

University of Wales Trinity Saint Davids

Wales Institute of Science and art



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

Application of CAE Processes

Structural & Fluid Analysis

Cody Surridge

2304028

14/12/2025

Table of Contents

1	Introduction	3
2	Validation	4
2.1	Linear Analysis	4
		4
2.1.1	Hand Calculations.....	5
2.2	Mesh Study	6
2.3	Thermal Validation.....	8
3	Kinematic Simulation & Structural Analysis	10
3.1	Rigid Dynamics.....	10
3.2	Steady State Thermal.....	11
3.3	Static Structural	12
4	Optimisation & Improvement	13
5	Conclusion.....	19
6	References.....	20

Table 1-Cantilever Beam Dimension & Properties.....	5
Table 2-Thermal Validation Geometry and Material Properties (Dlubal, 2025)	8
Table 3- Aluminium 6061-T6 Material Properties (ANSYS (Granta), 2015).....	10
Table 4- Aluminium 6262-T9 Properties (ANSYS (Granta), 2015)	13
Table 5- Aluminium 2618-T6 Properties (ANSYS (Granta), 2015)	13
Table 6- Stainless Steel AISI 410 Properties (ANSYS (Granta), 2015)	14

Equation 1- Maximum Displacement.....	5
Equation 2-Inertia	5
Equation 3-Maximum Stress (Von Misses).....	5
Equation 4-Moment	5

Figure 1- Static Structural Analysis Setup Box.....	4
Figure 2-Cantilever Load	4
Figure 3-Cantilever Beam.....	4
Figure 4- ANSYS Fixed Support.....	4
Figure 5-Cantilever with Maximum Stress =1.5314e+009 Pa	4
Figure 6- Cantilever with Maximum Deformation =9.3085e-002m.....	5
Figure 7-Cantilever Beam with the Dimensions 10X10X250 mm	5
Figure 8- A Graph Showing the Maximum Stress Vs Mesh Size	6
Figure 9-ANSYS Simulating High Level of Stress in the Corners	7
Figure 10-A Graph Showing Maximum Displacement Vs Mesh Size.....	7
Figure 11- Thermal Validation in ANSYS.....	8
Figure 12- Piston Assembly	10
Figure 13- Graph of the Loads applied to the Gudgeon Pin to Piston Connection	11

Figure 14- 6061-T6 Thermal Study.....	11
Figure 15- 6061-T6 Deformation.....	12
Figure 16- 6061-T6 Stress.....	12
Figure 17- Graph Showing Max & Min Temperature	14
Figure 18-Graph showing Max Stress.....	15
Figure 19-2618-T9 Stress	15
Figure 20-6262-T9 Stress	16
Figure 21- AISI 410 Stress.....	16
Figure 22-Graph showing the max deformation of each material.....	16
Figure 23-2618-T6 Deformation.....	17
Figure 24-6262-T9 Deformation.....	17
Figure 25-AISI 410 Deformation.....	18

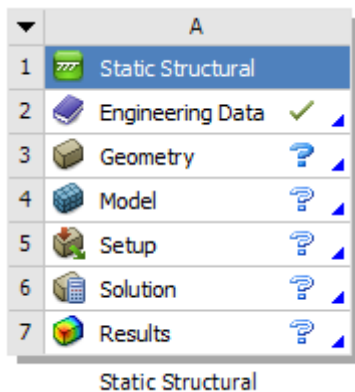
1 Introduction

In this report I will be covering validation examples of both a cantilever beam and bimetallic strip to identify how effective ANSYS is in producing static structural and thermal loads compared to results gathered from hand calculations. Using the validation example from the cantilever beam I will conduct a mesh study to assess the impact reducing the mesh size has upon the simulated result. Next, I will be developing a fully linked system in ANSYS using rigid dynamics, static structural and steady-state thermal. I will use this system to understand how well 6061-T6 Aluminium is suited in use of a piston assembly and optimise and improve the assembly by changing the material.

2 Validation

2.1 Linear Analysis

For my validation example of the static structural analysis system, I will be modelling a basic cantilever beam in SolidWorks. This allows me to control the known dimensions. From there, the model will be run in ANSYS where the maximum displacement as well as von misses stress will be extracted and compared to hand calculations.



The next step is to run the cantilever beam in ANSYS using the static structural analysis system. Setting up the system involves setting up the engineering data by inputting material properties of Aluminium 6061-T6 from the provided data sheet, importing the cantilever geometry with dimension listed in table 1 , setting up the model by applying fixed geometry and loads to the end of the cantilever beam, applying a suitable mesh to prevent anomalous results that ANSYS calculates incorrectly. Finally, running the model will allow myself to note down the maximum displacement and stress of 1000N acting upon the cantilever beam.

Figure 1- Static Structural Analysis Setup Box

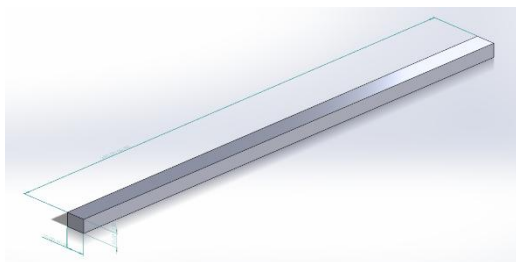


Figure 3-Cantilever Beam

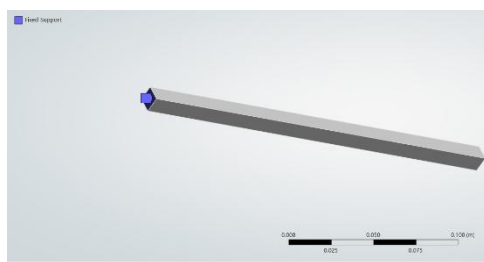


Figure 4- ANSYS Fixed Support

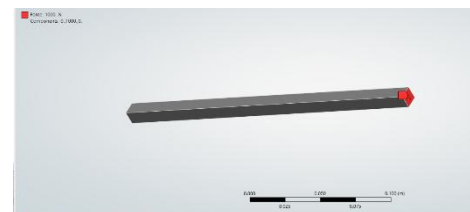


Figure 2-Cantilever Load

Running the cantilever give data showing the range of displacement as well as the stresses applied along the beam. Using the found maximum values I will then compare to calculation done by hand to prove the accuracy of the Static Structural system in ANSYS.

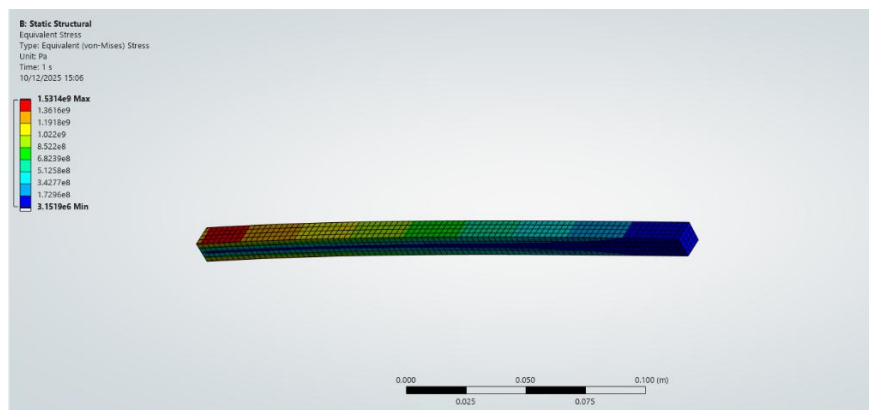


Figure 5-Cantilever with Maximum Stress =1.5314e+009 Pa

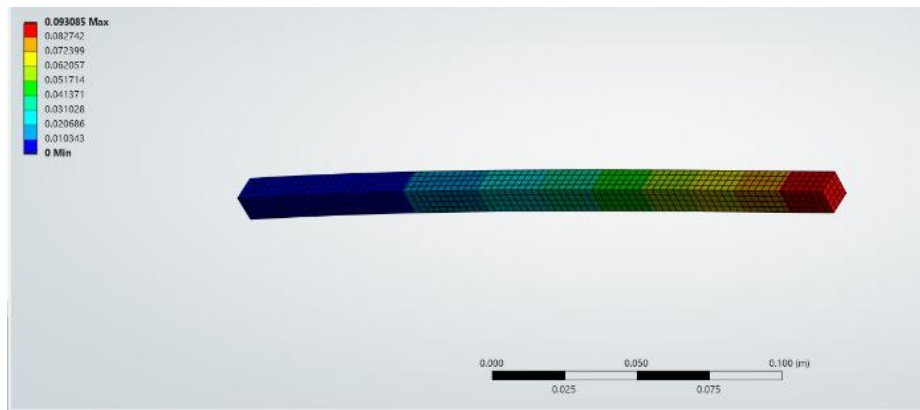


Figure 6- Cantilever with Maximum Deformation =9.3085e-002m

2.1.1 Hand Calculations

2.1.1.1 Equations and values

Equation 1- Maximum Displacement

$$\delta b = \frac{FL^3}{3EI}$$

Equation 2-Inertia

$$I = \frac{bh^3}{12}$$

Equation 3-Maximum Stress (Von Misses)

$$\sigma = \frac{yM}{I}$$

Equation 4-Moment

$$M = F * L$$

Figure 7-Cantilever Beam with the Dimensions

Table 1-Cantilever Beam Dimension & Properties

Force (F) -N-	Length (L) -mm-	Young's Modulus (E) -GPa -	base (b) -mm-	height (h) -mm-	y -mm-
1000	250	67	10	10	5

2.1.1.2 Finding δ , I and σ

$$I = \frac{10mm * 10mm^3}{12}$$

$$I = \frac{10000mm^4}{12}$$

$$I = 833.33mm^4$$

$$\delta b = \frac{1000N * 250mm^3}{3 * 67 \times 10^9 * 833.33mm^4}$$

$$\delta b = 93.283582mm$$

$$\delta b = 0.0932836m$$

$$M = 1000N * 250mm$$

$$M = 250000Nmm$$

$$\sigma = \frac{5mm * 250000Nmm}{833.33mm^4}$$

$$\sigma = 1500.06Nmm^{-2} \text{ or } \sigma = 1.50 \times 10^9 Nm^{-2}$$

Comparing against the simulated result from ANSYS, the maximum displacement and stress in ANSYS is 9.3085e-002m and Cantilever with Maximum Stress =1.5314e+009 Pa whereas, the hand calculations give a maximum displacement of 0.0932836m and stress of $1.50 \times 10^9 Nm^{-2}$ this gives an error of 2.05% and 0.21% respectively. Meaning that in regard to the stress the it is very accurate but in a region before the mesh starts to over calculate results in the corners, whereas for the displacement the result is accurate up to 2 decimal places but inaccuracies have occurred due to mesh size being larger and therefore having less detail meaning calculations will be further away.

2.2 Mesh Study

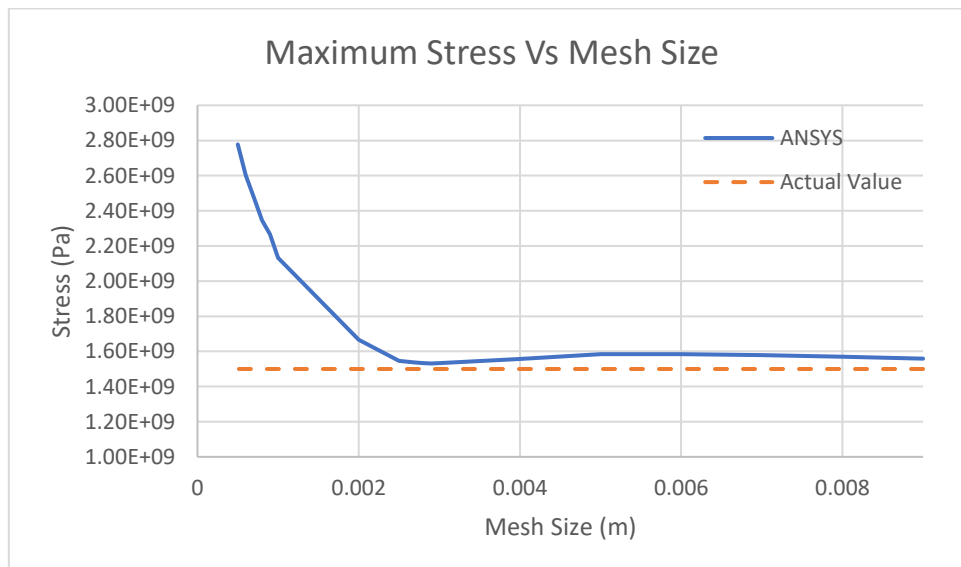


Figure 8- A Graph Showing the Maximum Stress Vs Mesh Size

Figure 8 shows that the mesh's accuracy is improving from 0.006m to 0.003m due to ANSYS being able to more accurately calculate stress acting upon the cantilever beam. Despite this, the mesh's accuracy begins to decline around 0.0025m and exponentially increase from around 0.001m. This means that the mesh is ineffective from this point onwards due to results being skewed. These errors are caused because of the mesh being finer as this increases the quantity of potential rounding

errors which would cause deviation from the actual result. This is then shown in the corners of the ANSYS simulation where there is area of significantly higher stress calculated than meshes with less detail.

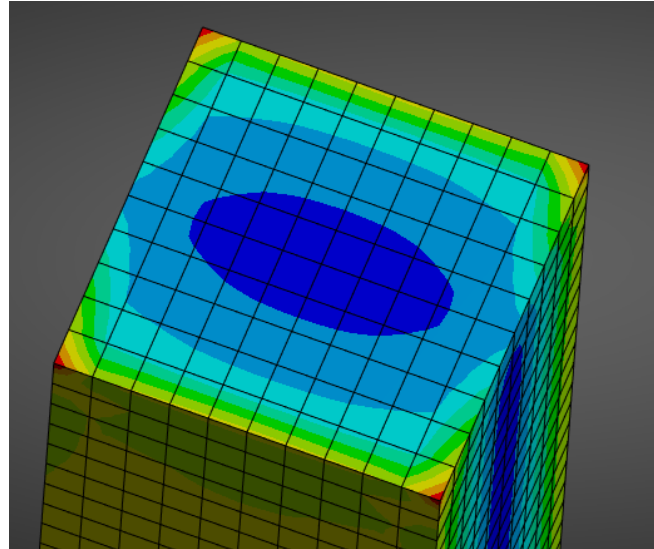


Figure 9-ANSYS Simulating High Level of Stress in the Corners

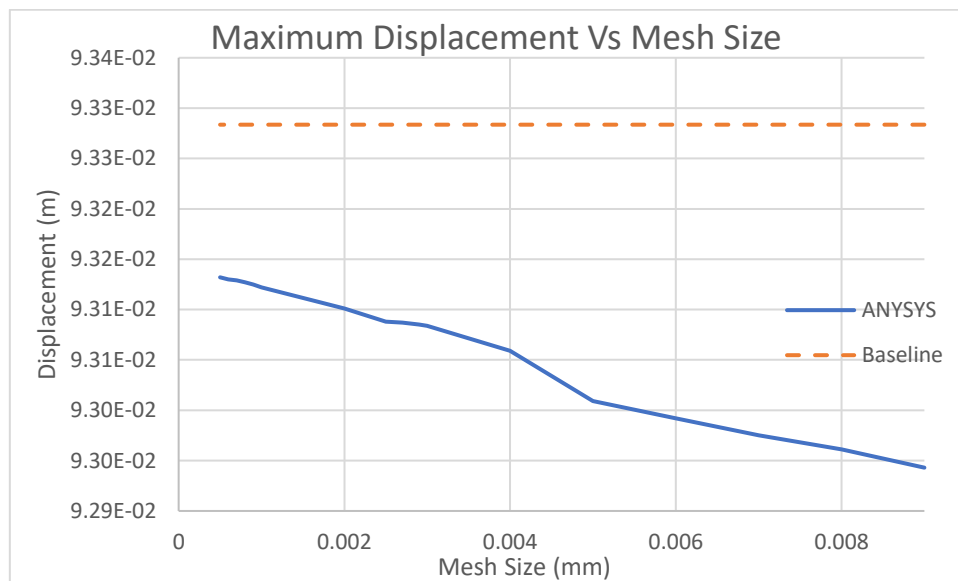


Figure 10-A Graph Showing Maximum Displacement Vs Mesh Size

Figure 10 shows the maximum displacement vs mesh size. Figure 10 shows that as the mesh size decreases the simulated displacement is becoming more accurate based on the calculated result. Despite this, the largest increase in mesh size occurs between 0.005m and 0.004m whereas the increase after 0.002 is significantly lower. This means that the effectiveness of decreasing the mesh size is reduced and may not be worth the extra time it takes to run.

Comparing Figures 8 and 10 the best mesh size for the cantilever beam is at 0.0029 where the percentage errors are 2.05% for the stress, which is the closest to the hand calculated value, Whereas

the displacement has an error of 0.21% which is the smallest error that occurs before the simulated stress begins to significantly decrease.

2.3 Thermal Validation

Table 2- Thermal Validation Geometry and Material Properties (Dlupal, 2025)

Material	Invar	Modulus of Elasticity	E _i	137000	MPa
		Poisson's Ratio	ν _i	0.28	-
		Coefficient of Thermal Expansion	α _i	1,200 e-6	°C-1
	Copper	Modulus of Elasticity	E _c	130000	MPa
		Poisson's Ratio	ν _c	0.354	-
		Coefficient of Thermal Expansion	α _c	2 e-5	°C-1
Geometry		Cross-Section Width	w	5	mm
		Layer Thickness	t=t _i =t _c	1	mm
		Length	L	100	mm
Load		Thermal Loading	T _c	100	°C

For the thermal validation I have used material data values from figure 11 to run a static structural analysis of a bimetallic strip consisting of Invar and Copper. The methodology of this begins the same as the linear analysis where I have inputted the engineering data, imported geometry with the dimensions listed, generated a suitable mesh and applied a fixing. However instead of applying a force on the free end a thermal loading of 100°C was applied being careful to adjust the temperature from an ambient of 22°C to 122°C to simulate the full 100°C of temperature increase.

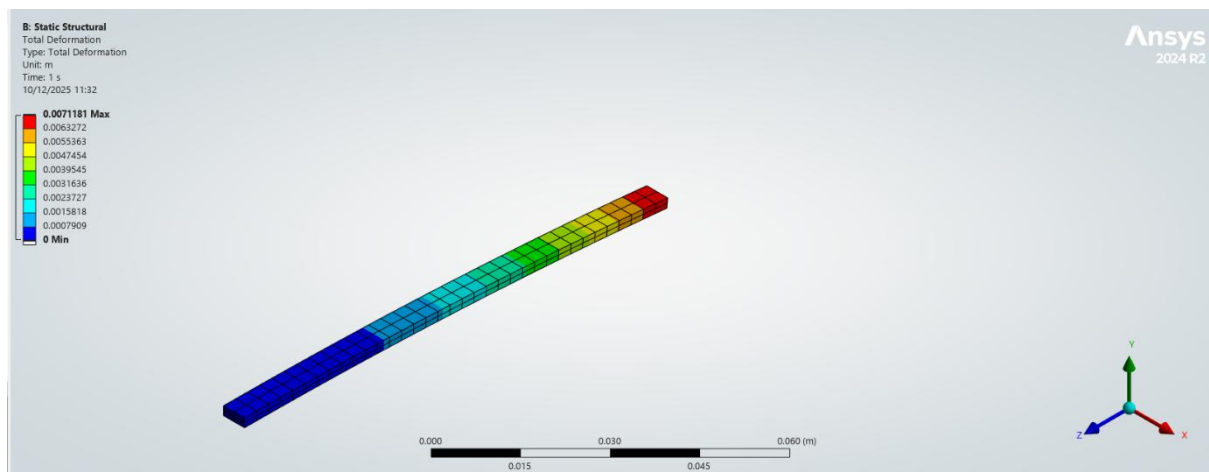


Figure 11- Thermal Validation in ANSYS

Running the geometry in ANSYS has given a simulated deflection due to thermal loading of 0.0071181m or 7.1181mm. This compares to the calculated value of 7.049mm to give a percentage error of 0.97% this means that ANSYS is accurate predicting the deformation due to thermal loading, however the result given through the simulation could be made more accurate. This could happen by reducing the mesh's size as this helps by causing more calculations to occur, despite it also could cause more errors by increasing the quantity of rounding errors.

3 Kinematic Simulation & Structural Analysis

3.1 Rigid Dynamics

To run the kinematic simulation and structural analysis in ANSYS I have had to set up and use three different analysis setup boxes. These are rigid dynamics, steady-state thermal and static structural.

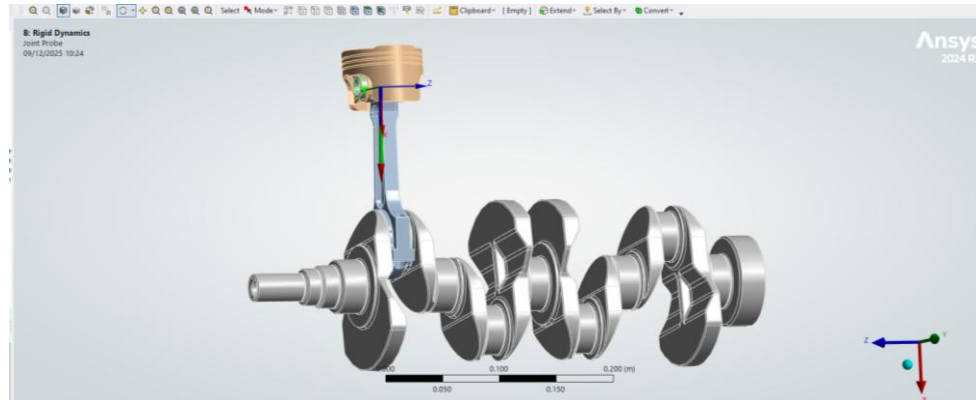


Figure 12- Piston Assembly

I conducted a kinematic simulation of the piston assembly (shown above) that was provided by using the rigid analysis setup in ANSYS. Setting up the analysis system involves, applying a different material to the piston head which in the baseline test is Aluminium 6061-T6, with material properties shown in table 3 This is because the piston is the component in the assembly where the load is applied.

Table 3- Aluminium 6061-T6 Material Properties (ANSYS (Granta), 2015)

6061-T6			
	Temperature (°C)	Low	High
Young's Modulus (Gpa)	22	69.1	69.1
	100	66.4	66.4
	150	64.6	64.6
	200	62.6	62.6
	250	60.4	60.4
	300	57.5	57.5
	350	53.8	53.8
	400	49.3	49.3
	450	42.8	42.8
	500	35.7	35.7
Density	2730		
poisson's Ratio	0.335		
thermal coefficient	2.39E-05		
Thermal Conductivity	169		

Next, I have deleted the automatically generated contacts and turned the connections auto detection to off as this prevent unwanted connections from generating in incorrect locations during later analyses. As a result, I have had to apply my own connections to act and boundary conditions. These are body-body revolute contacts between the crankshaft and conrod, gudgeon pin to conrod and gudgeon pin to piston. Furthermore, I have applied a revolute body-ground contact on the crankshaft and a general contact on the piston as this restricts the pistons movement to be only in the z axis and therefore simulating being confined in a cylinder.

After applying the connections, I then told ANSYS I wanted to look at the load on the gudgeon pin to piston connection when a rotational load of 6000RPM is applied to the crankshaft.

Simulating this has given a graph showing the load applied to the connection against time. Using this graph I have exported motion loads from the peaks of the graph below as these are the points at which the applied load is largest and therefore the largest stress and deformation would occur at. This then allows me to simulate the worst-case scenario in which the assembly would go through.

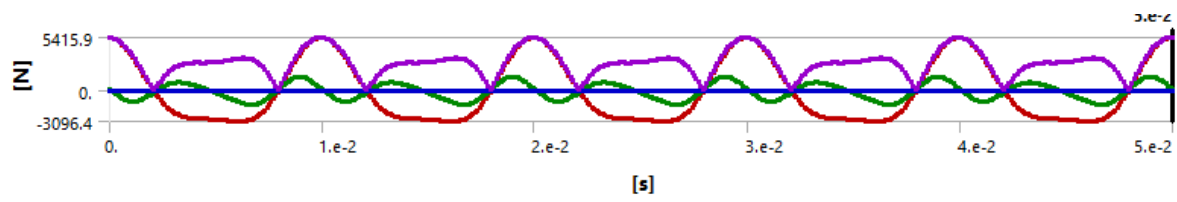


Figure 13- Graph of the Loads applied to the Gudgeon Pin to Piston Connection

3.2 Steady State Thermal

To start with the thermal analysis, I have suppressed every component bar the piston. This causes ANSYS to only simulate the thermal loading that would be acting upon the piston. The next step was to generate a suitable mesh for the piston geometry. For the simulation I have used a 4mm mesh.

The final step is to apply the required boundary conditions for the thermal simulation. In this instance the boundary conditions applied are convections based on the data given through the piston thermal study sheet provided.

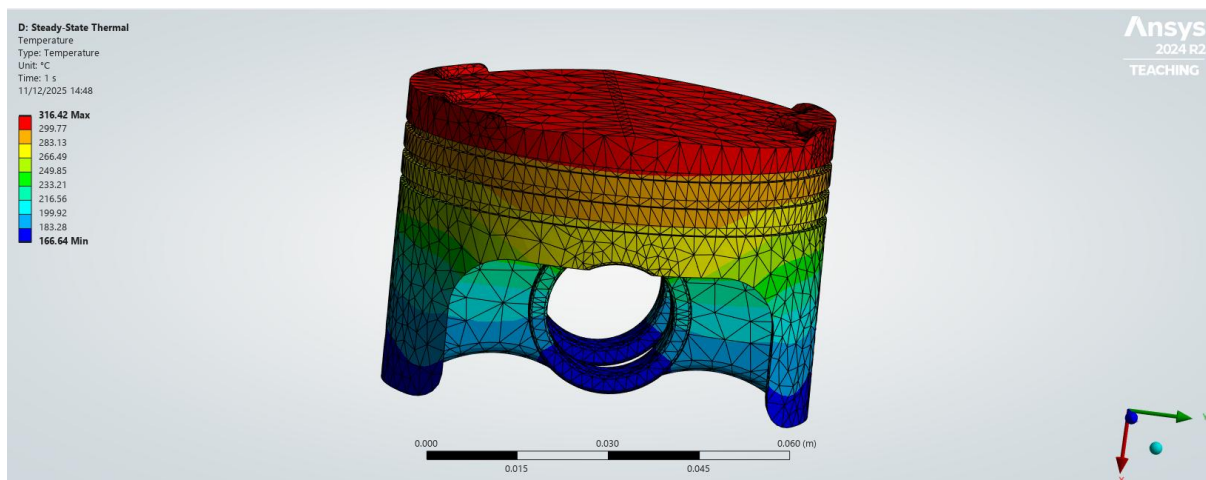


Figure 14- 6061-T6 Thermal Study

The simulation then shows that the highest temperature is on top of the piston at a maximum of 316.42°C with the temperature decreasing as it gets further to the skirt of the piston with a minimum of 166.64°C

3.3 Static Structural

Firstly, I had to change the piston be a flexible component. Similar to, the steady state thermal I have had to generate a 4mm mesh, however unlike the thermal simulation I have applied a sizing of 2mm to generate a mesh look greater detail to get a more accurate result on top of the piston. Next, I have turned on weak springs. This is so that the piston is simulated to be in a fixed position as well as preventing it from deflecting to infinity by having some small force applying at every point. The final step before running the simulation is to apply a boundary condition on top of the piston to simulate pressure inside.

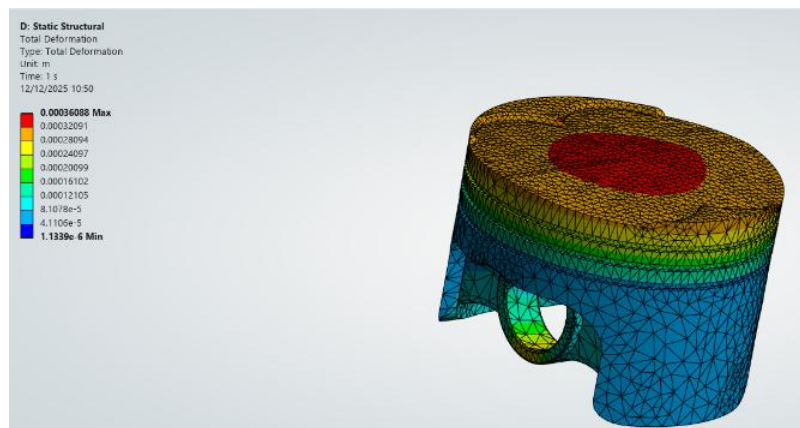


Figure 15- 6061-T6 Deformation

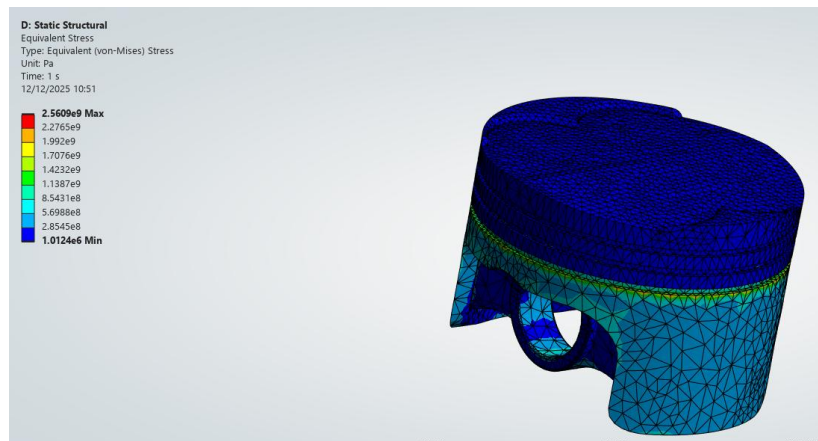


Figure 16- 6061-T6 Stress

Figure 15 and 16 shows the deformation and stress of the piston when the material is 6061-T6 aluminium. Figure 15 shows that there is a large area of deformation in the centre of the piston's top as well as small area of high area in the connection with the gudgeon pin. Furthermore, the deformation decreases towards the bottom of the piston skirt with the minimum deformation being below the bottom groove. Figure 16 shows that there the area of least stress is on top of the piston and the high stress is caused around the piston rings

4 Optimisation & Improvement

Using the 6061-T6 aluminium as a baseline material I can look at ways in which I can improve the stress and deformation of the piston. The way I have chosen to do this is by changing what material the piston is made from. The main locations on the piston I am looking to improve are the area of high deformation on top of the piston while reducing the overall deformation, as well as attempting to reduce the stress applied around the piston's rings.

Table 4- Aluminium 6262-T9 Properties (ANSYS (Granta), 2015)

6262-T9			
	Temperature (°C)	Low	High
Young's Modulus (Gpa)	22	70.1	70.1
	100	67.3	67.3
	150	65.5	65.5
	200	63.5	63.5
	250	61.3	61.3
	300	58.3	58.3
	350	54.6	54.6
	400	50	50
	450	43.4	43.4
	500	36.2	36.2
Density	2730		
poisson's Ratio	0.335		
thermal coefficient	2.54E-05		
Thermal Conductivity	179		

Table 5- Aluminium 2618-T6 Properties (ANSYS (Granta), 2015)

2618-T6			
	Temperature (°C)	Low	High
Young's Modulus (Gpa)	22	74.1	74.1
	100	71.2	71.2
	150	69.3	69.3
	200	67.1	67.1
	250	64.8	64.8
	300	61.6	61.6
	350	57.7	57.7
	400	52.9	52.9
	450	45.9	45.9
	500	38.3	38.3

Density	2780
poisson's Ratio	0.335
thermal coefficient	2.26E-05
Thermal Conductivity	154

Table 6- Stainless Steel AISI 410 Properties (ANSYS (Granta), 2015)

AISI 410, Hard Temper			
	Temperature (°C)	Low	High
Young's Modulus (Gpa)	22	212	212
	100	207	207
	150	203	203
	200	199	199
	250	195	195
	300	190	190
	350	185	185
	400	180	180
	450	174	174
	500	169	169
Density	7850		
poisson's Ratio	0.285		
thermal coefficient	1.10E-05		
Thermal Conductivity	27		

The materials I have chosen are a similar material with 6262-T9 Aluminium which has a similar young's modulus to the 6061-T6 but different thermal properties, 2618-T6 aluminium which as a greater young's modulus at all temperatures but has worse thermal properties and finally AISI 410 steel which has a significantly larger young's modulus with a lot smaller thermal values.

FOR ALL GRAPHS 6061-T6=1, 6262-T9=2, 2618-T6=3, AISI 410=4

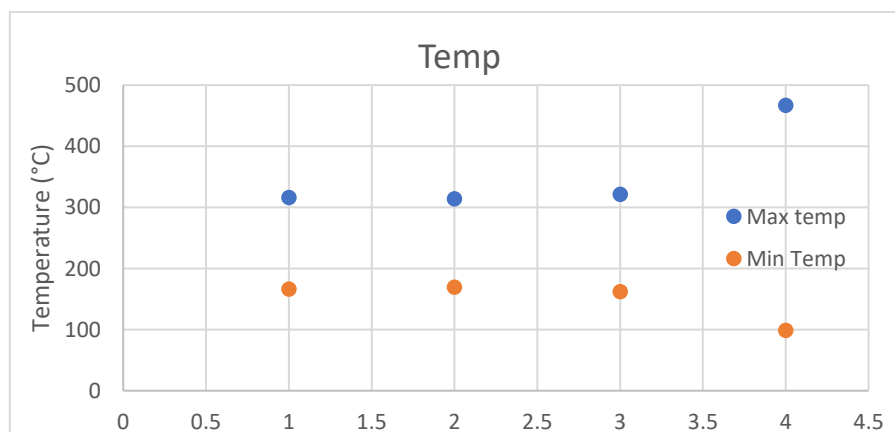


Figure 17- Graph Showing Max & Min Temperature

Figure 17 shows the maximum and minimum temperature for all four materials. The graph shows that 6262-T9 has lower max temperature but a higher minimum temperature than the 6061-T6. This is in contrast to the 2618-6 which has a lower minimum temperature and higher maximum temperature. Finally, it shows that steel's minimum temperature is the lowest while having the highest maximum temperature. This is a result of the AISI 304 having the worst thermal conductivity meaning that it struggles to dissipate changes in temperature

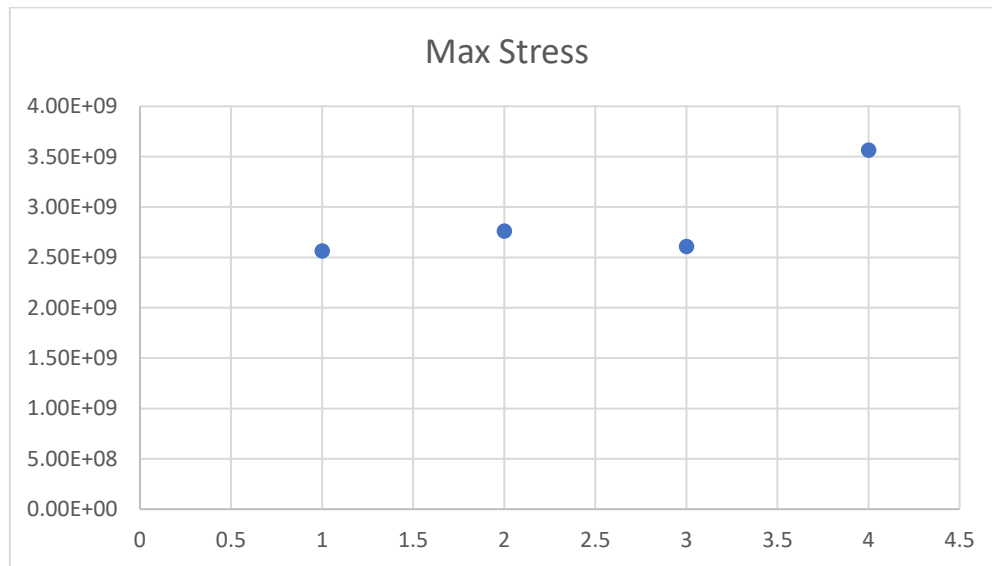


Figure 18-Graph showing Max Stress

Figure 18 is a graph showing the maximum stress that is applied for the piston when the material is changed. The graph above is showing that all three of the other materials have a higher maximum stress than the 60601-T6 but with the AISI 304 steel having a lot higher max stress. The area of highest stress on the 6061 is around bottom ring with frequent high areas.

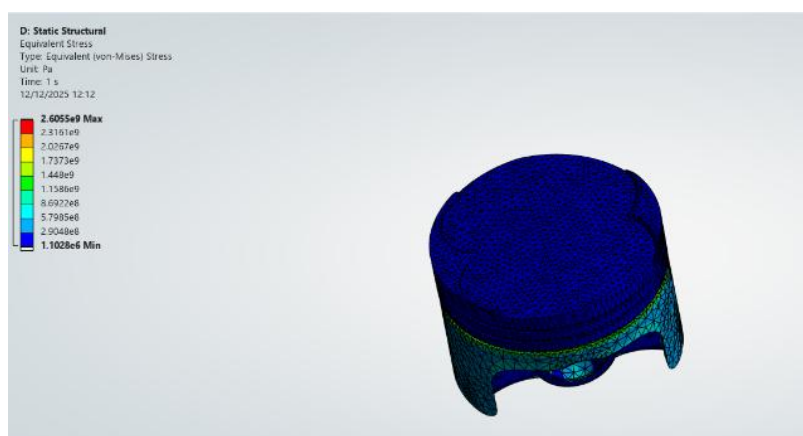


Figure 19-2618-T9 Stress

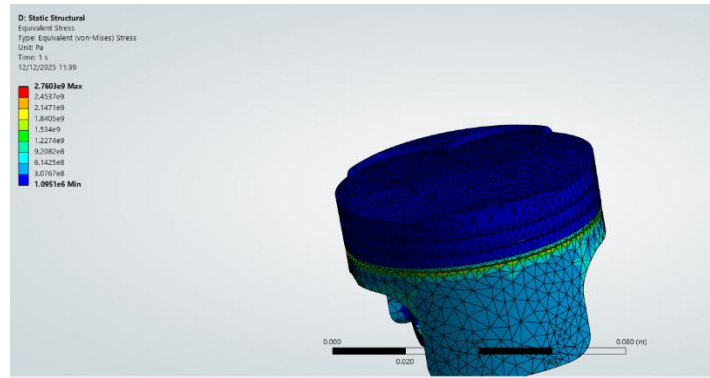


Figure 20-6262-T9 Stress

Figures 19 and 20 are the simulated stress of both 2618-T6 and 6262-T9. Both have higher max stress to the 6061-T6, but the areas of high stress are fewer and smaller than the 6061-T6. This is ideal as reducing the quantity of high stress areas may reduce deformation under piston rings helping the piston remain in its needed shape.

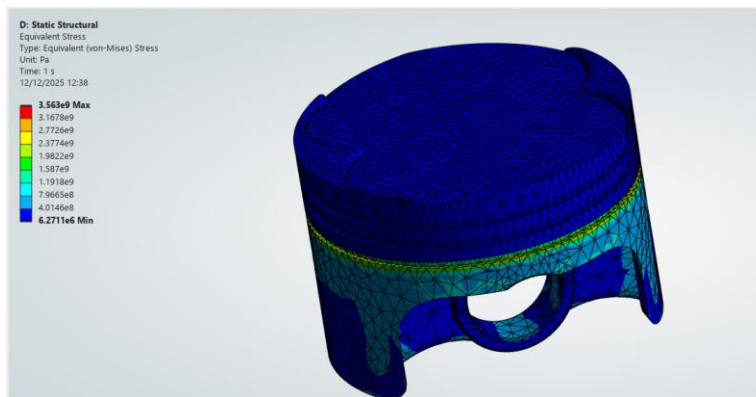


Figure 21- AISI 410 Stress

AISI 410 has a significantly higher max, and the stress is a lot more dissipated around the ring meaning more effective than the other materials.

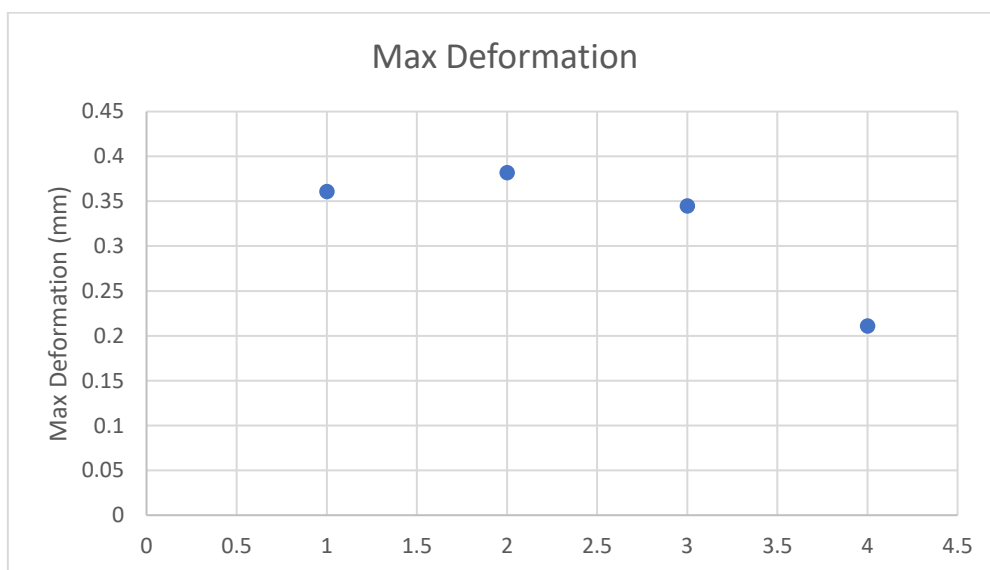


Figure 22-Graph showing the max deformation of each material

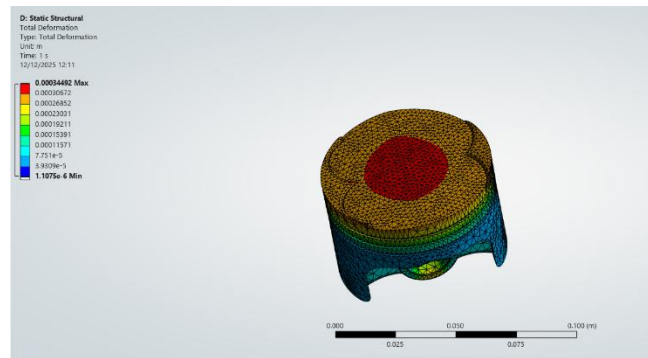


Figure 23-2618-T6 Deformation

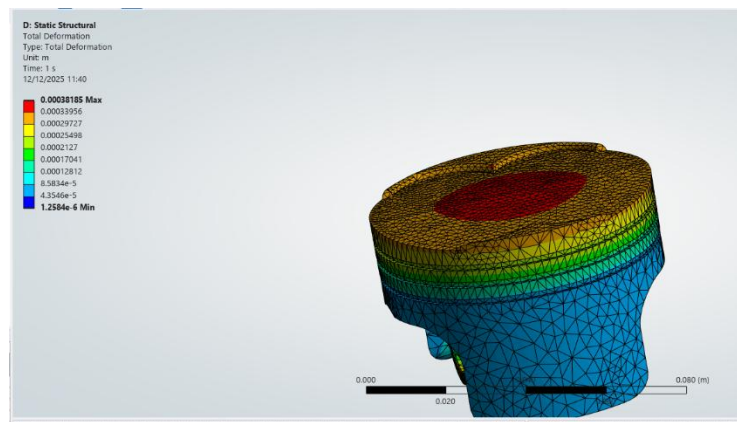


Figure 24-6262-T9 Deformation

The 6061-T6's max deformation occurs in the centre of piston head with small area of maximum deformation occurring on the curve of the piston head. This is like the 6218 and the 6262 where the maximum deformation occurs in the same place. Despite having higher stress, the 6128 has lower max displacement compared to 6061 in the same places, whereas the 6262 has a larger displacement which reduces its effectiveness for use as a piston.

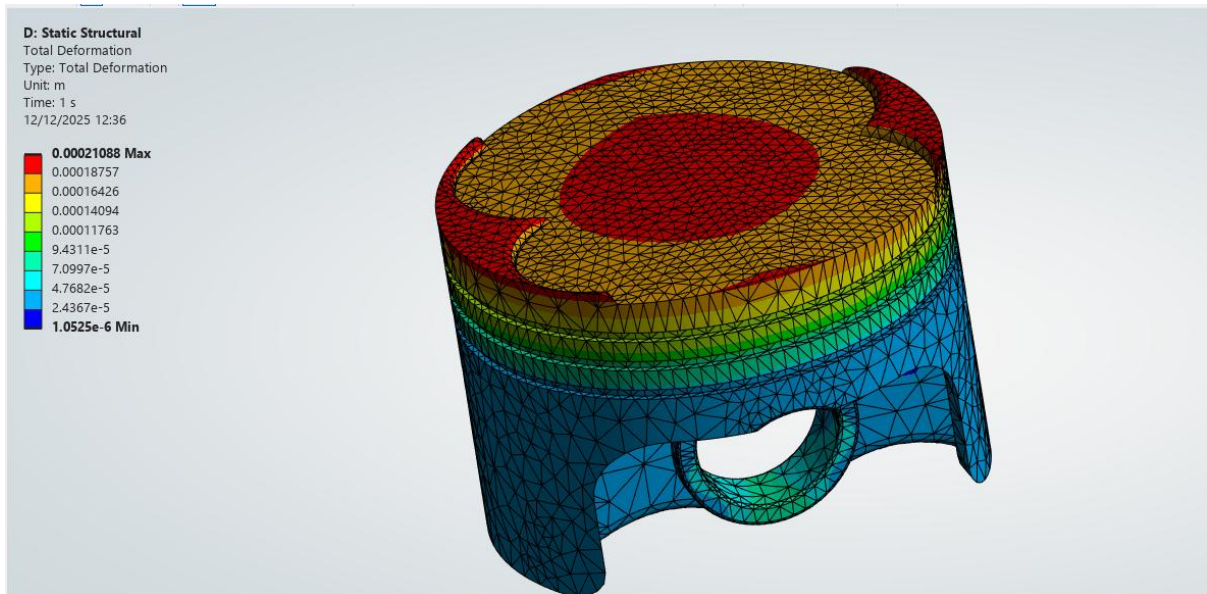


Figure 25-AISI 410 Deformation

Steel has significantly less maximum deformation meaning that the piston's shape remains mostly the same however, on the piston the area at which maximum displacement occurs is substantially larger which may cause issues to the overall piston geometry.

Considering this the improvement I would make is changing the material to aluminium 2618-T6 this is because despite having a slightly higher stress the 2618 has a lower deformation and in the same area as the 6061 making it more ideal for use in the piston. The steel would be better if it was not as heavy and the deformation didn't occur all over the top face.

5 Conclusion

To conclude, ANSYS is accurate in producing simulated results for both the static structural and thermal loadings, however, accuracy could be improved by reducing mesh size but in some case reducing the mesh size can lead to errors due to calculation errors due to rounding. The fully linked system determined the 6061-T6 is suitable for the job as a piston however could be improved upon by changing the material to one such as the 2618-T6 which has similar thermal properties and yield strength but has quite a lot less deformation occurring under the same temperature.

6 References

ANSYS (Granta), 2015. *Aluminium,2618,T6*, N/A: ASM International.

ANSYS (Granta), 2015. *Aluminium,6061,T6*, N/A: ASM International.

ANSYS (Granta), 2015. *Aluminum,6262,T9*, N/A: ASM International.

ANSYS (Granta), 2015. *Stainless Steel, martensitic,AISI 410, Hard Temper*, N/A: ASM International.

Dlupal, 2025. *VE0078 / Bimetallic Strip*. [Online]

Available at: <https://www.dlupal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/009078>

[Accessed 11 12 2025].