

University of Wales Trinity Saint David

Wales Institute of Science & Art



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

Application of Computer Aided Engineering Processes

Structural Design and Fluid Analysis (ACHA6010)

Dora Boszormenyi
1900966

December 2022

Table of Contents

1	Introduction	3
2	Validation Study.....	4
3	Structural Analysis.....	11
3.1	Design Improvement.....	16
4	Conclusion.....	17

1 Introduction

Computer-aided engineering (CAE) is a field that involves the use of computer software to assist in the design, analysis, and manufacturing of products. It plays a crucial role in the engineering process as it allows for the creation of virtual prototypes, which can be tested and evaluated before physical prototypes are built. This can significantly reduce the time and cost of product development, as well as improve the quality and performance of the final product.

CAE processes are used in a variety of industries, including aerospace, automotive, and biomedical, to name a few. They can be applied to a wide range of products, from simple mechanical components to complex systems. By using CAE techniques, engineers can analyse and optimise the design of a product to meet specific performance criteria, such as strength, durability, and efficiency. Use of CAE processes is essential for modern engineering and product development. It allows for the efficient and cost-effective design and analysis of products, leading to improved quality and performance. ('DESIGN AND ANALYSIS OF THE PISTON BY USING FIVE DIFFERENT MATERIALS 1 PAGADALA SIDDIRAJU 2 KOPPULA VENKATESWARAREDDY', no date; Wang, Liu and Shi, 2010)

There are several benefits to using CAE as a design tool. One of the main advantages is the ability to simulate and analyse a design before it is built or manufactured. This can save time and money by identifying potential problems early in the design process, allowing for quicker and more cost-effective fixes. CAE software can also help optimize a design by allowing engineers to test multiple design variations and quickly compare the results.('Thermal And Static Structural Analysis On Piston', no date)

It can also be used to analyse and test the performance of a product under various conditions, such as different loads or environmental conditions. This can be especially useful for safety-critical systems, as it can help identify potential failure modes and help design engineers implement measures to mitigate these risks.((PDF) DESIGN AND ANALYSIS OF PISTON WITH VARIOUS MATERIALS USING SOLIDWORKS AND ANSYS SOFTWARE, no date)

Despite the many benefits of CAE software, there are also limitations to its use as a design tool. One limitation is that the accuracy of the results obtained from CAE simulations depends on the quality and completeness of the input data. If the input data is incomplete or inaccurate, the results of the simulation may also be inaccurate. These limitations will be discussed in the main body of work(Wang, Liu and Shi, 2010)

2 Validation Study

Validation studies are important for several reasons. They help ensure the accuracy of the results and help identify any issues with the simulation process. By comparing the results obtained from a CAE simulation with experimental data or analytical data, it is possible to determine the accuracy of the simulation results. This helps to ensure that the results can be trusted and relied upon. If results don't match or they have a large percentage error difference, it may indicate that there are issues with the process, such as incorrect boundary conditions, material properties, or modelling assumptions. Identifying these issues can help to improve the accuracy and reliability of the simulator.

Validation can help identify limitations of the simulation progress as well, allowing users to understand the conditions under which results are most accurate and reliable. If it demonstrates that it can simulate studies accurately and reliably it can also help build confidence in the software used.

In order to validate the suitability of both SolidWorks and ANSYS Workbench, simple test cases will be utilised to find analytical solution which will then be used to measure the accuracy of the CAE results.

Example 0042 from Dlubal(*Structural Analysis Software | Dlubal*, no date) will be used to validate the static structural study simulated in Solidworks and Ansys. The example defines a cantilever beam that is fully fixed on one end and is loaded by a transversal and an axial force on the opposite end. Table 1 shows the parameters which were used to design the cantilever beam and the material properties.

Table 1- Material Data

Material	Steel	Modulus of Elasticity	E	210000.000 MPa
		Poisson’s Ratio	ν	0.300
Geometry	Cantilever	Length	L	10.000 m
		Cross-section width	w	0.080 m
Load		Axial Force	F_x	1.600 kN

For this example, the axial deformation is calculated using well-known formulae.

$$u_{x,max} = \frac{F_x L}{EA}$$

$$A = 80 \text{ mm} * 80 \text{ mm} = 6400$$

$$u_{x,max} = \frac{1600 \cdot 10000}{210 \times 10^3 \cdot 6400} = 0.011904$$

$$\therefore u_{x,max} = 0.012 \text{ mm}$$

Then, the cantilever beam deformation was simulated in both ANSYS and SolidWorks. Figure 1 and 2 show the results, respectively

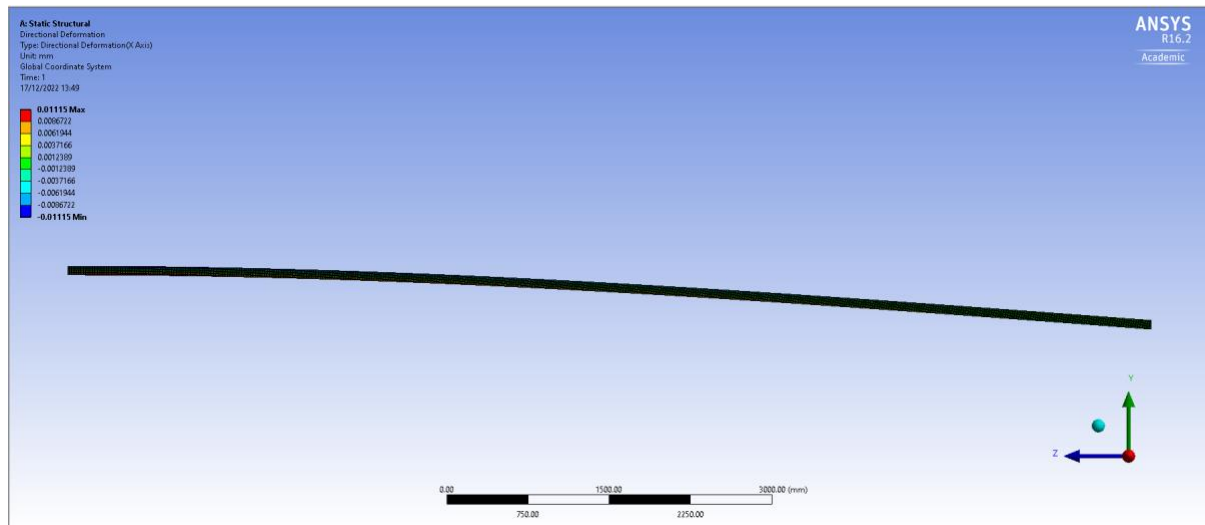


Figure 1 - Ansys Deformation

As shown in Figure 1 , the deformation of the cantilever beam was 0.01115 mm, whilst the analytical solution showed it should have 0.012 mm of deformation. There is a 0.0001 mm difference between the analytical solution and the simulation, that equates to a percentage error of 8.33%.

However, the deformation figure shown in Figure 2 is 0.009907 which has a percentage error of 17.5 %.

It is worth noting that SolidWorks exaggerates the deformation a lot more than ANSYS does, in order to be able to observe the changes visually. Otherwise, such small changes would not be visible.

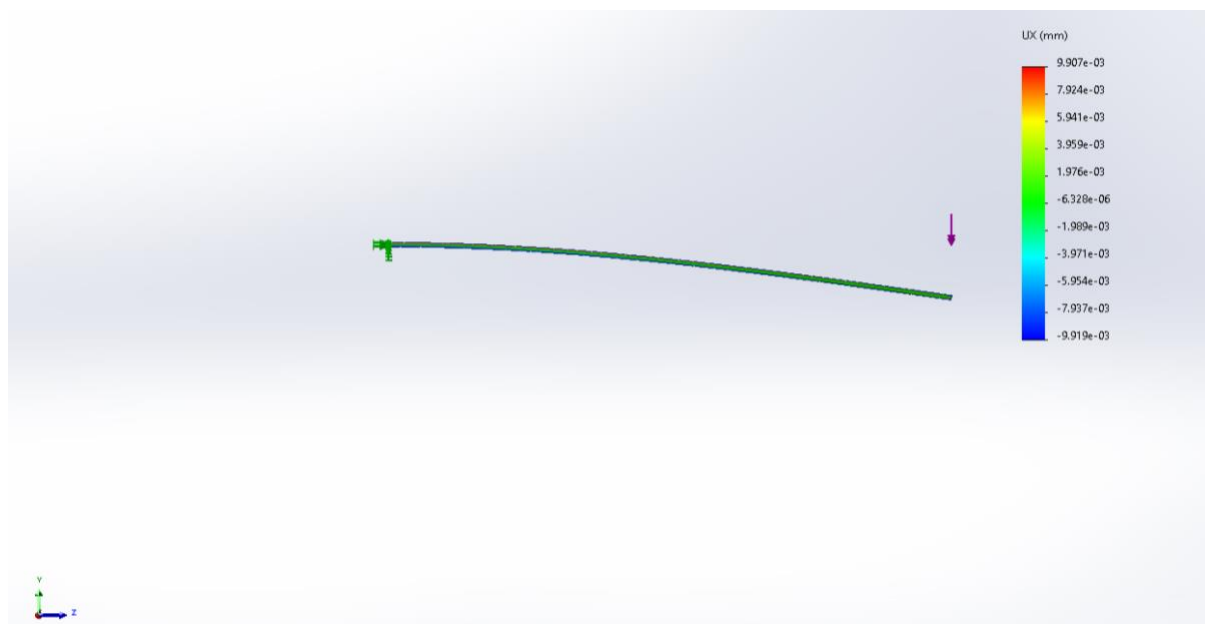


Figure 2 - Solidworks Deformation

It is difficult to make a general statement about which software is better for structural analysis, as the choice of software depends on the specific requirements and needs of the user. Both ANSYS Workbench and SolidWorks have their own strengths and capabilities, and it is important to consider the specific requirements of each project in order to choose the software that is best suited for the required results.

That being said, ANSYS Workbench is a powerful finite element analysis (FEA) software that is widely used for structural analysis, and it is often considered to be a leader in the field. It offers a wide range of features and capabilities for structural analysis, including the ability to analyze complex and nonlinear systems, and it is highly customizable, allowing users to tailor the software to their specific needs. (Mereuta, 2018; Pachpande, Patil and Student, 2019)

SolidWorks, on the other hand, is a 3D CAD software that is primarily used for design and engineering, and it offers a range of tools for structural analysis. However, it may not have the same level of advanced features and capabilities for structural analysis as ANSYS Workbench. (*Simulation and Analysis of Thermo-Mechanical Coupling Load and Mechanical Dynamic Load for a Piston* | IEEE Conference Publication | IEEE Xplore, no date)

As seen in the examples above, whilst a small difference between the results, there is a much greater percentage error in the simulation carried out by SolidWorks compared to Ansys. There could be a multitude of reasons as to why, including discretisation error, boundary conditions and even material properties. In order to obtain the most accurate and reliable results, it has been determined that ANSYS should be utilized for the purposes of this investigation. This decision was made based on the software's demonstrated ability to produce more accurate and precise results in structural analysis scenarios. As an example on how the two softwares treat designs differently, Figures 3 and 4 show the meshing on Ansys and Solidworks. Ansys used hexahedral mesh whereas SolidWorks used Tetrahedral.

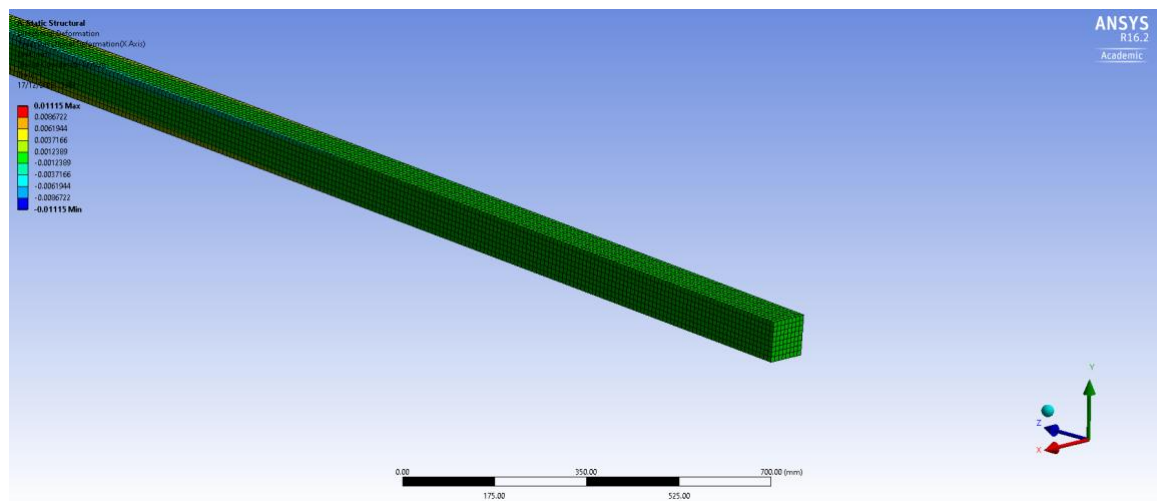


Figure 3- Ansys Mesh

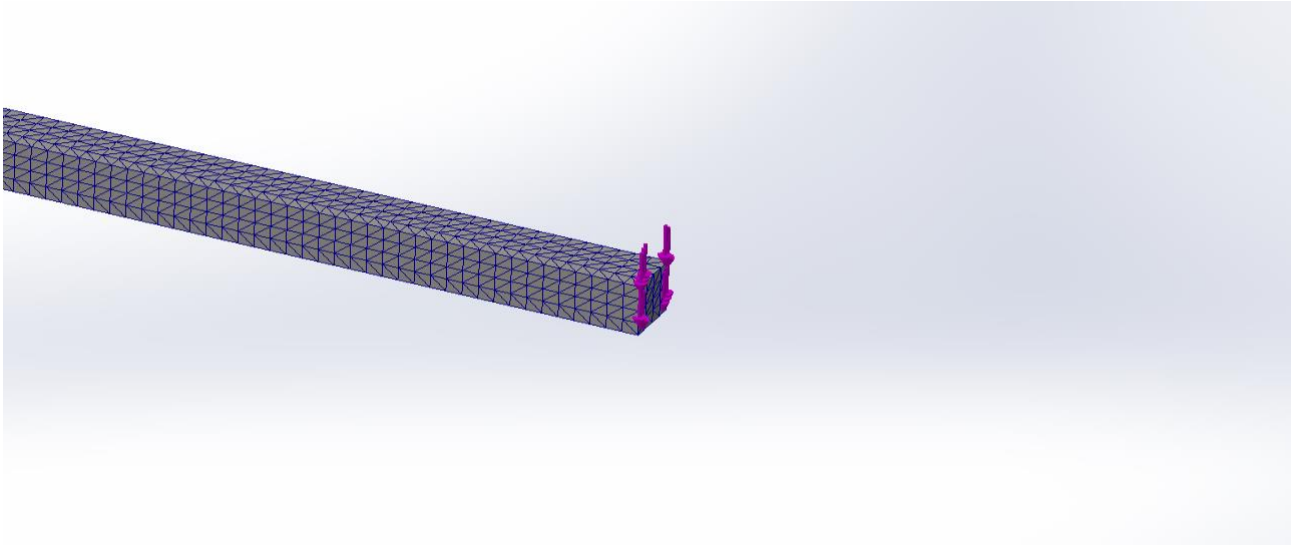


Figure 4 - Solidworks Mesh

Usually, tetrahedral meshes are good for more complicated geometries, however as this was a simple geometry, tetrahedral mesh wasn't required.

A secondary example was used to investigate the deflection of a bimetallic strip. Using the parameters supplied by Dlubal (Example 0078)(*Structural Analysis Software | Dlubal*, no date), a bimetallic strip, composed of copper and invar, was created and tested in Ansys. Similar to the cantilever beam, the bimetallic strip was fixed on one end with the other left free of any fixtures. The strip was loaded by the temperature difference shown in Table 2

Table 2 Material Properties

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 x 10 ⁻⁶ °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 x 10 ⁻⁵ °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 °C

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

$$K_1 = 4 + 6 \frac{1}{1} + 4 \left(\frac{1}{1} \right)^2 + \frac{130000}{137000} \left(\frac{1}{1} \right)^3 + \frac{137000 \cdot 1}{130000 \cdot 1}$$

$$K_1 = 4 + 6 + 4 + \frac{130}{137} + \frac{137}{130}$$

$$K_1 = \frac{285009}{17810} \approx 16.00275 \dots$$

$$u_{z,max} = \frac{6(\alpha_i - \alpha_c)(t_i + t_c)T_c L^2}{t_s^2 K_1} \frac{1}{2}$$

$$u_{z,max} = \frac{6(1.2 \times 10^{-6} - 2 \times 10^{-5})(1 + 1)100}{1^2 \cdot \frac{285009}{17810}} \cdot \frac{100^2}{2}$$

$$u_{z,max} \approx -7.049 \text{ mm}$$

For this example, only Ansys was used for validation against the analytical result. That is due to the aforementioned reasons above.

Figure 5 shows the deformation caused to the bimetallic strip by the thermal loading. As seen in Figure 5, the strip was deformed by 7.1203 mm, which is 1% more than in the analytical solution. This not only reinforces that Ansys is an appropriate tool to validate studies, but it also shows just how closely the results are.

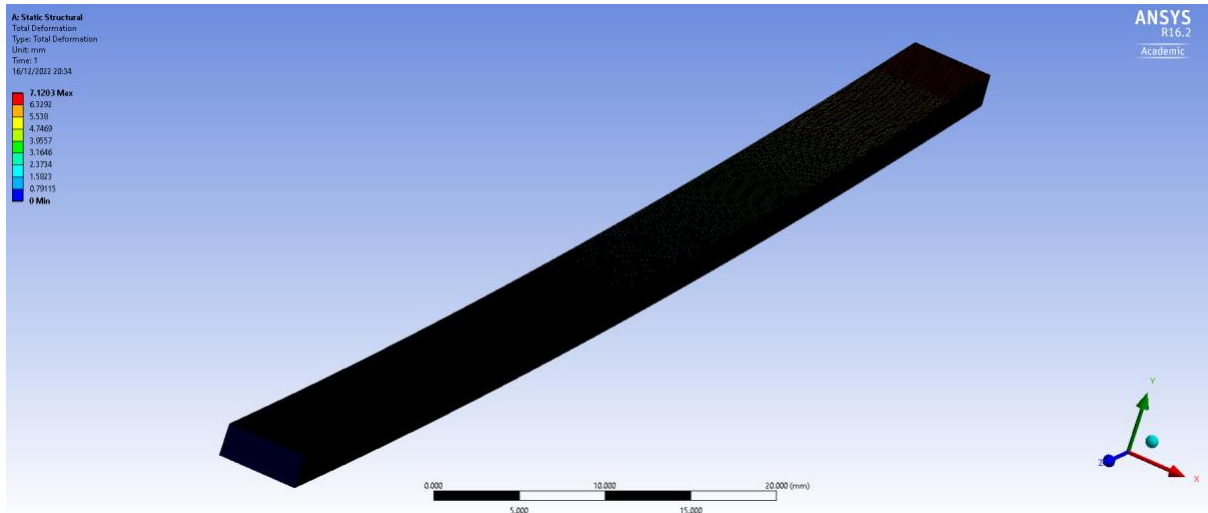


Figure 3- Bimetallic Strip Deformation

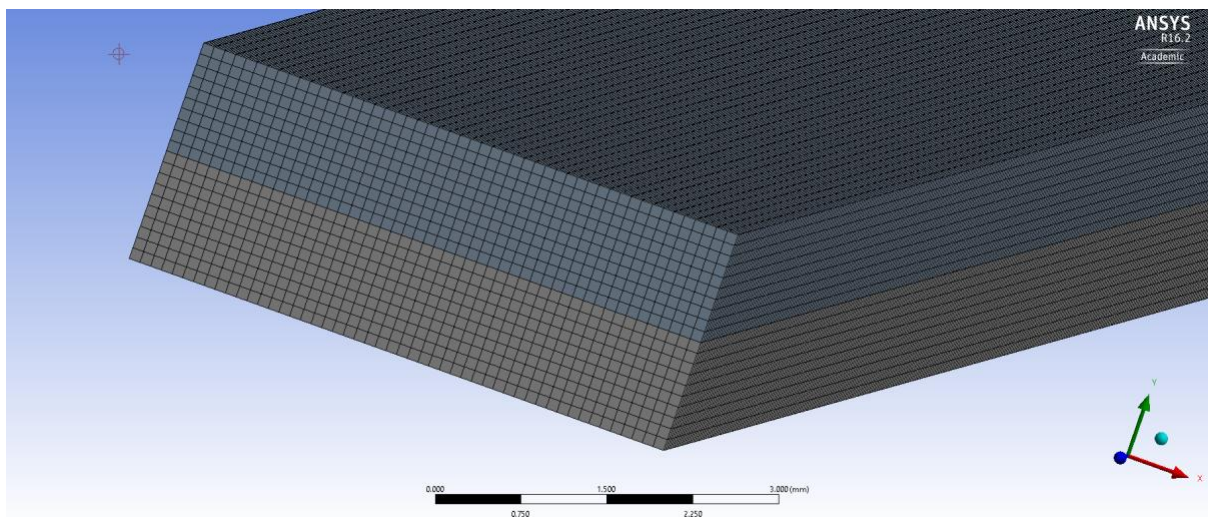
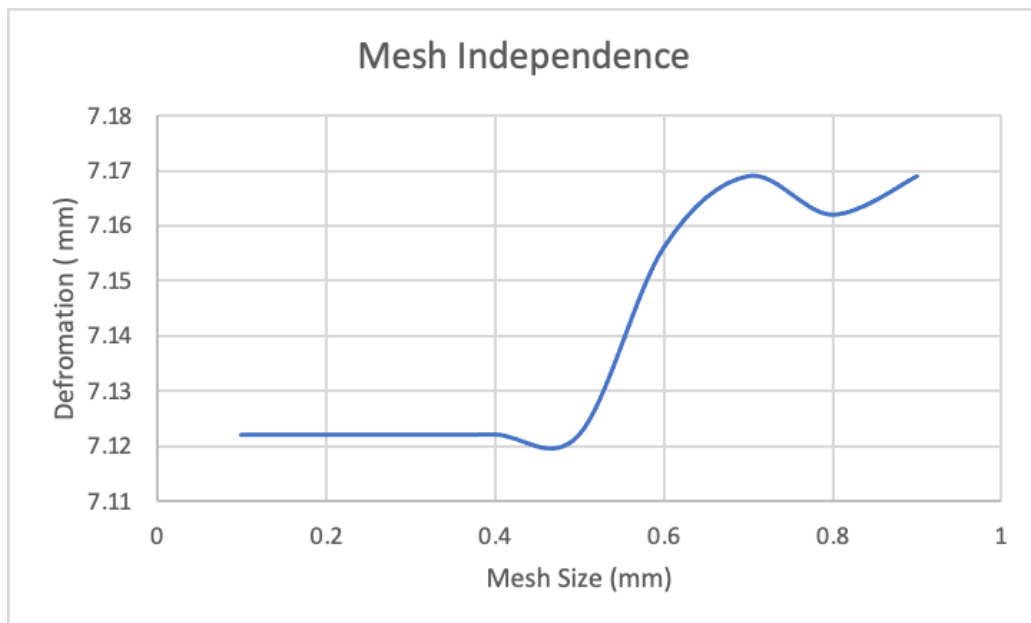


Figure 4- Bimetallic Strip Mesh

Another study that has been carried out is a mesh independence study, that is to ensure that results aren't affected by a mesh that's too small or too large. Graph 1 shows the mesh sizing plotted against the results of the bimetallic strip. It is obvious to see that a mesh larger than 0.7mm affects the results greatly. However, a mesh that is too fine could also mean that the computation time increases more and more and the difference in results would not change. Here it's seen that the results plateau from 0.5 to 0.1mm and it was decided that at that point, the results won't be affected by the mesh anymore.



Graph 1 - Mesh Independence Study

3 Structural Analysis

After validating the FEA process, an assembly, including a piston, crankshaft, connecting rod and gudgeon pin, was chosen and a kinematic, thermal and structural analysis was carried out.

Figure 7 shows the assembly created in CAD that was used to create the simulations mentioned above.

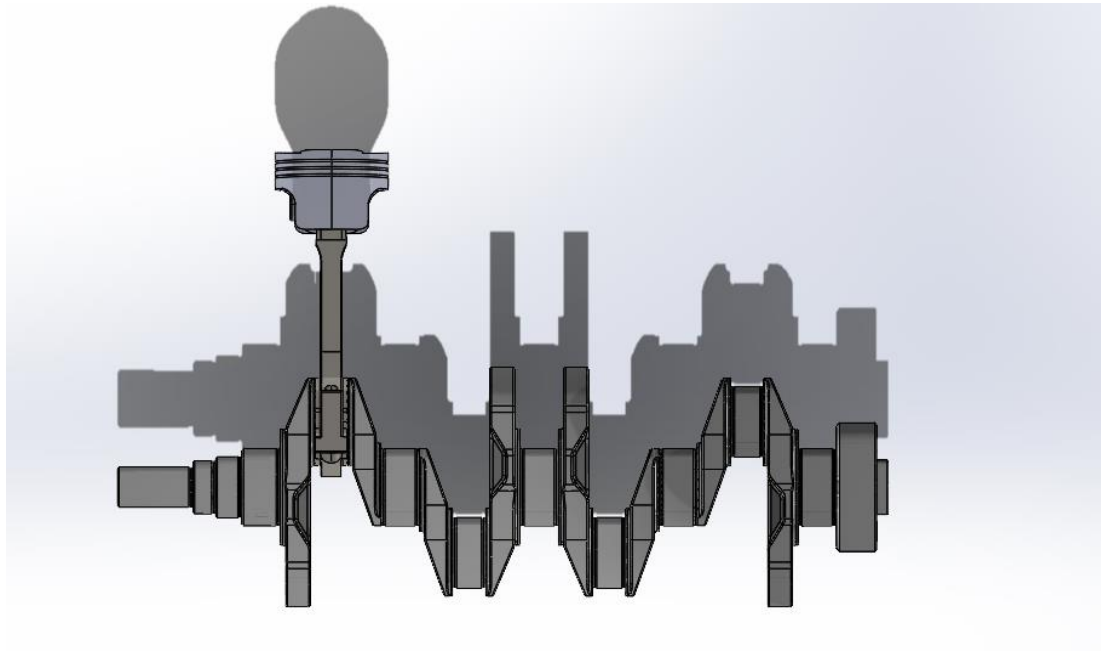


Figure 5- CAD Model

In order to make the results of the structural analysis as realistic as possible, realistic boundary conditions were used in the study along with realistic properties of the materials used.

The materials used in this analysis are 6061-T6 Aluminium for the piston head, and all the other elements were left as forged steel as there was no investigation carried out on them and they did not affect the results of the analysis.

The properties of 6061-T6 Aluminium are shown in Table 3, and the Young Modulus temperature dependant values in Table 4.

Table 3- Aluminium Properties

Poisson's Ratio	0.33
Coefficient of Thermal Expansion (22°C)	2.36E-05 1/°C
Thermal Conductivity	167 W/m-K
Specific Heat Capacity	0.896 J/g- °C

Table 4- Young Modulus Temperature Dependent Values

Temperature °C	Young's Modulus Value (GPa)
22°C	67 GPa
100°C	70 GPa
150°C	68 GPa
200°C	64 GPa
250°C	56 GPa
300°C	48 GPa
350°C	41 GPa
400°C	32 GPa
450°C	21 GPa
500°C	12 GPa

These were added onto Ansys's Engineering data channel and used throughout the studies as the material for the piston.

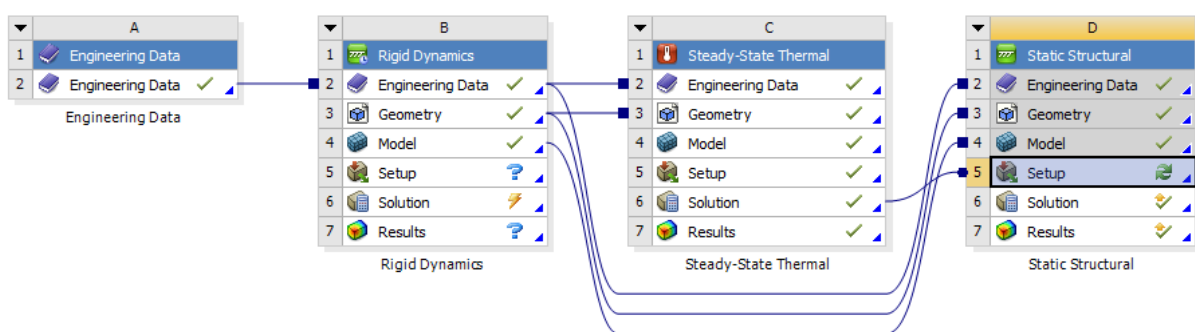


Figure 6 - Ansys Simulation Architecture

The architecture of how the structural analysis simulation was built can be seen in Figure 8. The material data was input into 'Engineering Data' and then shared across all the different streams.

The 'Rigid Dynamics' section was where the kinematics study was carried out. This was used to create a realistic set of loading conditions which could then be used in the structural study.

It was in this study where all the constraints of the system were applied:

- A revolute joint between the crankshaft and connecting rod
- A revolute joint between the connecting rod and the gudgeon pin

- A revolute joint between the gudgeon pin and the piston
- A revolute joint between the 'ground' and the crankshaft (to keep it in position)
- A general joint between the piston and 'ground to only move in the Z direction.

A mesh is not required in this stage as it is only a kinematics study, and no actual forces are being applied to the piston. Then a rotational velocity of 4000 RPM is applied to the crankshaft and the motion is simulated

The results show in Figure 9 show the force loads that the piston is experiencing at 4000 RPM. The maximum force load is then found and exported in order to apply it into the structural study. The reason why the maximum force load is exported is because then the 'worst case scenario' can be loaded into the structural study and investigated. If, for example, a fatigue study was done then at this stage both the maximum and minimum loadings would be exported for two different studies, so the cyclic loadings on the component can be seen.

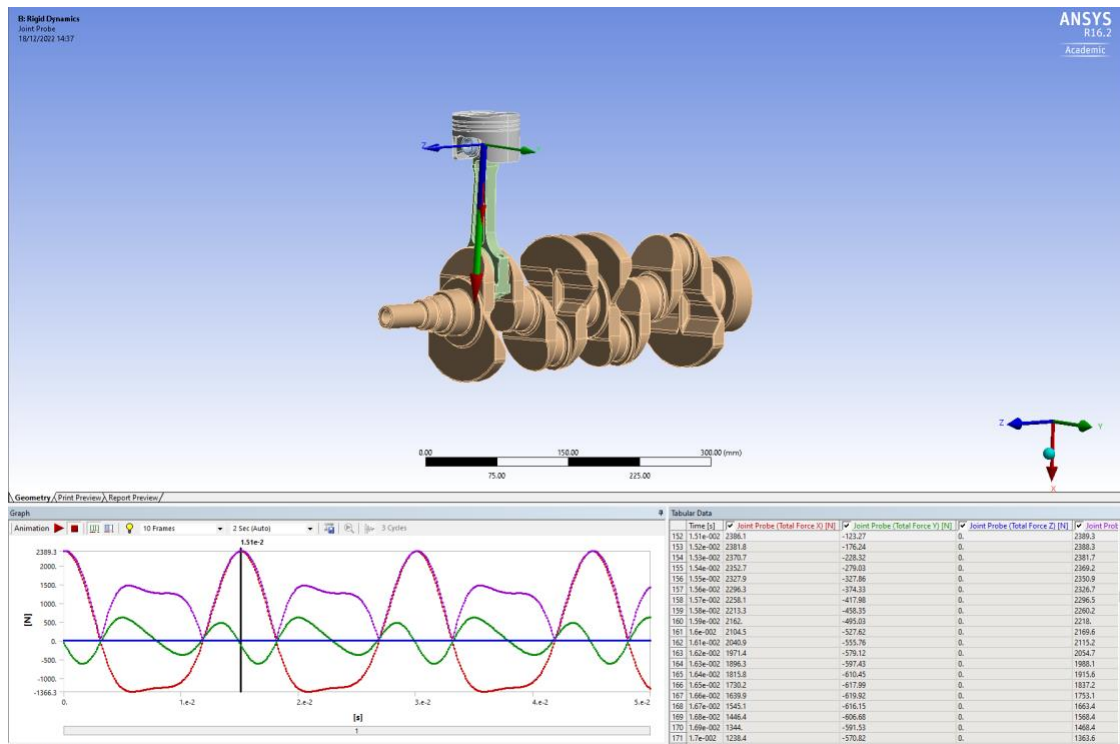


Figure 7 - Kinematics Study

After the kinematics study has been finished, the geometry and engineering data from the 'Rigid Dynamics' section was imported into a 'Steady-State Thermal' study. Here, a temperature profile is created for the piston using the boundary conditions found in Table 5.

Table 5- Boundary Conditions

Part	Heat Transfer Coefficient (W/m ² K)	Temperature (°C)
Crown	497.5	928.5
Crown Side	447.5	425
Top Groove	437.5	190
Middle Groove	437.5	190
Oil Groove	338.75	163.75
Underside	675	91.25
Skirt	462.5	107.5

Then, the system was solved, and a temperature profile was created for the piston. These can be seen in Figures 10 and 11.

It's clear to see that the temperature is greatest at the top of the piston where combustion happens and then the temperature gradually decreasing as it travels through the piston.

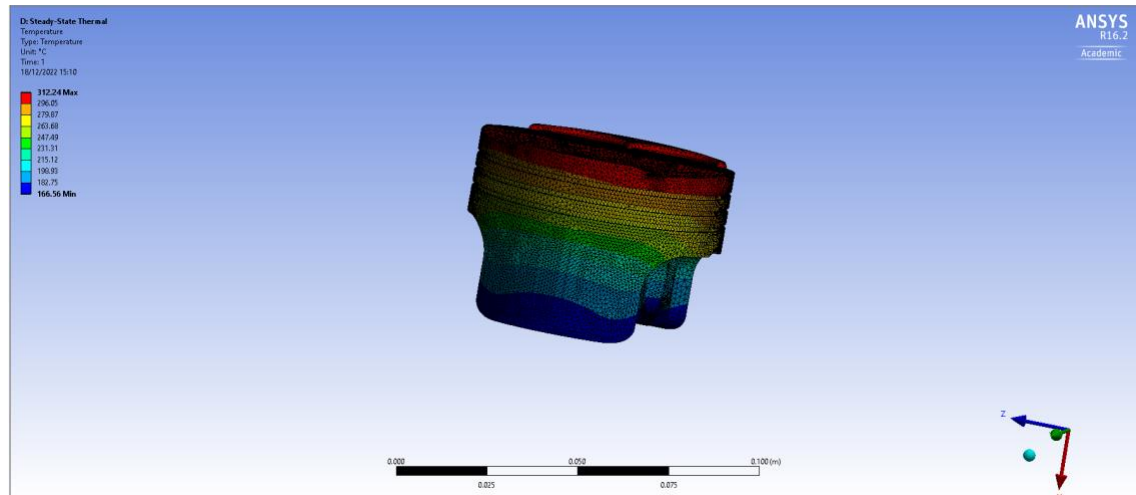


Figure 8 - Temperature Profile

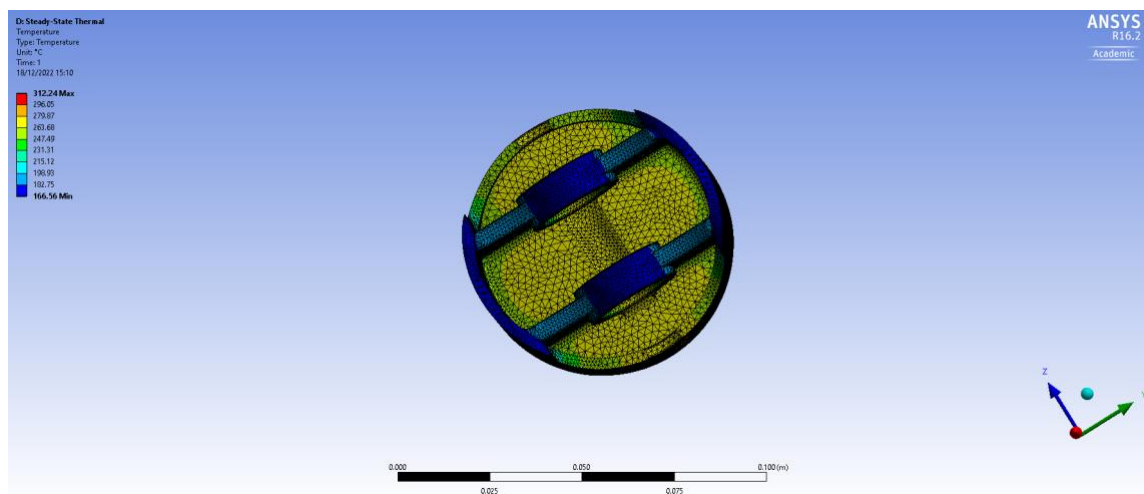


Figure 9 - Temperature Profile

The temperature profile was then linked into the 'Static Structural' study and is used as a boundary condition to determine how force loads affect the structure when it's heated.

Here, the force loads, and the thermal study are applied on the piston along with a pressure to replicate combustion.

Figures 12 and 13 show the temperature profile on the piston along with the total deformation caused by the input loads and temperature. It's clear to see where the highest temperature is, is where combustion happens, then as the temperature travels through the piston it cools down and the deformation is less. Figure 14 shows the total stresses on the piston, again, the highest being on the crown due to combustion

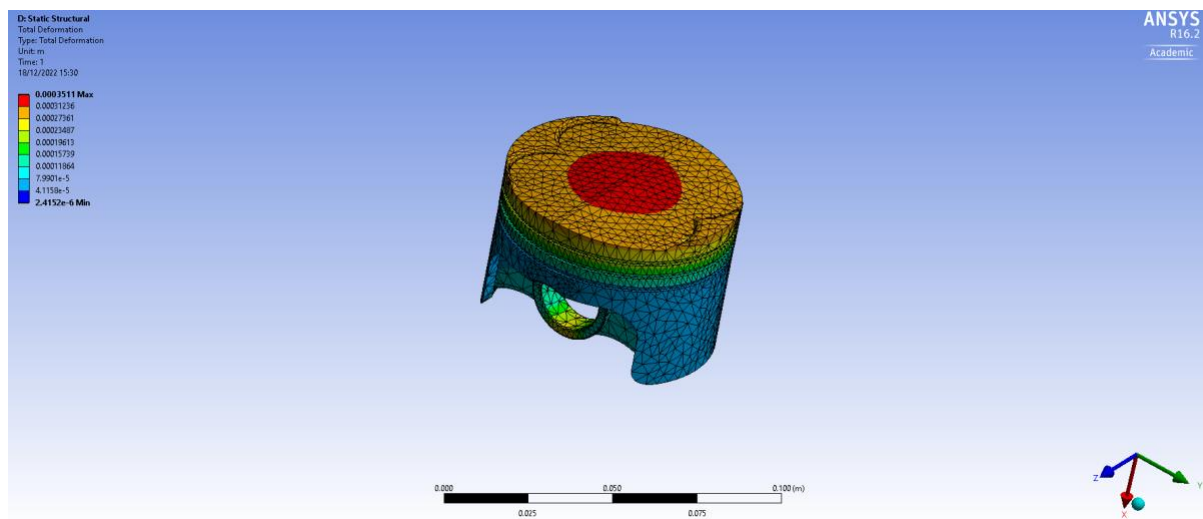


Figure 10 -Total Deformation

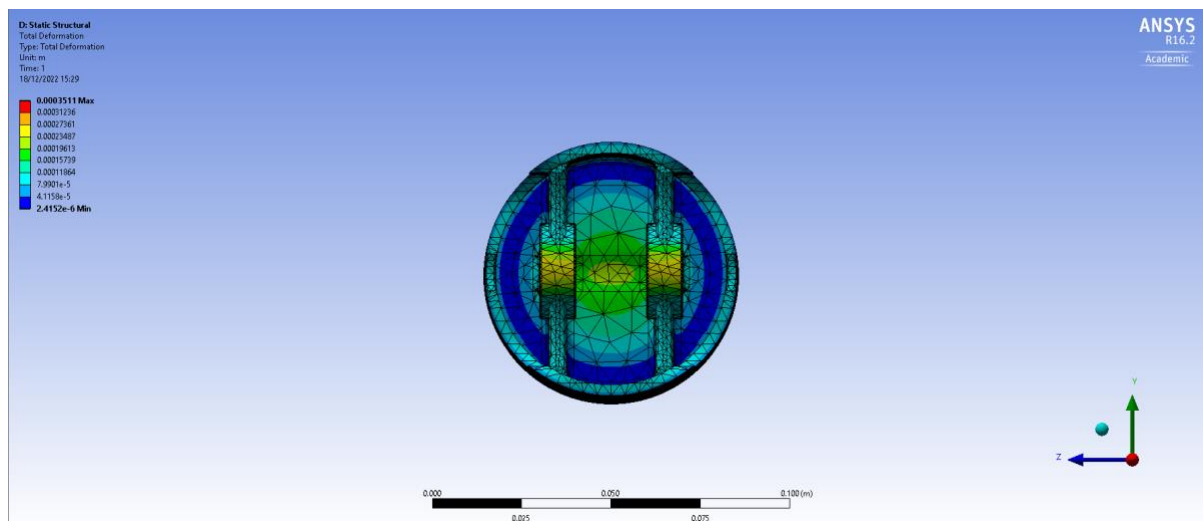


Figure 11 – Total Deformation Underside

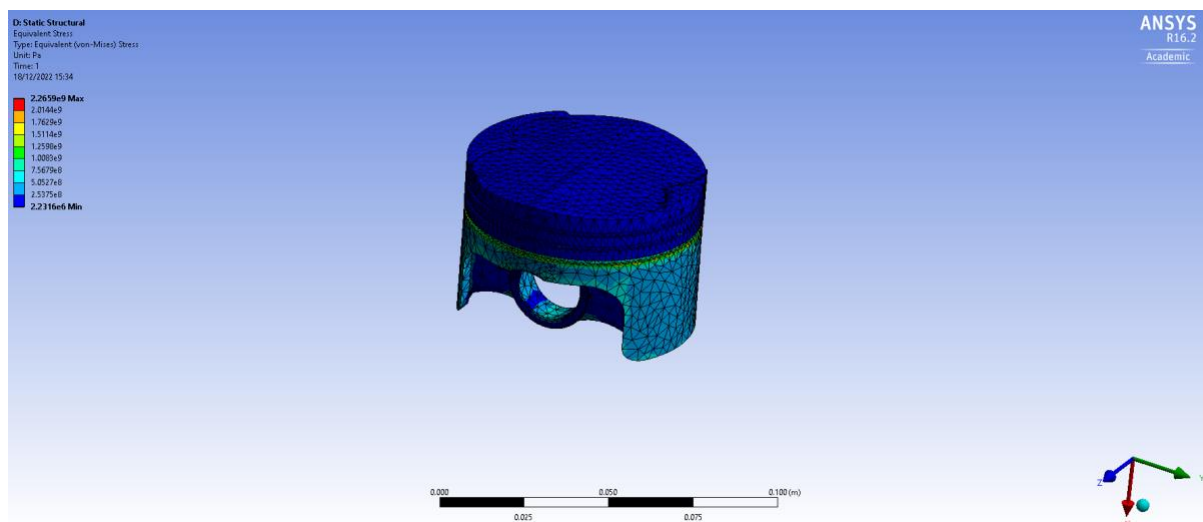


Figure 12 - Total Stresses

3.1 Design Improvement

One way in which this design could be improved upon would be to simply use a different material. The deformation and stresses on the piston depend not only on the loads that are being applied onto but also on the type of material used.

A different material, like 7075- T6 Aluminium, that has a higher tensile stress than 6061- T6 could see less deformation than 6061-T6. It could possibly withstand fatigue stress much better than 6061-T6 as well.

Another area of improvement would be on the piston's crown, as seen in Figure 14, the total load on the crown is lower than those on the side skirt.

4 Conclusion

Computer-aided engineering (CAE) and finite element analysis (FEA) simulations are useful because they allow engineers to analyse and optimize the performance of a product or system before it is physically built or tested. One of the main benefits of CAE and FEA simulations is that they allow engineers to analyse the behaviour of a product or system under various conditions, including load, stress, temperature, and vibration. This can help design products that are stronger, more durable, and more reliable. Another benefit is that they help optimize the design of a product or system for specific performance criteria, such as weight, cost, or energy efficiency.

This can save not only time but money by identifying and fixing potential problems early in the design process.

Despite the many benefits of CAE and FEA simulations, there are several limitations, including the need for accurate input data, the limitations of the computer hardware and software used to perform the simulations, and the complexity of the system being analysed.

In addition, these simulations are only as accurate as the assumptions and approximations that are made in the analysis and may not accurately predict the behaviour of a system under all conditions. If the input data is incomplete or inaccurate, the simulation results may not be reliable.

These simulations can be time-consuming and computationally intensive, especially for large or complex systems.

In summary, CAE and FEA simulations are useful tools for analysing and optimizing the performance of products and systems, but they are not a substitute for physical testing. They can help identify and fix problems early in the design process, but they are not always accurate and may require additional validation through physical testing.

5 References

'DESIGN AND ANALYSIS OF THE PISTON BY USING FIVE DIFFERENT MATERIALS 1 PAGADALA SIDDIRAJU 2 KOPPULA VENKATESWARAREDDY' (no date). Available at: <https://doi.org/10.51319/2456-0774.2021.2.0012>.

Mereuta, V. (2018) 'Static and Thermal Analysis of Piston using FEM Analysis', *International Journal for Research in Applied Science and Engineering Technology*, 6(1), pp. 201–206. Available at: <https://doi.org/10.22214/IJRASET.2018.1032>.

Pachpande, L.R., Patil, R.Y. and Student, P.G. (2019) 'Review on Design and Comparative Analysis of Piston by Ansys'. Available at: <http://ijesc.org/> (Accessed: 18 December 2022).

(PDF) *DESIGN AND ANALYSIS OF PISTON WITH VARIOUS MATERIALS USING SOLIDWORKS AND ANSYS SOFTWARE* (no date). Available at: https://www.researchgate.net/publication/353480940_DESIGN_AND_ANALYSIS_OF_PISTON_WITH_VARIOUS_MATERIALS_USING_SOLIDWORKS_AND_ANSYS_SOFTWARE (Accessed: 18 December 2022).

Simulation and Analysis of Thermo-Mechanical Coupling Load and Mechanical Dynamic Load for a Piston / IEEE Conference Publication / IEEE Xplore (no date). Available at: <https://ieeexplore.ieee.org/abstract/document/5421507/metrics#metrics> (Accessed: 18 December 2022).

Structural Analysis Software / Dlubal (no date). Available at: <https://www.dlubal.com/en> (Accessed: 18 December 2022).

'Thermal And Static Structural Analysis On Piston' (no date).

Wang, Y., Liu, Y. and Shi, H. (2010) 'Simulation and analysis of thermo-mechanical coupling load and mechanical dynamic load for a piston', *ICCMS 2010 - 2010 International Conference on Computer Modeling and Simulation*, 4, pp. 106–110. Available at: <https://doi.org/10.1109/ICCMS.2010.131>.