

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 & Fluid Analysis (ACHA6010)

Lewis Cox

2004098

Deadline Date 18-Dec-2022

Contents

1. Introduction.....	3
2. Validation of the Software Model	4
2.1. Impact of Mesh Sizing – Bimetallic Strip.....	5
3. Crankshaft & Piston Study Limitations	6
3.1. Rigid Dynamics Study	7
3.2. Steady-State Thermal Study.....	8
3.3. Static Structural Study.....	12
3.4. Fully Linked Study	13
4. Iterative Design of a Piston	14
5. Conclusion	18
6. References	19
7. Appendix.....	21
7.1. Tabulated Mesh Independence Values.....	21
7.2. Provided Boundary Conditions	21

1. Introduction

CAE (Computer-Aided Engineering) processes are an integral part of the automotive industry, growing in use since the 1960's when General Motors Research Laboratories created the first CAD (Computer-Aided Design) system to use interactive graphics (Digital School, 2021). By the 1980's, modelling had started to evolve from 2D to 3D, and during the 1990's PCs were powerful enough to run AutoCAD. Presently, CAE is an important tool that reduces physical experiment time and cost, allowing prototype design iterations to be conducted more efficiently (Solmaz, 2022). For example, the initial design of a part may be well below specification, failing with a minimal stress applied. CAE analysis could give an indication of this prior to a test rig being prepared, which for a large part, that may need to be tested at an increased temperature or pressure environment (e.g., a high-altitude aeroplane fuselage), prevents wasting considerable amounts of time and budget testing a design that was far from meeting the specification.

For reliable simulation results, Verification and Validation (V&V) models are used. The first standards for V&V were introduced in 1987 and 1988 by the American Nuclear Society and the American Institute of Aeronautics and Astronautics (Solmaz, 2022); industries where safety is paramount. Verification assessments are conducted to find programming errors and ensure that the mathematics used to solve the simulation are correct. Verification is performed prior to validation assessments, since if the verification fails the validity also fails. Validation assesses the CAE results by comparing them to physical experiments (Solmaz, 2022) (Slater, 2021). This process is performed in section 2 of the report.

CAE is also beneficial for iterative design. Small modifications can be made to parts without the associated material and manufacturing costs. The design steps can also be automated to allow the software to search for the optimal design. User input for boundary conditions is required, however within this area, a design of experiments approach can be used with techniques such as a cuckoo search (Jalal and Goharzay, 2018.)

The accuracy of the simulation is rarely 100 % compared with physical results. Specifications with acceptable tolerances for the CAE results are therefore important and are dependent on the application of the part.

2. Validation of the Software Model

Simulation software results need validation prior to application of the results to real-world scenarios. Fundamentally, the validation aims to show that the software can simulate scenarios and output results that are approximate to results calculated through fundamental calculations, or experimental results. The simulated design following expected trends is not an indication that the magnitude of the trends is representative of a real-world scenario. For example, an object showing a cooler thermal map after modification may be the expected outcome, however the result may have been exaggerated or underestimated by the software. This becomes even more critical when investigating deformation and stresses of structures under load.

For the purposes of this report, a validation was performed using a bimetallic strip case example created by Dlubal Software, 2018. Equation 1 and 2 are the calculations used (Young W. et al., 2011).

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

$$u_{z,max} = \frac{6(\alpha_i - \alpha_c)(t_i + t_c)T_c L^2}{t_s^2 k_1} \frac{1}{2} \quad (2)$$

Given the provided variables (temperatures and material data), the result is ≈ 7.049 mm using a minimum element size of 0.250 mm. The validation example was performed using Dlubal Software RFEM 5. All the simulation performed for this report used Ansys 2021 R1 Workbench. Validation across different software packages is equally important to validation against fundamental calculations. Engineering projects may involve several engineering groups that use different software packages. Whilst direct file sharing between the software packages may not be possible, simulation results should be approximate to each other, which is expected if each package is validated against the fundamental calculations. Figure 1 shows the visual results of the validation example from the Ansys software, using a mesh size of 0.250 mm. A maximum deformation of 7.121 mm was produced, falling 1.03 % away from the validation example.

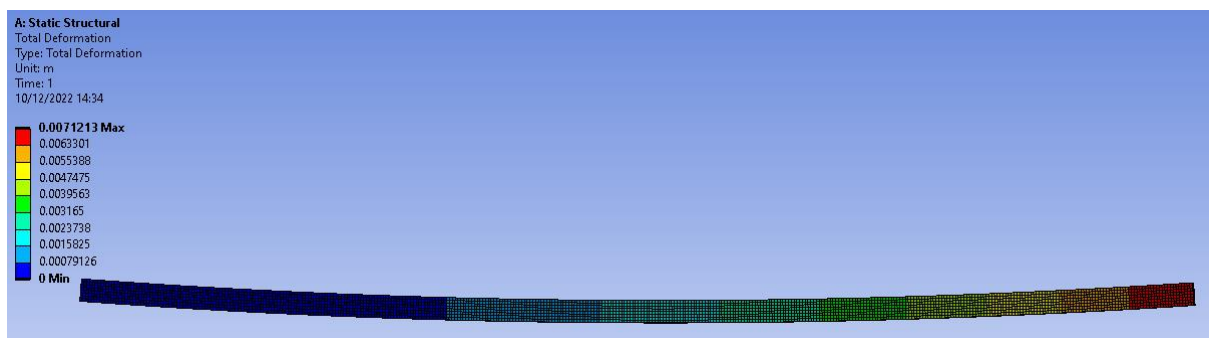


Figure 1 - The bimetallic strip validation example results.

2.1. Impact of Mesh Sizing – Bimetallic Strip

The mesh sizing was varied on the bimetallic strip study. The results are shown in Table 1. For the bimetallic strip, the software would not solve when the mesh sizing was increased beyond 0.005 m. At 0.005 m mesh sizing, the discretisation is minimal with only 496 nodes. Additionally, at this mesh size, the software removes nodes from the faces of the strip. Comparatively, 0.004 m mesh size increases to 976 nodes and includes the extra information from the faces of the strip as Figure 2 shows. On this simple example, it shows the importance of the mesh size in terms of the solvability of the solution.

Increasing the number of nodes through mesh size reduction did not produce significantly different results. Decreasing the mesh size by a factor of 10 influenced the maximum deformation by 0.04 % as shown in Table 1. Further, the solving time for the smaller mesh was increased by a factor of 160 %.

Table 1 - Impact of mesh sizing on bimetallic strip deformation results.

BiMetallic Strip Deflection Validation Example				
Example Validation Result =		7.049 mm		
Recreation of Validation Example with Ansys				
Mesh Size (m)	Average Deformation (mm)	Max Deformation (mm)	% Difference from Example	Elapsed Time (s)
0.00500	Failed To Solve	Failed To Solve	N/A	N/A
0.00100	2.3942	7.1233	1.05%	4
0.00050	2.3895	7.1230	1.05%	9
0.00025	2.3864	7.1213	1.03%	52
0.00020	2.3858	7.1210	1.02%	86
0.00010	2.3845	7.1203	1.01%	640

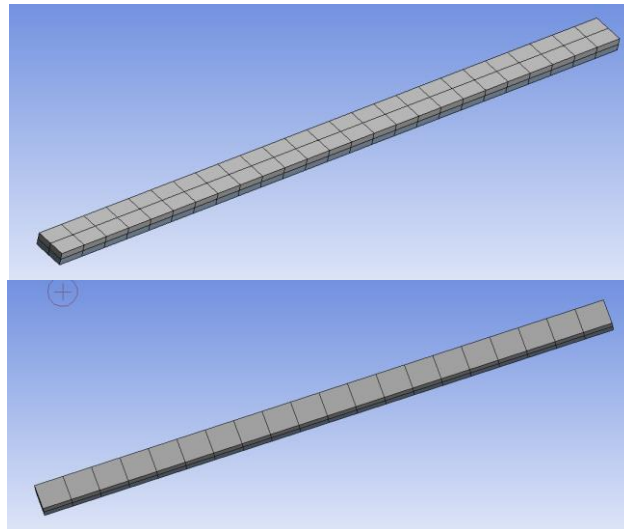


Figure 2 - A 0.004 m mesh (top) and a 0.005 m mesh (bottom). The reduction in discretisation of the face is visible.

3. Crankshaft & Piston Study Limitations

A crankshaft & piston assembly was assembled in Solidworks before being imported into Ansys Workbench as shown in Figure 3. A linked rigid dynamics, steady-state thermal and static structural study was performed. The scope of the deformation analysis was only the piston. The piston material was input into the Ansys Software as 6061 T6 Aluminium. The material property of the other components remained as default. These components remained in the rigid state and so could not deform. This section of the report looks at limitations encountered whilst performing the study. Material property data and boundary conditions provided by A. Noble are shown in Appendix 7.2.

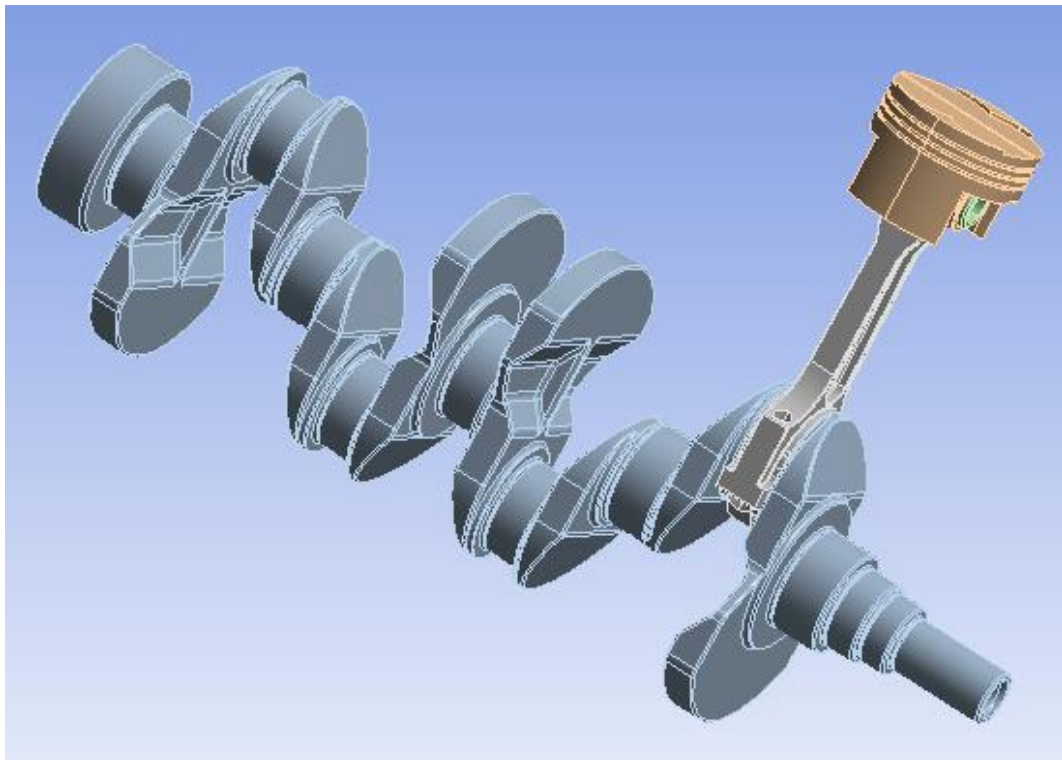


Figure 3 - The studied assembly in Ansys.

3.1. Rigid Dynamics Study

The rigid dynamics module is used to study dynamic responses of a system. It outputs forces, moments, displacements, velocities and accelerations (Ansys, 2021). The assembled system is imported into the Ansys software, in this case from Solidworks. It was found that the assembly ideally needs to be in the exact configuration needed for the analysis prior to importing into Ansys. In one instance the piston was tilted in relation to the conrod when imported. In a second instance, the conrod was offset and in contact with the crankshaft counterweights. These problems could not be corrected within Ansys, even with the configure option. The piston being tilted would impact deformation and stress results, particularly once a combustion pressure was applied to the crown. More significantly, the conrod being in contact incorrectly with the crankshaft caused a solving error for most of the motion, as rotation was restricted by the contact. In order to correct both errors, the assembly required updating in Solidworks, before being re-imported to Ansys and connections / joints being re-assigned.

For a simple assembly, positioning parts and connecting parts is relatively straight forward. Applying connections to a larger assembly, or an assembly containing intricate parts may become increasingly time consuming. The requirement for two individual programs adds cost onto a project, in addition to a decreased user experience. User training requirements are increased as being proficient in multiple software packages is required. User training increases cost in budget and time. It is unavoidable if the very best software and maximum functionality is needed, as each software package is created for a specific use allowing greater functionality within that area.

Joints and contacts also require realistic boundary conditions. For the crankshaft assembly, the piston must be fixed to restrict movement except in the vertical axis. Similarly, the crankshaft requires fixing to allow only rotation along its length. Incorrect joints caused incorrect results. Restricting only one half of the piston in the vertical direction (shown in Figure 4), caused a deformation gradient travelling horizontally, despite a combustion pressure of 100 bar applied to the top surface. The movement within the simulation is defined by the user, requiring an understanding of the dynamics of the assembly, and the knowledge of how the different connections available in Ansys interact prior to testing.

Transient studies require the user to set data recording points. The resolution of the data set relies upon this. Solving time is heavily impacted by this, potentially restricting the amount of data collected, particularly on PCs with less processing power.

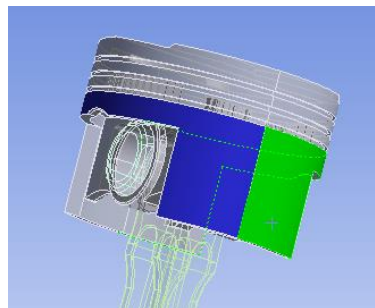


Figure 4 - The piston skirt joint assignment. Selecting only one half of the skirt produced erroneous results.

3.2. Steady-State Thermal Study

The piston was subjected to a steady-state thermal analysis. In order to conduct a thermal analysis, temperature boundary conditions were entered for each area of the piston. This information was accumulated experimentally, meaning the simulation results rely on the experimental data being valid and accurate. Additionally, the correct assignment of the boundary conditions to the part is important for correct results.

The piston was analysed with different mesh sizes, from 0.0050 to 0.0002 m. Similar to the Rigid Dynamics study, the mesh size has significant impact on solve time, but for the mesh sizes almost no impact on the deformation results. The average deformation experienced a change of 2.15 % with a mesh size 25 times smaller as shown in Table 2.

Table 2 - Impact of mesh sizing on the Steady-State Thermal study.

Piston Thermal-Static Structural Analysis					
Mesh Size (m)	Average Deformation (mm)	Average Deformation % Difference From 0.005 Mesh	Max Deformation(mm)	Max Deformation % Difference From 0.005 Mesh	Elapsed Time (s)
0.0050	0.75179	N/A	0.15167	N/A	4.00
0.0010	0.74821	-0.48%	0.15164	-0.02%	20.36
0.0005	0.75742	0.75%	0.15164	-0.02%	80.00
0.0002	0.76797	2.15%	0.15164	-0.02%	764.00

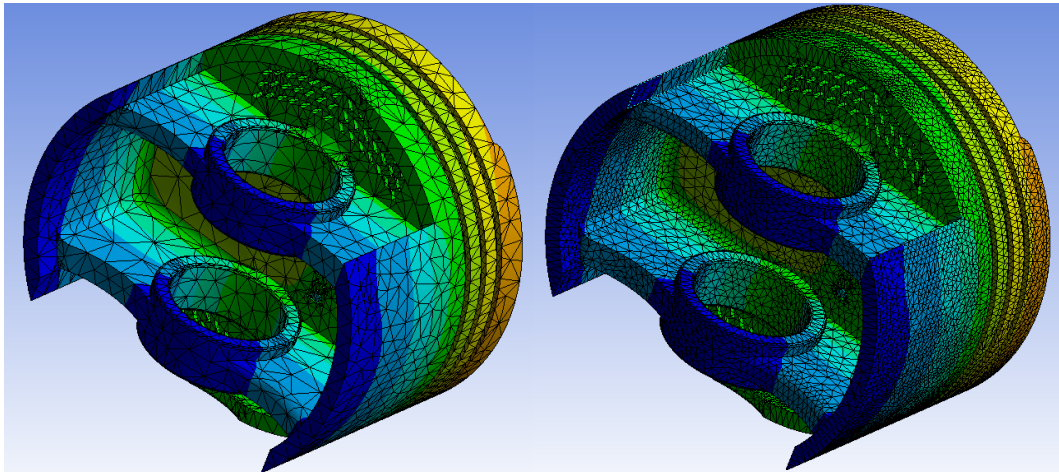


Figure 5 - Visual representation of mesh sizing showing a 0.005 m mesh (left) and a 0.00125 m mesh (right).

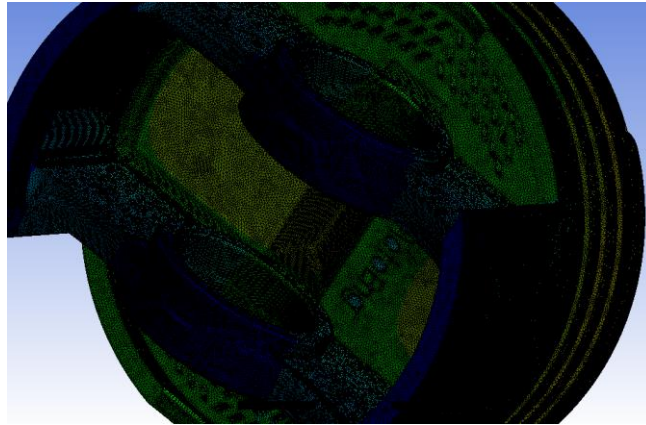


Figure 6 - A visual representation of a 0.0005 m mesh applied to a piston.

A mesh independence study was conducted on a second piston design, comparing node count to the maximum temperature. Only mesh size was varied, with other software settings remaining unchanged. A visual representation of discretisation is shown in Figures 5 and 6. The maximum and minimum node counts in the mesh study were 69964 (0.004 m mesh) and 4761497 (0.0002 m mesh). The maximum node count was restricted by the meshing and solve time.

Figure 7 shows the results of the independence study. Mesh independence can be said to be achieved at approximately 30000 nodes (0.00090 m mesh size). Whilst the maximum temperature value still has a variance of 0.03 %, refining the mesh further is a case of diminishing returns when solve time (shown in Figures 7 and 8) is considered. The largest node count of 4761497 with a mesh size of 0.0002 m was omitted from figure 7 as it skewed the graph, however shown in Figure 9 it can be seen that the large increase in node count produces small changes in maximum temperature (0.071 % from the next mesh size), but a large increase in solve time (see Appendix 7.1 for tabulated mesh independence study values).

Further, the solve time does not account for the initial generation of the mesh prior to starting the solve, which becomes a prohibitly long time. A point to note with meshing and solving is that the progress made by the software is not an indication of how much time is remaining ;the progress bar for meshing will hold at 5 %, but there is no way to know if it will progress further within seconds, minutes or days.

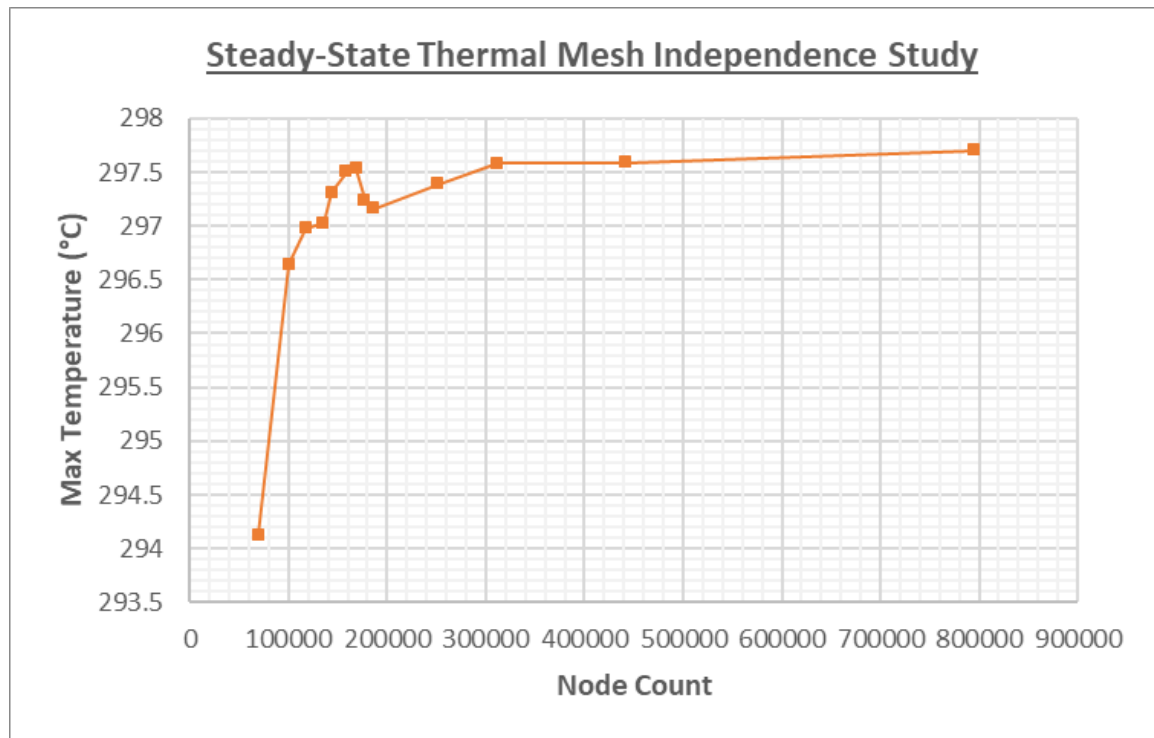


Figure 7- Node Count comparison to maximum temperature (largest node count omitted for graph clarity).

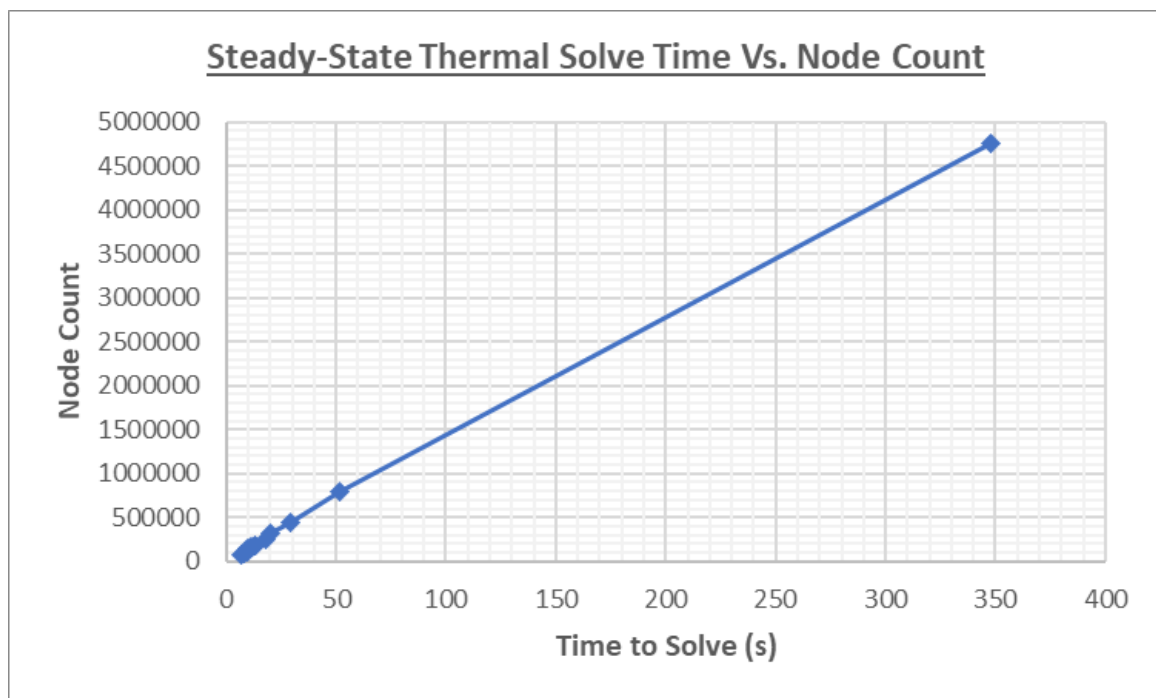


Figure 8 - Time to solve vs. node count. The time to solve dependent on node count.

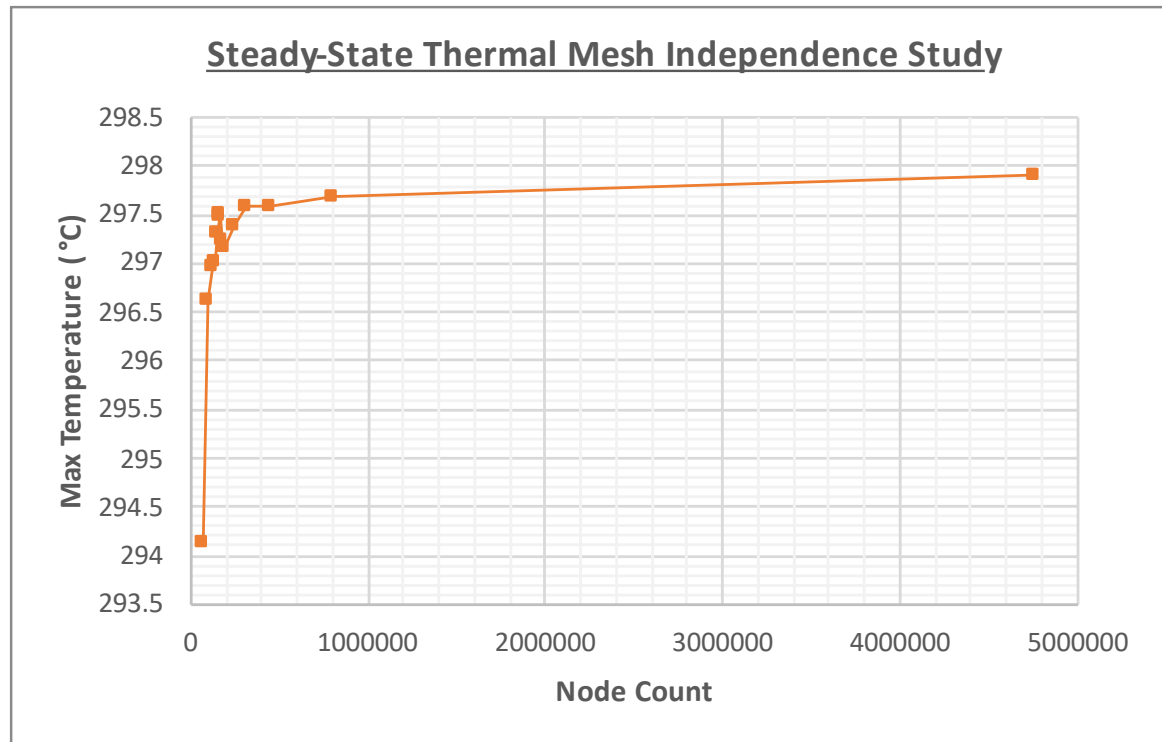


Figure 9 - Inclusion of the finest mesh size, 0.0002 m, 4761497 nodes, shows minimal impact on the result.

3.3. Static Structural Study

The Static Structural module determines the displacements, forces, stresses and strains in structures. In this module, the Ansys software assumes that the forces are applied slowly, without inducing damping or inertias. Parts set as rigid parts are viewed as point masses by the software. Components that are under analysis are set as flexible components. Additionally, if buckling or large deformations are expected, the “large deformation” setting must be selected in order to take into effect the changes in stiffness due to changes in shape (Ansys, 2021).

Selection of the correct large deformation setting is important to the result. The bimetallic strip validation example saw a 0.1 % change in maximum deformation. Solely, 0.1 % is not a large amount, but when combined with mesh sizing variances, the potential for errors to grow larger increase. Ultimately, it falls to the user to recognise the potential for deformation within the component.

Combustion pressure was applied to the piston crown. Similar to temperature conditions in the thermal study, applied pressure relies on experimental data for realistic values to be used. In this case 100 bar was used to simulate combustion pressure (engine simulations of a Ford Sigma and 4 motorcycles gave combustion pressures of 65 to 105 bar). In this scenario, user knowledge was required to apply the pressure to the correct areas of the piston. Separate analysis could determine combustion pressures using Ansys Chemkin if experimental data was difficult to capture or unavailable.

3.4. Fully Linked Study

The linked study combines the previous modules, combining the thermal deformation effects with the force deformations. Figure 10 shows how the modules are linked together. The engineering data is responsible for material data and so linking this forward (or downstream) reduces user work inputting information. Despite this, when creating a new module, it is important to check the material assignment of each component of the assembly as it is not always carried forward.

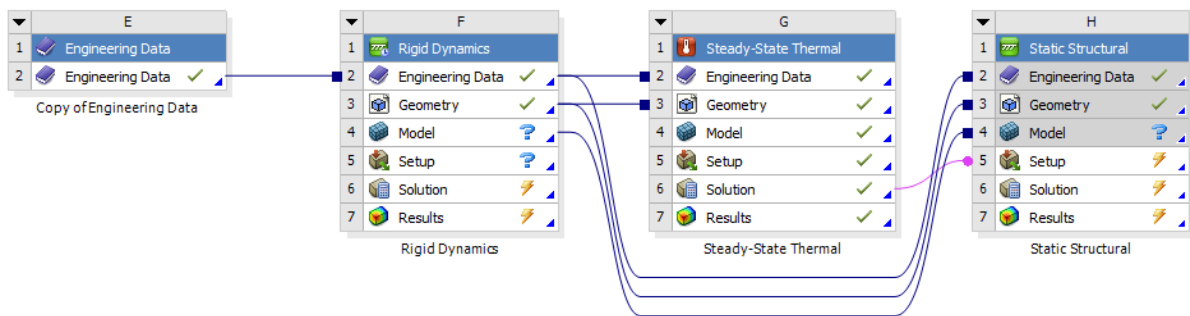


Figure 10 - The Ansys workflow. Modules are linked to allow results to be carried downstream.

To import the Rigid Dynamics study results to the Static Structural results, the linking technique does not work. A snapshot of the Rigid Dynamics result, in this case the maximum stresses, is exported as a .txt file. Opening the Static Structural module, the .txt file containing the results then needs to be imported. For a fatigue study, both maximum and minimum stresses would need importing, increasing potential for this step to be missed, therefore giving incorrect results. A change within the Rigid Dynamics module would also require a new .txt document to be saved and imported. The Steady-State Thermal solution can be updated automatically, it can be seen linked in Figure 10 which is more intuitive for the user.

At this stage of the fully linked system, making changes to geometry or to the Rigid Dynamics module became very difficult, usually resulting in completely restarting the analysis. The links between modules could not be removed.

4. Iterative Design of a Piston

The provided piston design gave a thermal profile as shown in Figure 11. The temperature peaked at 345.07 °C on the crown of the piston. The mass of the provided piston was 0.2614 kg. The aim of redesigning the piston was to reduce weight and temperature. A lighter piston reduces fuel consumption (Pachpande, 2019) and reduces reciprocating mass improving performance. A cooler piston can withstand greater stress from reciprocating motion and combustion pressures.

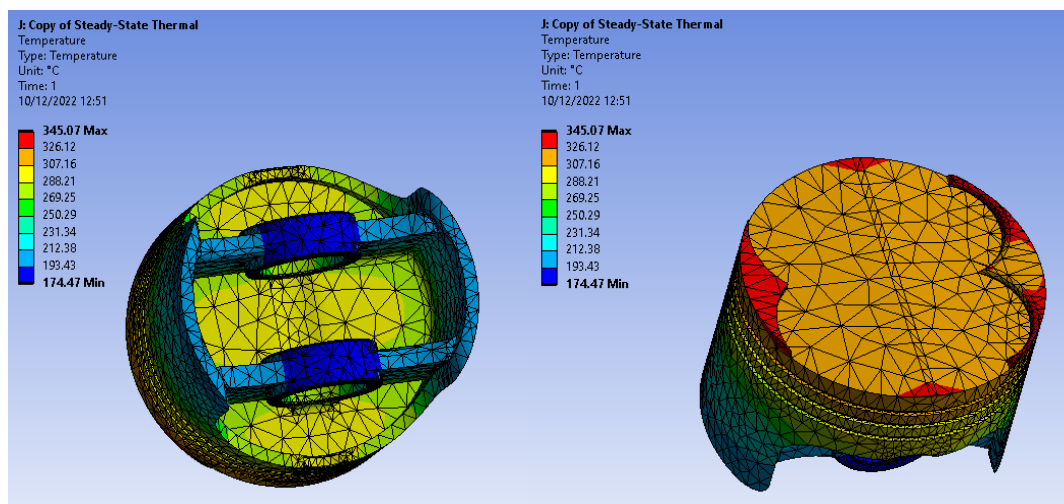


Figure 11 - A thermal study of the provided piston design.

The first design iteration removed mass from the piston and increased surface area with the aim of producing a heat sink (Arrow, 2019) on the underside of the piston as shown in Figure 12.

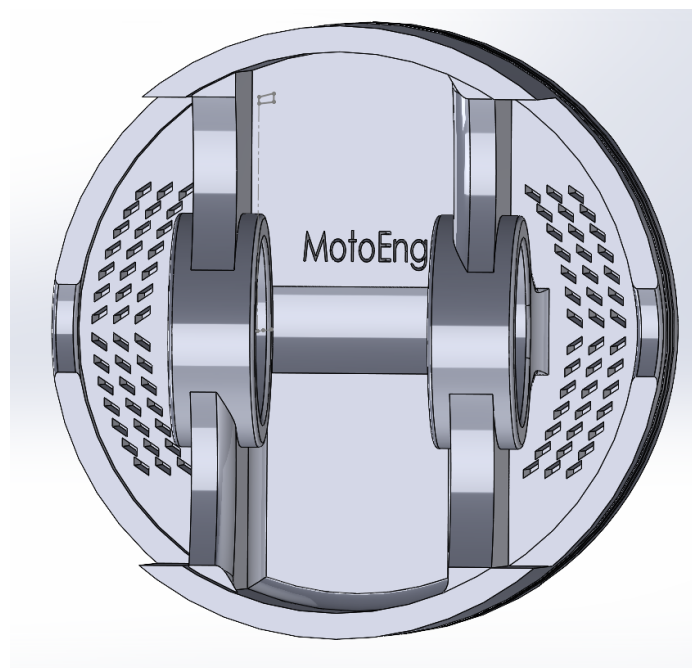


Figure 12 - The first design iteration with material cut away from the underside of the piston.

Figure 13 shows the results of the thermal analysis. It is immediately clear that the piston is cooler from the visual temperature gradient. Tabulated results are shown in Table 3. A 13.9 % reduction in peak temperature and a 19.9 % reduction in average temperature was achieved with a mass reduction of 1.89 %.

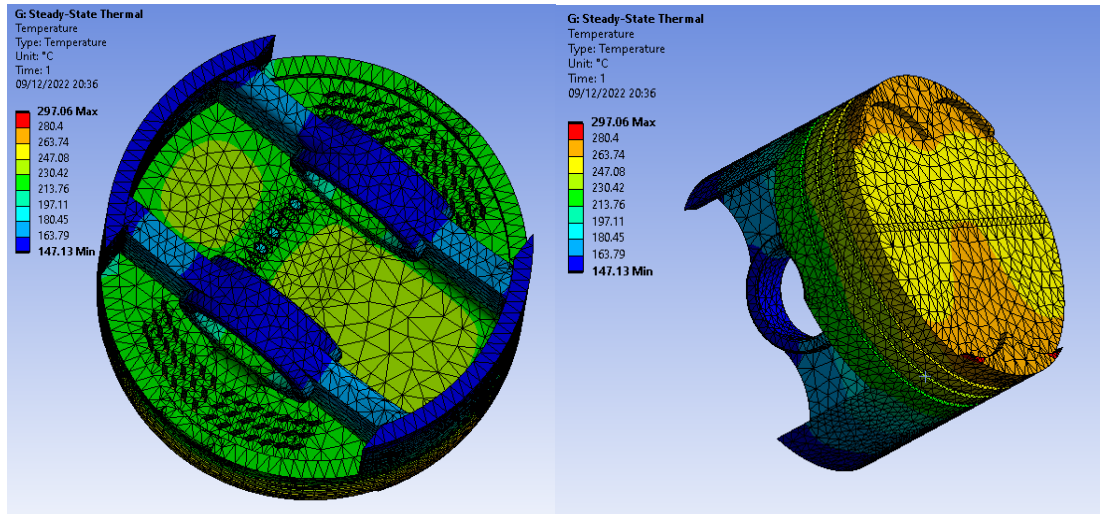


Figure 13 - Design iteration 1 thermal study, showing lower average and lower maximum temperatures.

Table 3 - Thermal comparison of the two pistons.

Piston Design Iteration 1		Thermal Study Results (°C)		
Mesh Size (m)	Mass (kg)	Max Temperature	Average Temperature	Nodes
0.004	0.25645	297.06	217.93	69964
Original Piston Design		Thermal Study Results (°C)		
Mesh Size (m)	Mass (kg)	Max Temperature	Average Temperature	Nodes
0.004	0.26140	345.07	272.40	40068

The mass reduction was minimal however, the minimal removal of material from the piston has made significant progress in thermal management. Progressing further with the study, it can be seen from the static structural study, Figures 14 and 15, how the maximum deformation of the new piston is reduced. A maximum deformation of 0.46142 mm was calculated for the original piston which reduced to 0.29502 mm in the new design (a 36 % reduction). This is summarised in Table 4.

During the setup of the new piston analysis, it was found that the more intricate part caused difficulties in assigning thermal boundary conditions. The heat sink on the underside of the piston and MotoEng text cut-outs were difficult to select with the cursor. Each face of each cut-out needed to be selected in order to apply the thermal conditions requiring the piston to be rotated multiple times and the zoom settings changed in order to be able to view and select the faces. This totalled 596 faces for the underside of the piston. If a second iteration was performed, these faces would need to be re-selected and assigned thermal boundary conditions. Missing these faces would provide inaccurate results. This also applies to joints in the Rigid Dynamics Study.

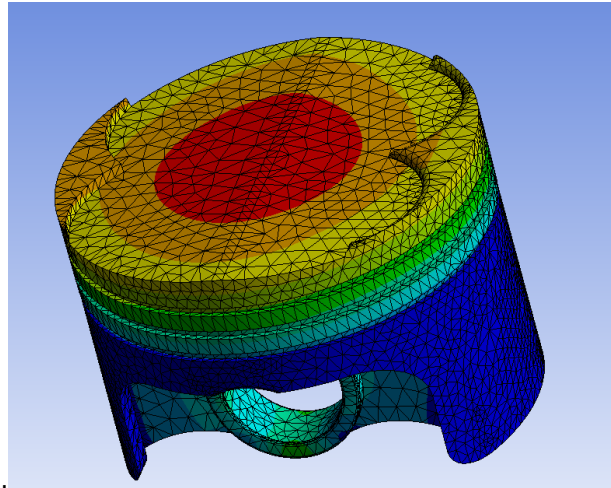


Figure 14 - The overall deformation of the original piston.

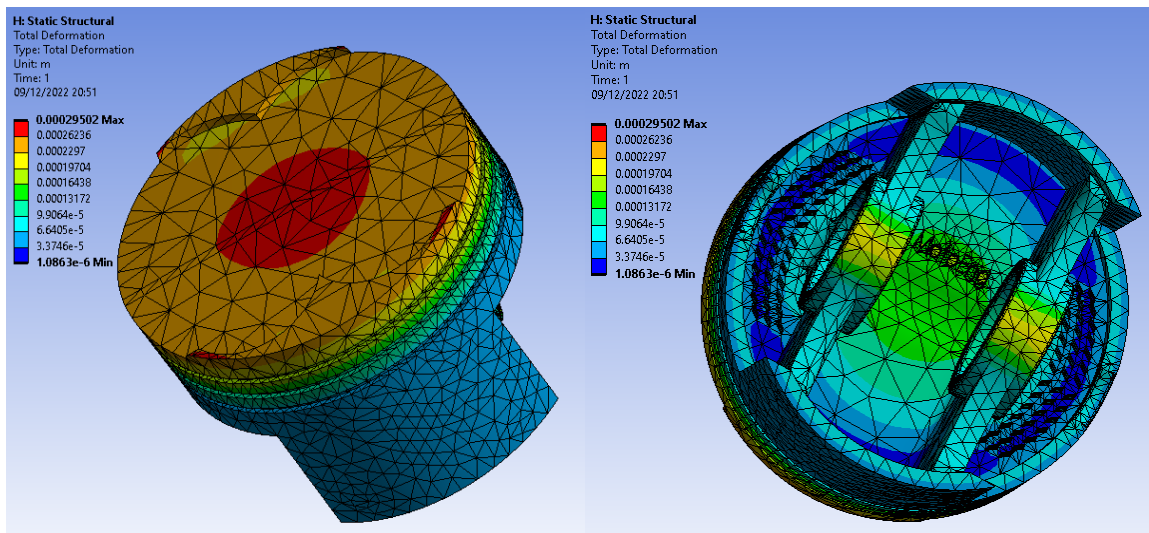


Figure 15 - The overall deformation of the newer piston design.

Table 4 - Summary of the deformation reduction between the original piston and the new design.

Fully Linked Piston Deformation Analysis	
<u>Original Piston Design</u>	
Max Deformation (mm)	Average Deformation (mm)
0.46142	0.15509
<u>New Piston Design</u>	
Max Deformation (mm)	Average Deformation (mm)
0.29502	0.11691
Reduction in Max Deformation (%)	Reduction in Average Deformation (%)
36%	25%

The next iteration of the piston design would aim to remove more weight. The analysis showed max deformation occurring at the top of the crown around the valve cut-outs. Smoothing or filleting these edges may lower the temperature and would be attempted in further iterations. Figure 16 shows further cut-outs made in the underside of the piston.

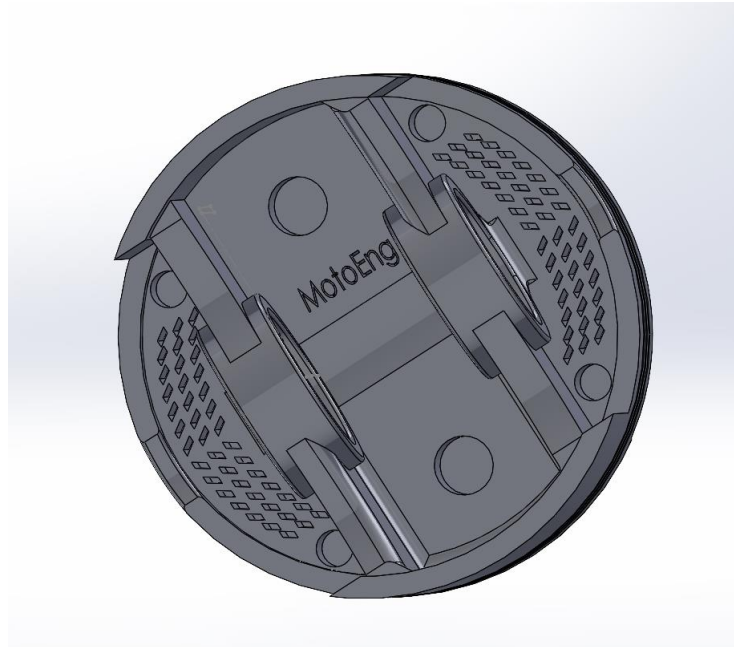


Figure 16 - A possible iteration of the design, removing more material to reduce mass.

5. Conclusion

CAE software is a powerful tool for engineers. It enables prototypes to be designed and tested without material and manufacturing costs which can also help a company promote sustainability, an increasingly important factor at all stages of the lifecycle of a component. Large amounts of component data can be generated relatively quickly by a single user at a PC workstation, cutting costs. The user must be trained in the CAE software, but this could be argued to be less costly than the training and experience needed to manufacture components and / or conduct physical testing. The accuracy of results relative to reality is high as shown by the bimetallic strip validation, but CAE relies on the accuracy of the input data and geometry of the component in question. Setting up a CAE analysis based on poor boundary conditions or without an understanding of the assembly dynamics, will not produce accurate results. The information the CAE software is fed, in addition to the proficiency of the user has a profound effect on the validity and accuracy of results.

6. References

Ansys 2021, "*Mechanical application 2021 R1, Mechanical User's Guide*", Ansys Inc., accessed 10-Dec-2022, available at [https://ansyshelp.ansys.com/account/secured?returnurl=/Views/Secured/corp/v211/en/wb2_help/wb2_h_transstructMBD.html?q=rigid%20dynamics].

Arrow 2019, "*Heat Sinks: Functions, Types & More*", Arrow.com, 31-Oct-2019, accessed 11-Dec-2022, available at [https://www.arrow.com/en/research-and-events/articles/understanding-heat-sinks-functions-types-and-more].

Dake M.R., FitzWilliam J., Henderson, M., Shaw, J. et al., "*Considerations for CFD Simulations of a Refueling Pump Nozzle with Application to the Computer Aided Engineering of a Vehicle Refueling System*", SAE Technical Paper 2018-01-0489, 2018, doi:10.4271/2018-01-0489

Digital School 2021, "*CAD: A Brief History*", Digital School Technical Design College, article modified 01-Mar-2021, Accessed 11-Dec-2021, available at [https://www.digitalschool.ca/cad-a-brief-history/].

Dlupal Software 2018, "*Verification Example – 0078 Bimetallic Strip*", accessed 10-Dec-2022, available at [https://www.dlupal.com/-/media/B437DA0115FB496C9C7BA58F08960A5E.ashx], provided by A. Noble for the purposes of this report.

Featips 2022, "*What is a Mesh Independence Study?*", featips.com, article modified 11-Oct-2022, accessed 10-Dec-2022, Available at [https://featips.com/2019/03/31/what-is-a-mesh-independence-study/].

Jalal M., Goharzay M. 2018, "*Cuckoo search algorithm for applied structural and design optimization: Float system for experimental setups*", Journal of Computational Design and Engineering, Vol. 6, Issue 2.

Pachpande L., Patil R. 2019, "*Review on Design and Comparative Analysis of Piston by Ansys*", Department of Mechanical - SGDCOE Jalgaon Maharashtra India, IJESC, Vol. 9, Issue 6, ISSN 2321 3361.

Slater J. 2021, "*Overview of CFD Verification and Validation*", NPARC Alliance CFD Verification and Validation Web Site, NASA, article modified 10-Feb-2021, available at [https://www.grc.nasa.gov/WWW/wind/valid/tutorial/overview.html]

Solaz S. 2022, "*Verification and Validation in Computer-Aided Engineering – Explained*", Simscale.com, article modified 19-Sep-2022, available at [https://www.simscale.com/blog/verification-and-validation-cae/].

Winship J. 1967, "*Designing an Automotive Engine Piston*", 1967 Automotive Engineering Congress and Exposition, Technical Paper 670020, ISSN 0148-7191.

Young W., Budynas, R., Sadegh, A. 2011, "*Roark's Formulas for Stress and Strain*", 8th Edition, McGraw-Hill Education, 2011.

Solidworks parts (crankshaft, conrod, gudgeon pin and piston) provided by A. Noble.

Experimental data for thermal boundary conditions and material data provided by A. Noble.

7. Appendix

7.1. Tabulated Mesh Independence Values

Piston Design Iteration 1		Mesh Independence Test - Steady-State Thermal	
Mesh Size (m)	Nodes	Max Temperature (°C)	Elapsed Time (s)
0.00400	69964	294.13	7
0.00200	100643	296.64	8
0.00180	117656	296.98	10
0.00160	135578	297.03	10
0.00150	144247	297.31	10
0.00140	158783	297.50	11
0.00130	169149	297.53	12
0.00125	176985	297.24	13
0.00120	186350	297.16	13
0.00100	251653	297.39	18
0.00090	312233	297.58	20
0.00070	442274	297.59	29
0.00050	795924	297.70	52
0.00020	4761497	297.91	348

7.2. Provided Boundary Conditions

Thermal Study Boundary Conditions

	Heat Transfer Coefficient					Temperature (Celsius)				
	1	2	3	4	Average	1	2	3	4	Average
Crown	600	350	500	540	497.5			928.5		928.5
Crown Side	500	540	350	400	447.5	250	650	500	300	425
Top Groove	300	650	200	600	437.5	180	150	180	250	190
Middle Groove	300	650	200	600	437.5	180	150	180	250	190
Oil Groove	260	520	175	400	338.75	165	150	170	170	163.75
Underside	1500	750	250	200	675	95	90	90	90	91.25
Skirt	1000	400	150	300	462.5	110	110	120	90	107.5

Structural Study Boundary Conditions

	Pressure	Inertia	Side
Max P	10537060	11371.29	7440.469
Max I	304222.2	18089.83	-1.10E-12
Max S	8907806	6954.198	8707.43

6061-T6 Aluminium Material Properties

Temp (°C)	Young's Modulus		Poisson's Ratio	0.33
	Value	Unit		
22	67	Gpa	Coefficient Thermal Expansion 22°C	2.36E-05 1/°C
100	70	Gpa	Thermal Conductivity	167 W/m-K
150	68	Gpa	Specific Heat Capacity	0.896 J/g-°C
200	64	Gpa		
250	56	Gpa		
300	48	Gpa		
350	41	Gpa		
400	32	Gpa		
450	21	Gpa		
500	12	Gpa		