

BEng Motorsport Engineering

Application of Computer Aided Engineering Processes

<i>Student Name</i>	Robin Thomas
<i>Student Number</i>	2306069
<i>Module Name</i>	Structural Design & Fluid Analysis
<i>Module Code</i>	ACHA6010
<i>Date</i>	17/12/23

School of Engineering
University of Wales Trinity St. David
2023-2024

Table of Contents

Table of Figures	2
Introduction.....	3
1: Finite Element Analysis as a Tool for Design.....	4
1.1: Principles of FEA	4
1.2: FEA Validations	4
1.3: Advantages of FEA.....	4
1.4: Limitations of FEA	5
2: ANSYS Validations.....	6
2.1: Continuous Load Cantilever Beam Validation.....	6
2.1.1: Analytical Derivation	6
2.1.2: ANSYS Validation	7
2.2: Bimetallic Strip Validation.....	9
2.2.1: Analytical Derivation	9
2.2.2: ANSYS Validation	10
3: Rigid Dynamics, Structural, and Thermal Engine Analysis	12
3.1: Geometry.....	12
3.2: Materials	12
3.3: Results and Analysis.....	13
3.3.1: Rigid Dynamics Study.....	13
3.3.2: Steady-State Thermal Study	13
3.3.3: Static Structural Study.....	15
4: Performance Optimisation	17
4.1: Crankshaft Balancing.....	17
4.1.1: Geometry	17
4.1.2: Results and Analysis.....	18
Conclusion	20
References.....	21

Table of Figures

Figure 1 – Continuous Load Beam Geometry	7
Figure 2 - Continuous Load Beam ANSYS Conditions	7
Figure 3 - Continuous Load Cantilever Beam Deflection (0.020m)	8
Figure 4 – Continuous Load Beam Mesh Independence Study	8
Figure 5 – Bimetallic Strip Geometry	10
Figure 6 - Bimetallic Strip ANSYS Conditions	10
Figure 7 – Bimetallic Strip Deflection (0.00025m).....	11
Figure 8 - Bimetallic Strip Mesh Independence Study.....	11
Figure 9 - Geometry for the one cylinder engine	12
Figure 10 - Force acting on the piston/gudgeon pin joint throughout 0.05 seconds of run time at 4000RPM.....	13
Figure 11 - Mesh of the piston for the thermal and structural studies	14
Figure 12 - Steady-state thermal conditions input.....	14
Figure 13 - Thermal study results.....	15
Figure 14 - Total Deformation - Isometric and Underside views	15
Figure 15 - Von-Mises Stress - Isometric and Underside Views	16
Figure 16 - Geometry comparison showing the centre of mass and the cut material	17
Figure 17 - Geometry comparison showing the shift in centre of mass	17
Figure 18 - Force acting on the crankshaft/conrod joint throughout 0.05 seconds of run time at 4000RPM	18
Figure 19 - Force acting on the crankshaft/conrod joint throughout 0.05 seconds of run time at 4000RPM with the updated crankshaft geometry	18
Figure 20 - Deformation compared between the original (L) and modified (R) crankshafts.....	19
Figure 21 – Von-Mises Stress compared between the original (L) modified (R) crankshafts.....	19

Introduction

In the following report, the use of Computer Aided Engineering will be discussed and investigated, supported by the conduction of a series of tests in FEA studies. CAE and FEA's use as a tool for design, as well as its advantages and limitations, will be considered, and two validations will also be carried out to this end. Furthermore, a connected study of rigid dynamics, steady-state thermal, and static structural will be carried out on an engine piston assembly, before possible improvement to the crankshaft is investigated using the FEA results and the rapid design and testing benefits of the CAE process.

1: Finite Element Analysis as a Tool for Design

1.1: Principles of FEA

Finite element analysis is a process that simulates physical phenomena using the Finite Element Method (FEM), which is a mathematical method of predicting how geometry will react when subjected to various physical conditions (University of Victoria, 2015). The method divides an engineering problem into a finite number of elements by discretising the problem (Bhavikatti, 2005). The properties of each discrete element can then be calculated, the results of which are visualised by the FEA software, usually via a colour scale (Simscale, 2022). This means that the accuracy of the result of the simulation is affected by the quality of the mesh used for the test. It is important to use a mesh fine enough to ensure an accurate simulation (Bhavikatti, 2005), however not so fine that computing time is wasted or inaccuracy is introduced due to compounding mathematical inaccuracies within the simulation due to idealised conditions (Mit.edu, 2017).

1.2: FEA Validation and Verification

In order to verify the results of CAE programs, standard validations are used. These are simple problems used to compare the results with an analytical solution, in order to determine the error within the method and to easily analyse the effects of parameter changes with a relatively fast solve time, owing to the simplicity of the problem. Validations help to increase the integrity and reliability of the CAE software itself and allow for greater trust in the end result of a CAE simulation (Srinivas, 2020).

For CAE problems, mesh studies are also conducted, for a similar purpose as validations – to ensure the validity of the generated result. Studies are ran at different mesh densities in order to ascertain the effect of the change of mesh density on the solution of the run. When the study has reached mesh independence, it means that the fineness of the mesh is independent of the results of the study (Kelley, et al., 2020), which results in greater accuracy (Xu, Li, & Lin, 2019). These principles are detailed further in section

2: ANSYS Validations.

1.3: Advantages of FEA

FEA software improves the design process by quickening the prototyping and optimisation phase of product design, by reducing the number of required physical prototypes and tests. It brings the testing phase of product design in far earlier than conventional design and prototyping, and so allows for less investment to be necessary in designs that are ultimately found to not be viable (Manor Tool & Manufacturing Co., 2019). FEA processes are mainly used for improving product safety, testing real-world conditions and stresses on designs, and using this data to therefore evaluate and optimise designs and materials. This has the effect of reducing the cost and time of production (Simscale, 2022) (Sieminski, 2001).

FEA is often used where analytical analysis is unfeasible due to the complexity of the geometry or conditions being tested. To this end, modern FEA also allows the testing of entire systems of components working in conjunction with each other, as well as with a variety of conditions, a selection of which are demonstrated below in section 3: Rigid Dynamics, Structural, and Thermal Engine Analysis. This further reduces development costs and streamlines the process. The breadth of parameters that FEA processes can solve for also makes FEA universally applicable to a range of industries and scenarios, making it an invaluable tool for engineering design (Bhat, 2021).

Another main strength of using FEA is its data-driven iterative design capability. This entails using the resulting data from a simulation to inform the next iteration of a product, be it changes to geometry or material (Orr, 2012). An example of this process is given below in section 4: Performance Optimisation.

1.4: Limitations of FEA

One limitation of Computer Aided Engineering methods is the computational cost of highly complex simulations (Park, Chin, Cha, & Lim, 2022). Complex simulations require significantly more time and more powerful computers to run to completion, which can be infeasible in a scenario where deadlines must be met. It is for this reason that much research is carried out into the optimisation of the FEA process, to mitigate this limitation. One such method is to utilise adaptive meshing, which creates a coarse base mesh and re-meshes the areas of concern for the simulation, which significantly reduces solve time at a negligible cost to accuracy compared to a full detailed mesh (Suresh & Regalla, 2014). It is also for this reason that the thermal and structural analyses detailed below are steady state and static simulations, as while it is possible to run an analysis at each step of the transient study, it is a waste of resources to do so as parts are engineered to withstand the harshest conditions that they will be subjected to when in use (Zhou, Panetta, & Zorin, 2013). Misuse of FEA can also lead to misguided design decisions if too much trust is placed in them, leading to expensive redesigns later in the design process (SAE International, 2023). As FEA processes are, by their very nature, approximations, this can also

lead to inaccuracy and subsequent design errors if certain parameters are ignored or overlooked, either by mistake, for simplicity's sake, or to reduce solve time (Liu & Quek, 2014). This is especially common due to the continuing development of FEA software packages with more convenient user interfaces, which makes pulling data out of a simulation far easier than pulling accurate solution data for the same problem (Liu & Quek, 2014).

2: ANSYS Validations

2.1: Continuous Load Cantilever Beam Validation

The cantilever beam validation is carried out according to the VE 0016 Continuous Load validation from Dlubal Software (Dlubal Software, 2021).

2.1.1: Analytical Derivation

2.1.1.1: Material Properties

Modulus of Elasticity (E) = 210000MPa

Poisson's Ratio (ν) = 0.0

Shear Modulus (G) = 105000MPa

2.1.1.2: Beam Dimensions

Length (L) = 1m

Width (w) = 0.05m

Thickness (t) = 0.005m

2.1.1.3: Load

Pressure (p) = 2750Pa

2.1.1.4: Calculation

To calculate deflection, the following equation is used:

$$u_{z,max} = \frac{3pL^4}{2Ew^2t}$$

Substituting in the variables:

$$\frac{3 \times (2.75 \times 10^3) \times 1^4}{2 \times (210 \times 10^9) \times (0.05)^2 \times 0.005} = 1.57143 \times 10^3 \text{ m}$$

$$\therefore 157.143 \text{ mm } z_{max}$$

2.1.2: ANSYS Validation

The constant pressure beam problem was simulated in ANSYS to compare to the analytical solution and verify the CAE method's accuracy. A mesh independence study has also been conducted to further validate the CAE method, by proving that the result of the simulation was independent of mesh element size (Gardiner, 2017).

2.1.2.1: Parameters

The beam geometry was created in SolidWorks and the material, support, and force conditions were applied with the values specified above.

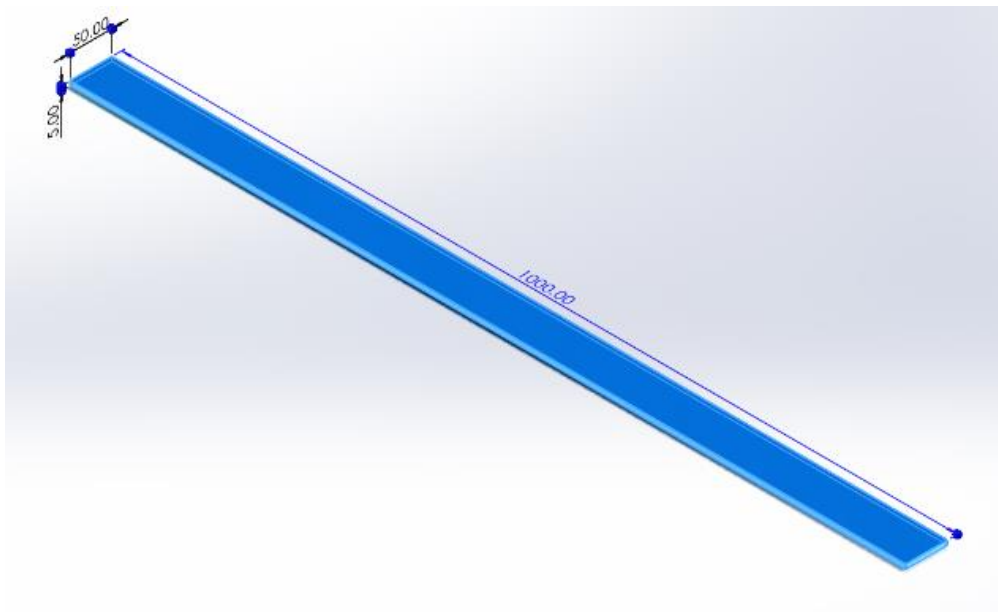


Figure 1 – Continuous Load Beam Geometry

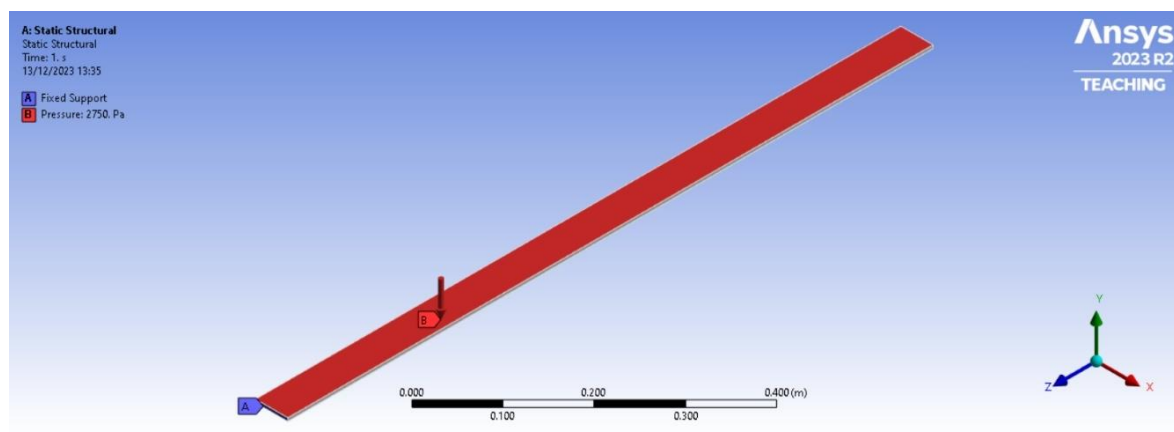


Figure 2 - Continuous Load Beam ANSYS Conditions

2.1.2.2: Validation

The simulation was initially carried out with an element size of $0.020m$, as in the standard validation (Dlubal Software, 2021).

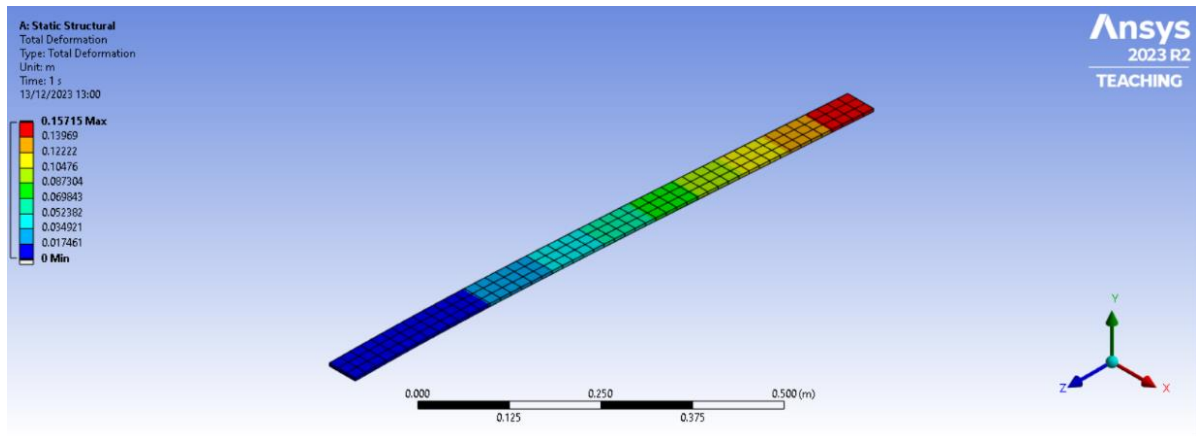


Figure 3 - Continuous Load Cantilever Beam Deflection (0.020m)

The maximum deflection of this simulation is 0.15715m, or 157.15mm. The analytical verification yielded a result of 157.143mm, which means the ANSYS simulation is inaccurate by a measure of $\frac{157.15}{157.143} - 1 = 0.0000445\%$.

2.1.2.3: Mesh Independence Study

A series of simulations were carried out at element sizes ranging from 0.01m - 0.04m.

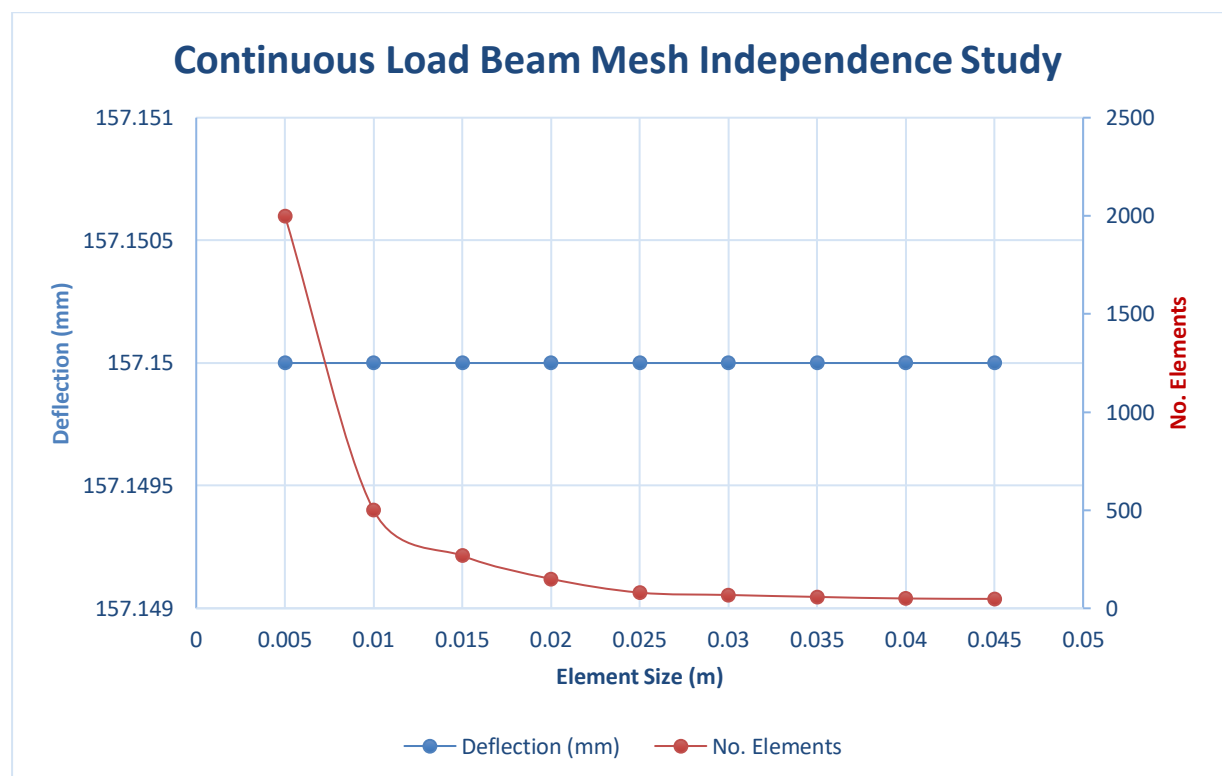


Figure 4 – Continuous Load Beam Mesh Independence Study

As can be seen from the graph, the element size, and therefore number of elements, has no effect on the deflection of the beam. This means the simulation has reached mesh independence.

2.2: Bimetallic Strip Validation

The bimetallic strip was analysed for deflection when subjected to a temperature increase due to differing coefficients of thermal expansion. It was conducted based on the VE 0078 Bimetallic Strip validation from Dlubal Software (Dlubal Software, 2017)

2.2.1: Analytical Derivation

2.2.1.1: Material Properties

Invar

Modulus of Elasticity (E_i) = 137000MPa

Poisson's Ratio (ν_i) = 0.280

Coefficient of Thermal Expansion (α_i) = $1.200 \times 10^{-6} \text{ }^\circ\text{C}^{-1}$

Copper

Modulus of Elasticity (E_c) = 130000MPa

Poisson's Ratio (ν_c) = 0.354

Coefficient of Thermal Expansion (α_c) = $2.000 \times 10^{-5} \text{ }^\circ\text{C}^{-1}$

2.2.1.2: Beam Dimensions

Length (L) = 100.00mm

Cross-section Width (w) = 5.00mm

Layer Thickness ($t_i = t_c = t$) = 1.000mm

2.2.1.3: Load

Thermal Loading (T_c) = 100°C

2.2.1.4: Calculation

To calculate deflection, the following equation is used:

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

This gives the absolute value: 7.049mm.

2.2.2: ANSYS Validation

The bimetallic strip problem was simulated in ANSYS to compare to the analytical solution and verify the CAE method's accuracy, as with the cantilever beam. A mesh independence study has also once again been carried out.

2.2.2.1: Parameters

The strip geometry was created in SolidWorks and the material, support, and force conditions were applied with the values specified above.

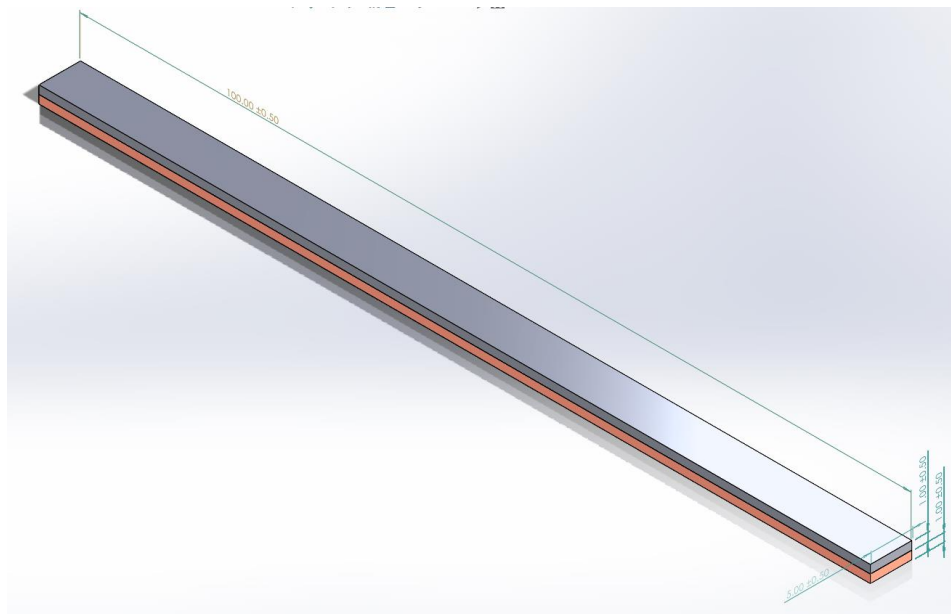


Figure 5 – Bimetallic Strip Geometry

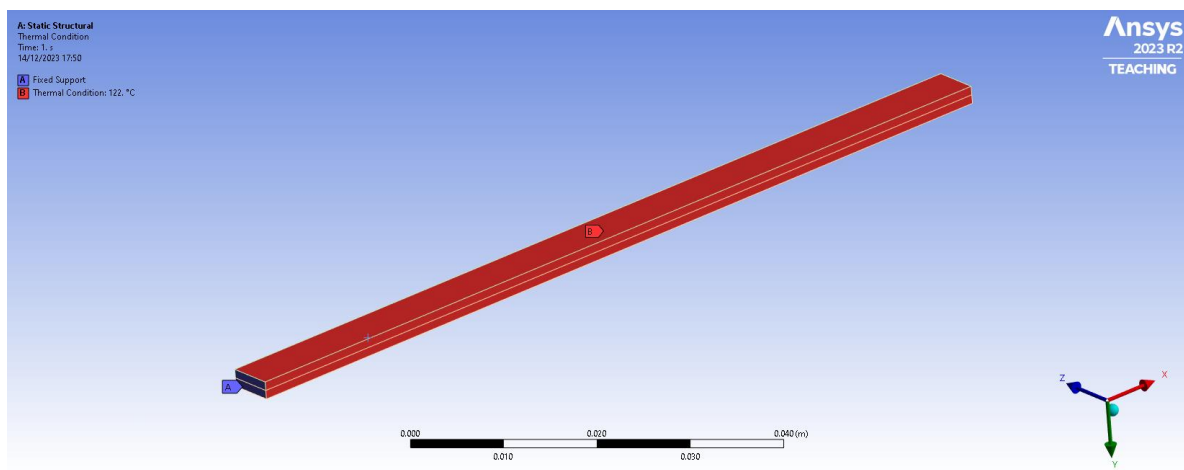


Figure 6 - Bimetallic Strip ANSYS Conditions

2.2.2.2: Validation

The simulation was initially carried out with an element size of $0.00025m$, as in the standard validation (Dlubal Software, 2017).

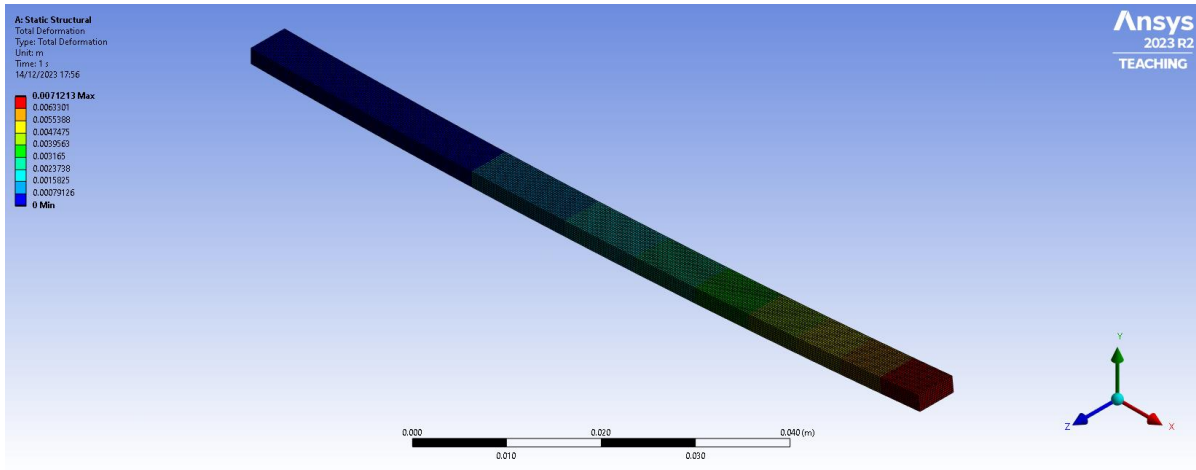


Figure 7 – Bimetallic Strip Deflection (0.00025m)

The maximum deflection of this simulation is 0.0071213m , or 7.1213mm . The analytical verification yielded a result of 7.049mm , which means the ANSYS simulation is inaccurate by a measure of $\frac{7.1213}{7.049} - 1 = 0.0102568\%$.

2.2.2.3: Mesh Independence Study

A series of simulations were carried out at element sizes ranging from 0.0001m - 0.004m .

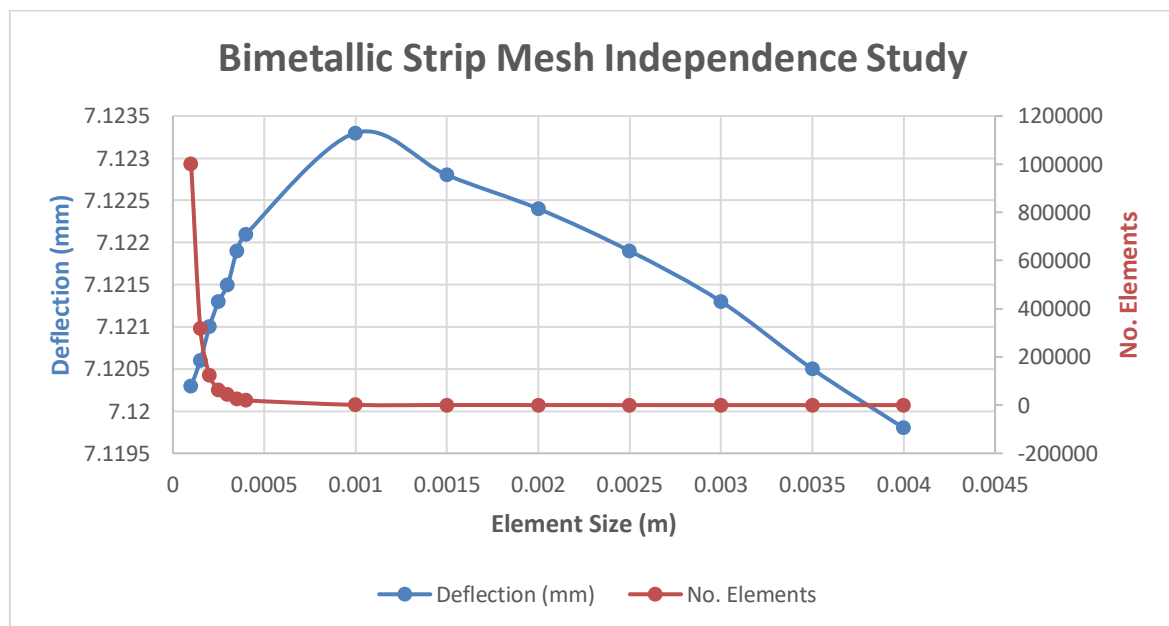


Figure 8 - Bimetallic Strip Mesh Independence Study

As can be seen from the above mesh independence study, the deflection results continues to vary with mesh quality. This means the study has not reached mesh independence, and therefore is not as accurate as the result from the continuous load validation. It should be noted that the results range from 7.1198mm to 7.1203mm , which results in an error range from 0.0100439% to 0.0101149% , which remains a very small margin of error compared to the analytical result.

3: Rigid Dynamics, Structural, and Thermal Engine Analysis

3.1: Geometry

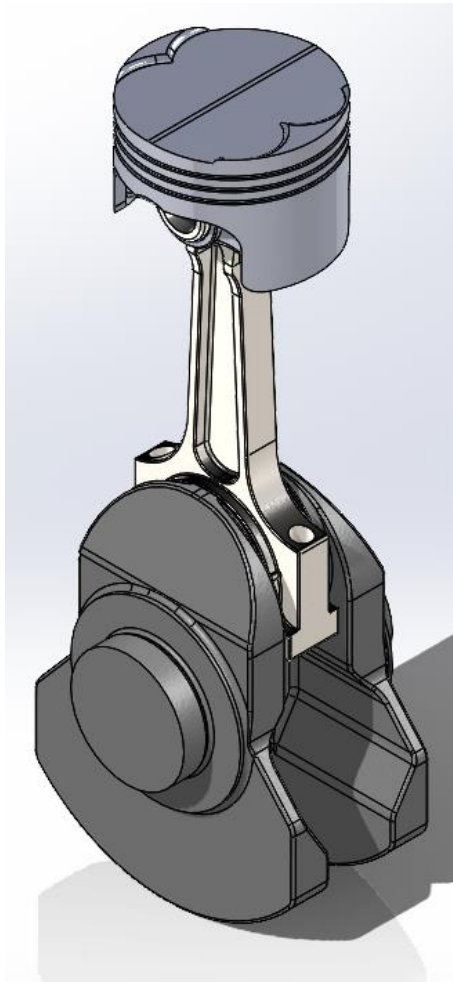


Figure 9 - Geometry for the one cylinder engine

The geometry represents a standard one cylinder engine, and includes the accurate modelling of a crankshaft, conrod, gudgeon pin, and piston.

3.2: Materials

The materials for each component were selected according to the surrounding literature on the selection of materials for engine internals. The piston material selected was 6061 T6 Aluminium alloy, due to its high strength, lower density than its steel contemporaries, and its high machinability and low thermal expansion, allowing for smaller tolerances and higher precision (Goyal & Nema, 2016). For the other three components, 42CrMo4 steel alloy was used, which is commonly used for crankshafts (Stahlhandel Gröditz GmbH, 2022). It's also common for gudgeon pins to be a similar or equal material to the crankshaft, (Ward, 2012), and for conrods to be made of the same steel alloys (Precious Industries, 2020). This is because the high strength and surface hardness are desired qualities, with nitride

hardening also being often used with an added beneficial effect of improving fatigue resistance (Ward, 2012).

3.3: Results and Analysis

3.3.1: Rigid Dynamics Study

The rigid dynamics study studies the inertia forces generated by the mass and velocity of each component, rather than studying the components via a mesh. A joint probe was placed on the joint between the gudgeon pin and the piston, to measure the loads imparted through this joint. A rotational velocity was set to 4000RPM at the crankshaft, and the simulation was ran for a time period of 0.05s.

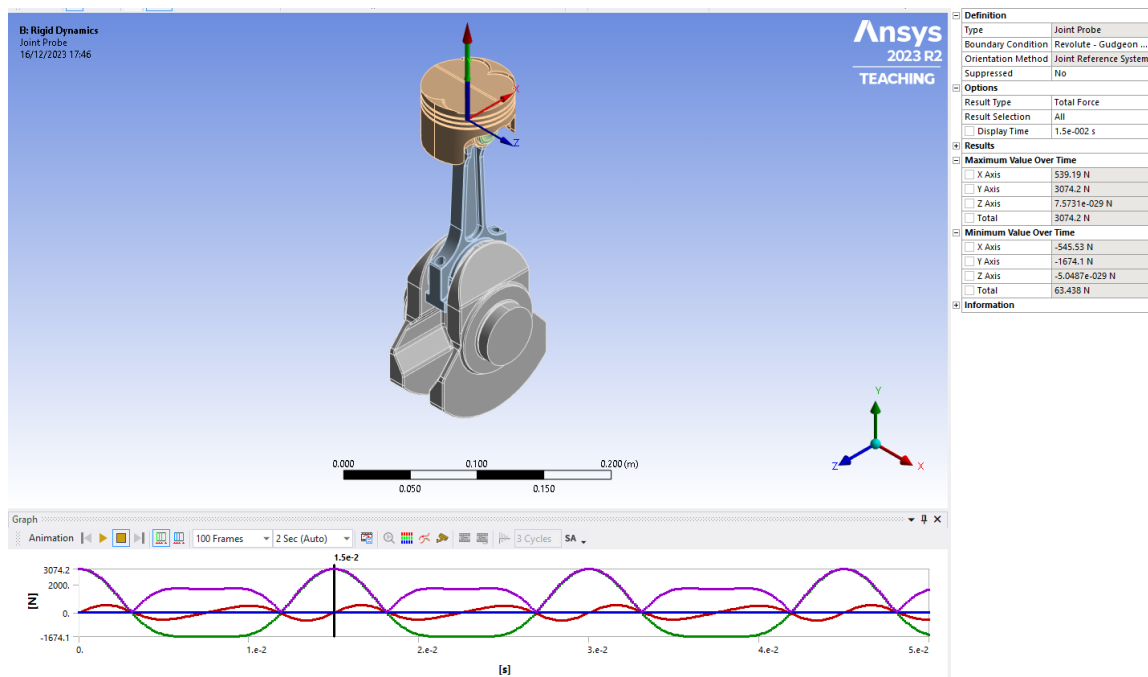


Figure 10 - Force acting on the piston/gudgeon pin joint throughout 0.05 seconds of run time at 4000RPM

The forces generated in each direction were graphed by the software and colour coded to each direction of the global co-ordinate system. This can be seen in the bottom of the above figure. The peak force measured was 3074.2N.

3.3.2: Steady-State Thermal Study

For the thermal study, the piston was investigated, and a temperature map was generated for use in the structural study. Temperatures were input for various faces of the piston. This also required the creation of a mesh, that the rigid study did not require.

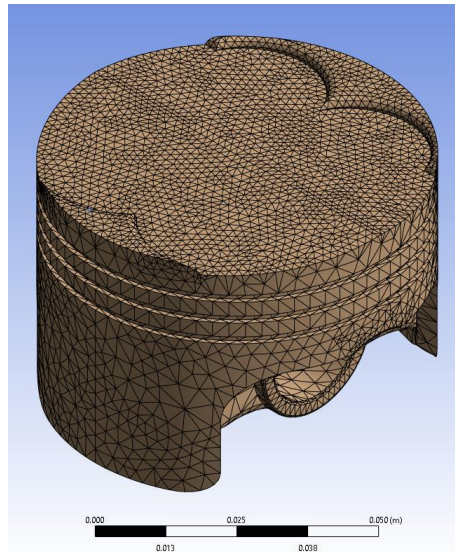


Figure 11 - Mesh of the piston for the thermal and structural studies

As can be seen above, an overall mesh size of 0.003m was set with an additional sizing constraint of 0.0015m for the crown faces of the geometry.

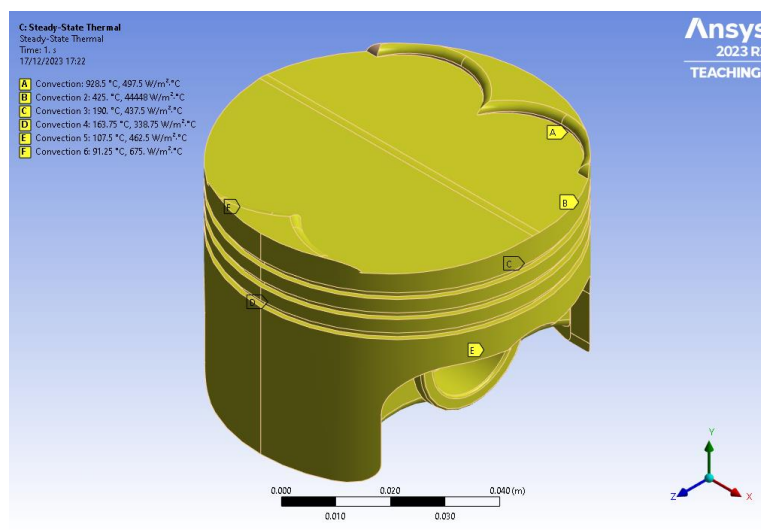


Figure 12 - Steady-state thermal conditions input

The thermal conditions were applied to each set of piston faces, including the crown, piston rings, oil ring, skirt, and underside faces.

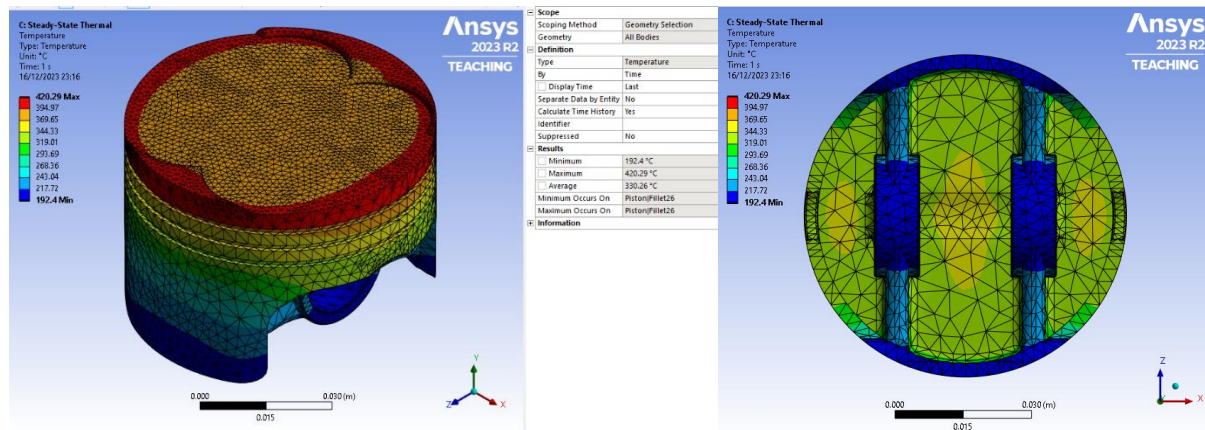


Figure 13 - Thermal study results

A maximum temperature of 420.29°C was seen at the edge of the piston crown, with the lowest temperature of 129.4°C being recorded on the bottom of the gudgeon pin mounts. This result is mainly used to inform the structural study. The fact that FEA programs can simulate for different variables, and feed these variables into other studies, makes FEA processes and software a highly versatile and valuable tool for engineering design (SAE International, 2023).

3.3.3: Static Structural Study

The forces acting on the piston from the gudgeon joint were imported into the structural study as input forces, as well as the temperature profile gleaned from the thermal study. A pressure of 1.05e7Pa was also added to the crown surfaces to roughly simulate the forces from the gas expansion acting on the piston during engine operation.

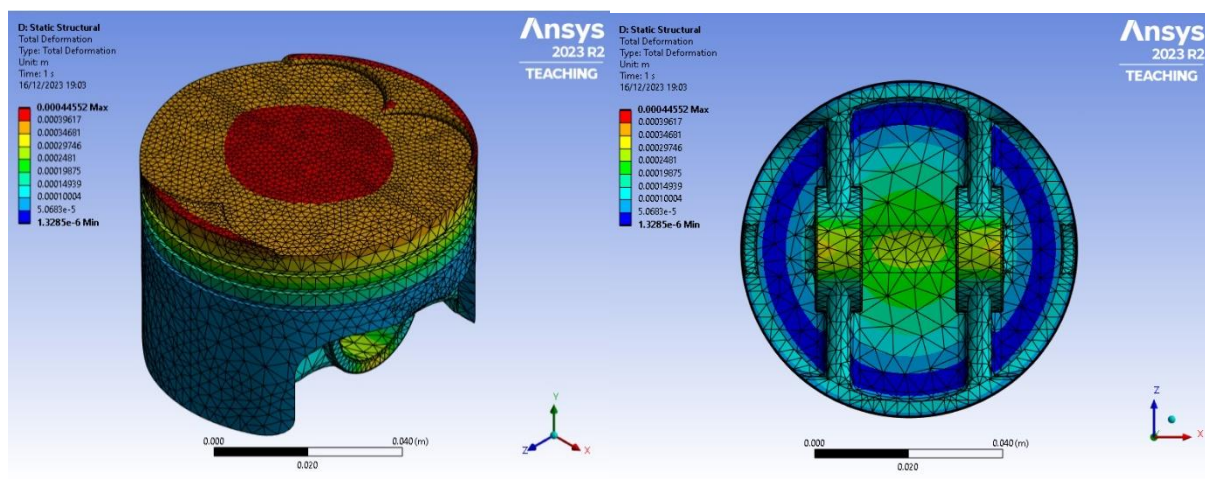


Figure 14 - Total Deformation - Isometric and Underside views

The deformation plot somewhat matches with the thermal expansion given from the temperature study input, with greater deformation around the gudgeon pin as this is where the motion loads from the rigid dynamics study act upon.

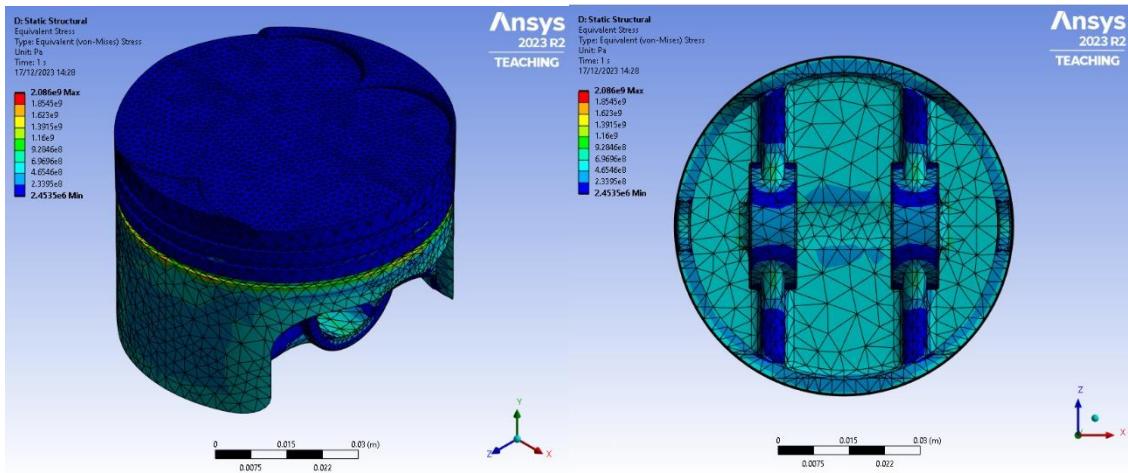


Figure 15 - Von-Mises Stress - Isometric and Underside Views

The peak stress acts around the oil ring, as this is the point at which the temperature of the piston drops sharply, and so the expansion reduces significantly as well, as can be seen in the deformation plot. Stress is also significant in the underside of the piston, due to the inertial forces acting through the gudgeon pin and the deformation of the crown impacting the underside of the piston.

4: Performance Optimisation

4.1: Crankshaft Balancing

The objective of the modification was to bring the centre of mass of the crankshaft closer to its axis of rotation and investigate the effect on deformation and stress that this gives. The balancing of one-cylinder engine crankshafts in reality is far more complex (Foale, 2007), however this serves as an example of the FEA-driven design process.

4.1.1: Geometry

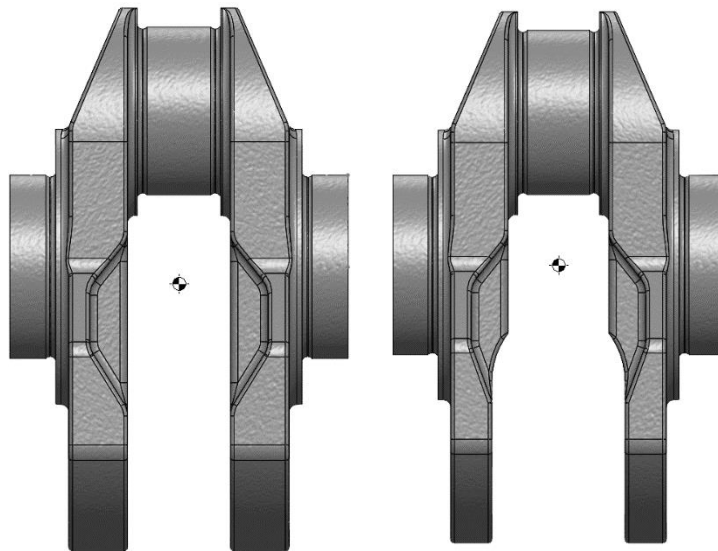


Figure 16 - Geometry comparison showing the centre of mass and the cut material

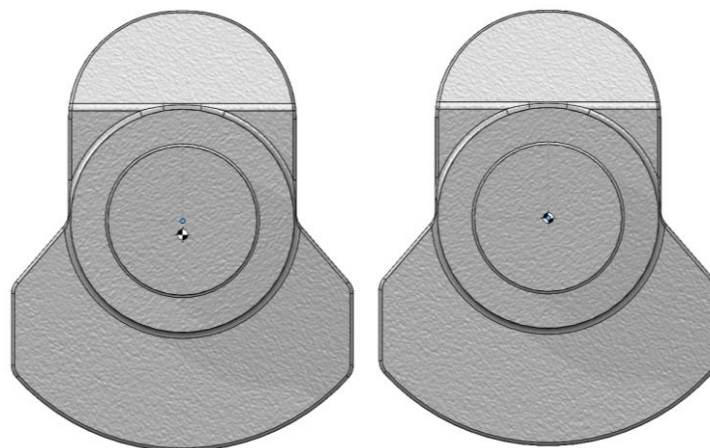


Figure 17 - Geometry comparison showing the shift in centre of mass

Figure 15 shows the material cut out from the crankshaft. This reduced the mass from 3060.61g to 2782.89g, by 277.72g. The centre of mass was shifted to the centre of the main journal to investigate the effect on stress and deformation.

4.1.2: Results and Analysis

4.1.2.1: Rigid Dynamics

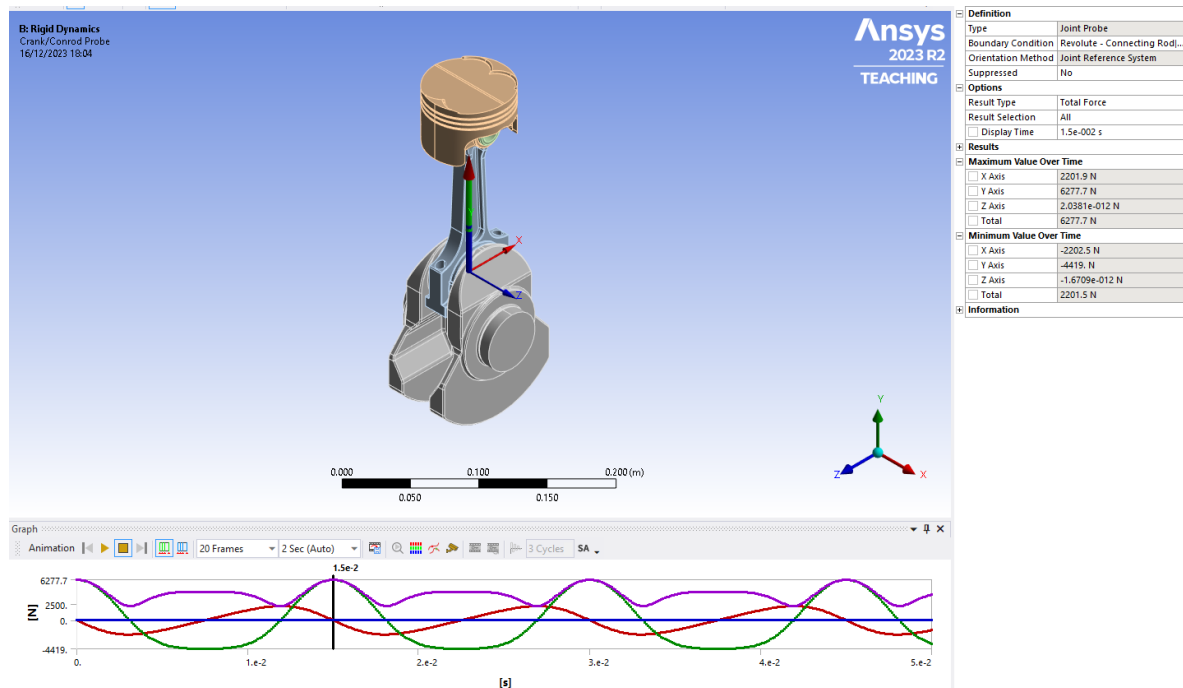


Figure 18 - Force acting on the crankshaft/conrod joint throughout 0.05 seconds of run time at 4000RPM

The same post-processing as in the previous study was carried out for the crankshaft-conrod joint probe. The peak force measured was 6277.7N, with the unmodified geometry.

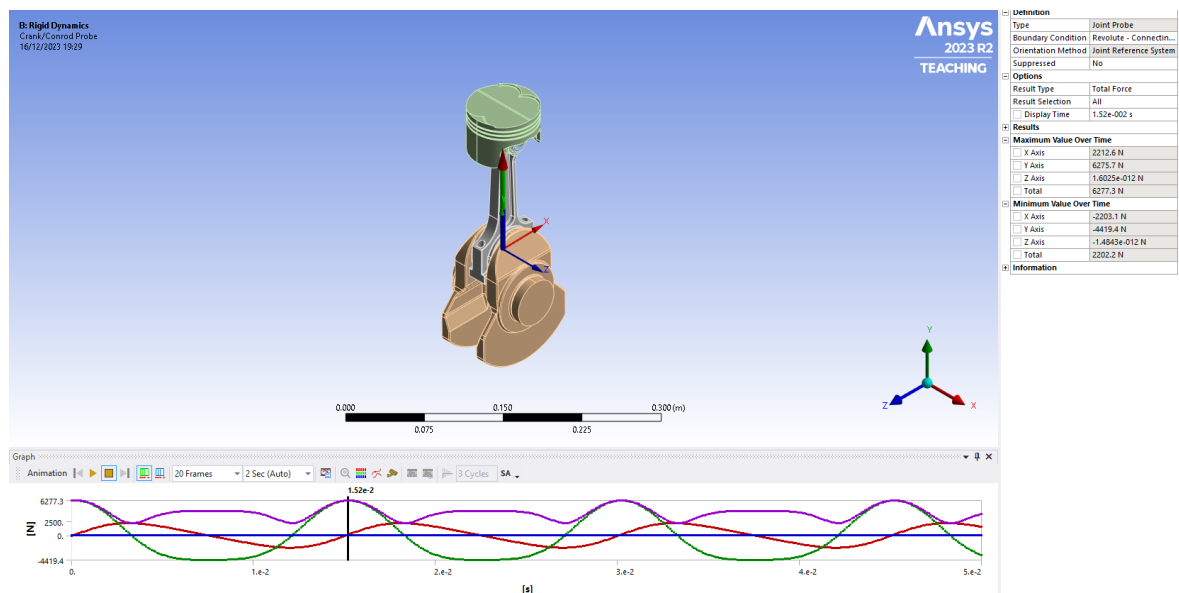


Figure 19 - Force acting on the crankshaft/conrod joint throughout 0.05 seconds of run time at 4000RPM with the updated crankshaft geometry

For the updated crankshaft geometry, the maximum force measured was 6277.3N, marginally lower.

4.1.2.2: Structural

The simulation was run using the imported joint loads from the rigid dynamic study, and an approximated constant temperature of 95°C.

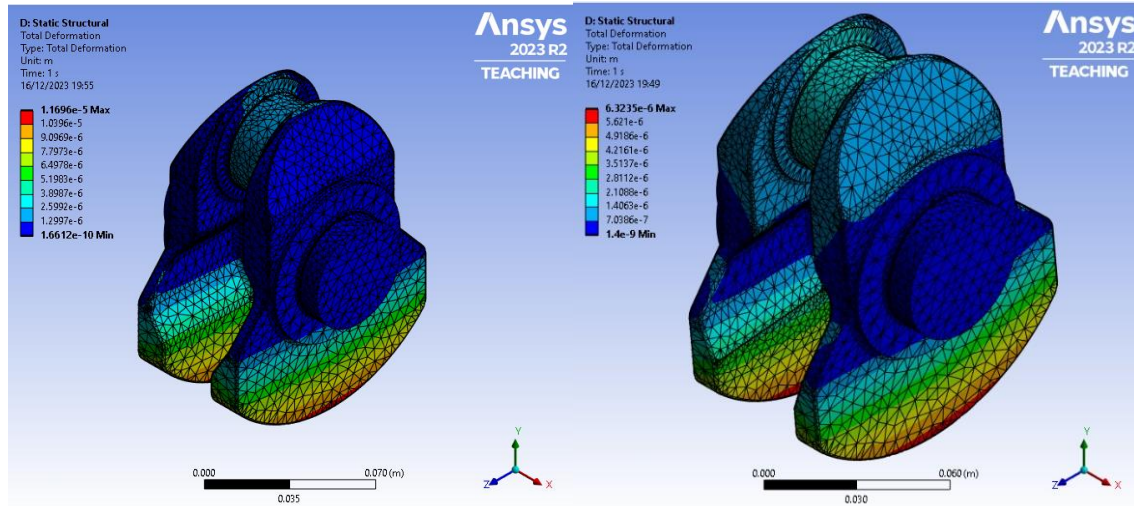


Figure 20 - Deformation compared between the original (L) and modified (R) crankshafts

The maximum deformation in the updated geometry can be seen to be lower, by $5.336 \times 10^{-3} \text{ mm}$. This may be because the maximum deflection can be seen to be where the material was cut away. The deformation also has a relative increase around the rod journal.

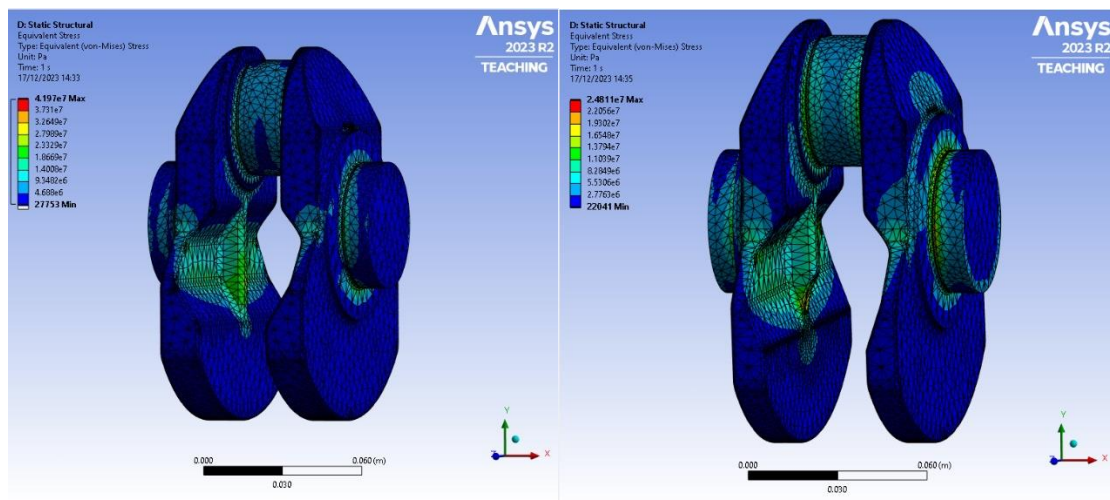


Figure 21 – Von-Mises Stress compared between the original (L) modified (R) crankshafts

The stress is decreased in the modified geometry by $1.7159 \times 10^7 \text{ Pa}$, or 17.159 MPa . A peak point of stress was also created where the rib was filleted away, which could be refined away in further testing. It also could be that the deformation and stress is decreased simply due to the decreased mass of the part, as the forces acting on the part are purely motion forces derived from the inertia of the system's rotation.

Conclusion

In conclusion, the use of FEA in the engineering design methodology is widespread and for good reason, making the design process faster and cheaper with less physical prototyping and testing necessary. However, it is also important to consider the limitations of FEA processes, including the approximated nature of CAE and the computer resources necessary. This analysis was supported and exemplified by two FEA validations, a constant pressure beam and a bimetallic strip, both measuring deflection due to a force applied and a thermal condition respectively. A combined study of rigid dynamics, steady-state thermal, and static structural analyses was also carried out, with the motion loads and temperature profile generated from each study feeding into a final structural analysis. The effect of a geometry change on the crankshaft of the assembly was also investigated, as an example of the data driven analysis possible in FEA design processes.

References

- Bhat, F. Z. (2021, January 17). Application of Finite Element Analysis. Retrieved from https://www.researchgate.net/publication/352017598_Application_of_Finite_Element_Analysis
- Bhavikatti, S. S. (2005). *Finite Element Analysis*. New Delhi: New Age International (P) Ltd.
- Dlupal Software. (2017, October 12). *VE 0078 Bimetallic Strip*. Retrieved from Dlupal Software: <https://www.dlupal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/000078>
- Dlupal Software. (2021, January 20). *VE 0016 | Elastic Bending – Continuous Load*. Retrieved from Dlupal Software: <https://www.dlupal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/009016>
- Foale, T. (2007). *Some science of balance*. Retrieved from Motochassis: <https://motochassis.com/Articles/EngineBalance/EngineBalance.pdf>
- Gardiner, J. (2017, March 27). *Finite Element Analysis Convergence and Mesh Independence*. Retrieved from XCEED Engineering: <https://www.xceed-eng.com/finite-element-analysis-convergence-and-mesh-independence/#:~:text=Mesh%20Independence%3A%20Following%20convergence%2C%20additional,are%20independent%20of%20the%20mesh.>
- Goyal, N., & Nema, T. (2016). Design and Optimization of Piston used Uncoated Aluminium Alloy-6061 and Coated with Ceramic Material using CATIA & ANSYS. *International Research Journal of Engineering and Technology*, 131-137.
- Kelley, C. T., Bernholc, J., Briggs, E. L., Hamilton, S., Lin, L., & Yang, C. (2020). Mesh independence of the generalized Davidson algorithm. *Journal of Computational Physics*.
- Kurowski, P. (2002, November 15). *When Good Engineers Deliver Bad FEA*. Retrieved from MahineDesign: <https://www.machinedesign.com/3d-printing-cad/fea-and-simulation/article/21831586/when-good-engineers-deliver-bad-fea>
- Liu, G. R., & Quek, S. S. (2014). Chapter 13 - Using FEM Software Packages. In G. R. Liu, & S. S. Quek, *The Finite Element Method (Second Edition)* (pp. 397-421).
- Manor Tool & Manufacturing Co. (2019). *The Benefits of Finite Element Analysis in Manufacturing*. Retrieved from ManorTool: <https://www.manortool.com/finite-element-analysis>

- Mit.edu. (2017). *Mesh Convergence*. Retrieved from MIT.edu: <https://abaqus-docs.mit.edu/2017/English/SIMACAEFSARefMap/simagsa-c-ctmmeshconverg.htm>
- Orr, B. (2012, December). *An Iterative Approach to Automating Design Optimization*. Retrieved from Scholarworks California State University: <https://scholarworks.calstate.edu/concern/theses/sf268774b>
- Park, S.-H., Chin, J.-W., Cha, K.-S., & Lim, M.-S. (2022). Deep Transfer Learning-Based Sizing Method of Permanent Magnet Synchronous Motors Considering Axial Leakage Flux. *IEEE Transactions on Magnetics*, 1-5.
- Precious Industries. (2020). *Materials Used in Manufacturing of Connecting Rod?* Retrieved from Precious Industries: <https://www.preciousind.com/connectingrod-manufacturing-materials.php>
- SAE International. (2023). *Finite Element Analysis (FEA) for Design Engineers*. Retrieved from SAE International: [https://www.sae.org/learn/content/pd531241/#:~:text=Finite%20Element%20Analysis%20\(FEA\)%20is,widely%20implemented%20by%20automotive%20companies.](https://www.sae.org/learn/content/pd531241/#:~:text=Finite%20Element%20Analysis%20(FEA)%20is,widely%20implemented%20by%20automotive%20companies.)
- Sieminski, D. (2001). Race Car Design Takes Finite Element Analysis. Nora Engineering.
- Simscale. (2022, October 29). *What Is FEA | Finite Element Analysis?* Retrieved from Simscale: <https://www.simscale.com/docs/simwiki/fea-finite-element-analysis/what-is-fea-finite-element-analysis/>
- Srinivas, K. (2020, May). *Verifications and Validations in Finite Element Analysis (FEA)*. Retrieved from ResearchGate: https://www.researchgate.net/publication/341727057_Verifications_and_Validations_in_Finite_Element_Analysis_FEA
- Stahlhandel Gröditz GmbH. (2022). *1.7225 (42CRMO4) | 1.7227 (42CRMOS4) MATERIAL DATA SHEET*. Retrieved from Stahlhandel Gröditz: <https://www.stahlportal.com/en/stock/17225-42crmo4-17227-42crmoss4/#:~:text=42CrMo4%20%7C%201.7225%20is%20a%20versatile,rods%20are%20manufactured%20from%20it.>
- Suresh, K., & Regalla, S. P. (2014). Effect of mesh parameters in finite element simulation of single point incremental sheet forming process. *International Conference on Materials Processing and Characterisation*, (pp. 376-382). Hyderabad.

University of Victoria. (2015). Retrieved from Lectures:
https://www.engr.uvic.ca/~mech410/lectures/FEA_Theory.pdf

Ward, W. (2012, July 3). *Piston pins - material choices*. Retrieved from High Power Media:
<https://www.highpowermedia.com/Archive/piston-pins-material-choices>

Xu, X., Li, H., & Lin, Y. (2019). Mesh-Order Independence in CFD Simulation. *IEEE Access*, 119069-119081.

Zhou, Q., Panetta, J., & Zorin, D. (2013). *Worst-case Structural Analysis*. Retrieved from NYU.edu:
<https://cims.nyu.edu/gcl/papers/zhou2013wcs.pdf>