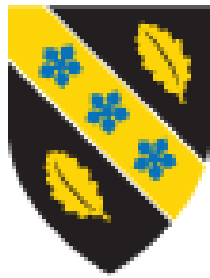


University of Wales Trinity Saint David

School of Engineering



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

Application of Computer Aided Engineering Processes

ACHA6010

Jacob Quarry

(2000885)

18/12/2022

Contents

1	Introduction	Error! Bookmark not defined.
1.1	Aims.....	1
1.2	Objectives.....	1
2	Validation	2
3	Simulation	4
4	Analysis and Iteration	7
5	Conclusion	10
6	References	11
7	Appendix	11

1 Acknowledgement of

Thanks to Andrew Noble for providing all of the 3D resources used for this simulation.

2 Introduction

Computer-Aided Engineering (CAE) is key to the optimization of components. It allows a part to be simulated across a range of loadings and external effects across its lifetime to see if it will operate in a safe predictable manner.

This project revolves around simulating a piston, rod and crankshaft assembly in Ansys. This assembly was chosen as the great heat of combustion will impact the strength and other properties of the materials involved. As a result, it is a good example of both high loads and unfavourable conditions.

2.1 Aims

- Better understand the loads acting on a system
- Use this understanding to design components

2.2 Objectives

- Validate the operation of Ansys
- Simulate the loads acting upon a piston
- Use those results to improve the design

3 Validation

Validation is important to verify that the CAE solver operates in a predictable and accurate manner. As a result, there are published results to replicate and compare the reported findings to those compared. Given how this project requires heat as well as force loading then a bimetallic strip example was used. Appendix 1 (Dlupal, 2022) shows the material properties and sizing, along with the expected deformation of 7.049mm and how it could be calculated.

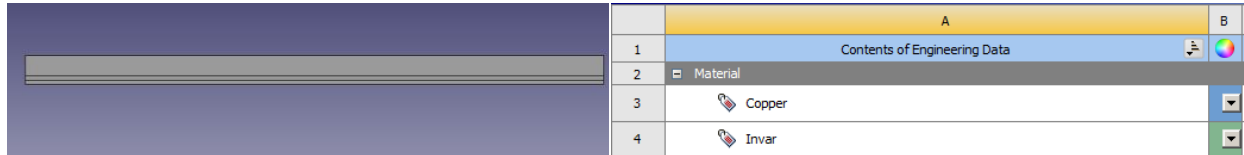


Figure 1 - The bimetallic strip being tested and the materials applied

The bimetallic strip was created in Solidworks and then opened in Ansys where the material properties were applied. It was then rigidly affixed at one end and the temperature was ramped up to 100 degrees above ambient. The first mesh tested was 1mm and reported deformation 1% higher than the calculated value. To test if this was by nature of the way Ansys calculated it or if it was mesh dependent; a range of meshes were applied from 2mm to 0.1mm.

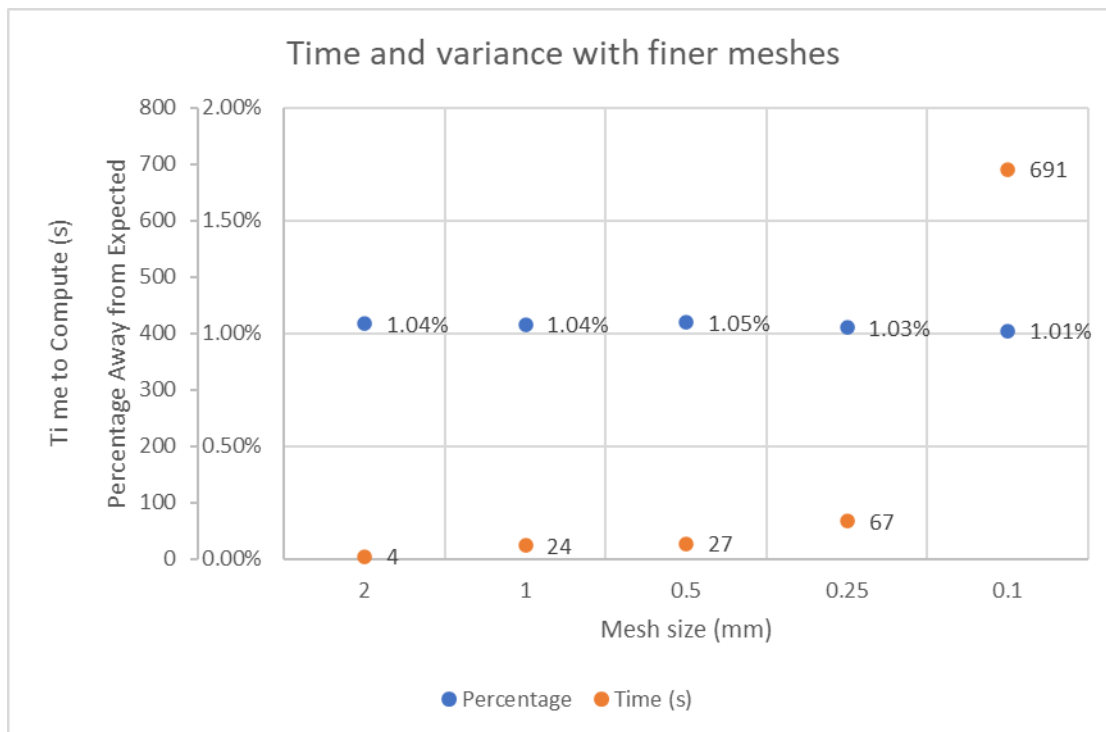


Figure 2 - Comparison of mesh size against accuracy and solve time

Its clear in Fig.# that the 1% over the calculated value was consistent across the range of meshes tested. There was a slight improvement in accuracy as the mesh got finer from 0.5mm through to 0.1mm, although only 0.04%. This is inconsequential, especially when compared to the required processing time increasing by over 25 times. To see if the trend continued, even finer meshes were tried. These just caused Ansys to crash however. It is clear that across the range of meshes tested that the Ansys deflection is very comparable to the validation values.

4 Simulation

To test the assembly in Ansys the components were made into an assembly in Solidworks. This can then be saved as a STEP file to be imported into Ansys. The materials used were structural steel for the crank, gudgeon pin and conrod and 6061 aluminium for the piston. Both materials are poor approximations of those actually used: the steel is less of a concern however as the piston is the focus of all the studies past the rigid dynamics and even in that initial study the material of the steel components will have little effect. The aluminium does however matter more as it's not reflective of the alloys that are used. 6061 is liable to expand more with temperature than other alloys and will lose its heat treatment properties at the expected temperatures. Given the nature of the study however, such as the fact that clearances to the gudgeon pin or cylinder clearances aren't being considered, it is a reasonable approximation.

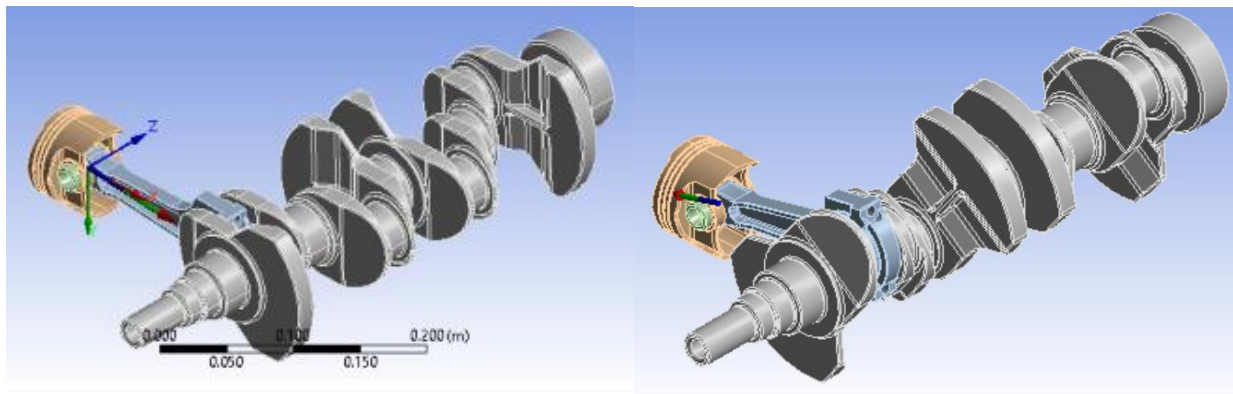


Figure 3 - Crankshaft assembly in Ansys and as it rotates

The dynamics simulation was conducted at 4000 RPM. For a more reasonable upper limit of loading it was intended to run the simulation at 8000 and then 6000 RPM. At this point however the limitations of Ansys on the university systems was discovered as Ansys failed to compute above 4000 rpm and was crashing. Although not ideal, it isn't a major concern as the iterative designs will be conducted at the same RPM so the results will be comparable.

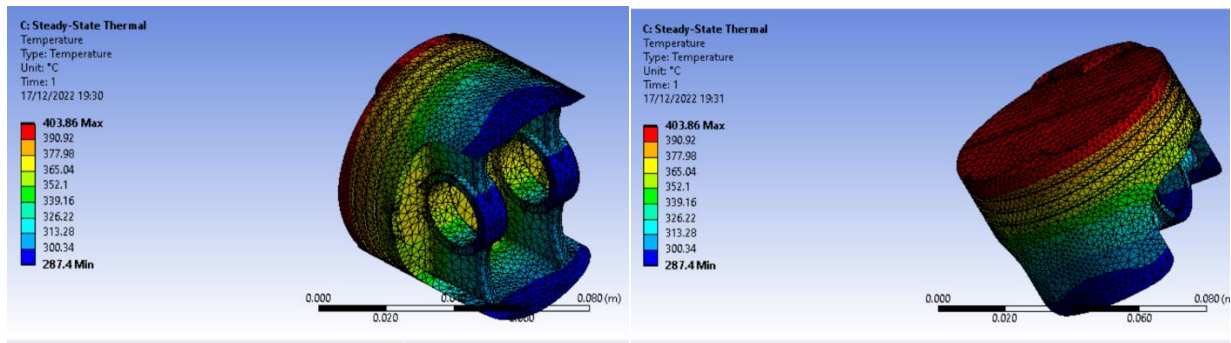


Figure 4 - Two angles of the generated heat map

After the dynamics analysis, heat loading representative of the conditions inside the engine were applied (Noble, 2022). This was done with a 2mm as again trying to mesh any finer caused issues with the solver. This showed a peak temperature at the crown of 404°C however even the coolest part was 287°C. This shows the importance of the thermal study as the temperatures observed are substantial and would impact the strength of the material. This is very important for the future iterative design as it informs where material could be saved. If material was to be removed from around the crown (say thinning it) it would reduce the local thermal mass and higher temperatures would be likely. As a result, not only would there be less material for strength in the area but what was left would be weaker. This is representative of one of the biggest advantages of CAE, it allows the modelling of a highly complex system that would be very costly to test otherwise. It would also allow testing of the extremes, such as the level of overheating that would be required to cause clearance issues or failure of the material.

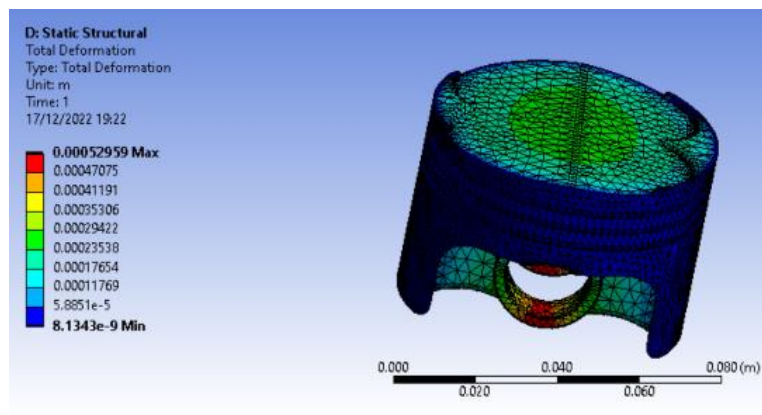


Figure 5 - Deformation under load

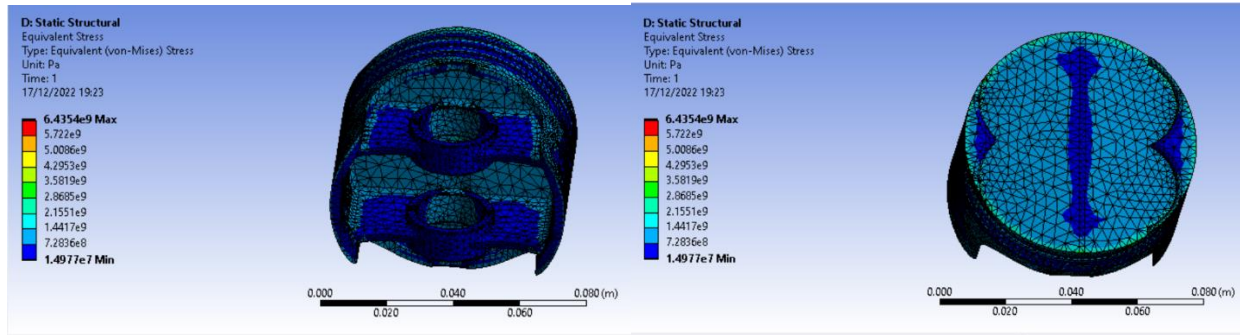


Figure 6 - Two angles of von-Mises stress

A 2mm mesh was again used for the final static simulation with the net results of the dynamic and thermal studies. The results that were considered were the Von-Mises stress and the deformation across the piston. To simulate combustion pressures a pressure of 105 bar was applied to the piston crown. To support it during simulation weak springs were used as opposed to constraining at a particular point. This allows it to deform as it would without constraints affecting the results. This was particularly beneficial as it can be seen that the peak deformation of 0.5mm was seen at the base of the gudgeon pin bore. As a result, this should be reinforced or the loading on the part reduced. For the iterative design the latter option was chosen, the mass of the piston will be reduced to try and reduce some of the loading and avoid adding material. It can also be seen that the von-Mises colouration is not particularly useful as it is biased towards the blue lower stress side. To test if this was meshing related a few other meshes were tested, a 1mm mesh was attempted but didn't solve, a 4mm and a 4mm with a 1mm specific sizing on the crown down to the top compression ring land.

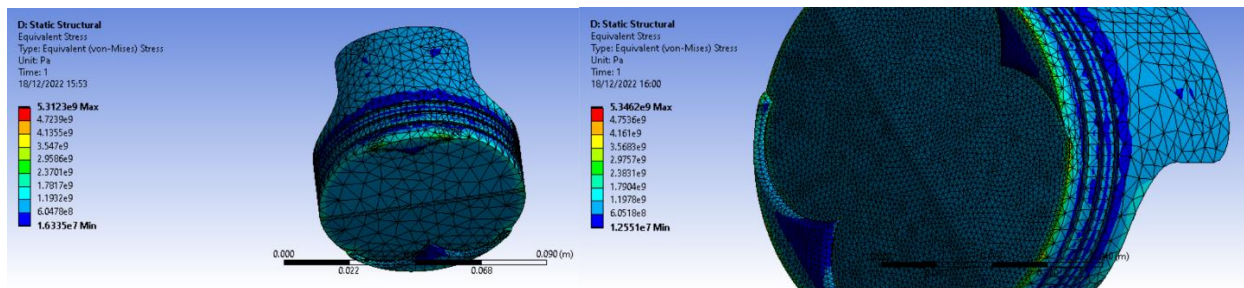


Figure 7 - Stress concentration with 4mm mesh and 1mm specific sizing on the right.

It became clear that the peak stress was higher when a 2mm mesh was used and when using the 1mm specific sizing the stress increased from a pure 4mm mesh. This indicates that some type of runaway condition is happening as the mesh gets finer causing the stress to rise. As expected the calc time increased as the mesh got finer with the 4mm taking 74 seconds, the 4mm with 1mm on the crown taking 275 seconds and the full 2mm mesh taking 423 seconds.

5 Analysis and Iteration

Considering the results from Ansys, it was key to take material from the piston to reduce the force experienced when accelerating the piston downwards. It was also highlighted by the thermal and structural results that this couldn't be taken from the piston crown as that is subject to a lot of load. Also, the simulation was limited in its scope as it didn't account for the loading on the side skirt by the cylinder during use. As a result of this, it was determined that windowing the piston skirts, similarly to what is applied on two strokes, would be likely to get the best results.

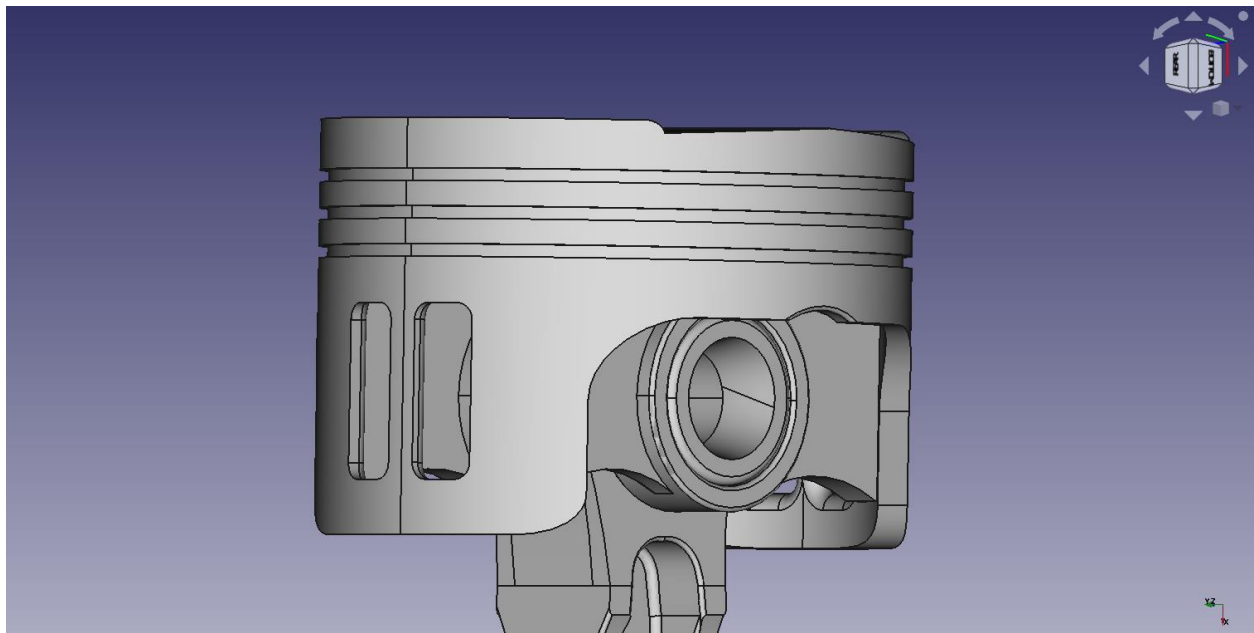


Figure 8 - Windowed Piston

To preserve the skirt stiffness a 4mm wide central channel was retained. The windows themselves are 20mm long and reach until the cross pieces that the gudgeon pin passes through. To mitigate stress concentrations then the windows have a 2mm radius on each corner and a 2mm fillet on the reverse side to blend into the rest of the piston. Alongside this, it was clear that most of the peak stress was from the area around the valve reliefs so the fillet on them was increased from 1.5mm to 2.5mm.

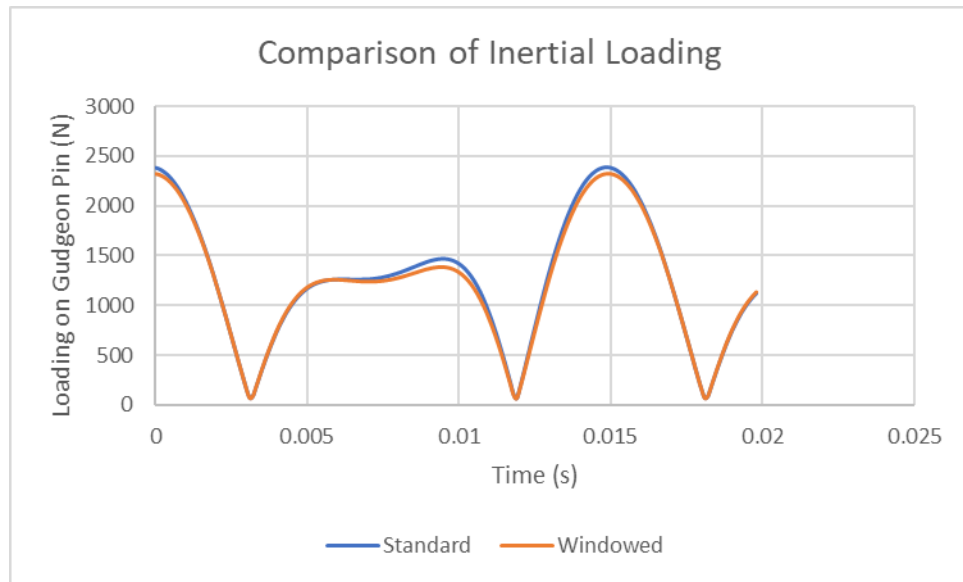


Figure 9 - Comparison of joint loadings from inertia

The rigid dynamics data from Ansys shows marginal savings on the joint loading. At peak a saving of 80N is possible and this is a respectable 6.5% of the joint loading. The inertial savings over the entire cycle however average out to only 30N.

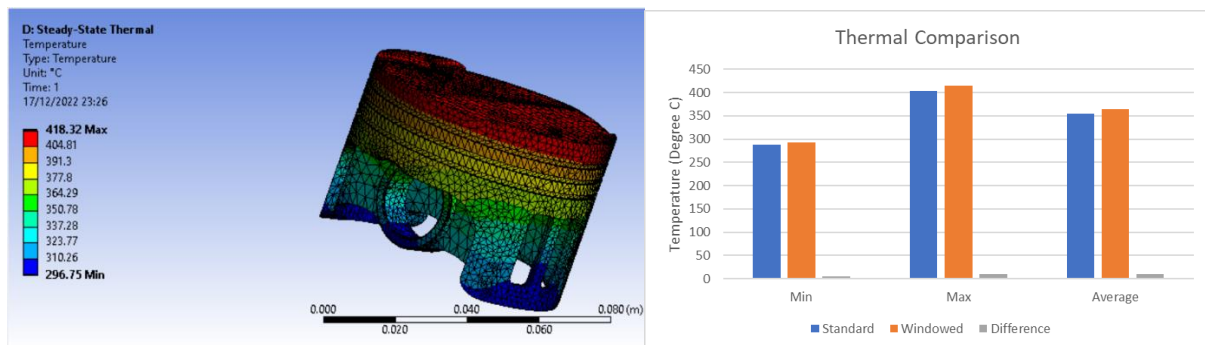


Figure 10 - Thermal Map of piston and temperature comparison

The thermal analysis showed that the windowed piston got hotter than the standard piston due to the reduced thermal mass. This was around 10°C hotter on both the average and maximum temperatures, That however only a 3% temperature increase against the standard component.

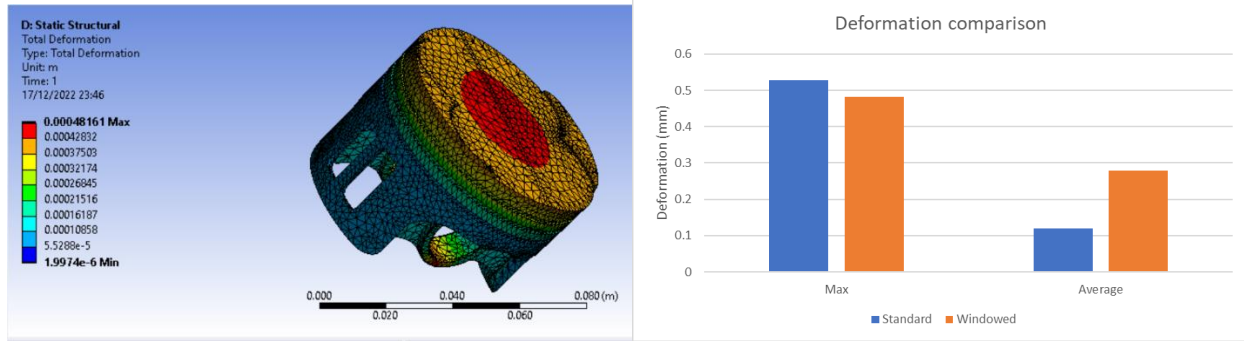


Figure 11 - Deformation of piston

The deformation results of the static structural analysis showed that the maximum deflection had decreased by approximately 0.05mm. However, the average deflection increased by approximately 1.5mm. Reducing the mass was successful as the heatmap shows the peak deformation has changed from the gudgeon pin bore to the piston crown.

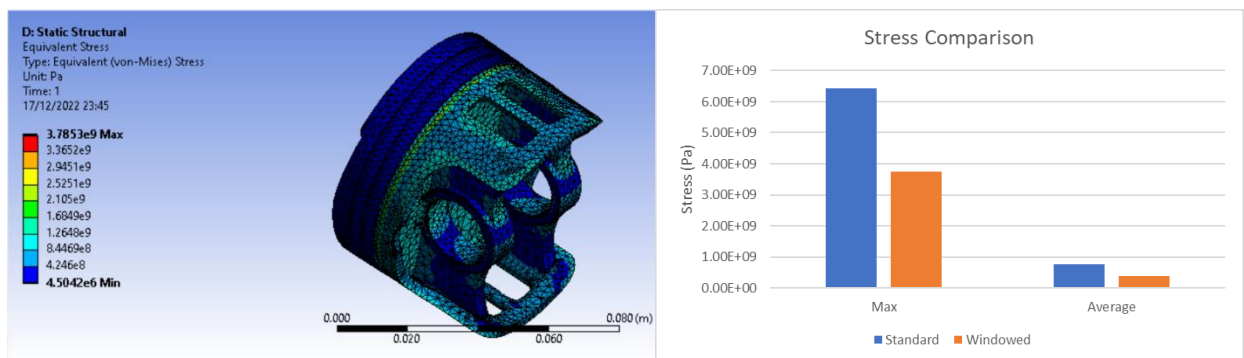


Figure 12 - Stress map and comparison

The modifications successfully reduced the overall stress in the part. As can be seen from the map there was increased stress around the windows but the fillets worked as intended to stop it raising too much. Alongside this the fillets at the valve reliefs helped to reduce the peak stress. As shown by the graph the modifications reduced the peak stress by 42% and halved the average stress of the component. This could however have been further reduced by chamfering or filleting the sharp 90° angles that little the part: good examples would be the edge of the piston crown, the ring lands and the edges of the piston skirt, although the latter will have a minimal effect due to the reduced stress in the region.

It is clear that reduction of mass had the intended effect by reducing the loading that the piston was exposed to. This is especially applicable as the force massively rises with rpm so any reduction will only compound. Future steps could involve eliminating stress concentrations and further reducing mass in the piston skirt through shortening or thinning.

6 Conclusion

Computer-aided engineering is a very powerful tool that allows mass testing and prototyping that could not be achieved in another manner. It is however very dependent on both the input values and computing resources available.

A comprehensive understanding of all the effects and forces at play are absolutely essential. For example, this study did not account for any clearances in any of the bores that are relevant in the top end assembly. This severely limits the usability of the study as it means there could be an unconsidered force acting on the components that could prove detrimental, such as fatigue on the de Havilland Comet. The modification of the piston with the skirts being windowed is limited on its transmissibility to the real world for example as there's no interaction with the cylinder. There's no telling if the piston would begin to move in an unintended way with shortened skirts as it travelled along the bore or if the bore clearance would reduce to an unacceptable level as the material expands a higher amount due to the increased temperatures. If that was all accounted for however there would be further issues, such as the material choice and specific alloys. Fundamentally to get usable data then a lot of work is required into the exact conditions that will be expected from each and every component.

Which leads onto the second issue, by nature of how CAE works, it is very intensive on computing resources. As that accuracy is continually chased and meshes get more intricate and more components get added the computing and solve time requirements increase at a rate much greater than linearly. This limits the accuracy of the testing that can be conducted on a smaller scale, such as this project where Ansys frequently failed to solve.

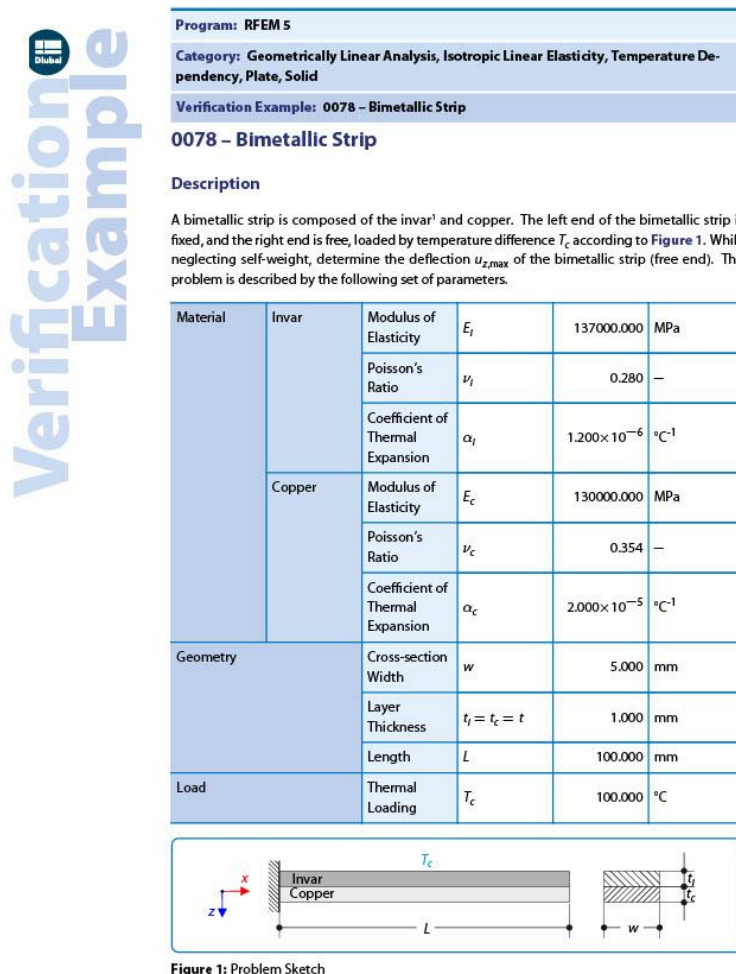
It is however incredibly powerful and essential for the modern design process. With it, incredible designs can be achieved with the minimum material in the correct places so everything operates safely and reliably whilst still being lightweight.

7 References

Dlubal (2022) *Verification Examples*. Available at: <https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/000078> (Accessed: 11 December 2022)

Noble, A. (2022) *Thermal Study Boundary Conditions*.

8 Appendix



¹ Invar is an alloy of iron and nickel with very low coefficient of thermal expansion.

Figure 13 - Bimetallic Strip Example (Dlubal, 2022)

Verification Example: 0078 – Bimetallic Strip

Analytical Solution

The bimetallic strip is made of two metals with different coefficients of thermal expansion. They are frequently used in instruments to sense or control temperatures. When the ambient temperature is changing, the bimetallic strip is bending. That is caused by the different elongations of the used metals.

The analytical solution is based on the approach introduced in [1]. The following formula gives the dimensionless coefficient K_1 which can be used for the calculation of the equivalent stiffness

$$K_1 = 4 + 6 \frac{t_c}{t_i} + 4 \left(\frac{t_c}{t_i} \right)^2 + \frac{E_c}{E_i} \left(\frac{t_c}{t_i} \right)^3 + \frac{E_i t_i}{E_c t_c} \quad (78-1)$$

The maximum deflection of the bimetallic strip is then defined by the following formula

$$u_{z,max} = \frac{6(\alpha_i - \alpha_c)(t_i + t_c)T_c L^2}{t_i^2 K_1} \approx -7.049 \text{ mm.} \quad (78-2)$$

RFEM 5 Settings

- Modeled in RFEM 5.16.01
- The minimum element size is $l_{FE} = 0.250 \text{ mm}$
- Isotropic linear elastic model is used

Results

Structure Files	Program	Entity
0078.01	RFEM 5	Plate
0078.02	RFEM 5	Solid

Model	Analytical Solution	RFEM 5	
	$u_{z,max}$ [mm]	$u_{z,max}$ [mm]	Ratio [-]
RFEM 5, Plate	-7.049	-7.047	1.000
RFEM 5, Solid		-7.064	1.002

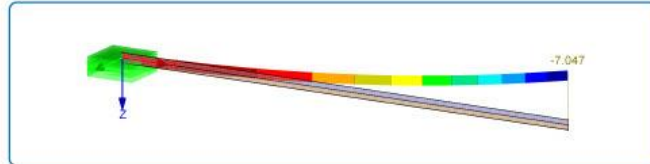


Figure 2: RFEM 5 results – Plate model

References

- [1] YOUNG, W., BUDYNAS, R. and SADEGH, A. *Roark's Formulas for Stress and Strain, 8th Edition*. McGraw-Hill Education, 2011.