magnitude (MPa or N/m ²)	w- Beam Width (m)
L- Beam Length (m)	t- Beam Thickness (m)

The analytical method results obtained using the beam size and the material properties provided can be seen below:

$$U_z max = \frac{3 \times 2750 \times (1^4)}{2 \times (2.1 \times 10^6) \times (0.05^2) \times 0.005} \quad <=>$$
$$<=> U_z max = 157.143mm$$

Subsequently, a static structural analysis on Ansys was used to conduct the required validation. The plate was modelled and imported. The material properties were also set accordingly to the simulation requirements. Further, the fixture and continuous load were also applied to the plate (figure 1).

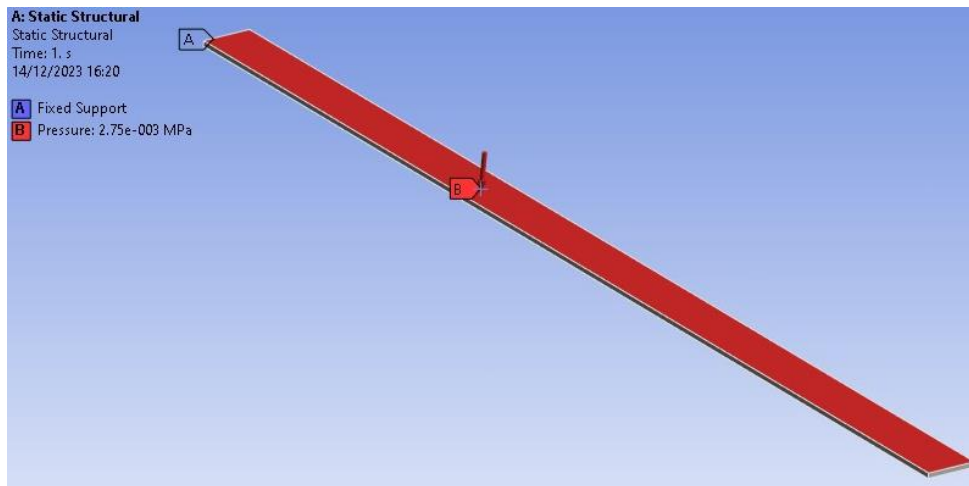


Figure 1- Continuous Load Validation Plate Fixture and Load

Moreover, the default mesh element size was used. The simulation was then conducted, providing a maximum deflection of 156.2mm, being marginally lower than the value provided by the analytical solution (figure 2).

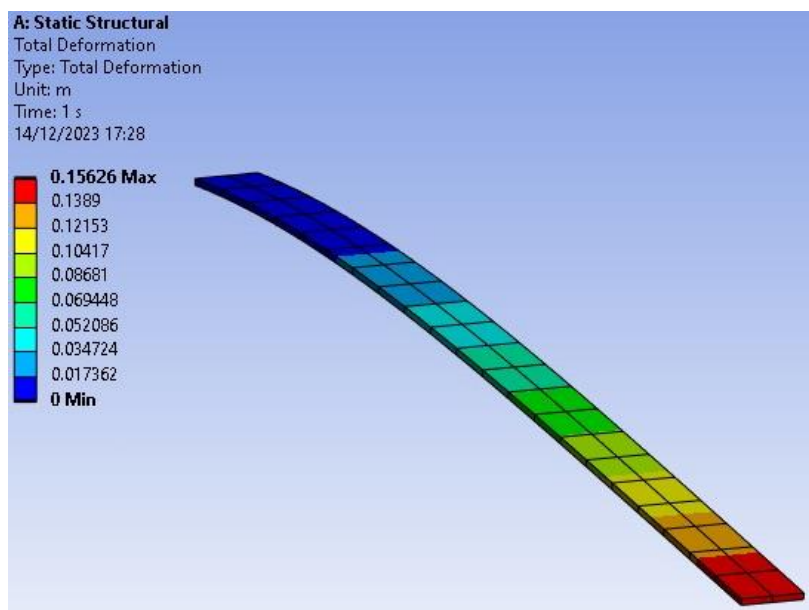


Figure 2- Continuous Load Plate Default Mesh Element Size Results

A mesh element size value of 0.02m was then applied to the beam (Dub, 2021). This displayed a maximum deflection value of 156.7mm (0.443mm lower than aim). The simulations were then conducted with reduced mesh element size in 0.005m increments, down to 0.005m (figure 3). The mesh refinement did not affect the final deflection value. Lopes et al. (2020, p.5), state that mesh dependency is localised when a h-refinement method discretization is used for a free or fixed element

mesh. Therefore, any further refinement of the mesh element size would not affect the overall results. Lastly, the continuous load validation using CAE software presented, shows a 0.282% error in comparison to the analytical method.

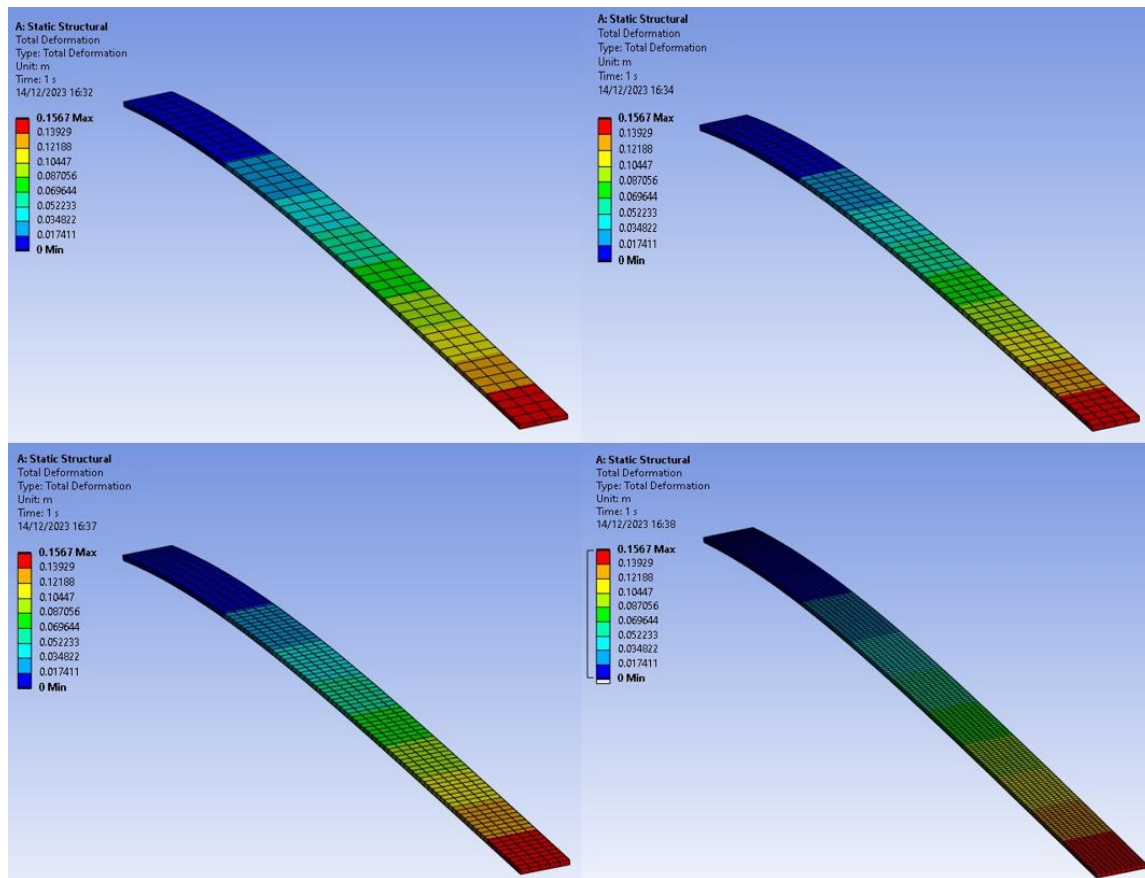


Figure 3- Continuous Load Plate 0.02-0.005m Mesh Element Size Results

Task 1.2.3- Bimetallic Strip Validation

This validation presented below involves a thermo-mechanical simulation within an assembly, composed of 2 metal strips with equal dimension, along with a respective material for each (copper and invar). The strips are fixed in one end and a thermal load of 100°C is applied to both bodies of the assembly. The aim of the validation conducted is to assess any result difference obtained by the analytical method and the simulation solved on Ansys. The leading value analysed is the maximum deflection acquired for both methods. The maximum deflection ($U_{2,max}$) for the bimetallic strip requires a constant, specified by the stiffness coefficient (K_1). The stiffness coefficient can be calculated using the formula seen below (Dub, 2017):

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

Where:

K_1 - Stiffness Coefficient (N/m)

t_c or t_i - Layer Thickness (m)

E_c or E_i - Young's Modulus of Copper/ Invar (MPa)

The essential maximum deflection (U_{zmax}) then be calculated using the following formula:

$$U_{zmax} = \frac{6(\alpha_i - \alpha_c)(t_i + t_c)T_c L^2}{t_s^2 K_1} \quad (3)$$

Where:

U_{zmax} - Maximum Deflection (m)

L - Strip Length (m)

α_i or α_c - Thermal Expansion Coefficient of Invar/ Copper ($^{\circ}C^{-1}$)

t_s - Strip Thickness (m)

t_i or t_c - Layer Thickness (m)

K_1 - Stiffness Constant (N/m)

T_c - Thermal Loading ($^{\circ}C$)

The solved equation displays a deflection value of -7.049mm. The assembly was then constructed, and the corresponding properties were set to both materials. In sequence, the fixture and thermal loading were applied to the designated faces and bodies (figure 4).

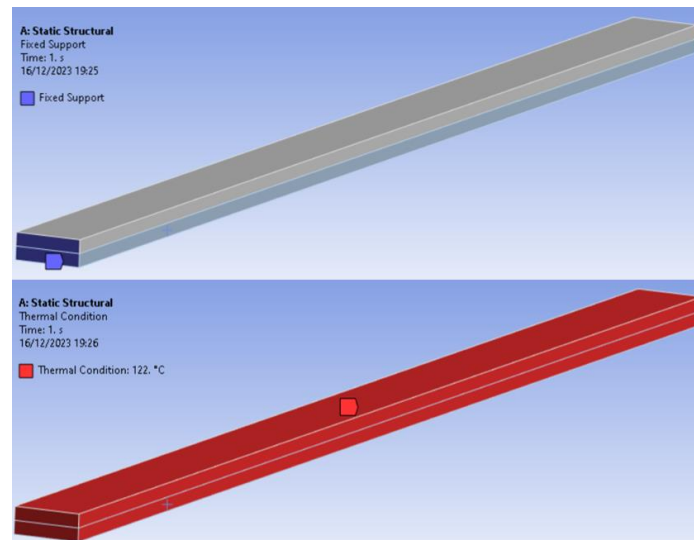


Figure 4- Bimetallic Strip Validation Fixture and Thermal Loading

The indicated mesh element size of 0.25mm was used for the simulation. Consecutively, the simulation was solved, in which, the maximum deflection value is displayed at 7.12mm (figure 5). This is 0.071mm higher than the analytical method solution. The percentage of error present is also higher than the other specified continuous load validation value. A 1.007% error can provide accurate results; however these should be analysed prior to validation. Lastly, other mesh element sizes were tested, however, anomalies were shown when solving the thermos-mechanical analysis, due to the total deformation results reaching a maximum of 0.0003mm.

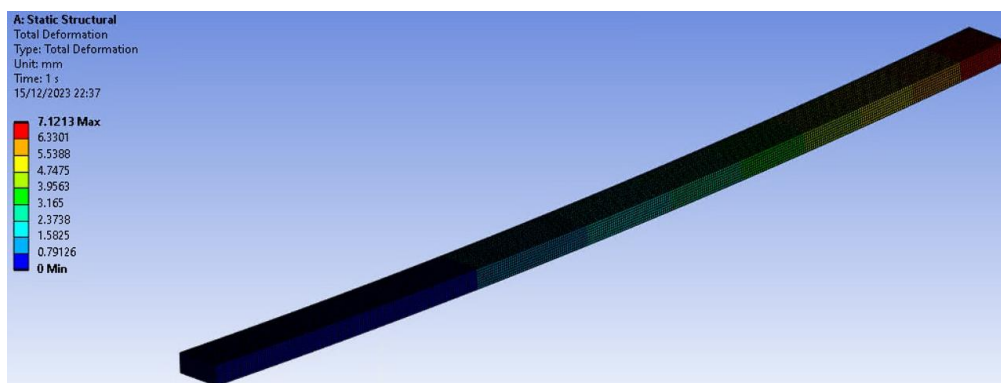


Figure 5- Bimetallic Strip Validation 0.25mm Mesh Element Size results

Task 2- Engine Assembly Simulation

Task 2.1- Engine Assembly Simulation Overview

An engine assembly was used for the required analysis using different simulation solvers within Ansys. The boundary conditions used throughout the simulations were kept realistic, with a focus on obtaining realistic results, which can then be approached with an enhancement using more desirable material properties, along with feature changes to conclude the projected overall weight reduction and improvement in component life.

Task 2.2- Engine Assembly Simulation Analysis

Task 2.2.1- Kinematics Analysis

In order to achieve accurate simulation results, a rigid dynamics study was set up. The leading maximum forces along each axis in correspondence to the degrees of freedom (DOF) of the engine, will then be used along other solvers. Moreover, the rigid dynamics solver does not require any component bodies to be flexible along the study. Awrejcewicz (2012, p.187-262) states that kinematics involves particle motion analysis in a geometric approach and located in a space, which do not consider any generating work done and overall forces. The initial piston material was changed to an aluminium alloy 6061 T6, with AISI 1060 plain carbon steel applied to the remaining assembly components. 6061 T6 alloy provides a good corrosion resistance factor and contains a medium fatigue strength (Aalco Metals Ltd, 2018). AISI 1060 also provides high strength and hardness, which makes the materials applicable for the proposed use.

Furthermore, the contacts enclosed by the assembly are set to distinct joints, which allow the components to rotate/ translate along the normal determined degrees of freedom (figure 6). The appointed load variable is designated by the rotational velocity of the crankshaft body to ground joint. This allows the ruling engine speed to be chosen for analysis. In sequence, a 5000rpm value was selected. Lastly, the desired contact force between the gudgeon pin and piston is set as a solution.

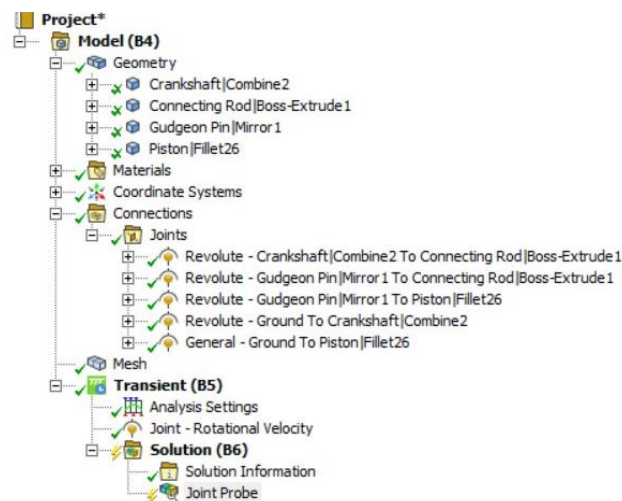


Figure 6- Engine Assembly Kinematics Analysis Simulation Tree

The simulation was then solved, and the axial forces were shown for a set number of cycles. As seen in figure 7, a maximum value of 3720.5N is registered piston to gudgeon pin contact. The force located in this contact region is high, and could lead to material failure if any high stress concentration is located. Therefore, any material options should be assessed in terms of thermal dependency of the

material and overall elastic properties. Ramamurti et al. (2011, p.86-87) analyse the stress concentration involved in a gudgeon pin and suggest that the lubrication efficiency of an engine has a substantial impact in the stress sustained by the gudgeon pin. Figure 7 implied that the forces exerted in the contact region are set uniformly, with no anomalies in data fluctuation seen. The maximum force results along the contact between the piston and gudgeon pin are then retrieved, and set as imported motion loads, which are then used as a boundary loads for subsequent analysis using different solvers.

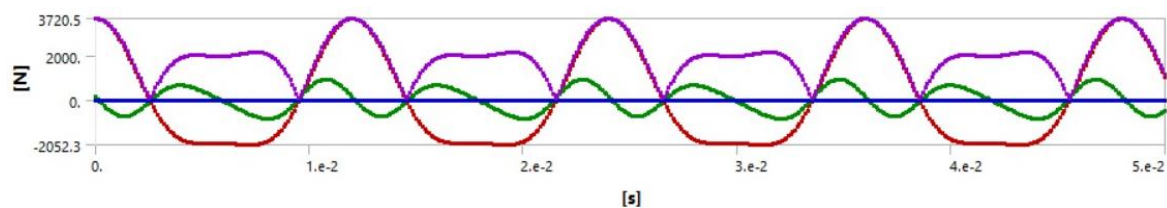


Figure 7- Engine Assembly Kinematic Analysis Axial Force Graph

Task 2.2.2- Thermal Influence Analysis

The thermal influence on the assembly is then simulated along the piston. All other bodies are suppressed from this solver. Moreover, a refined mesh sizing control was applied to the piston, due to the high temperatures faced, the results obtained in this simulation are crucial in terms of piston design, and may affect the range of pressures and fuel mixture regulation which can be used (figure 8).

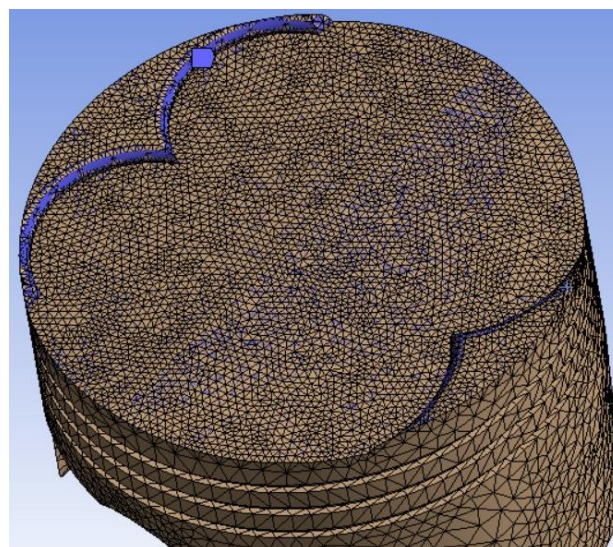


Figure 8- Engine Assembly Thermal Influence Analysis Mesh Sizing Controls

With the main purpose of simulating realistic conditions, the temperatures and film coefficients were assigned to the right areas of the piston as convection boundary conditions (figure 9). Kumar and Kumar (2022, p.1346-1369) assess a variety of piston coatings and state Nickel Chrome Boron Silicium (NiCrBSi) coatings can be heat treated to improve micro hardness, which may counteract seizures caused by contact.

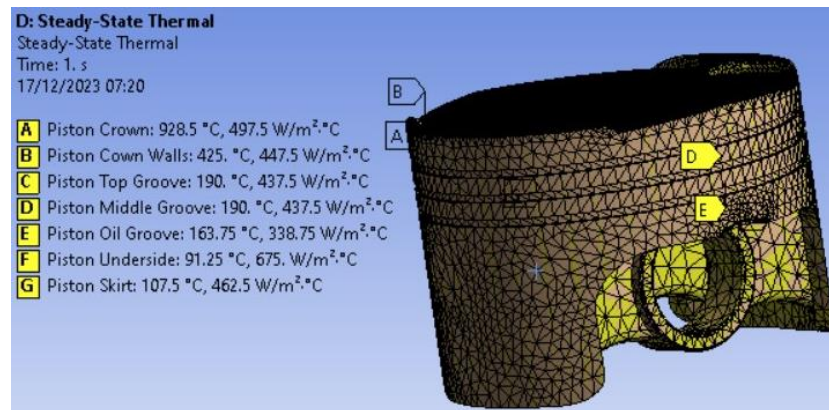


Figure 9- Engine Assembly Thermal Influence Analysis Convection Boundary Conditions

The overall temperature goal was then applied to the respective solution. As seen below in figure 10, the maximum temperature is located at the piston crown (304.01°C). The temperature is then gradually reduced moving to the skirt and underside of the piston, with a 166.24°C minimum recorded value located in this region. The underside extrude of the piston presents a higher value, due to the bulk temperature on the combustion reaction being rapidly transferred to this area. The piston boss does not endure a substantial temperature character, which preserves component life.

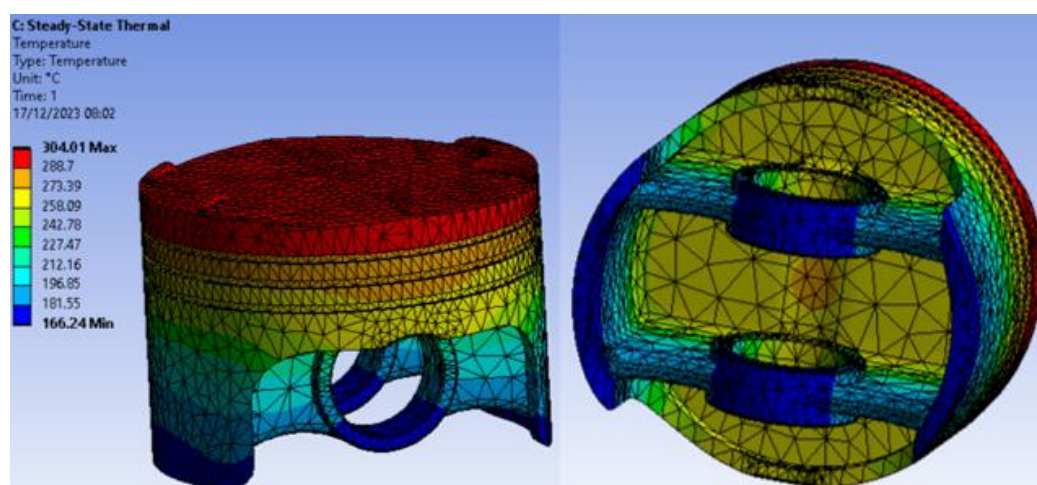


Figure 10- Engine Assembly Thermal Influence Analysis Results

Task 2.2.3- Structural Analysis

Lastly, the structural pressure loads are applied to the piston with focus on total deformation. The respective remote forces from the rigid dynamics analysis along with the temperature loads, are imported to the solver. Equivalent to the thermal influence analysis, mesh sizing controls on the piston crown were also added. The pressure value of 105bar is then applied to the piston crown (figure 11). This will provide a maximum deflection of the piston, which can be analysed and used for design reference.

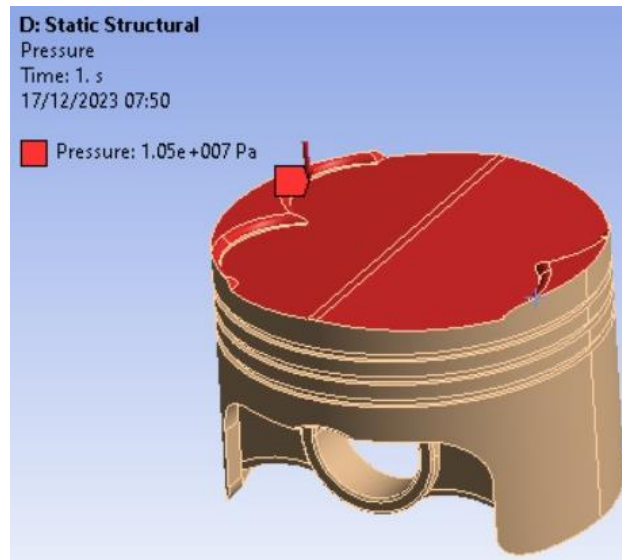
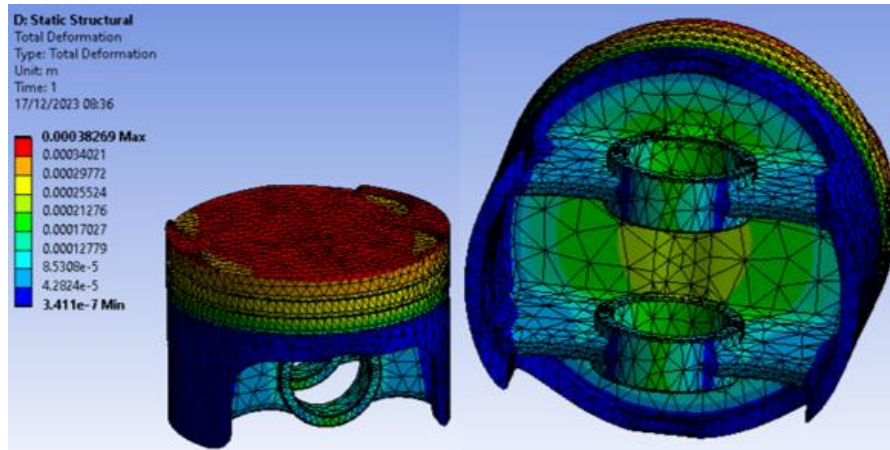


Figure 11- Engine Assembly Structural Analysis Pressure Load

The final simulation was then solved, displaying a maximum deformation value of 0.38mm detected in the piston crown (figure 12). This is also similar to the thermal influence results, when considering the locations which will have to sustain with the highest values seen in the leading aspects of temperature and overall deformation along with wear. The piston skirt shows a substantially low character for deflection. The underside extrude of the piston and piston boss will experience marginally higher deformation values due to the bulk material temperature and pressure applied, with the additional remote forces presented by rotational velocity of the crankshaft.



Task 2.2.4- Engine Assembly Modification and Analysis

With the aim of analyse response with material and feature changes. The leading piston skirt length was increased to 30mm (figure 13). The leading piston and connecting rod material were changed to aluminium alloy 2618 T61 and Titanium alpha-beta alloy (Ti-6Al-2Sn-2Zr-2Mo) solution treated and aged. 2618 T61 alloy along with the design change, show a 5.69g weight increase. However, it presents a higher temperature strength and fatigue life. Ti-6Al-2Sn-2Zr-2Mo also shows a relatively higher strength and corrosion resistance (Veiga, et al., 2012). Which can be directly compared with the previously used material.

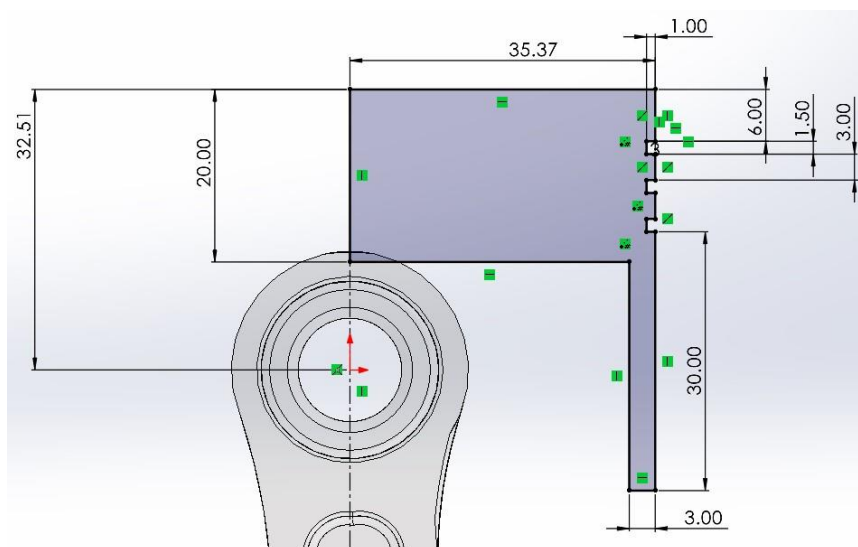


Figure 13- Engine Assembly Piston Dimension Modification

The corresponding geometry for the simulation was then changed to the updated assembly. The kinematic simulation was then solved using the same process mentioned above. As seen in figure 14, the results show a lower overall force in the contact region between the piston boss and gudgeon pin (833.02N). With a reduced cyclic load across the region, the overall component life is increase. On the hand, the force fluctuations seen, present a pattern, but are inconsistent. This could be due to geometry change, which has resulted in a change of the crankshaft balancing. “Crankshaft unbalance translates into high vibrations that are transmitted to the entire car” (Dagna, et al., 2021). This can also affect the depicted engine response.

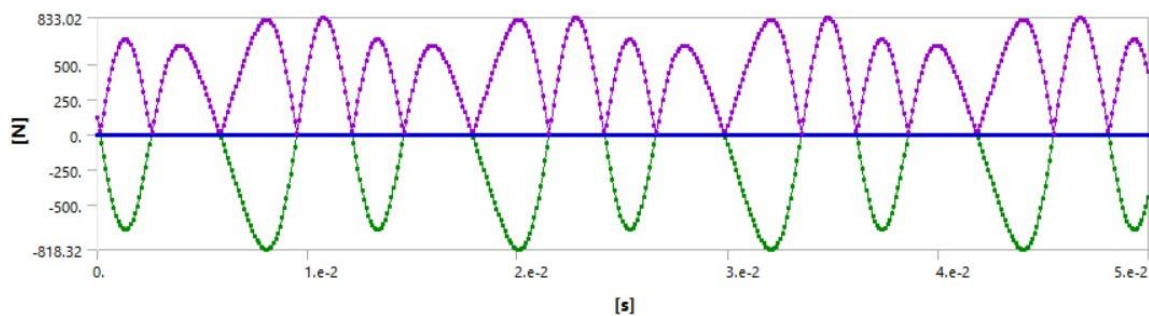


Figure 14- Modified Engine Assembly Kinematic Analysis Axial Force Graph

Further, the thermal influence of with the modified material and piston skirt was assessed. The temperatures along each area are identical. Figure 15 shows that the modification present a marginally higher maximum temperature along the piston crown 313.77°C. However, the range in temperature recorded with the modified assembly is higher at 154.9°C compared to 137.77°C seen in the base assembly. The piston skirt boss and underside temperatures are similar to the values displayed in the base assembly.

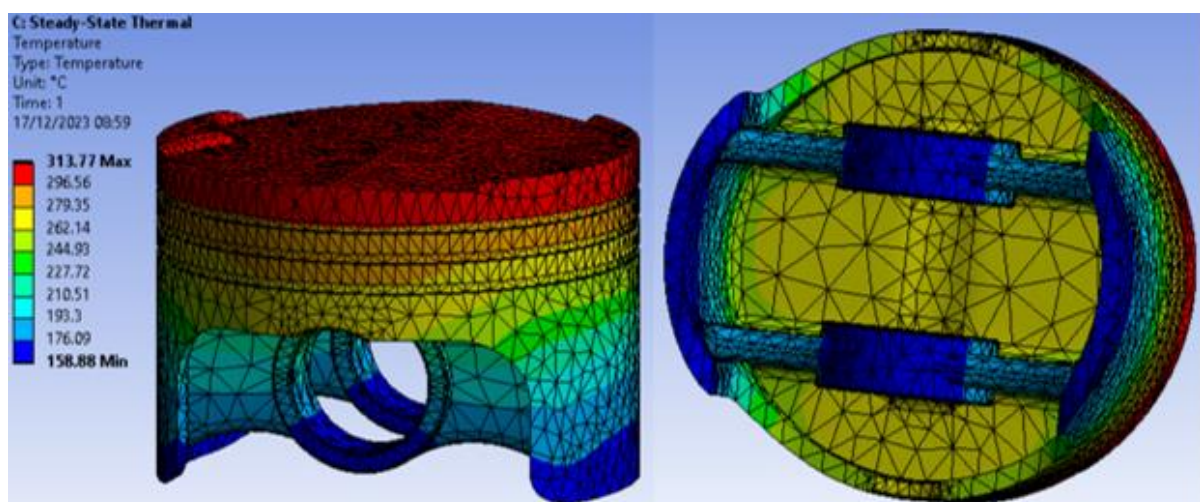


Figure 15- Modified Engine Assembly Thermal Influence Analysis Results

The remote forces and resultant temperatures were then imported to the static structural solver. In sequence, the key pressure load was applied to the piston crown. Ultimately, the simulation was solved, providing total deformation and respective von-mises stress. The total deformation presents a small change in the piston deformation of 0.34mm as seen in figure 16. These results are almost equivalent when in direct comparison. Both top and bottom sections present similar values.

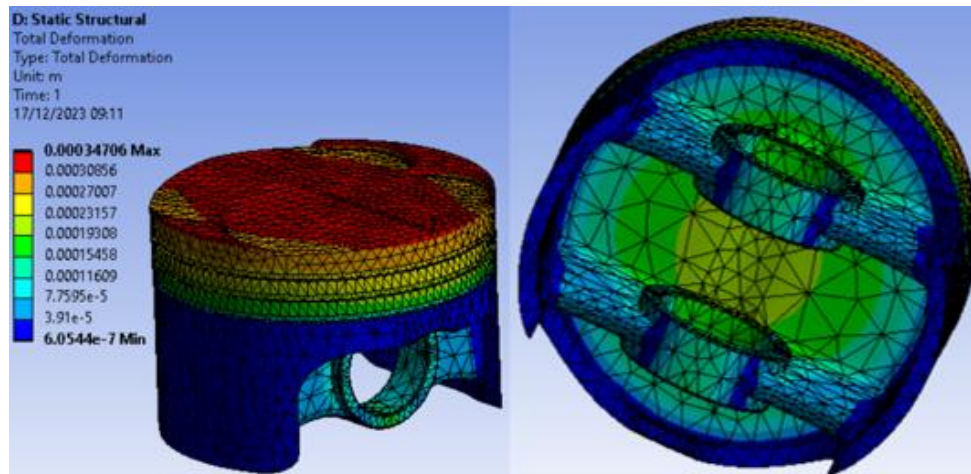


Figure 16- Modified Engine Assembly Structural Analysis Total Deformation Results

Lastly, the von-Mises stress plots along the piston surface show the regions of high stress concentrations. An example depicted, is the contact area of the piston boss and gudgeon pin, where the maximum recorded value is shown ($4.612 \times 10^8 \text{ Pa}$). The oil ring groove also shows high along the revolve of the piston. The piston crown walls also contain the lowest stress values recorded by the simulation (figure 17).

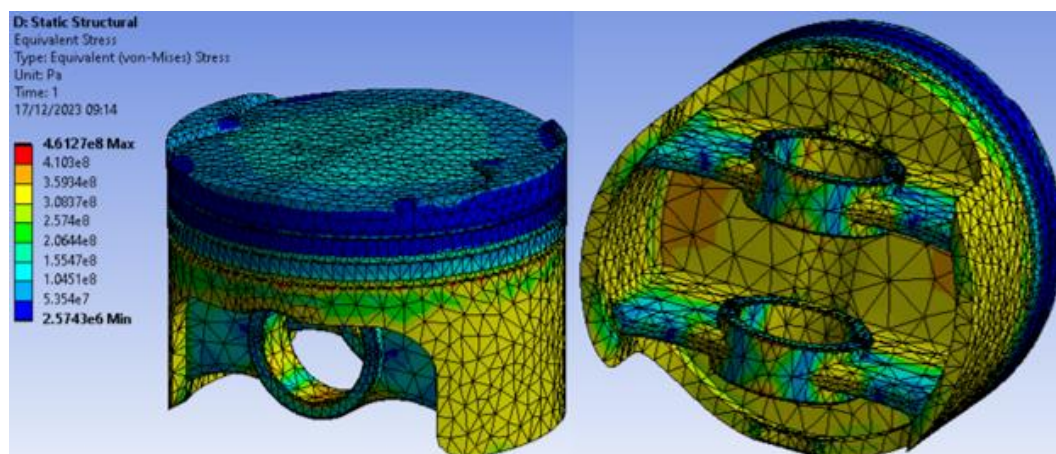


Figure 17- Modified Engine Assembly Structural Analysis von-Mises Stress Results

Conclusion

In conclusion, CAE can be used for a range of dynamic studies, where a validation is also applicable in order to assess the accuracy of the results provided. In sequence, an engine assembly was used to model realistic conditions which apply to the same, allowing for precise results within different simulation solvers. The analysis conducted also indicates that CAE can be used to optimise designs using iterative approaches, where the components are modified in accordance with the desired application, which can then be assessed along different parameters to select the most optimal design.

References

- Aalco Metals Ltd, 2018. *Aluminium Alloy 6061 - T6 Extrusions*, Wednesbury: Aalco Metals.
- Awrejcewicz, J., 2012. *Classical Mechanics: Kinematics and Statics*. 1st ed. ns.: Springer.
- Bi, Z., 2018. *Finite Element Analysis Applications*. 1st ed. London/ San Diego/ Cambridge/ Oxford: Elsevier.
- Dagna, A., C. D. & Gastaldi, C., 2021. A General Framework for Crankshaft Balancing and Counterweight Design. *Applied Sciences*, 11(8997), pp. 1-21.
- Dub, M., 2017. *Bimetallic Strip (Dlupal)*. [Online] Available at: <https://www.dlupal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/000078> [Accessed 16th December 2023].
- Dub, M., 2021. *VE 0016 | Elastic Bending – Continuous Load (Dlupal)*. [Online] Available at: <https://www.dlupal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/009016> [Accessed 14th December 2023].
- Halden Special Steel, ns.. *C60 Carbon Steel | DIN1.0601 | AISI 1060 | S60C*. [Online] Available at: <https://www.round-bars.com/products/c60-steel/> [Accessed 17th December 2023].
- Kumar, S. & Kumar, M., 2022. Tribological and Mechanical Performance of Coatings on Piston to Avoid Failure—A Review. *ASM International 2022*, 22(ns.), p. 1346–1369.
- Lee, H.-H., 2021. *Finite Element Simulations with ANSYS Workbench 2021*. National Cheng Kung University: SDC Publications.
- Lopes, B. et al., 2020. Assessment of mesh dependency in the numerical simulation of compact tension tests for orthotropic materials. *Composites Part C: Open Access*, ns.(ns.), pp. 1-14.
- Ramamurti, V. et al., 2011. Design considerations of Gudgeon pin in reciprocating air compressors by semi analytic approach. *Journal of Mechanical Engineering Research*, 4(3), pp. 75-88.
- Veiga, C., Davim, J. & Loureiro, A., 2012. *PROPERTIES AND APPLICATIONS OF TITANIUM ALLOYS: A BRIEF REVIEW*, Coimbra/ Aveiro: Advanced Study Centre Co.
- Young, W., Budynas, R. & A, S., 2011. *Roark's Formulas for Stress and Strain*. 8th ed. New York: McGraw-Hill Education.