



University of Wales Trinity Saint David

Prifysgol Cymru Y Drindod Dewi Sant

BEng Automotive

School of Engineering

Application of Computer Aided Engineering

Processes **ACHA6010**

PIERLUIGI BONAGURA

(2005700)

18/12/2022

Abstract

Aims

The aim of this technical report is to simulate and test a real engine into a computation environment. Additionally, 6 main tests have been analysed with the help of Ansys Workbench such as:

Simply Supported beam, where a beam has been designed in Solidworks and transferred to Ansys. The boundary conditions were applied such as fixed support and force to the beam and the results have been explained in the Literature review section.

Static structural analysis, thermal analysis, and rigid dynamic study: the piston has been tested under real parameter from different sources and the result and the discussion are highlighted in the Methodology section.

Kinematic study with a Fatigue analysis has shown in the Result and Discussion the piston force will be discussed and compared with different resource

Finally, the limitations of the computational software are discussed in the conclusion.

Table of Contents

1	Introduction	1
2	Literature review.....	2
2.1	Simply Supported beam analysis with a use of Ansys	2
2.2	Modelling and stress analysis of the simply supported beam.....	2
2.3	Thermal and static structure analysis	5
3	Methodology.....	7
3.1	Static structural and Thermal analysis of a single piston.....	7
3.2	Thermal Study	12
3.2.1	Temperature set up	12
3.2.2	Thermal studies results	13
3.3	Rigid dynamic study simulation of an engine	14
4	Results	21
4.1	Kinematic study results.....	21
5	Discussion.....	25
6	Conclusion.....	28
7	References	30

1 Introduction

Technology has moved to ensure that the studies of Engineers are improved. Back in the days, Engineering used to draw and doing calculations by hand, whereas, as the technology has improved, many of this process has been substituted by computer-aided design (CAD) (Chen & Liu, 2018).

In addition, as illustrated in Figure 1, CAD, computer-aided engineering (CAE) and computer aided manufacturing (CAM), has significantly changed the engineering landscape.

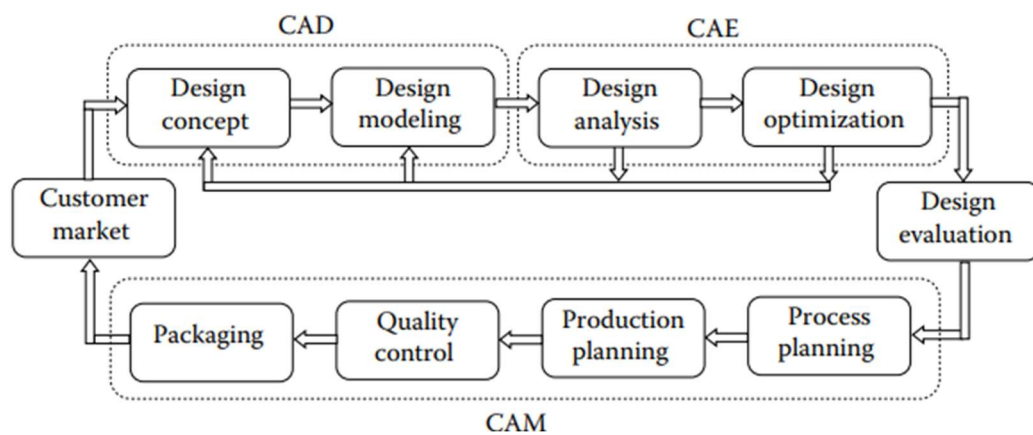


Figure 1. The fundamental process of CAD, CAE and CAM model (Chen & Liu, 2018, p.2)

For instance, previously in order to manufacture some components, used to take up to 6 years, whereas, now with the advantages of the technology will be done in less than a year. Therefore, the Finite Element (FEM) is a mathematical model where engineering solves many problems such as stress analysis and dynamic response from vehicle components. Moreover, the main applications that is being used for the FEA simulation is named ANSYS Workbench. Ansys Workbench is an easy-to-use platform, which was created to integrate ANSYS (Chen & Liu, 2018).

The main software that has been used for this technical report is ANSYS Workbench 19.1. The primary test is to highlight the main function of a stress and strain calculation using the CAE software and describing the stress distribution using the FEM method (Chougule & Khatawate, 2013).

For this technical report 6 main study have been considered. Firstly, the simply supported beam has been investigated with some math calculations and compare into Ansys Workbench, further, thermal and static structural have been analyse and another thermal study have been compared. Then a rigid dynamic study have been explained with a full engine with the joint connected. Finally, a kinematic study results has been explained and a Fatigue analysis has been explained.

2 Literature review

2.1 Simply Supported beam analysis with a use of Ansys

The Beam is a structural solid member which bend as load/force is applied to it, that give as a result a bending.

There are many different types of beams such as:

- Simple beam where the end of the two-support resting at each end. All parts between the supports are free to move in the vertical plane below, which effects the vertical loads.
- A constrained beam or restrained beam is a rigid body that is fixed at both ends and simply supported at each other.
- A continuous beam is a bar which rest in more than one support.
- The cantilever beam is a member which is attached at one end, and it is free to move between the free end and the support as evidenced in figure 2 (Kutz, 2015).

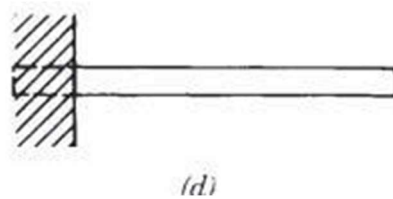


Figure 2. A cantilever beam (Kutz,2015,p.574)

2.2 Modelling and stress analysis of the simply supported beam

In this technical report, a simply supported beam has been used for the static structural analysis.

Firstly, the model has been created into Solidworks, following the dimensions in table 1, and some hand calculations have been done following the Verkel' s calculation (Verkel, 2017).

After that, the model has been moved into an Ansys Workbench 19.2, which has been tested through some stress deformation.

Table 1. Materials into Ansys Workbench 19.2

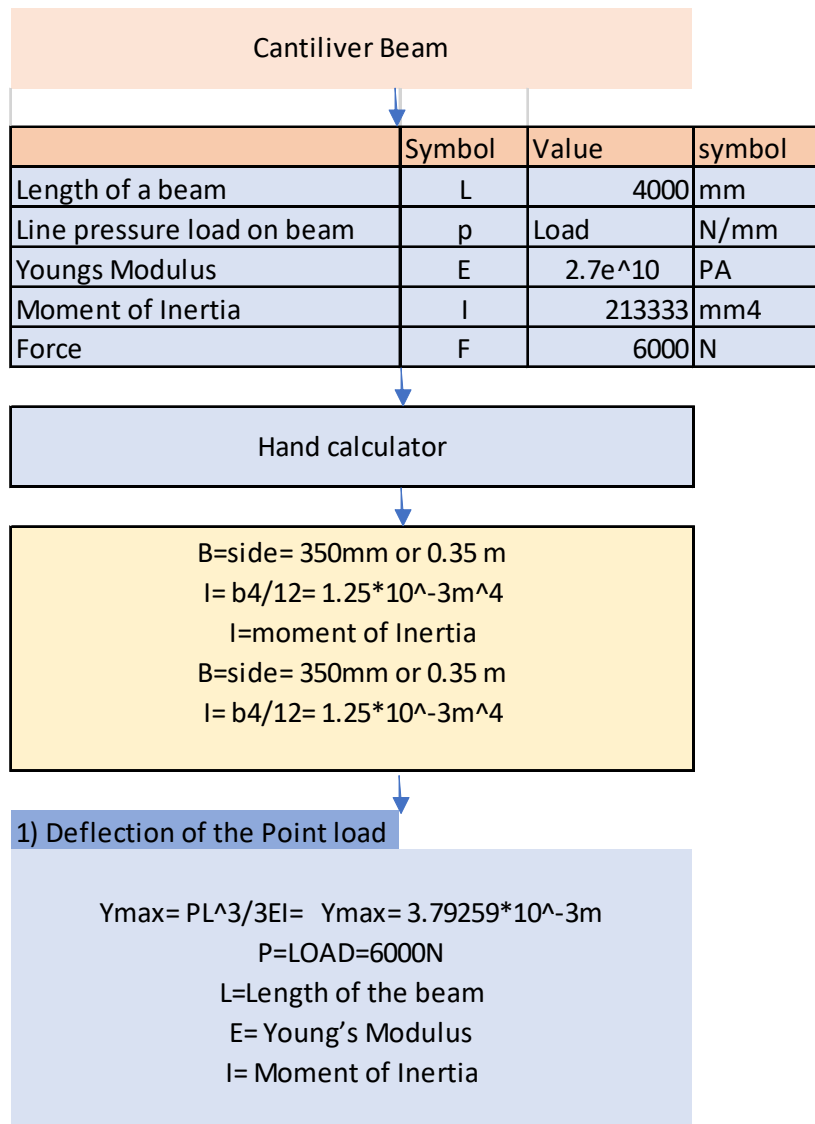


Figure 3 illustrates, the materials have been selected and edited, and a calculation have been made following the test of Verkel as shown in Table 1 (Verkel, 2017).

Properties of Outline Row 3: Steel				
	A	B	C	D E
1	Property	Value	Unit	
2	Isotropic Elasticity			
3	Derive from	Young's Modulus and...		
4	Young's Modulus	2.7E+10	Pa	
5	Poisson's Ratio	0		
6	Bulk Modulus	9E+09	Pa	
7	Shear Modulus	1.35E+10	Pa	
8	Field Variables			
9	Temperature	Yes		
10	Shear Angle	No		
11	Degradation Factor	No		

Figure 3. Steel material edited, of the Cantiliver beam

Subsequently, the Model has been created in Solidworks then moved into the Ansys workbench as illustrates in Figure 4.

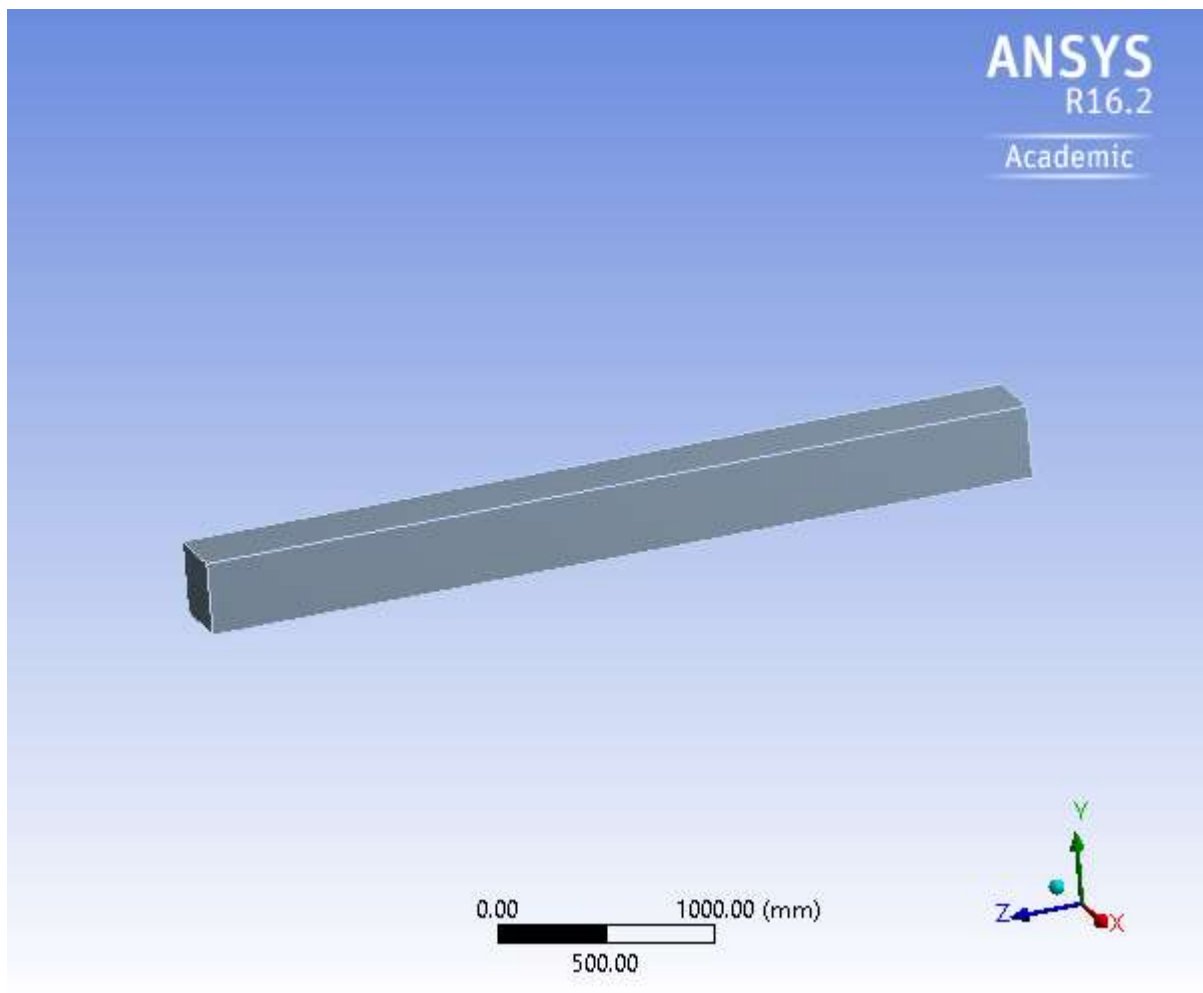


Figure 4. Cantilever Beam

Afterwards, the mesh has been set as a default and the boundary conditions with the Fixed support and Force are applied to the component as illustrated in figure 5.

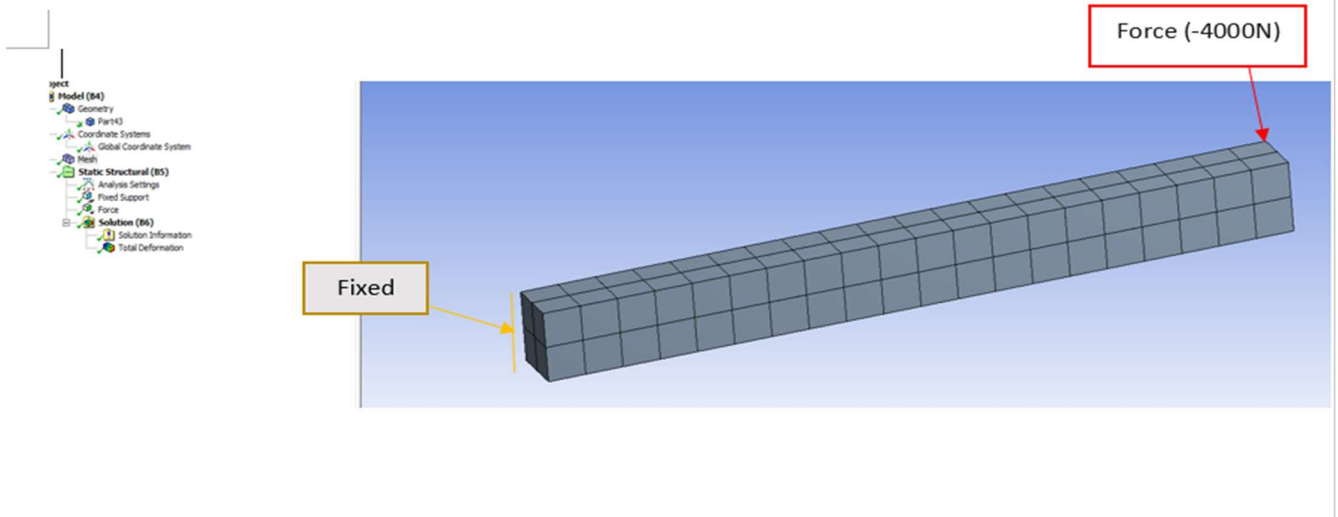


Figure 5. Cantilever beam with mesh, fixed support and Force applied to the free end

Once the simulation is ran , the max value of the Deformation of the Beam evidence to be 3.8165mm, whereas the analytic calculation shows 3.79mm as table 1, which is pretty close to the original expectation as illustrated in Figure 6 (Verkel, 2017).

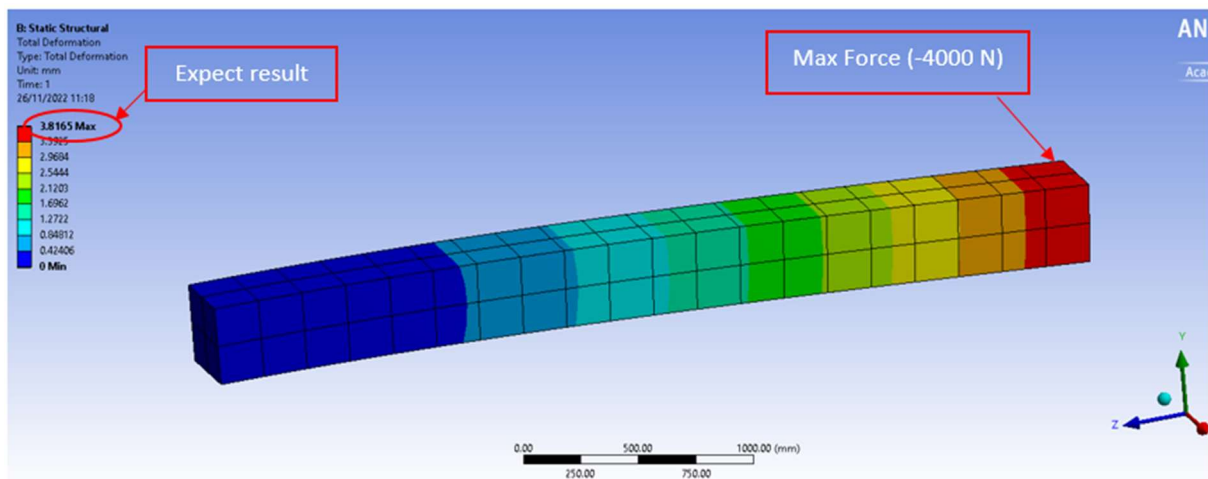


Figure 6. Final basic static structural of a Cantilever Beam

2.3 Thermal and static structure analysis

The main function of the pistons is to provide gas through the cylinders bore and move the piston up and down. Additionally, the piston allows the combustion process to be transferred to the connecting rods and then on to the crankshaft (Hillier & Coombes, 2012).



Figure 7. Single pistons from different engine ((Hillier & Coombes, 2012,p.64)

As illustrated in Figure 7, the top face of the piston is mainly exposed at high temperature from the burned gases during the combustion. That force (pressure) which performed in the cylinder, is transferred in the entire structure of the piston to the gudgeon pin and then through the connecting rod (Hillier & Coombes, 2012). The high temperatures and mechanical stress give a result of thermal and mechanical stress. Primarily, mechanical stress comes from the gas forces and thermal stress through the combustion process of the engine. As a result, the piston should be able to transfer the kinetic energy of the high gas pressure and temperature (Kumar, 2017).

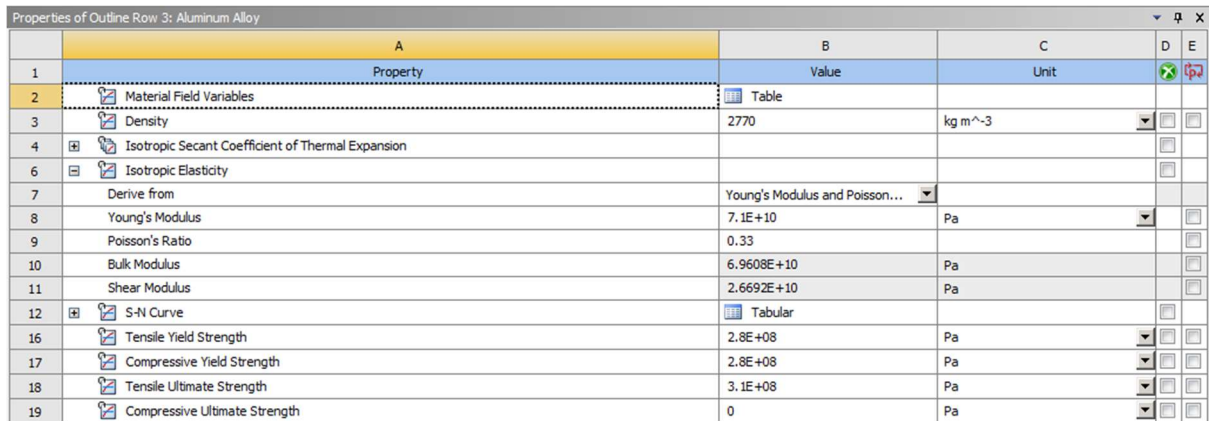
3 Methodology

Modern high-powered engines frequently use piston aluminum-silicon alloys because of their low specific gravity, excellent thermal conductivity, and superior castability. However, the Al-Si alloy's high-temperature strength is significantly decreased under the conditions of increased thermomechanical loads brought on by the growth in the specific power output of diesel engines and the working temperatures of pistons reaching 0.8–0.9 of the melting point. In this regard, it is necessary to more accurately predict the temperature and stress-strain condition of the pistons in order to give the requisite lifetime, taking into consideration two-frequency loading and inelastic deformation (Sivachev & Myagkov, 2019).

In addition, the piston is inserted in Ansys Workbench and Static Structural and Thermal analysis has been conducted and compared with other sources.

3.1 Static structural and Thermal analysis of a single piston

For the main thermal Analysis for this technical report, the piston has been chosen.



	A	B	C	D	E
1	Property	Value	Unit		
2	Material Field Variables	Table			
3	Density	2770	kg m ⁻³		
4	Isotropic Secant Coefficient of Thermal Expansion				
6	Isotropic Elasticity				
7	Derive from	Young's Modulus and Poisson...			
8	Young's Modulus	7.1E+10	Pa		
9	Poisson's Ratio	0.33			
10	Bulk Modulus	6.9608E+10	Pa		
11	Shear Modulus	2.6692E+10	Pa		
12	S-N Curve	Tabular			
16	Tensile Yield Strength	2.8E+08	Pa		
17	Compressive Yield Strength	2.8E+08	Pa		
18	Tensile Ultimate Strength	3.1E+08	Pa		
19	Compressive Ultimate Strength	0	Pa		

Figure 8. Aluminium alloy, piston selected

Figure 8 illustrates the main material, which is the aluminium alloy, selected for this thermal and static analysis in Ansys. As the piston is subjected to extremely high temperatures during the combustion in an internal combustion engine, and at the same time, high strength and a high resistance need to be lightweight (EUROPEAN_PATENT_APPLICATION, 1998). Also, the aluminium alloy is a very strong material, and it is a perfect barrier for the dispersion of the carbon dioxide (Callister & Rethwisch, 2020).

The first two main investigations have been conducted such as Static Structural and Steady-state Thermal in the ANSYS as evidenced in Figure 9.

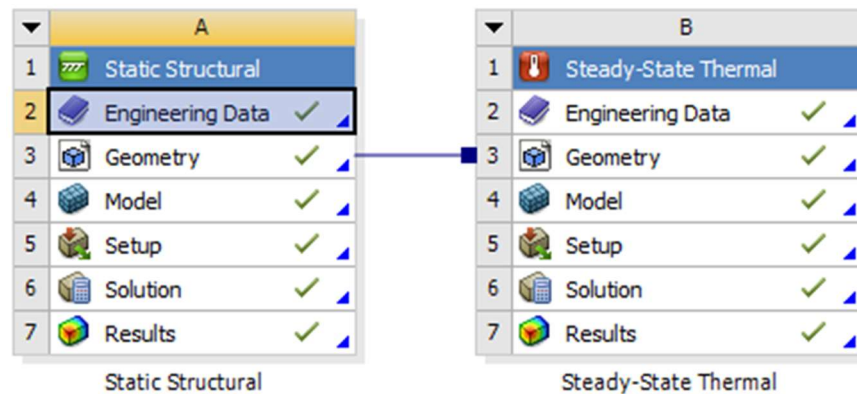


Figure 9. Static structural and Steady-state Thermal of a single piston

Table 2. Static structural and Thermal Analysis

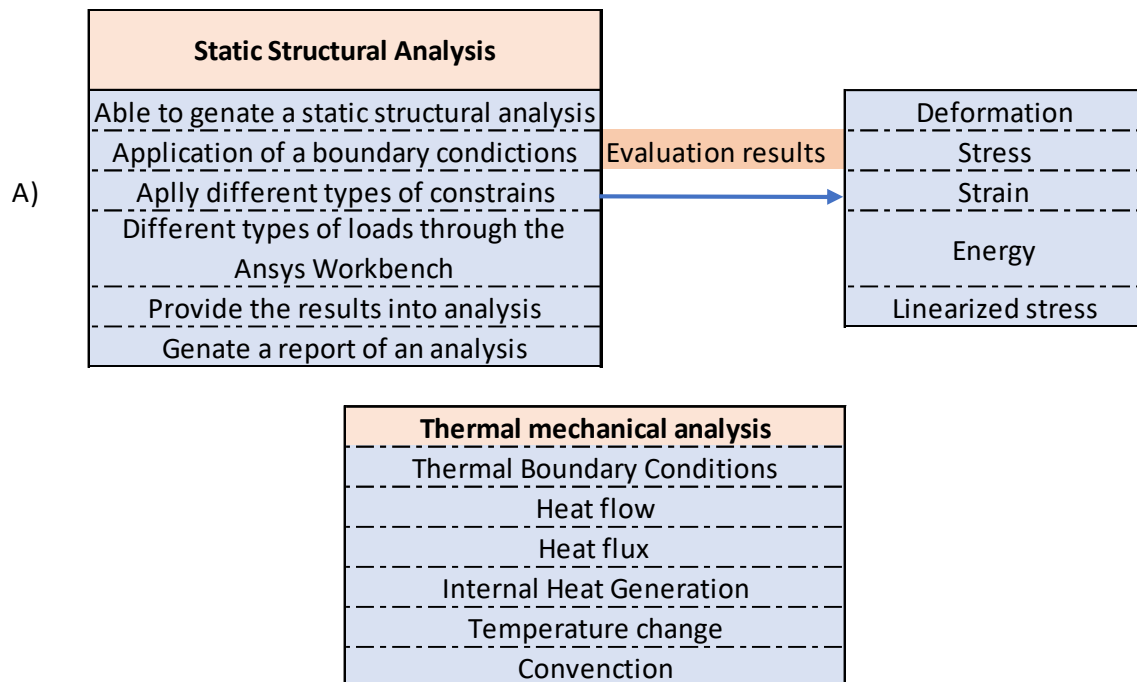


Table 3 evidences the major importance of using static structural analysis and the thermal mechanical analysis in the Ansys Environment and through this technical report (ANSYS, 2016).

Table 3. Mesh sizing static structural

Details of "Mesh"	
[-] Display	
Display Style	Use Geometry Setting
[-] Defaults	
Physics Preference	Mechanical
Element Order	Program Controlled
☐ Element Size	4.0 mm
[+] Sizing	
[+] Quality	
[+] Inflation	
[+] Advanced	
[+] Statistics	

The main part of an engine is the piston, Once the Aluminium Alloy has been set up correctly in the piston, the static Structural analyse has been started, where Table 4 illustrates the mesh size of 4 mm.

Table 4. Piston and Engine Specification

Features	Specifications
Make	Simple piston, assembled in Solidworks
No of cylinders in a total engine	4
head piston	54.5 (mm)
max speed	1500
cylinder diameter	100 (mm)
Connecting rod length	120(mm)

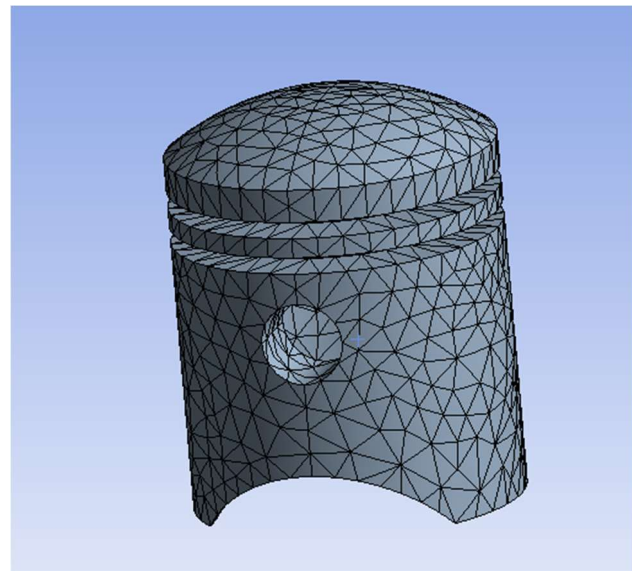


Figure 10. Piston mesh at 4 mm

Table 5 shown the measurements and Figure 10 illustrates the Piston which has been created in Solidworks and transferred into Ansys Workbench. Table 4 shows the mesh size of 4mm of the

piston, further, Figure 11 evidence the boundary conditions that have been applied such as (Ali, Suresh, & Surendar, 2018):

- Fixed support
- Pressure of 16 MPa

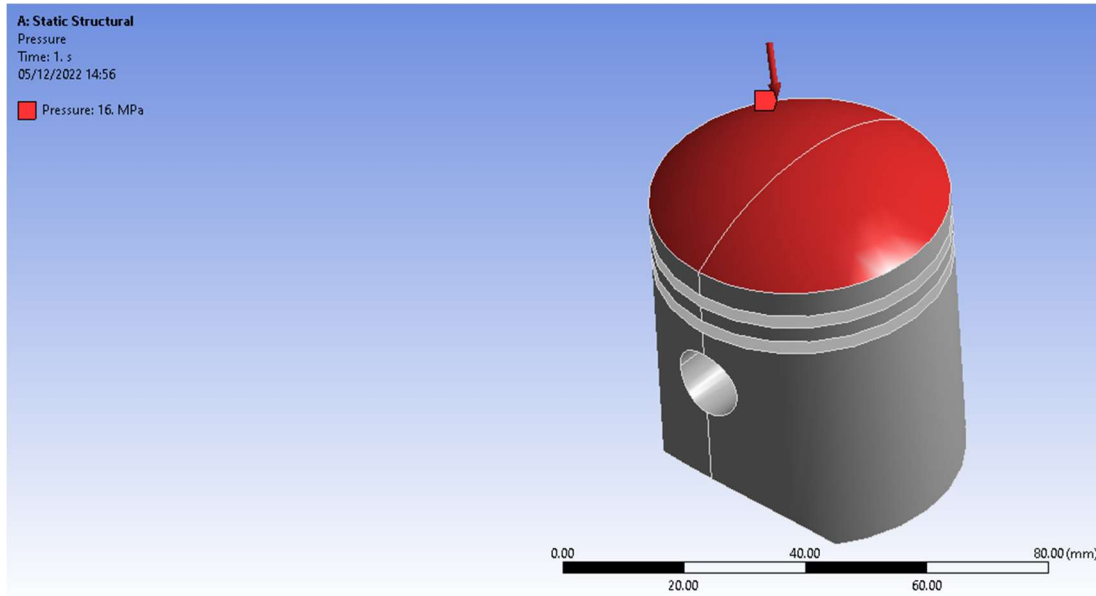


Figure 11. Static structural of a piston following the test

The realistic pressure of 16MPa, as illustrated in Figure 11, has been taken from the experimentation of Dagar, Rinawa, Gupta and Gupta (Dagar, Rinawa, S Gupta, & Gupta, 2022) .

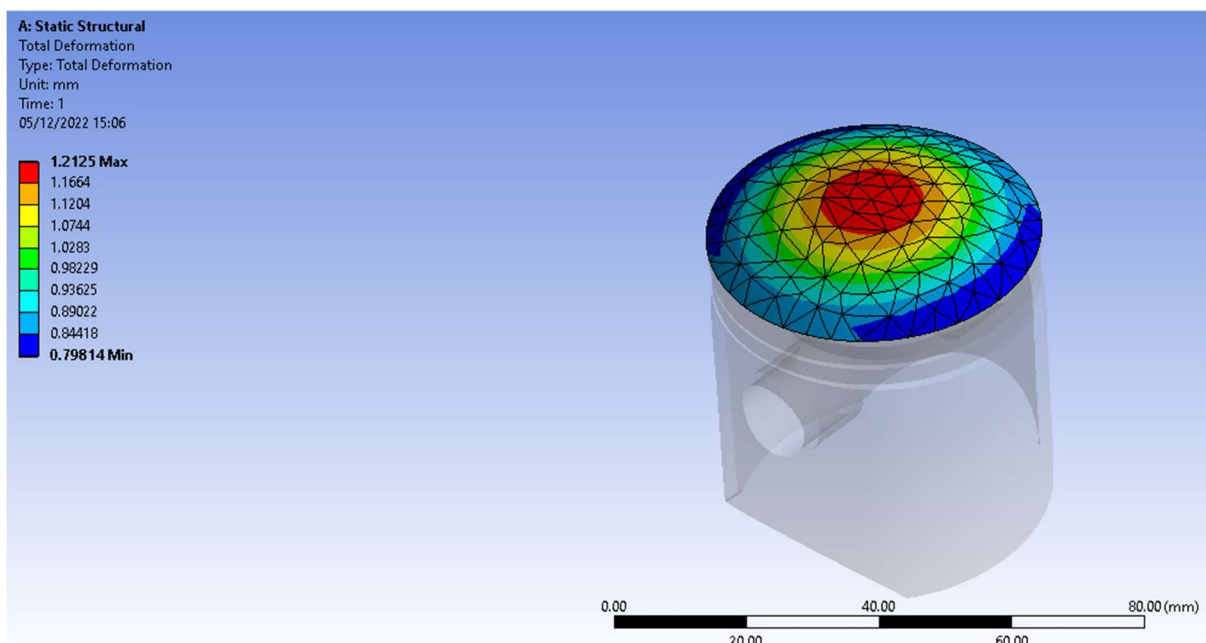


Figure 12. Total Deformation

On the other hand, for the static structural, the meshing size has been started from 4mm as illustrated the Figure 10, and the boundary conditions were applied to the components such as the total deformation. However, the Total Deformation is 0.17534 mm at 2 mm mesh deflection from the Dagar, Rinawa experimentation. Nevertheless, the actual model tested in Ansys evidences a value of 1.2125 mm at 4 mm mesh as illustrated in Figure 12.

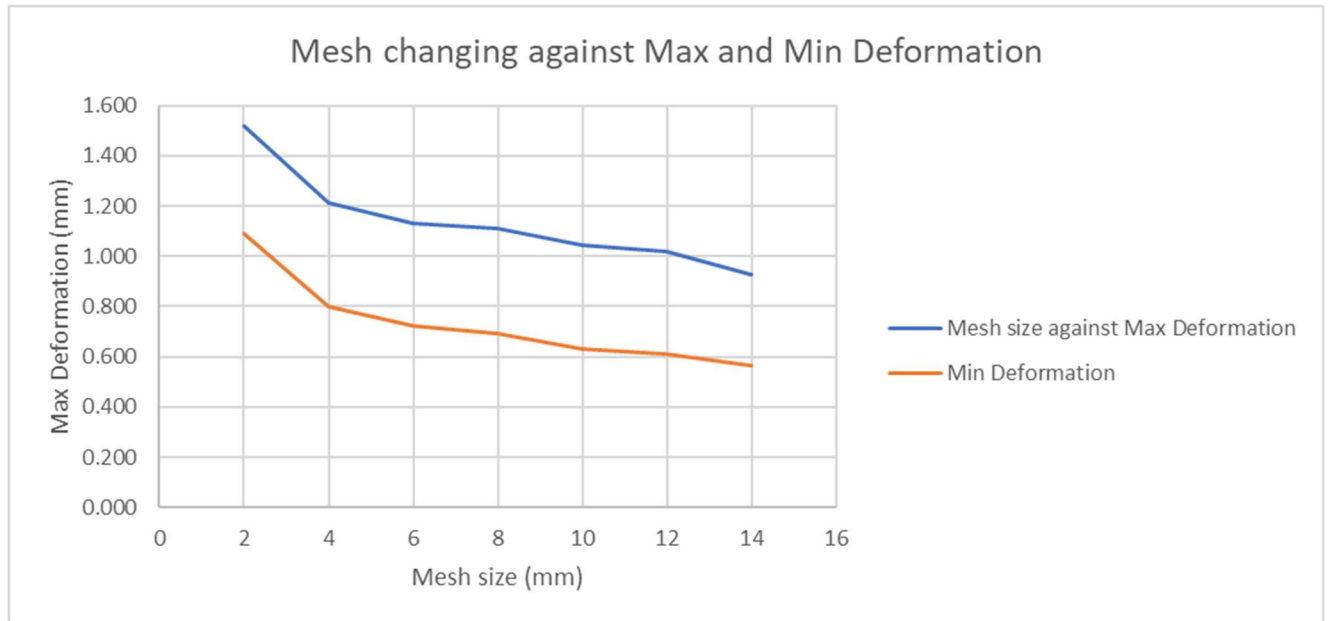


Figure 13. Mesh changing

Table 5. Table of value of mesh changing

mesh	Deformation Max	Deformation Min
mm	mm	mm
2	1.520	1.0907
4	1.213	0.7981
6	1.132	0.7254
8	1.113	0.6944
10	1.044	0.6321
12	1.021	0.6131
14	0.926	0.5659

The mesh in Ansys is an important aspect to consider through the software as illustrated in Figure 13 and Table 6. Meshes is a fundamental aspect to consider due to gives a much smoot elements, further, mesh is a crucial step to analyse, which give the components to be divided into small elements and nodes where Ansys is able to evaluate (Soni, et al., 2022).

Therefore, Figure 13 and Table 6, illustrates the importance of mesh variation in Ansys environment. The original test was 0.17534mm whereas in the actual test, with the same Aluminium alloy, there is an approximate error of around 1% which is still acceptable (Dagar, Rinawa, S Gupta, & Gupta, 2022).

3.2 Thermal Study

Another study has been conducted for this technical report which is a thermal study and compared with another test and compared

3.2.1 Temperature set up

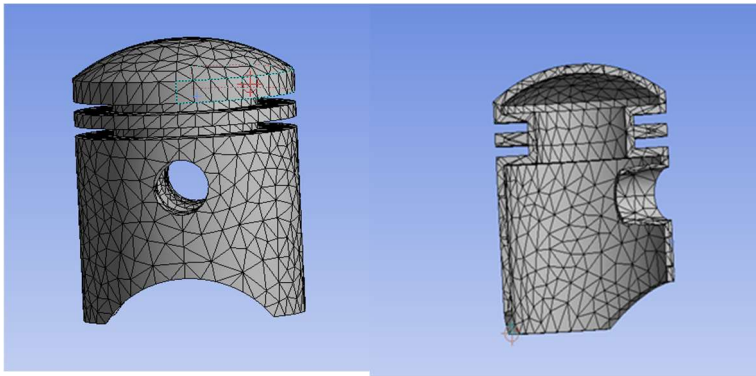


Figure 14. Mesh sizing 4mm external and internal of the piston

Once the mesh was set up as illustrated in Figure 14, the first steady-state thermal started and the boundary conditions were applied, the value was taken from the Soni, et al. (2022) test.

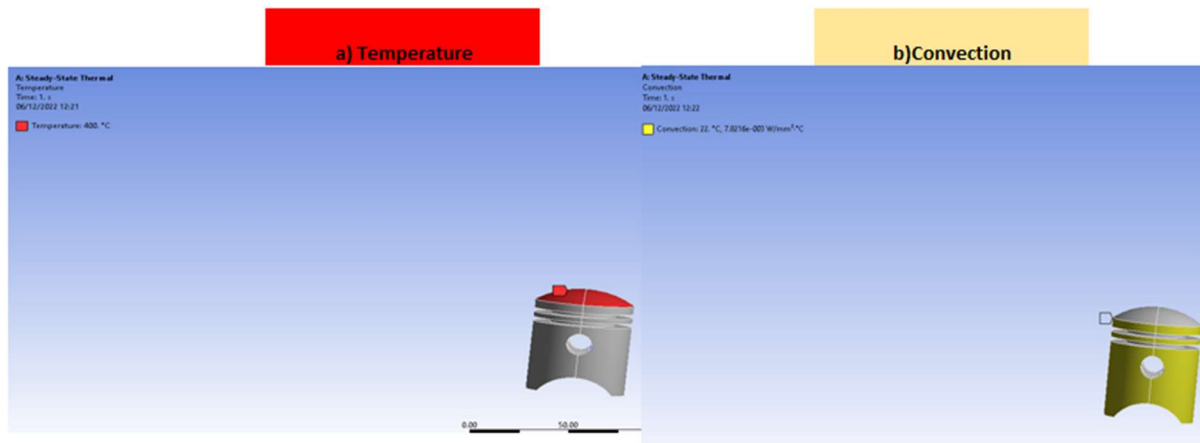


Figure 15. a) Temperature and b) Convection

The state temperature is applied on top of the head of the piston at 400°C as evidenced in Figure 15, a temperature of the piston, where b is the convection of the piston and the material selected is the Aluminium alloy from the Material Selection of Ansys software. As a result of the lubrication and air-cooling heat transfer which happens in the engine, the convection is given at the single piston. The convection is applied to the face of the piston.

The value of $7.8216 \times 10^{-3} \text{ W/mm}^2 \cdot ^\circ\text{C}$ was added to the convection phase (Soni, et al., 2022) . Once the boundary conditions have been set up the thermal results such as Thermal and Heat flux.

3.2.2 Thermal studies results

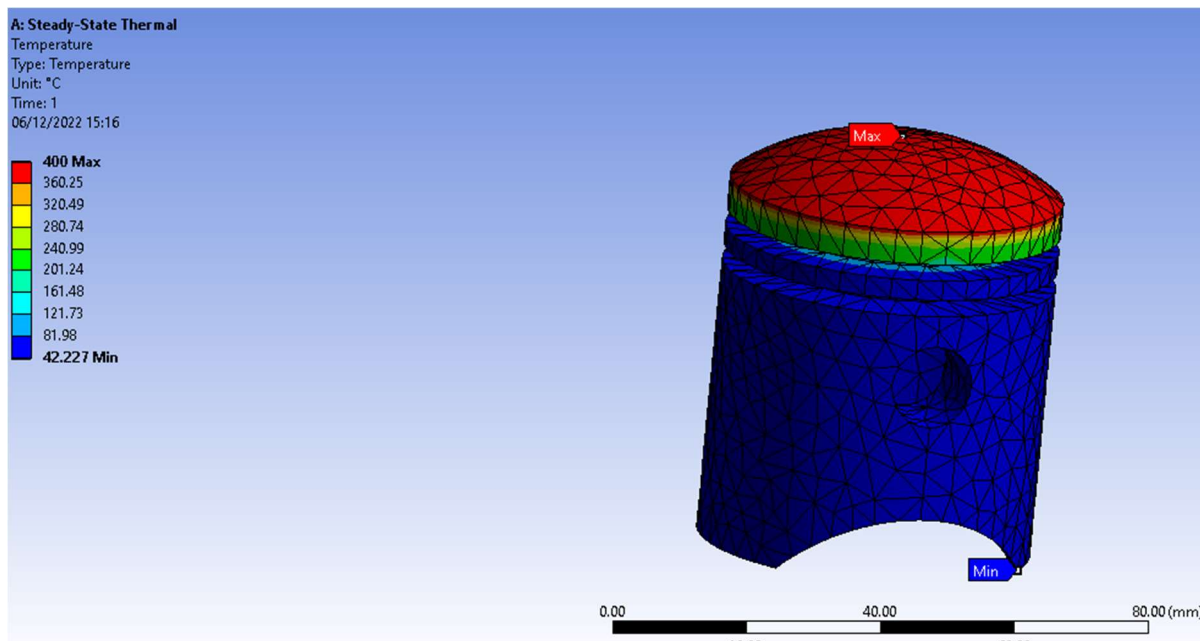


Figure 16. Temperature of the piston with a Max and Min value

Once the Max temperature of 400°C has been applied to the top head of the piston, the solutions illustrate the minimum value of 42.227°C as evidenced in figure 16.

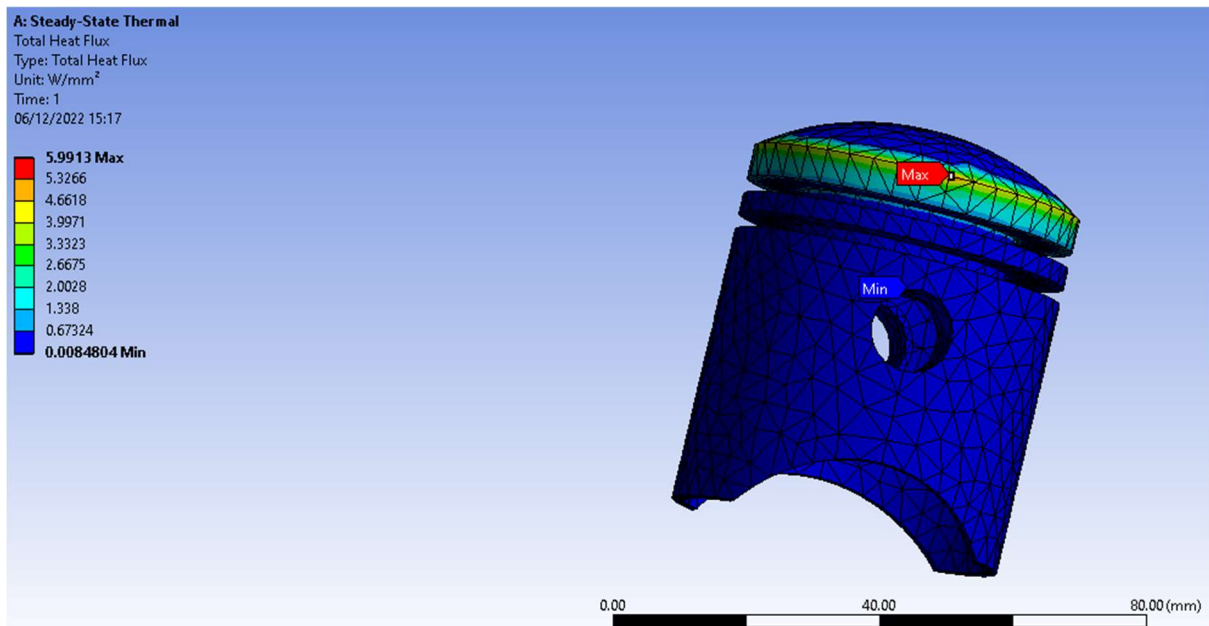


Figure 17. Heat Flux value

Once the value of convection and temperature have been applied to the boundary conditions, the piston of Figure 17 evidence the max value of 5.9913 (W/mm²) in comparison to the test in which the max value is 5.9121 (W/mm²) . Therefore, the difference in the max value is less than 1%, which is still an acceptable solution (Soni, et al., 2022).

3.3 Rigid dynamic study simulation of an engine

The primary component of an Internal combustion Engine is the crankshaft system. In addition, in the internal combustion engine reliability, vibration, noise and other factors are significantly influenced by the crankshaft's dynamic properties (You, Ma, & Wen, 2009). When modelling, simulating, and creating designed technical systems, the link between stresses and strains, displacements and forces, and stresses and forces is crucial (Neto, Roseiro, Cirne, & Leal, 2015). For the dynamic test, an Engine has been created following the measurements of CAD CAM TUTORIAL, and various components have been designed as illustrated in figure 18, such as (TUTORIAL, 2019):

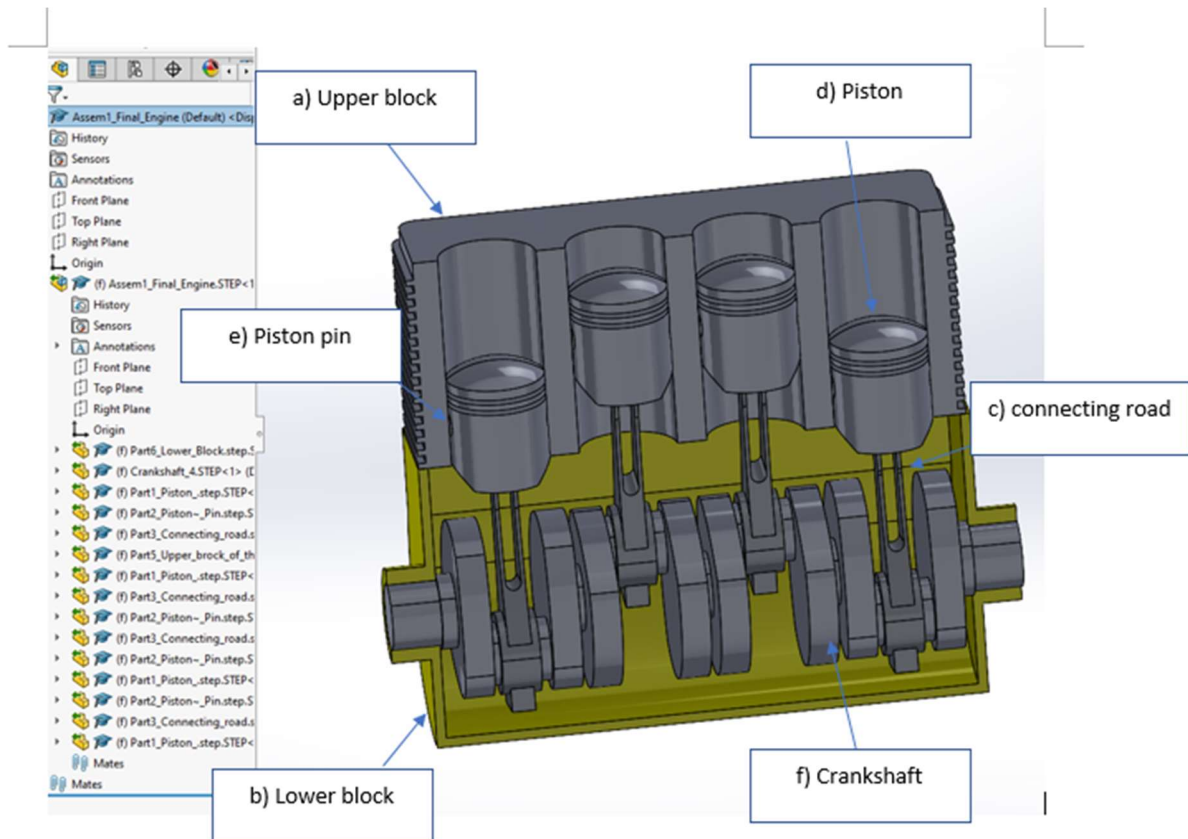


Figure 18. Engine components designed in Solidworks (TUTORIAL,2019)

- a) Upper block
- b) Lower block
- c) Connecting road
 - d) Piston
 - e) Piston pin
 - f) Crankshaft

Once the Engine components have been designed into Solidworks, the entire Engine components have been transferred into Ansys Workbench.

Ansys start up

The main reason for this technique is to regenerate a real engine into the software and make some Design improvements. In addition, the primary part of Ansys is to insert the correct material for each component, so the component can have the right mass and give a realistic velocity as evidenced by equation 1 (Chen & Liu, 2018).

$$F = m * a$$

Equation 1

F = Force applied on any components (Newtons)

m = mass of a component (kg)

$$a = \text{acceleration} \left(\frac{m}{s} \right)$$

In addition, considering the DOF model, the equation of motion can be rewritten as Equation 2 (Chen & Liu, 2018):

$$\mathbf{Mu} + \mathbf{Cu} + \mathbf{Ku} = \mathbf{f}(t) \quad \text{Equation 2}$$

u = Nodal displacement vector

m = Mass

c = damping of a model

K = Stiffness of a components

F = Force vector

The material that has been selected for the single-head piston is Aluminium alloy as evidenced in Figure 8 whereas other components of the engine have been given as Structural Steel by default of the Ansys software. In addition, Equations 1 and 2, are going to be fundamental to find the right Kinematic energy throughout the engine (Chen & Liu, 2018).



Figure 19. Full Engine from the kinematic study

Figure 19, shows the full Engine has been inserted in the Ansys Environment. Moreover, the piston head has been converted into an Aluminium alloy piston as evidenced the Figure 20.

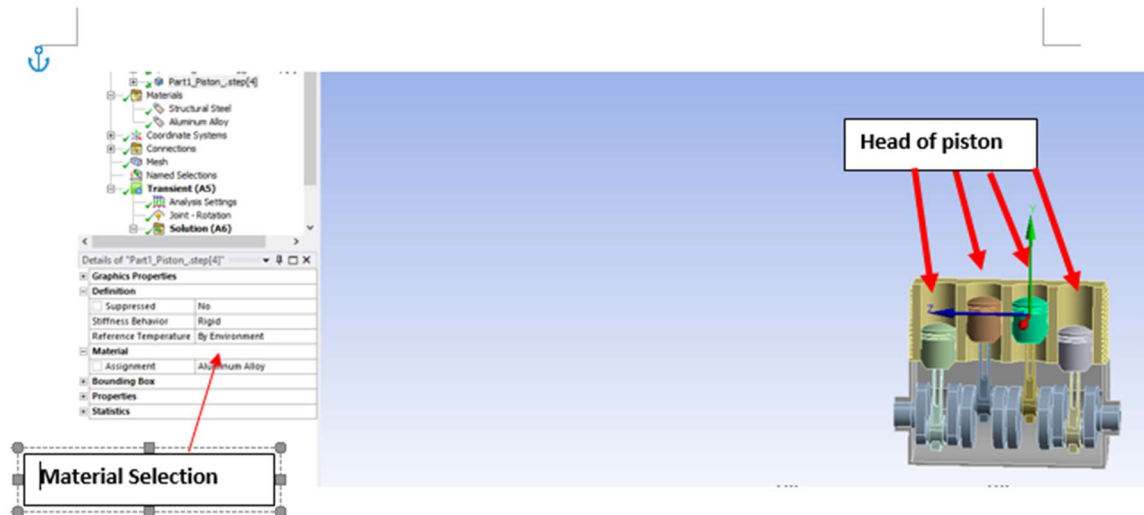


Figure 20. Material Selection of all 4 piston Engine

Once the material selection has been applied to the main head of the piston, the rigid dynamic analysis has started. As mentioned before, the main part of the engine is the piston which is attached to the gudgeon pin and connects to a connecting rod. In addition, it can transfer the energy of the combustion through the cylinder, in the crankshaft through the connecting rod, further, the piston is able to transfer the driving force and convert that energy into the crankshaft, which makes the crankshaft rotate clockwise(Gopal, Kumar, Reddy, Rao, & Srinivasulu, 2017).

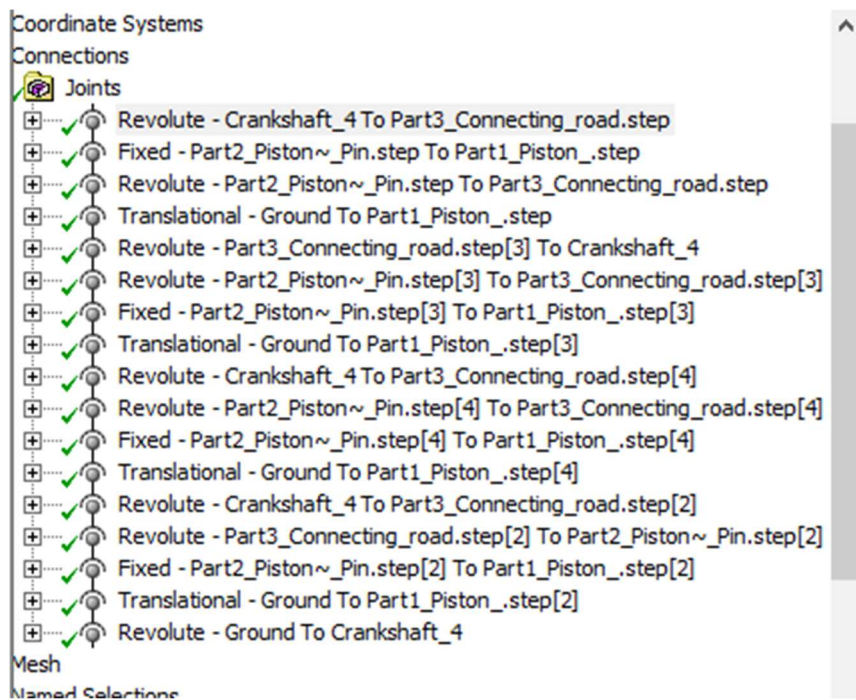


Figure 21. Connections joint of the entire Engine

Figure 21 highlights the entire joint connection which has been used for this kinematic analysis.

An idealised kinematic linkage that regulates the relative motion of two bodies is a joint. Fixed joint types are identified by their fixed translational and rotational DOF (Padmasri, Surendar, Suresh, Vidyasagar, & Design, 2018).

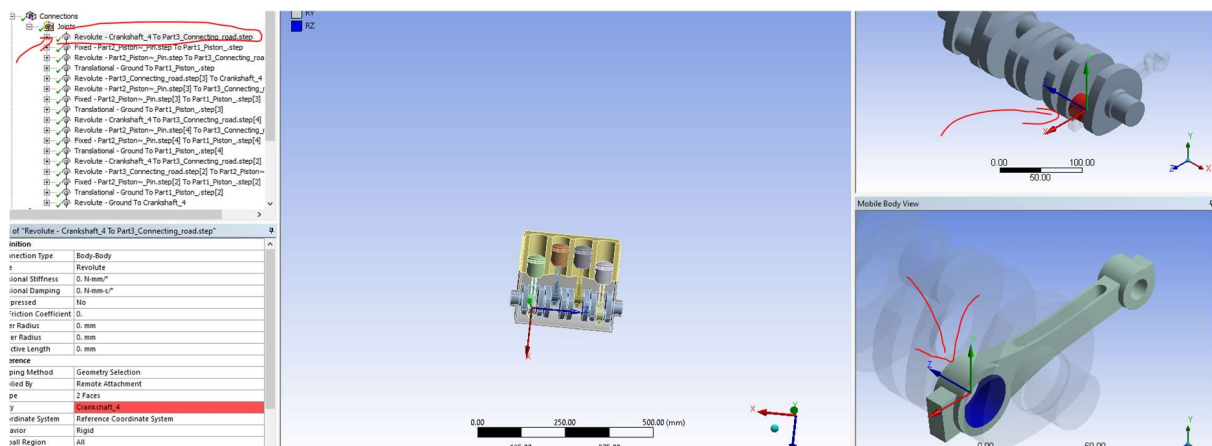


Figure 22. Camshaft attached to the piston

Figure 22 illustrates the revolute joint between the connecting road and the camshaft. The main function of the camshaft can convert the rotation up and down of the piston into a rotatory motion, that is the reason is used the function revolute in Ansys (Gopal, Kumar, Reddy, Rao, & Srinivasulu, 2017).

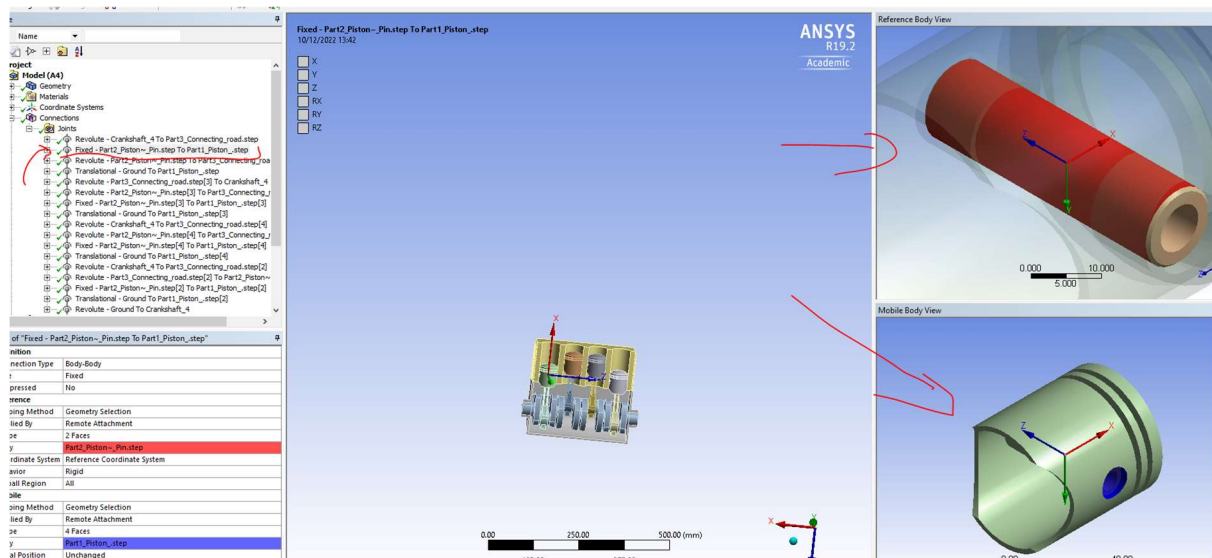


Figure 23. Gudgeon pin and piston fixed

Figure 23 illustrates the fixed components between the main piston and the piston pin, Ansys will be able to calculate the force which is evidenced in Equations 1 and 2. In addition, the pin-hole's surface has been provided with frictionless support. Because fixed support at the pin-hole will help the piston transition from TDC to BDC. As a result, whatever force the piston is under owing to the gas explosion leads the piston pin to fail (inducing bending stresses) (Havale & Wankhade, 2017).

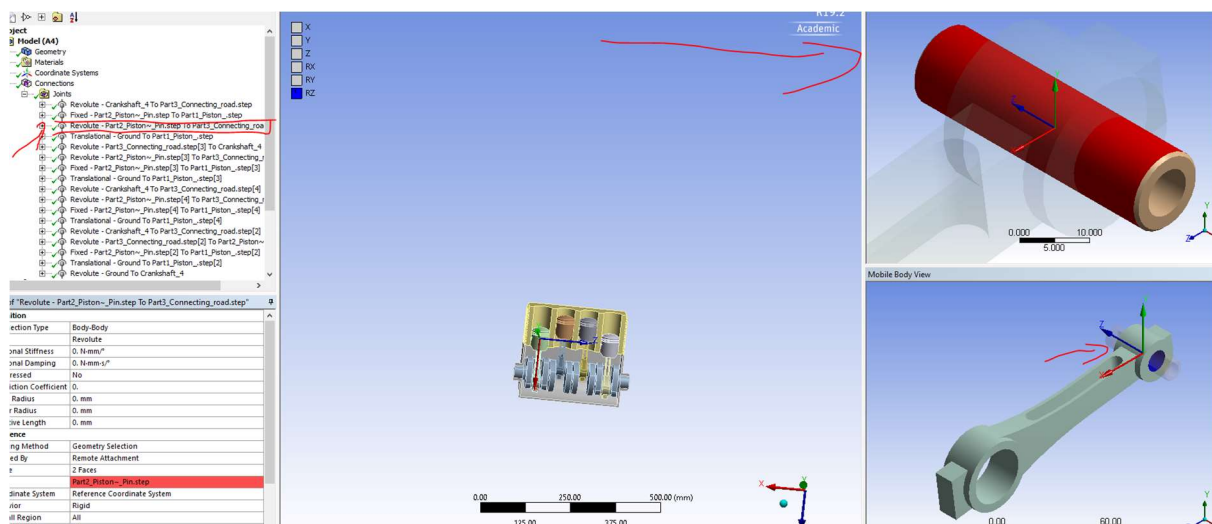


Figure 24. Revolute connected with a connecting rod

Whereas Figure 24 illustrates the connecting rods, the main function of connecting rod is to transfer energy in a rotational motion (Hillier & Coombes, 2012).

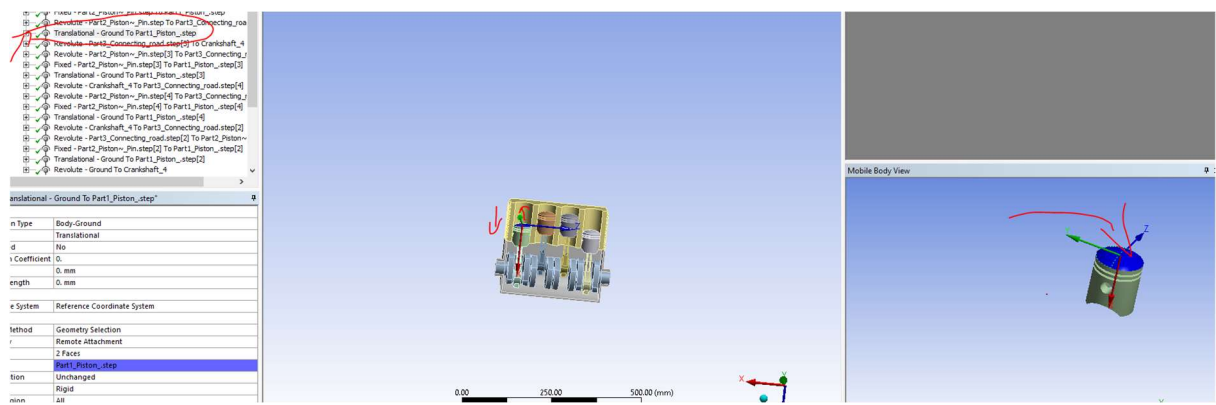


Figure 25. Translational motion through the cylinder

Another important function that has been added to Ansys to ensure the correct movement of the piston in the cylinder is the translation motion of the piston in the cylinder engine. The cylinder is a simple tube close to one end, and the piston moves up and down as highlighted in Figure 25 (Hillier & Coombes, 2012). Moreover, the other additional joint has been done to mate the other four pistons.

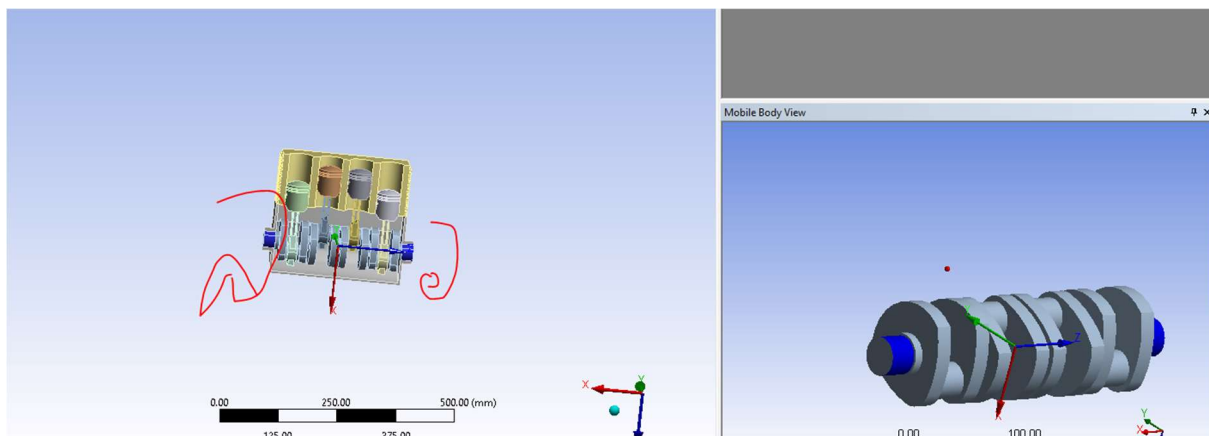


Figure 26. Crankshaft revolute joint

Finally, the crankshaft is one of the complicated parts of an Internal combustion Engine (Singh, Amarjeet, Mittal, & Angra, 2014). Figure 26, illustrates the crankshaft which rotates and it is set at both ends with a function revolute to ground, that can rotate as the piston is moved downwards when the piston reaches the lowest point and the highest point in the engine (Hillier & Coombes, 2012).

4 Results

4.1 Kinematic study results

Once the Rigid Dynamics have been set up, the study on the single piston and export of the maximum force that the piston is subject to during the travel into the cylinder has started (Andrew, 2022).

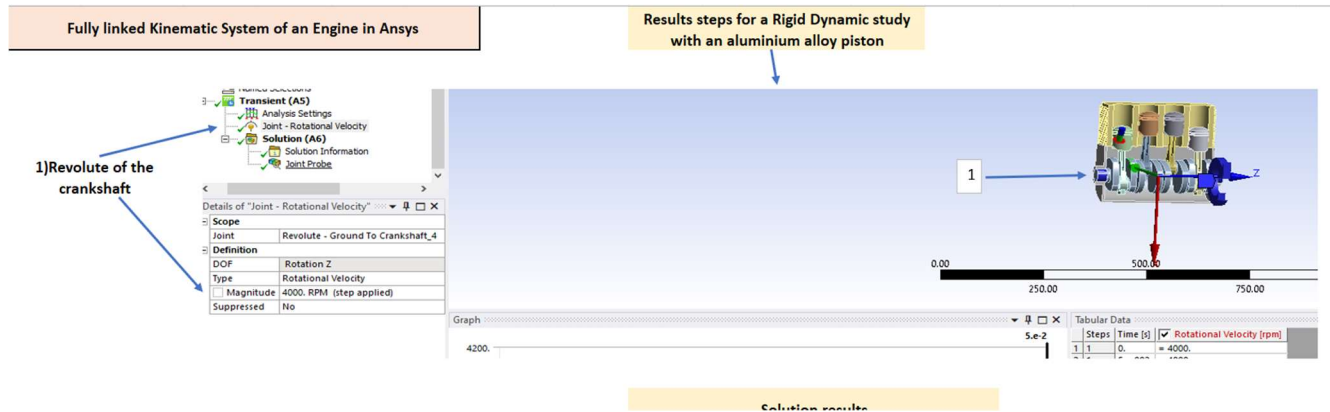


Figure 27. Camshaft of the Engine

Once the piston has been joined correctly, with the other components of the engine, the simulation on the single piston as evidenced in the Figure 27 has started (Andrew, 2022).

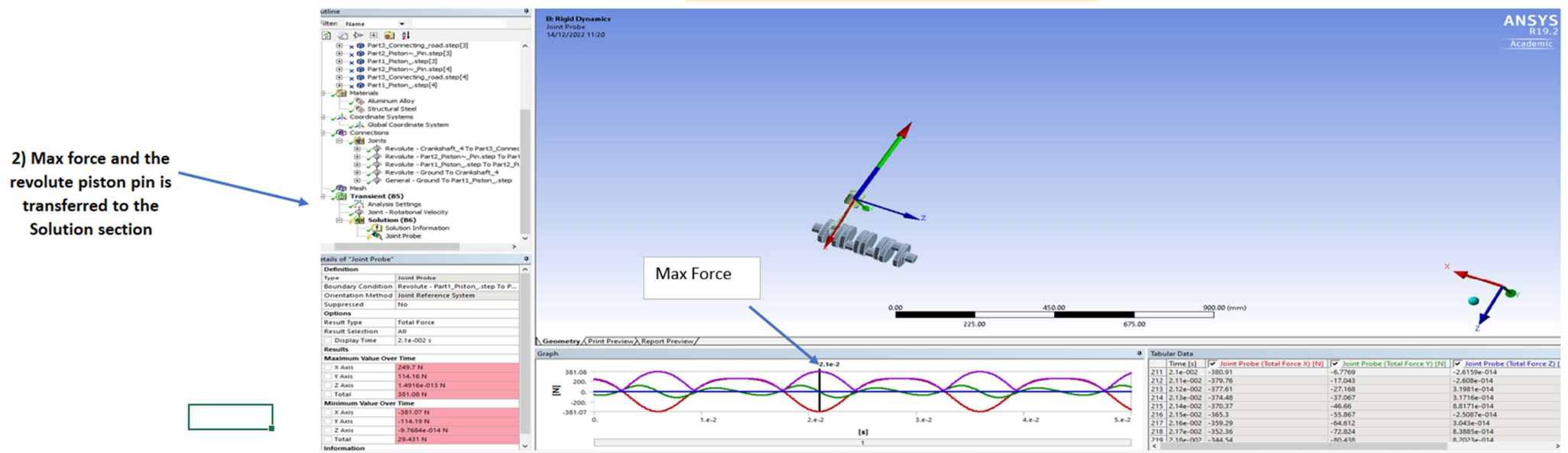


Figure 28. Max Force of the piston and transferred into the thermo study

Figure 28 highlights the Max force of the piston, which is subject during the combustion process, then as the demonstrator Andrew, those data are going to insert in the thermal analysis test (Andrew, 2022).

A			
Region	Region of Ansys Piston	Heat Transfer Coefficient (W/mmC)	Temperature Degrees
A	Top Head, 1	698	650
B	Ring lad, 2	470	500
C	Ring lad, 3	470	180
D	None	470	170
E	Piston skirt, 4	470	110
J	Piston inside, 5	900	95
I	Piston inside b, 6	240	95
F	Piston ring a	-	200
G	Piston rig b	-	180

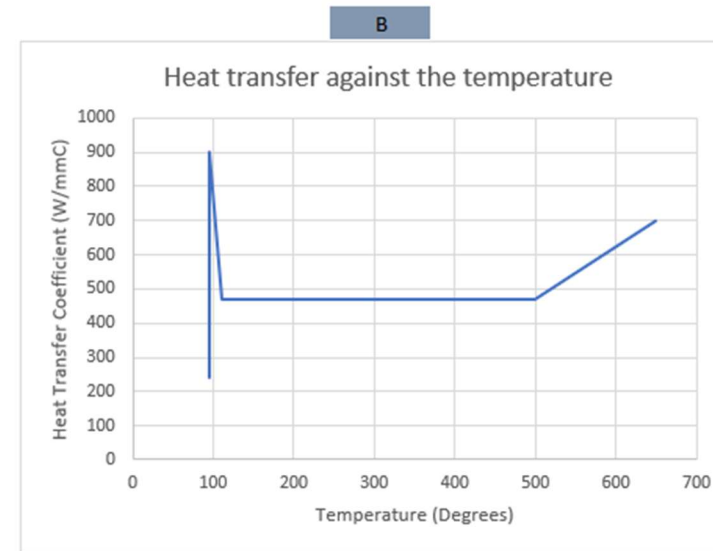
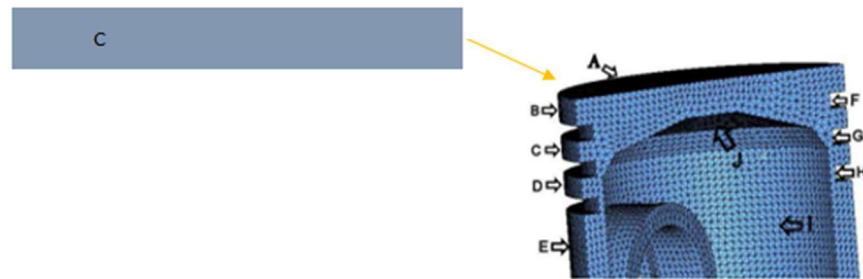


Figure 29. a) Table of Heat Transfer Coefficient and Bulk Temperature, b) graph plotted in Excel, c) Piston value regarding the table a (Mahajan & Deshmukh, 2016, p. 4)

Figure 29 illustrates the data from Mahajan & Deshmukh, the Heat transfer was collected, and the boundary conditions were applied to the entire piston as illustrated in Figure 30 (Mahajan & Deshmukh, 2016).

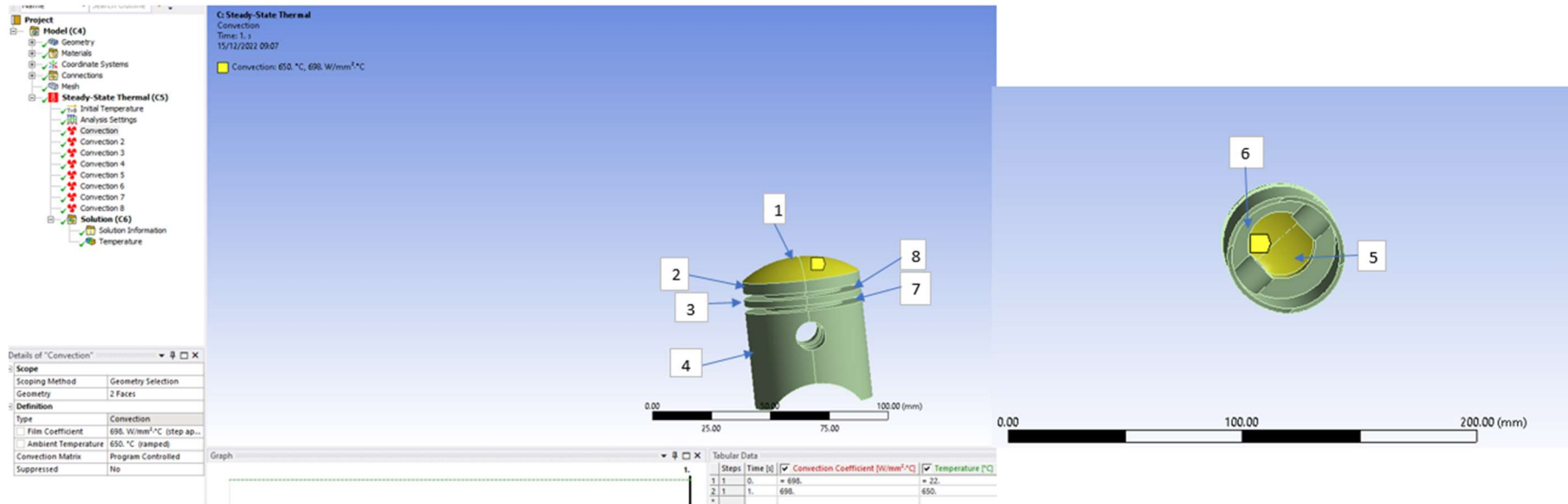


Figure 30. Temperature Convection of the piston taken from Mahajan & Deshmukh (Mahajan & Deshmukh,2016, p.4)

Moreover, Figure 30 illustrates the data taken from the real test of Mahajan and Deshmukh and insert in the Steady-State Thermal of Ansys. Also, the Data from Table a of Figure 29 were inserted into a piston of a Figure 30 (Mahajan & Deshmukh, 2016).

5 Discussion

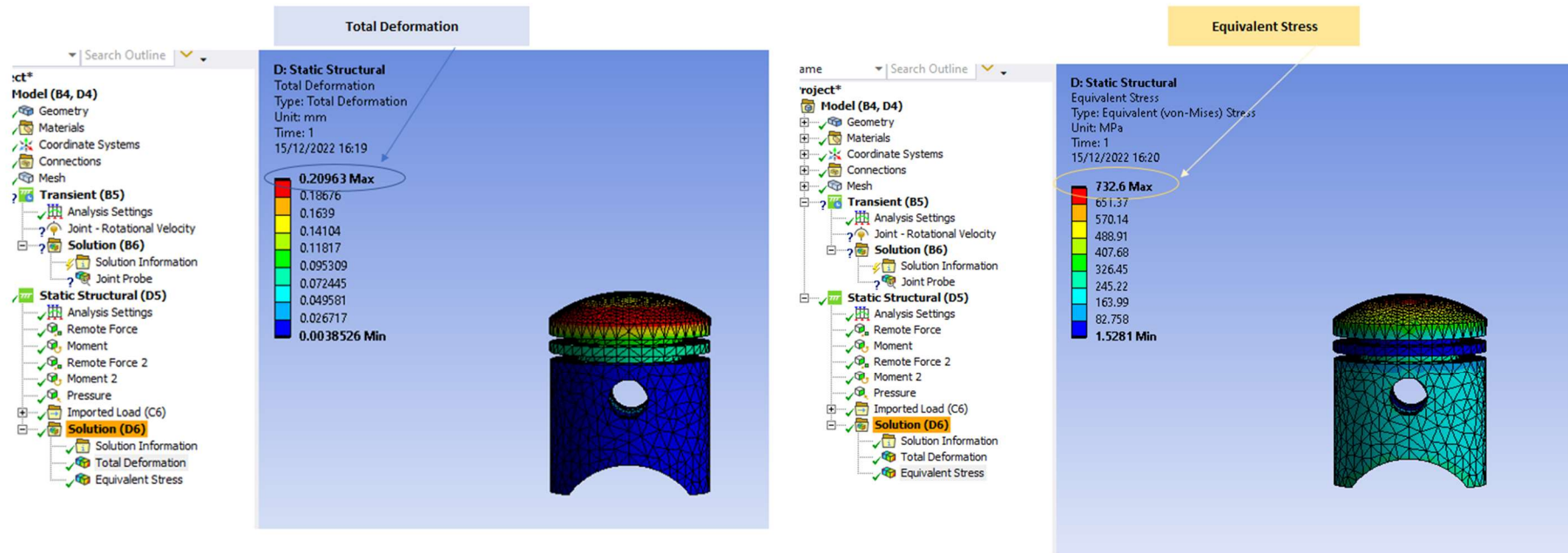


Figure 31. Final Solution of the piston

The stress deformation of Figure 31 illustrates to be at 0.20963 mm in comparison of the demonstrator of Andrew which is the max value of 0.34932 mm and has a variation of 1% which is still acceptable (Andrew, 2022).

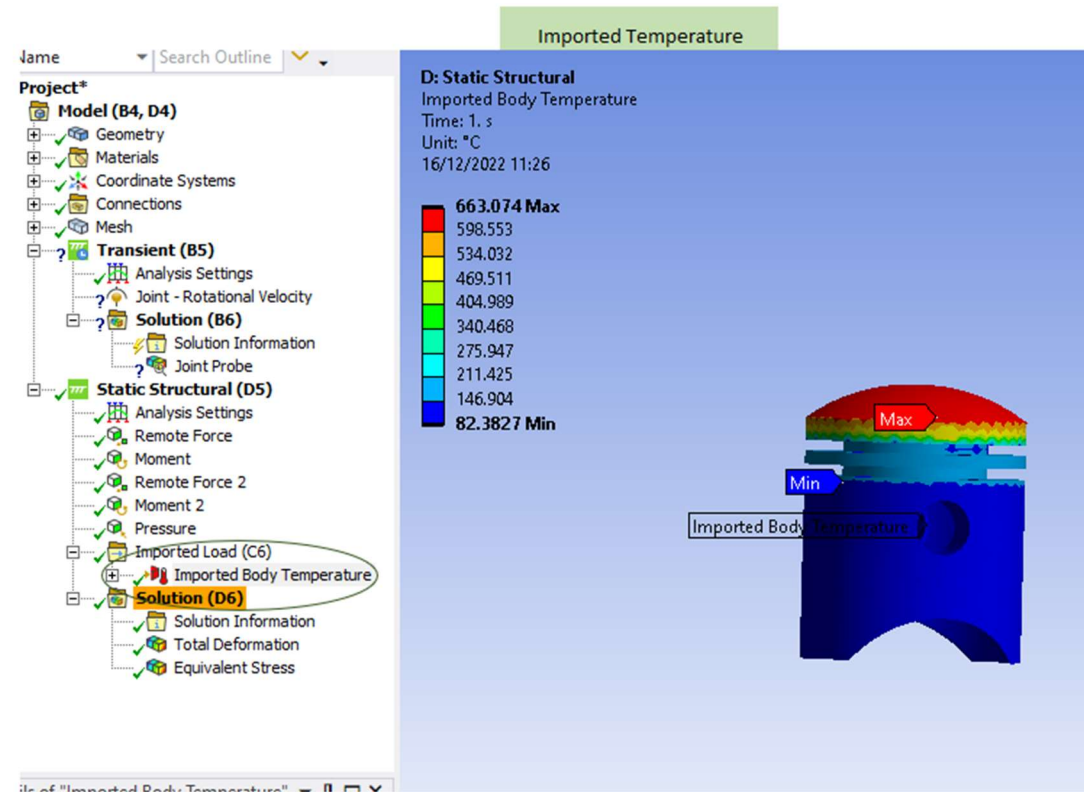


Figure 32. Imported Temperature

However, from the Figure 32 of imported Temperature is 663.074 degrees which were taken from Figure 30 and the Equivalent Stress of 732.6 Mpa , the value of the demonstrator are nearly double it, the reason behind that would be the design measurement, different materials and different pressure applied to the head piston of 1.3 MPA (Andrew, 2022).

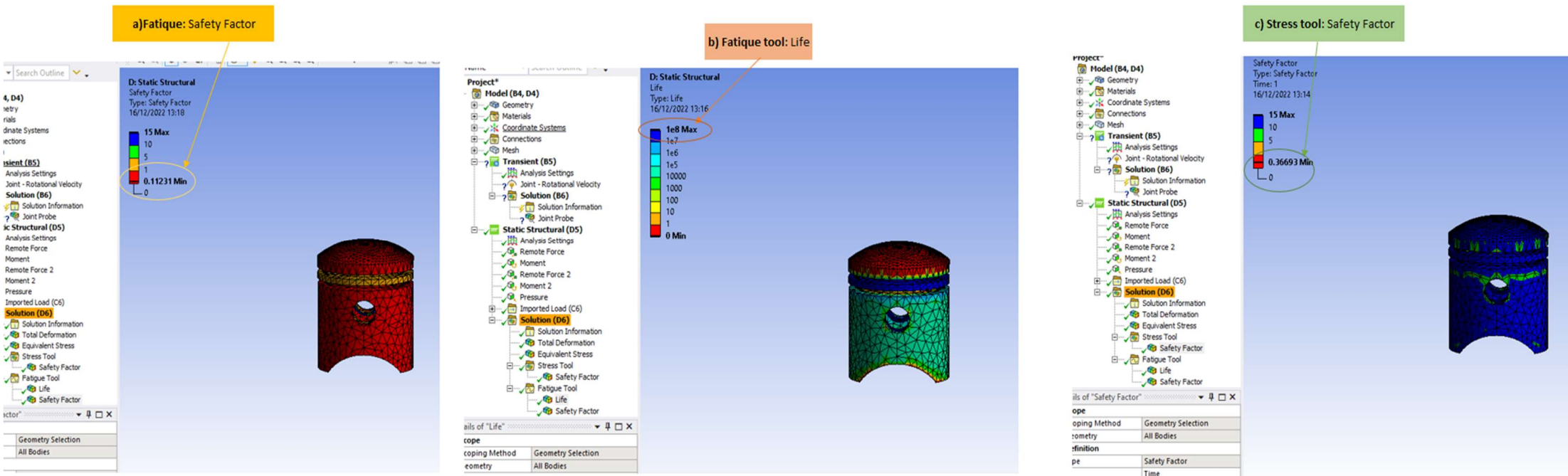


Figure 33. Final Fatigue Analysis, where a) consider the Safety factor, b) life cycle and c) Safety factor under a stress condition

Finally, Figure 33 illustrates the importance of having an Ansys Workbench, the fatigue safety factor of a minimum of 0.11231, the life testing is 10^8 , whereas the stress solution of min 3.6693. If the Final Fatigue analysis would be above 1 that would have an infinite life, however, a result above zero means that is safe to use 10^8 times.

6 Conclusion

In this technical report, 6 main studies have been conducted such as :

- **Simply supported beam:** The final result of the stress analysis on a simple beam is around 3.815 mm and the hand calculations show 3.79 mm which has only a 1% difference.
- **static structural analysis:** Once the Aluminium alloy has been inserted correctly, the Static Structural analysis and the thermal analysis have been considered and the boundary conditions have been inserted correctly. In addition, the realistic pressure of 16 Mpa and the different mesh sizes were evidenced in table 6 and Figure 13. From the original test, it was 0.17534 mm, although the mesh sizing from 2 to 14 mm gave a slight error of nearly 1%, therefore that is acceptable.
- **Thermal study:** For the first thermal study, once the boundary conditions of temperature and convection were applied correctly, the realistic temperature evidences the max value of 400 degrees. Moreover, the Heat flux was applied and the value of 5.9913 (W/mm²) with the real experiment of 5.9121 (W/mm²) had a margin of error lower than 1%.
- **Rigid dynamic study:** Once the model has been converted from Solidworks (CAD) into Ansys, the rigid dynamic study has been started. Following Andrew's experimentation has been important to understand the joint function of Ansys to replicate the engine simulation in software.
- **Kinematic study results:** The main two equations 1 and 2, have been fundamental to calculate the force of the guided pin in the head of the piston. Once the max force has been transferred in the Steady-state thermal, the real temperature has been inserted from Mahajan & Deshmukh (2016). Once the data have been imported in the Static Structural, it has been evidenced a total deformation of 0.20963 mm which is nearly close to Andrew's assessment and the Equivalent Max Stress of 732 Mpa.
- **Fatigue:** To conclude, the fatigue test highlights the Safety factor of the component which is evidenced in Figure 34 to be 0.11231 which means it is safe to be used. The Fatigue tool evidence that the head piston can be under those conditions and movements for about 10e8 of times.

During this analysis, there have been many limitations of the software, such as: saving and reusing, slightly different parameters when the values were used for a different test. Although the results were slightly different from other sources, the boundary conditions were inserted correctly but were still acceptable.

Finally, technology has moved on and can help Engineers to reduce the amount of work that back in the day required up to 2 years, however, there is still a margin of error of 1%, which means those software are not 100% accurate.

7 References

- Ali, S. A., Suresh, N., & Surendar, E. (2018). Static and thermal analysis of piston with different thermal coatings. *International Research Journal of Engineering and Technology (IRJET)*. Retrieved from <https://d1wqtxts1xzle7.cloudfront.net/57774675/IRJET-V5I10239-with-cover-page-v2.pdf?Expires=1669986416&Signature=doVWfNgAht~U0ioHVzDP7HrnqJoMLyR~HbqUaNHdAfT3kkroBJ--r9OnZNxO5Hf2TlaDUUW6FRuu4UOdyGKKliHqsM-0cglwEwn3qcJZ3wufkGXf~pcLSWvnTmA321F8DRkP9-jeNkr>
- Andrew, N. (Director). (2022). *Fully linked system* [Motion Picture]. Retrieved from <https://web.microsoftstream.com/embed/channel/0473ddb1-c257-4584-b3c1-10a50abf2a37?app=microsoftteams&sort=undefined&l=en-gb#>
- ANSYS. (2016). *Thermal Analysis*. Retrieved from Ansys Lecture 9: https://www.clear.rice.edu/mech517/WB16/lectures_trainee/Mechanical_Intro_16.0_L09_Thermal.pdf
- Callister, J. W., & Rethwisch, D. G. (2020). *Fundamentals of materials science and engineering: an integrated approach*. John Wiley & Sons. John Wiley & Sons.
- Chen, X., & Liu, Y. (2018). *Finite element modeling and simulation with ANSYS Workbench*. CRC press.
- Chougule, S. S., & Khatawate, V. H. (2013). Piston strength analysis using FEM. *International Journal of Engineering Research and Applications (IJERA)*, 1724-1731.
- Dagar, N. R., Rinawa, M. L., S Gupta, V. C., & Gupta, P. (2022). *Design and analysis of piston using aluminum alloy and composites in Solidworks and Ansys*. Materials Today: Proceedings 67.
- EUROPEAN_PATENT_APPLICATION. (1998, July 17). *Aluminium alloy for a piston and method of manufacturing a piston - European Patent Office*. Retrieved from patentimages: <https://storage.googleapis.com/93/40/1d/0604f606d405aa/EP0892075A1.pdf>
- Gopal, G., Kumar, L. S., Reddy, K. V., Rao, M. U., & Srinivasulu, G. (2017). Analysis of Piston, Connecting rod and Crank shaft assembly. 7810-7819. Retrieved from <https://www.sciencedirect.com/science/article/pii/S2214785317314116>
- Havale, S. A., & Wankhade, S. (2017). Design, Thermal Analysis and Optimization of a Piston using ANSYS. *International Research Journal of Engineering and Technology*. Retrieved from <https://d1wqtxts1xzle7.cloudfront.net/55496149/IRJET-V4I12241-with-cover-page->

v2.pdf?Expires=1670259337&Signature=gEZiKhW0e-
Joz8TKmnD9R7exMoHz06lanKcwOkby8rIF1~Af4WPL3z0IORF1f5OaLVIEwHHIA94wR7OKfOw
7imnBXRzSYx-cBGela-AGfNtuvMLZLQDv4RWY5IWNIf2a~8Eb3prQKMYX

Hillier, V. A., & Coombes, P. (2012). *Hillier's fundamentals of motor vehicle technology*. Nelson Thornes.

Kumar, M. (2017). *A finite element thermo-mechanical stress analysis of IC engine piston*. Technol. Retrieved from <https://d1wqtxts1xzle7.cloudfront.net/54173463/IRJET-V4I6376-with-cover-page-v2.pdf?Expires=1669648420&Signature=AAwJkqICdSu3M3oAXY-bBrxmuxl~akLrv2X~tcYjglPFkqL9IZ8fbm~2IBww8T1TTI7mtBYICdgDewrcsdSDlymAavOYJDv-Y28p8hJAAtIY7B2IIKL~W2KfHbcusYZrVaoJU62uRQUuBy5>

Kutz, M. (2015). *Mechanical engineers' handbook, volume 1: Materials and engineering mechanics*. John Wiley & Sons.

Mahajan, K. S., & Deshmukh, S. H. (2016). Structural and Thermal Analysis of Piston. *International Journal of Current Engineering and Technology*. Retrieved from <https://inpressco.com/wp-content/uploads/2016/06/Paper522-29.pdf>

Neto, M. A., Roseiro, A., Cirne, L., & Leal, R. (2015). *Engineering computation of structures: the finite element method*. Springer International Publishing.

Padmasri, V., Surendar, E., Suresh, N., Vidyasagar, M., & Design. (2018). Design, Multi Rigid Body Dynamic and Modal Analysis of a Centrifugal Governor by using ANSYS. *International Journal of Engineering Research & Technology (IJERT)*. Retrieved from https://d1wqtxts1xzle7.cloudfront.net/62966581/design-multi-rigid-body-dynamic-and-modal-analysis-of-a-centrifugal-governor-by-using-ansys-IJERTV7IS11004920200415-30313-g1vjvl-libre.pdf?1586962271=&response-content-disposition=inline%3B+filename%3DDesign_

Singh, Amarjeet, Mittal, V. K., & Angra, S. (2014). Strength analysis and optimization methods for four cylinder engine crankshaft based on CATIA and ANSYS. *Trans Tech Publications Ltd*, 1789-1793.

Sivachev, S. M., & Myagkov, L. L. (2019). *Thermomechanical fatigue analysis of diesel engine piston: finite element simulation and lifetime prediction technique*. In *International Conference on Industrial Engineering*. Springer, Cham. Retrieved from https://link.springer.com/chapter/10.1007/978-3-030-22041-9_13

- Soni, A. K., Godara, S. S., Gade, R., Brenia, V., Shekhawat, R. S., Saxena, K. K., & Prasad, R. (2022). Modelling and thermal analysis for automobile piston using ANSYS. *International Journal on Interactive Design and Manufacturing*.
- TUTORIAL, C. C. (Director). (2019). *Solidworks tutorial Design and Assembly of Engine* [Motion Picture]. Retrieved from <https://www.youtube.com/watch?v=Gsa7jnI5D50&t=899s>
- Verkel. (2017). *Tutorial 3:- Cantilever Beam Problem Using Ansys Workbench*. Retrieved from https://www.youtube.com/watch?v=I6_kHiE0oVw&t=521s
- You, X., Ma, X., & Wen, B. (2009). Analysis on dynamic characteristics of internal combustion engine crankshaft system. *IEEE*, 742-745.

