

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)

Christopher Bell

2003884

December 2022

Contents

1	Introduction	2
2	Literature Review	3
3	Methodology.....	4
3.1	Validation	4
3.1.1	Bimetallic Strip	4
3.1.2	Cantilever Beam	7
3.1.3	Transient Validation	9
3.2	Full Assembly	11
3.2.1	Material Study.....	11
3.2.2	Materials Discussion	14
4	Conclusion.....	15
5	References	16
6	Appendix	17
6.1	Validation Mesh and Results.....	17
6.1.1	Bi-Metallic Strip.....	17
6.1.2	Cantilever	17
6.1.3	Rigid Dynamics	18
6.2	Material Properties	19
6.3	Results.....	20
6.3.1	Magnesium Results	20
6.3.2	Aluminium Piston Results	21
6.3.3	Low Carbon Steel Piston	22
6.4	Results Graphs	24

1 Introduction

This assignment will cover the use of computation analysis on thermal studies, static structural and transient studies. It will include how to validate all three studies using a validation found from existing literature and using analytical methods. This validation will also compare how changing the mesh size will affect the result. Finally, an assembly will be used to show how all three studies can be used in conjunction together and how distinctive design process can be used to improve the design.

2 Literature Review

(mac Donald 2011) explains the use of numerical solutions to calculate forces that a component may see. It also explains how using a correct factor of safety should be used so that the component will be strong enough but to not also have too much mass.

(Saxena and Sahay 2005) discusses the use of computer aided engineering and how it can be applied to a component to visually see the effects. It explains that more complex cases can be solved quicker than just using analytical methods, and how the results can be seen on screen where stress concentration are allowing for a redesign of the component in a shorter time.

(Srinivas 2020) states that there are errors in solvers which can cause issues in an assembly, the effect mesh sizing can have on accuracy & how incorrect boundary conditions will invalidate a study. It then goes on to say that validation can help to spot these problems so that they can be changed before a final assembly.

(Liu and Coordinator 2013) explains the effect that mesh sizing has on the result and the computational time. The paper states that the finer the mesh the lower error % is calculated however there is a significant increase in computational time. However, it does mention that accuracy of the analysis is the goal and therefore time taken is not an issue.

(Wynn and Eckert 2017) collates other authors discussions on why the iterative design process is a valuable tool to have. Key points include that it is easy to change condition if circumstances change, it improves concurrency due to the number of times the process is ran. How it can be used to elaborate ill-structured problems and how to elaborate on designs and improve them.

(GmbH 2016) discusses the use of different piston materials. This includes what the requirements are for a piston which are that it should be lightweight but strong enough for the forces applied. The materials that it discusses are aluminium metals but also the use of both steel and iron. (Rao Koppuravuri Associate Professor et al. 2020) on the other hand discusses the use of a magnesium piston.

3 Methodology

3.1 Validation

3.1.1 Bimetallic Strip

To ensure a study is accurate a validation case should be used so that the solver is proven to work. To validate the thermal study a bimetallic strip is used, this case is found at (Dlubal Software 2018). This case uses two strips of dissimilar materials bonded together while a thermal condition is applied, and a displacement is calculated.

The strips have a length of 100 mm, a width of 5 mm and a thickness of 1 mm. The two strips are identical in all dimensions and then are stacked on top of each other. The strips were made in solidworks before being exported into Ansys. In Ansys the two strips are connected using a bonding condition. This means that the faces that are connected will have the same displacement if a force is applied to one and not the other.

The materials that are used in this study are invar and copper. The difference in materials will mean that one strip should expand more than the other forcing the whole strip to bend. The properties that are used for copper include a modulus of elasticity of 130000 MPa, a Poisson's ratio of 0.354 and a coefficient of thermal expansion of $2 \times 10^{-5} \text{ }^{\circ}\text{C}^{-1}$. The invar on the other hand has material properties with a modulus of elasticity of 137000 MPa, a Poisson's ratio of 0.280 and a coefficient of thermal expansion of $1.2 \times 10^{-6} \text{ }^{\circ}\text{C}^{-1}$.

These properties are then imputed into the engineering data within Ansys to be used later. Once this is setup a thermal study can be setup. The material properties can be either set up within the engineering data in the thermal or as a separate entity before being linked to the thermal study.

Now that that has been setup the geometry can be imported from solidworks. Once done the setup for the study can begin. As stated, before the two strips will need to be connected using a bonded connection. After this is complete, using the geometry tree the materials will need to be selected for each strip with the top strip being selected as invar and the bottom as copper.

Then the strip will need to be constrained from one end other wise the strip will become shaped like a banana as shown in figure 1. Once complete the thermal condition can be applied which in this example is an increase in 100 degrees c, because the ambient temperature is 22 degrees c this means that a condition of 122 degrees c will be applied.

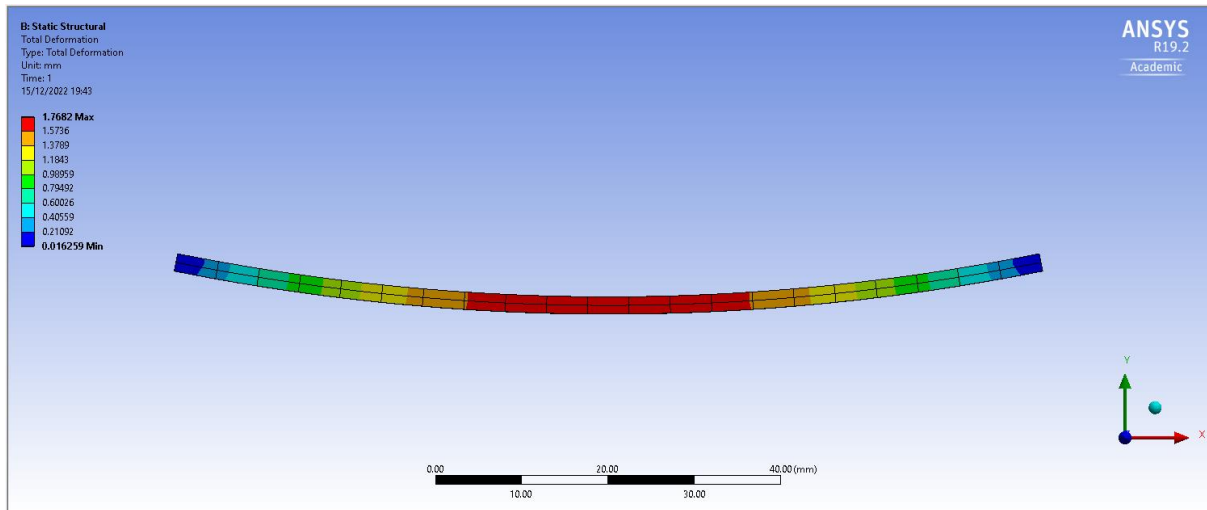


Figure 1 showing an unconstrained strip

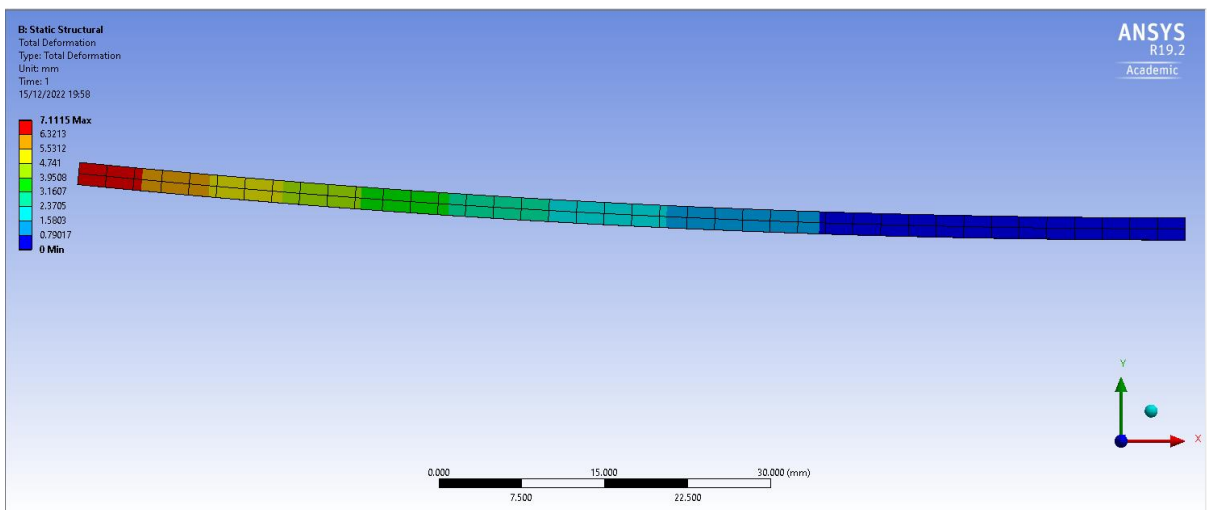


Figure 2 showing a fully constrained model and its deflection

Now the strips can be meshed and the solved. A starting mesh size of 2.5mm is used which gave a result of 7.1199 mm. The target value given from the validation is 7.049mm which give an error of 0.996 %. While this is an acceptable error different mesh sizes can be tried to see whether this will have a minor effect.

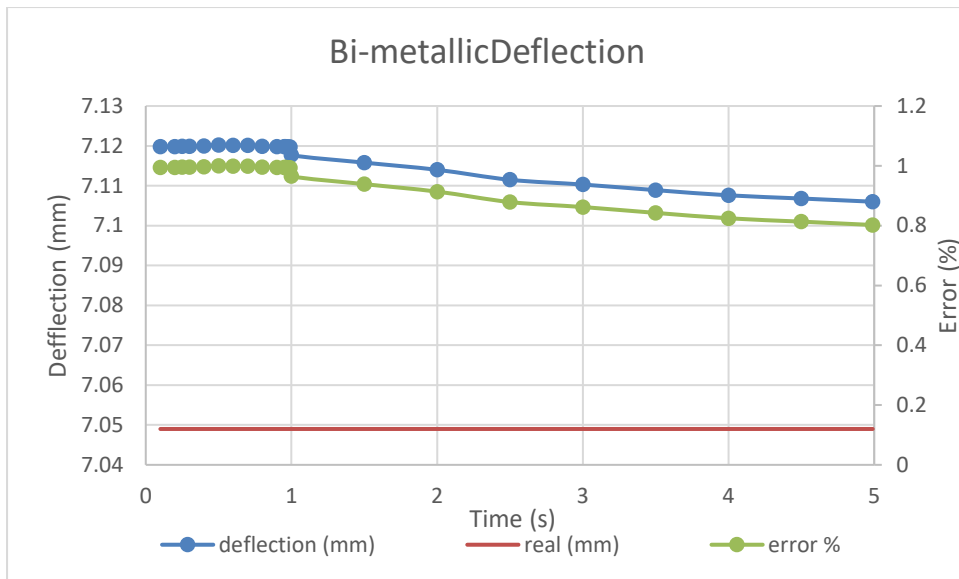


Figure 3 showing the deflection of the bi-metallic strip.

As shown above in figure 3 a mesh study has been undertaken looking at 23 different size meshes and their affect on error. What can be implied is that a lower mesh size has a lower error which could be due to any round error form the amount of decimal places will carry over form the start. This rounding error will have a greater affect with the more elements because the error is used in the calculation several times multiplying the issue.

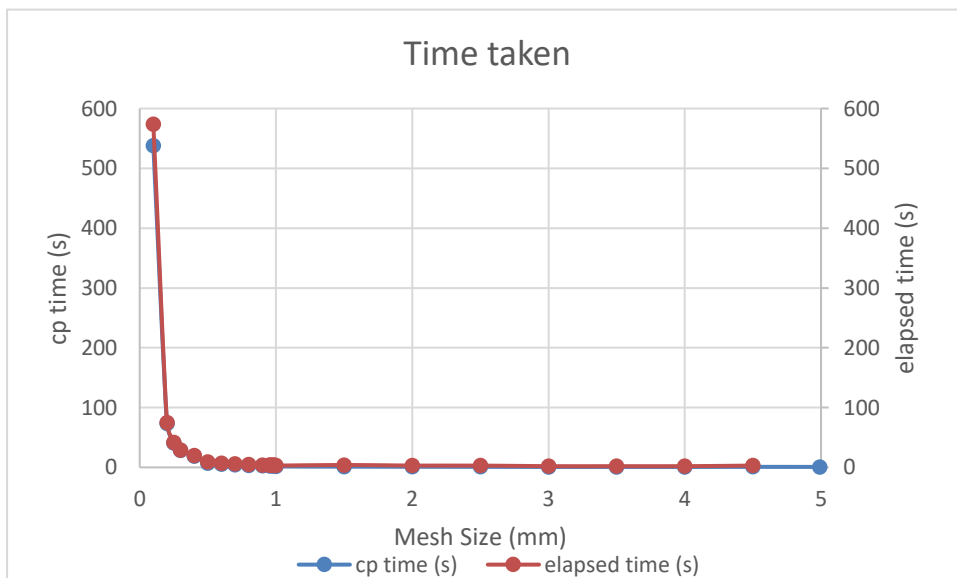


Figure 4 showing the time taken in comparison to mesh size

Figure 4 implies that to compute a study using a mesh size of 0.1 mm it would take 9 and a half minutes while a study using a mesh size of 0.2 mm would only take 13 % of that time. And using data from figure 3 it has the same output, meaning that it is not time effective to use this small of a mesh size. On the other side it shows that any mesh size between 1 and 5 mm will have the same time taken of only 4 seconds but will have a change in deflection.

3.1.2 Cantilever Beam

To validate a static structural study a simple cantilever beam supported at one end with a universally distributed load can be used. The case used here can be found at (Dlubal Software 2022). The beam that is used has a length of 1m a width of 0.05 m and a height of 0.005 m. The material used has material properties that consist of 210000 MPa, a Poisson's ratio of 0 and a shear modulus of 105000 MPa.

The part was again drawn up in solidworks before being exported to Ansys where the above material properties will have been inputted into the engineering data. The once into the setup area this material can be applied to the beam. Once done boundary conditions will need to be set, in this case these are a constraint at one end of the beam so that it does not move and the universally distributed load and can be seen below in figure 5

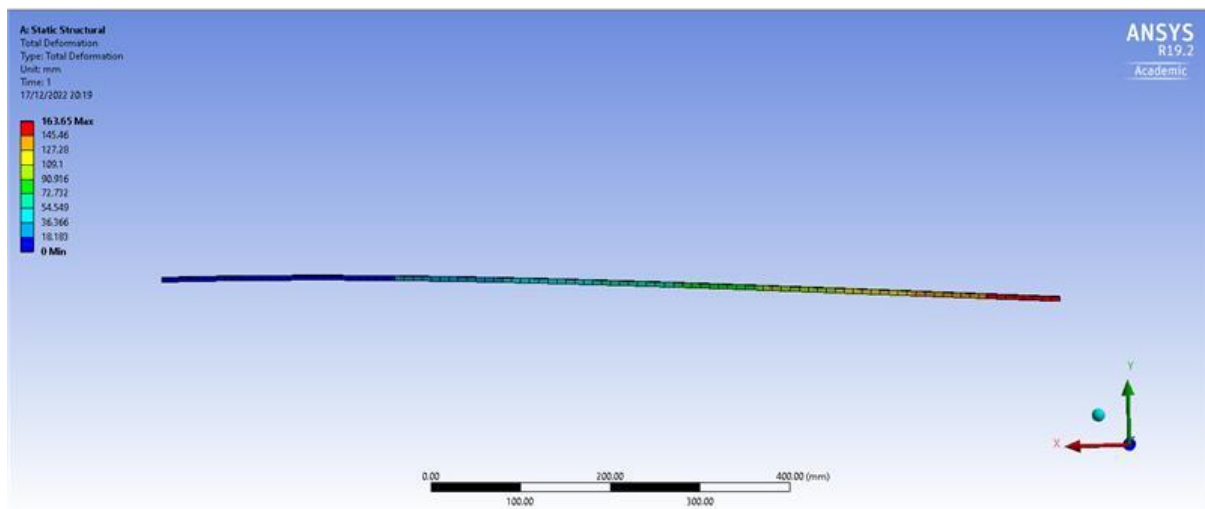


Figure 5 showing the cantilever beam deflection

This load is a pressure of 2750 Pa that act across the whole of the top face of the beam. Then a mesh will need to be generated, a starting mesh of 2.5 mm is again used. This study can then be solved, and a defamiation can be calculated.

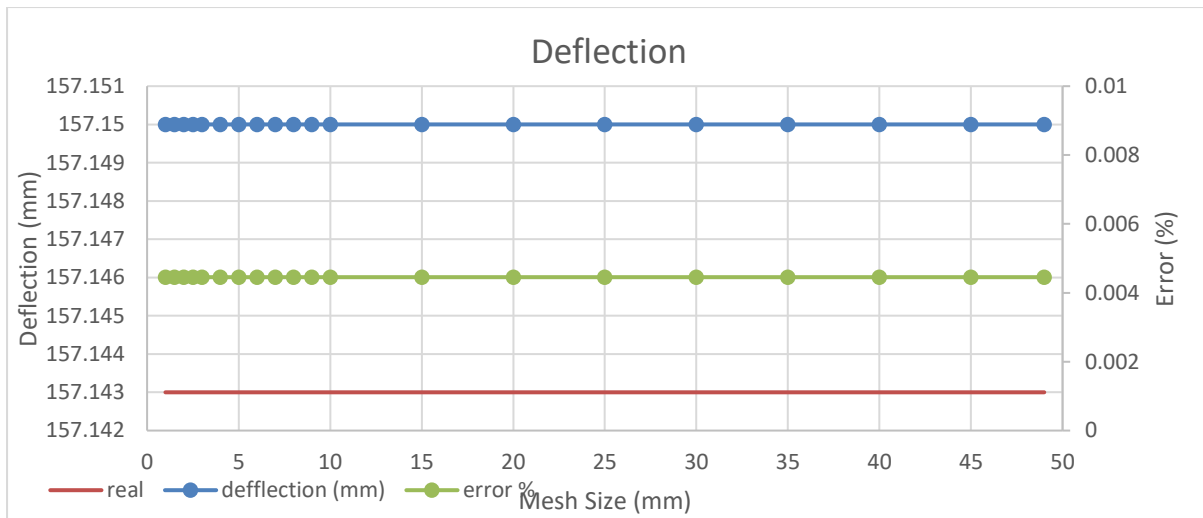


Figure 6 shows a graph comparing real deflection compared to calculated and error

As shown above 19 studies using different mesh sizes were completed to see the affect on deflection. From the graph, it can be interpreted that the mesh size has no effect on measured deflection. This is good as it means that there can be a reasonably large mesh used when using larger parts which will help with solving times for more complex geometry.

It can also be compared analytically by using the equation $y = \frac{wL^4}{8EI}$, entering the values for this will give the equation $y = \frac{(2750 \times 0.05) \times (1^4)}{8 \times (210000 \times 10^6) \times (\frac{0.05 \times (0.005^3)}{12})}$ which is equal to 0.157143 m or 157.143mm. This means that the study is now accurately analytically as well as computationally.

3.1.3 Transient Validation

To prove that correct forces are being calculated when using a transient study, a simple case using a block rotating about a centre point can be used. This case was worked out analytically only. For this study, a simple cube with a height of 0.1 m, a length of 0.1 m and a width of 0.1 m. This block is then connected to a beam with a length of 1 m which is connected to a centre point and then rotated around with a velocity of 500 rad/s.

To work out the centripetal force the mass of the block must be known, to do this the mass must be calculated, the area of the cube is calculated as 0.001 m^3 . The material used was titanium which has a density 4510 kg/m^3 , a Poisson's ratio of 0.36 and a young modulus of 1.02×10^{11} . Using the above area, the mass can be calculated as 4.51 kg.

Using these values in the centripetal force equation above give $F = \frac{4.51 \times 500^2}{1}$ which when calculated gives a force of $1.1275 \times 10^6 \text{ N}$ or 1127.5 kN. When the same inputs are used in Ansys a force of 1183.9 kN is calculated. This gives a percentage error of 5% which is acceptable. The Ansys setup can be found below in figure 7.

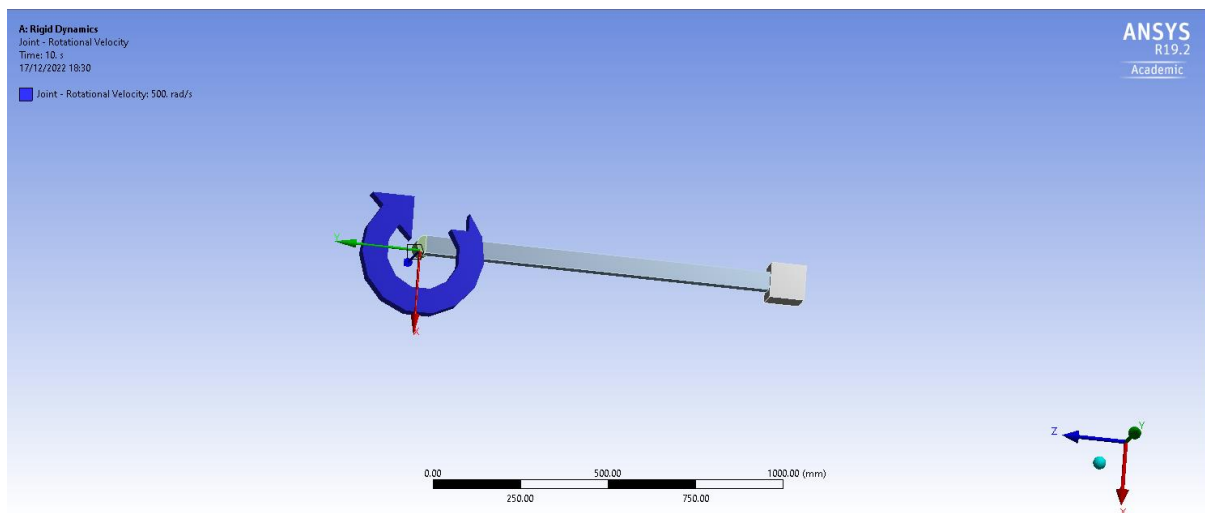


Figure 7 showing the transient study setup

To prove that this will create the correct displacement on the beam a static structural study was then run to see how much the beam would deform when the load was applied at the end of the beam. This was first done by calculating the strain by using force over area. The height and width of the beam are 0.5m for both. This creates an area of 2.5×10^{-3} . The force is taken from the above analytical solution rather than the computational, this was also used in the Ansys study as well. The strain was calculated to be 450.8 GPa.

Then the strain will be calculated using the stress over young's modulus. This beam uses the same properties as titanium, so the equation becomes 450.8 GPa over 1.2×10^{11} . Finally, the deflection can be measure by using change in length is equal to strain multiplied by the length of the beam. This

change in length is calculated to be 4.42mm. An Ansys study was then run as shown below in figure 8 using the same parameters as above but starting with an initial 1 mm mesh.

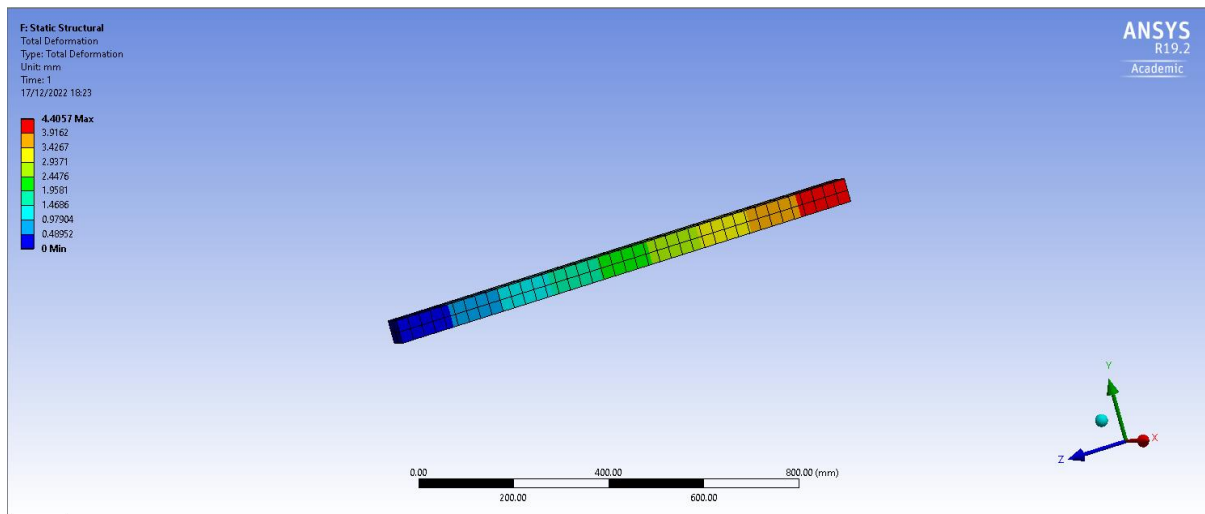


Figure 8 showing an Ansys study on beam extension using a 25 mm mesh.

Then a mesh study was used to see what affect the size of the mesh has on the calculation result. This study used mesh sizing that ranges from 1mm to 49.5 m, this is the maximum mesh size because Ansys cannot run a mesh size that is equal to the maximum width so the closest measurement possible was used.

As shown below in figure 9 the most accurate mesh size is 1mm as it has the lowest percentage error. However as discussed earlier these smaller mesh sizes mean that the studies take longer, and this follows the same pattern with this study as a mesh using an element size of 0.2m takes 44 seconds compared to 723 for the 0.1mm mesh size.

All results and mesh sizing for the above validations can be found in appendix 6.1

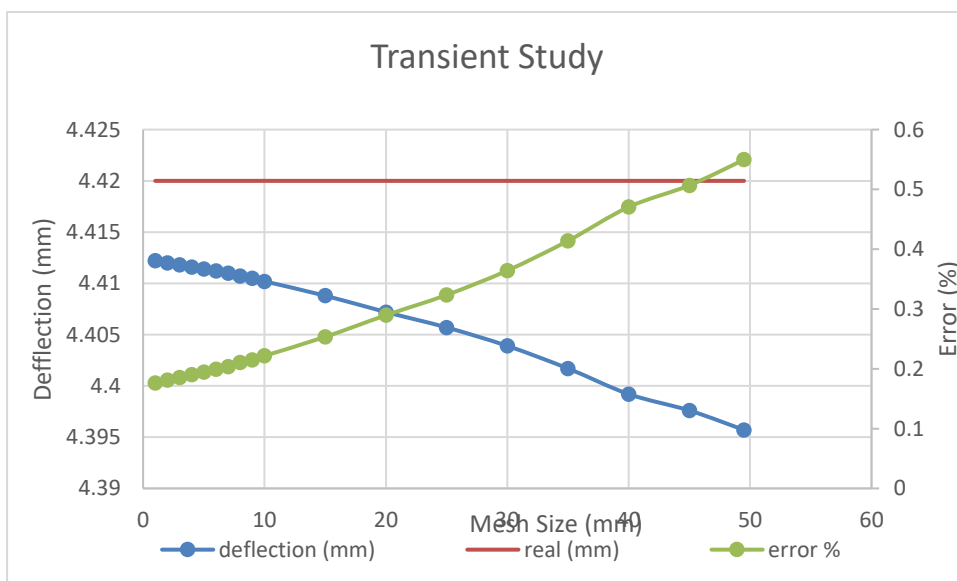


Figure 9 showing the effect on mesh sizing compared to deflection

3.2 Full Assembly

3.2.1 Material Study

Now that the CAE software has been validated the process to start a complete study linking more than one study can be used. The example used here is a piston, crankshaft, conrod and gudgeon pin assembly. The assembly used was provided from (Noble 2022) and can be seen below in figure 10. This is used because it allows for all the above systems to be implemented with a relation to real systems.

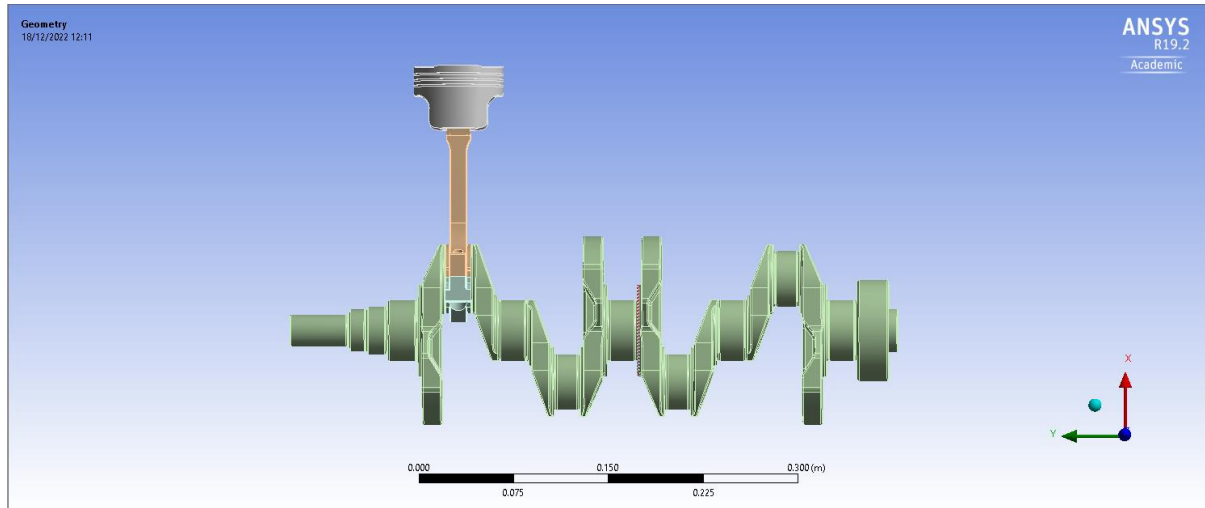


Figure 10 showing the crankshaft assembly which consist of a crankshaft, piston, gudgeon pin, the bottom half of the conrod and top half of the conrod.

The first initial step is to set up the rigid dynamic study by first setting up the materials that will be used. To do this the inbuilt granta materials pack within Ansys this is used because it has different materials with correct parameters that need to be used in the studies. The material that will be used are low carbon steel for the crankshaft and gudgeon pin, titanium for the conrod. However, the piston will have three different materials that are low carbon steel, aluminium alloy, and a magnesium alloy. The starting material was the magnesium alloy & the properties can be found in appendix 6.2.

Then the geometry will need to be imported and set up so that the geometry moves in such a way that is accurate to a real-life crankshaft assembly. To complete this a joint between the two halves of the conrod is done so that they are essentially one piece. There are then revolute joints put in where the gudgeon pin connects to the piston and conrod. The crankshaft then has a revolute joint that only allows for it to rotate in one dimension thus fixing in place but allowing it to rotate, finally the conrod parts are then connected via a revolute joint to the crankshaft.

Now the crankshaft can be given a rotational velocity and a joint probe setup on the connection between the gudgeon pin and piston. This will allow the load measure on this joint to be shown and

then exported for the final deformation and can be seen below in figure 11 as an example, before being used later in the static study.

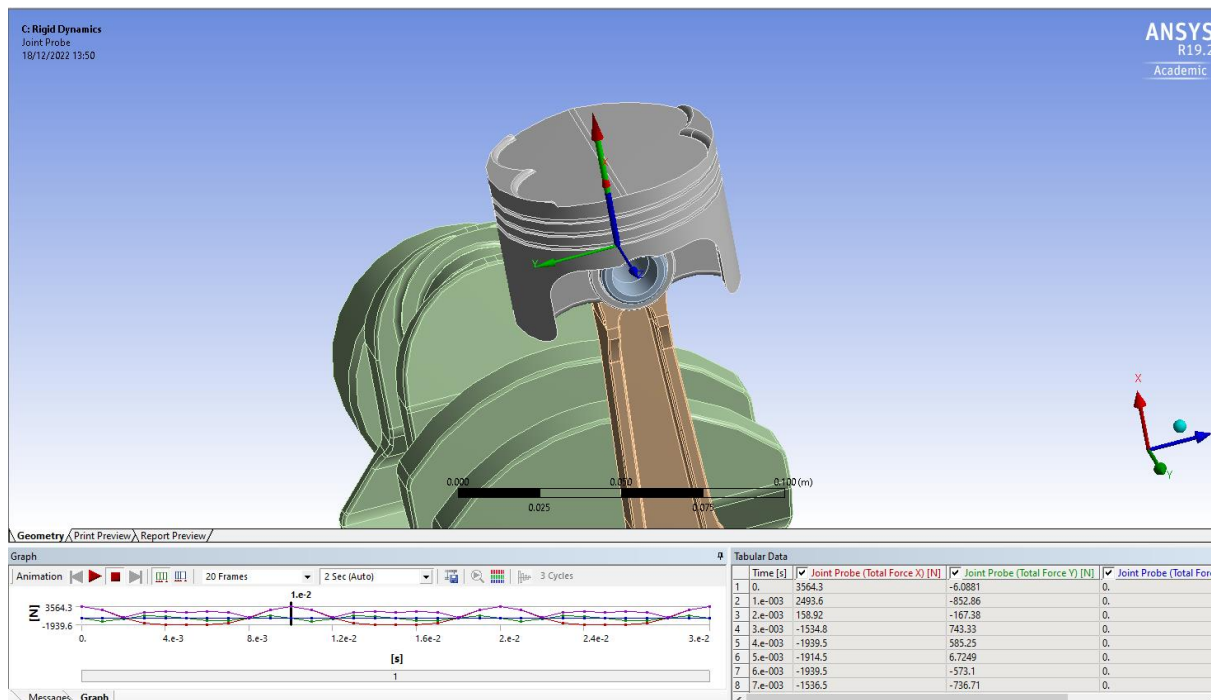


Figure 11 showing the joint loads for a magnesium piston

Now the thermal loadings can be used in the study, the study can be linked in as shown below in figure 12. The thermal loads that are used are taken from (Cioatǎ et al. 2017) and can be seen in figure 13 are applied to each relevant part on the piston. The piston is then run at a 2mm mesh because as discussed earlier a 1mm would be very time consuming.

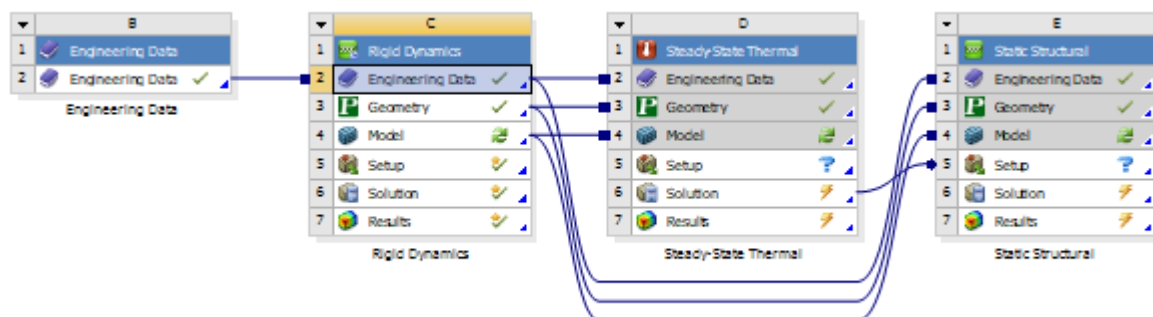


Figure 12 showing how all the studies are linked together

The results for the magnesium can also be seen below in figure 14 which shows that the temperature is far greater on the top with a max temp of 248 and a min of 113. However, these higher temps are seen closer to the centre of the crown of the piston. This is because magnesium has good thermal conductivity which allows the heat to dissipate better.

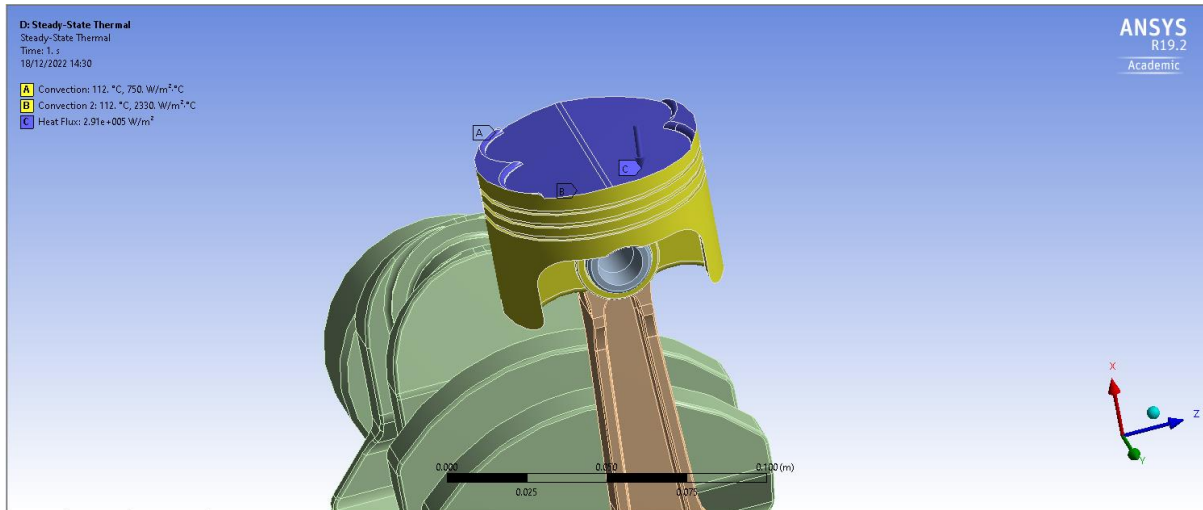


Figure 13 showing the thermal loading on the piston

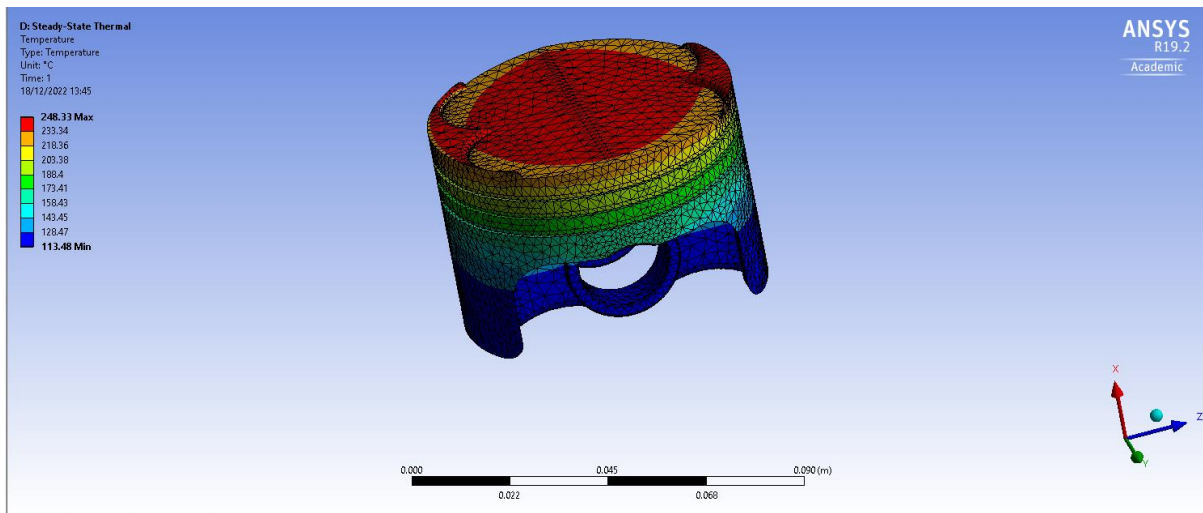


Figure 14 showing the temperature profile on the piston

Once done the final static study can commence which combines the rigid dynamic load, temperature profile and a pressure which can be added to the piston to simulate the combustion pressures. The pressure that was used in this study was found in (Rajasekhara Babu 2015), and has a value of 10.936 MPa. This was applied across the top of the piston crown evenly as shown below in figure 15. A mesh of 2mm was again used due to the above discussed points. The study was then run combining all the loads and temperatures. The deformation can be seen below in figure 16 while the stress and strain can be seen in appendix 6.3.1. this shows that there is a max deflection of 0.276 mm, an equivalent stress of 1.266 Gpa and strain of 0.0275. The next step is complete the studies again with different materials. These results can be seen in appendix 6.3.2 and 6.3.3.

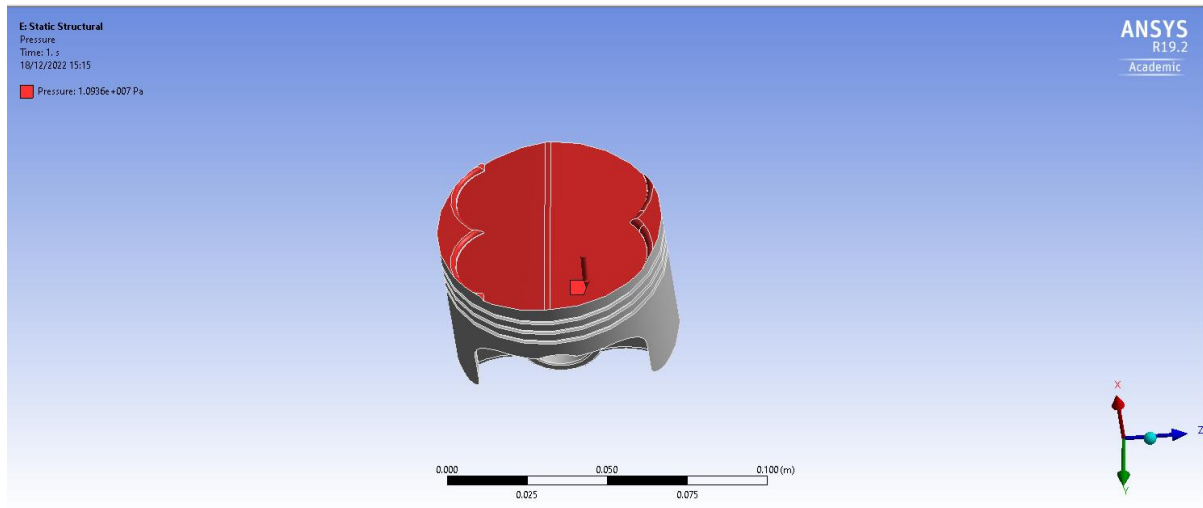


Figure 15 showing the pressure distribution for the static structural study.

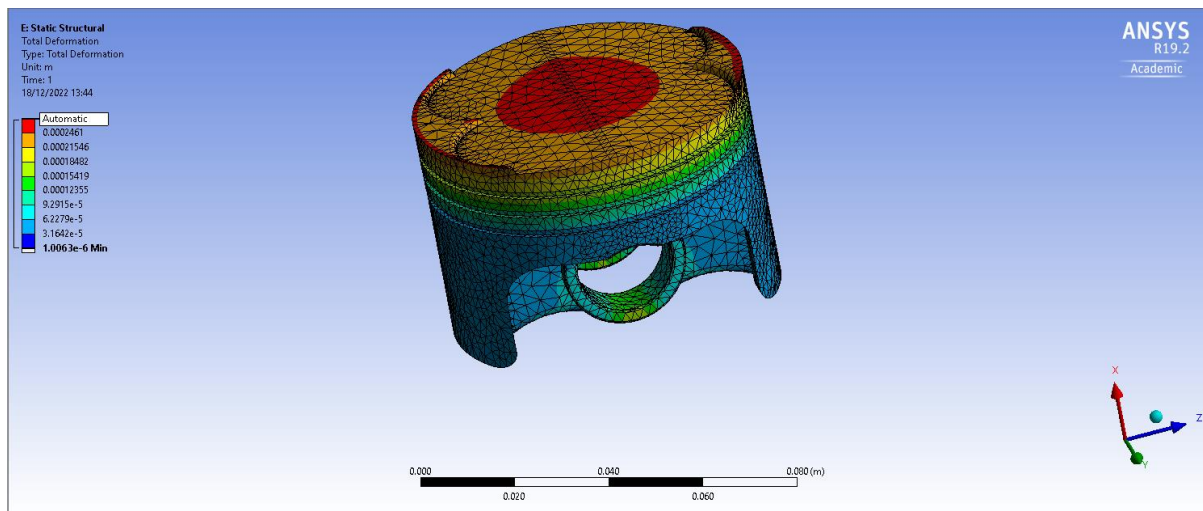


Figure 16 showing the deformation of the piston

3.2.2 Materials Discussion

Magnesium		Aluminium		Steel	
mesh (mm)	2	mesh (mm)	2	mesh (mm)	2
deformation (mm)	0.276	deformation (mm)	0.194	deformation (mm)	0.134
stress GPa	1.266	stress GPa	0.994	stress GPa	1.796
strain	0.0275	strain	0.0136	strain	0.00856
Temp cc	248.33	Temp c	203.67	Temp c	280.17

Table 1 showing the comparison between the different materials with graphical representation in appendix 6.4

Form the comparison above the most suitable material is aluminium because it allows for the heat to dissipate better and so it has a lower max temp which will help to keep cylinder temperature lower. It also has a lower deformation which means it will keep its shape for longer.

4 Conclusion

To conclude the use of FEA is an important tool to have as it allows for large assemblies or components to be solved in a quicker time period. The use of validation becomes important as mesh sizes can have a large impact on the outcome of the study. There are many validations out there that can be used so it is important to use ones that are relevant to the studies being undertaken, but also to make sure all subsystems are validated.

If problems arise from the validation or the error is greater than wanted, note what changes are needed to improve the accuracy and then implement these on the final assembly. When completed the final assembly it is important to constrain the model in an accurate and desired way.

Reading of the data and correctly interoperating this so that improvements can be made is an important tool to have so that the correct changes can be made.

Aluminium was chosen as the Ideal material during the iterative design phase because a lower deformation compared to that of magnesium, while the steel had a lower deformation than aluminium it had a higher max temperature which could mean that the cylinder temperature could get higher than needed. Aluminium is also a lighter material so it would be able to provide more power in the engine because it will contribute to a lower rotational inertia.

5 References

- Cioată, V. G., Kiss, I., Alexa, V. and Rațiu, S. A., 2017. Mechanical and thermal analysis of the internal combustion engine piston using Ansys. In: *IOP Conference Series: Materials Science and Engineering*. Institute of Physics Publishing.
- Dlubal Software, 2018. *Verification Example 0078-Bimetallic Strip Description* [online]. Available from: <https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/000078> [Accessed 14 Dec 2022].
- Dlubal Software, 2022. *Verification Example Program: RFEM 5, RFEM 6 Category: Geometrically Linear Analysis, Isotropic Linear Elasticity, Plate Verification Example: 0016-Elastic Bending-Continuous Load 0016-Elastic Bending-Continuous Load Description* [online]. Available from: <https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/000016> [Accessed 14 Dec 2022].
- mac Donald, B. J., 2011. *Practical Stress Analysis with Finite Elements*.
- GmbH, M. I., 2016. *Pistons and engine testing*. Springer.
- Liu, Y. and Coordinator, G., 2013. *CHOOSE THE BEST ELEMENT SIZE TO YIELD ACCURATE FEA RESULTS WHILE REDUCE FE MODEL'S COMPLEXITY Wave Energy Converter Project View project Mechanical Systems Design View project CHOOSE THE BEST ELEMENT SIZE TO YIELD ACCURATE FEA RESULTS WHILE REDUCE FE MODEL'S COMPLEXITY*.
- Noble, A., 2022. Crankshaft Assembly.
- Rajasekhara Babu, K., 2015. *Management and Applied Sciences* *www.ijetmas* [online]. International Journal of Engineering Technology. Available from: www.ijetmas.com.
- Rao Koppuravuri Associate Professor, S., Podugu, S., Sai Sathwika, S. and Balgar, S., 2020. Design and Analysis of Piston Made of Magnesium Silicon Carbide (MGSIC). *International Journal of Advanced Research in Engineering and Technology* [online], 11 (8), 273–284. Available from: <http://iaeme.com><http://iaeme.com>.
- Saxena, A. and Sahay, B., 2005. *Computer Aided Engineering Design* - Saxena, Sahay (Springer).
- Srinivas, K., 2020. *Verifications and Validations in Finite Element Analysis (FEA)* [online]. Available from: <http://www.advances.com>.
- Wynn, D. C. and Eckert, C. M., 2017. Perspectives on iteration in design and development. *Research in Engineering Design*, 28 (2), 153–184.

6 Appendix

6.1 Validation Mesh and Results

6.1.1 Bi-Metallic Strip

mesh (mm)	deflection (mm)	real (mm)	cp time (s)	elapsed time (s)	error %
0.1	7.1198	7.049	537.781	574	0.99441
0.2	7.1198	7.049	73.156	75	0.99441
0.25	7.1199	7.049	40.656	42	0.995801
0.3	7.1199	7.049	28.156	29	0.995801
0.4	7.12	7.049	18.859	20	0.997191
0.5	7.1202	7.049	6.938	9	0.999972
0.6	7.1201	7.049	5.344	7	0.998581
0.7	7.1201	7.049	4.094	6	0.998581
0.8	7.1199	7.049	3.266	5	0.995801
0.9	7.1198	7.049	2.562	4	0.99441
0.95	7.1198	7.049	2.422	4	0.99441
0.96	7.1198	7.049	2.422	4	0.99441
0.97	7.1197	7.049	2.438	4	0.993019
0.98	7.1197	7.049	2.578	4	0.993019
0.99	7.1197	7.049	2.219	4	0.993019
1	7.1177	7.049	1.641	3	0.965199
1.5	7.1158	7.049	1.297	4	0.938756
2	7.114	7.049	1.078	3	0.913691
2.5	7.1115	7.049	0.984	3	0.878858
3	7.1103	7.049	0.734	2	0.86213
3.5	7.1089	7.049	0.75	2	0.842606
4	7.1076	7.049	0.703	2	0.82447
4.5	7.1068	7.049	0.891	3	0.813306
4.99	7.106	7.049	0.75	4	0.802139

Table 2 showing results for the bi metallic strip validation

6.1.2 Cantilever

mesh (mm)	deflection (mm)	time taken (s)	cp time (s)	real	error %
1	157.15	148	144.047	157.143	0.004454
1.5	157.15	57	54.578	157.143	0.004454
2	157.15	28	25.547	157.143	0.004454
2.5	157.15	15	13.312	157.143	0.004454
3	157.15	12	9.5	157.143	0.004454
4	157.15	8	5.75	157.143	0.004454
5	157.15	4	2.078	157.143	0.004454
6	157.15	4	1.812	157.143	0.004454
7	157.15	4	1.75	157.143	0.004454
8	157.15	4	1.484	157.143	0.004454
9	157.15	4	1.594	157.143	0.004454
10	157.15	3	1.391	157.143	0.004454

15	157.15	3	1.078	157.143	0.004454
20	157.15	4	1.016	157.143	0.004454
25	157.15	3	1.172	157.143	0.004454
30	157.15	3	1.141	157.143	0.004454
35	157.15	3	1.016	157.143	0.004454
40	157.15	3	0.953	157.143	0.004454
45	157.15	4	1.062	157.143	0.004454
49	157.15	4	0.969	157.143	0.004454

Table 3 showing the results for the cantilever beam

6.1.3 Rigid Dynamics

mesh (mm)	deflection (mm)	real (mm)	cp time (s)	elapsed time (s)	Column1	error %
1	4.4122	4.42	714.062	723	0.176471	0.176471
2	4.412	4.42	41.938	44	0.180995	0.180995
3	4.4118	4.42	13.453	16	-0.18552	0.18552
4	4.4116	4.42	6.6062	7	0.190045	0.190045
5	4.4114	4.42	3.5	5	-0.19457	0.19457
6	4.4112	4.42	2.453	4	0.199095	0.199095
7	4.411	4.42	1.984	3	-0.20362	0.20362
8	4.4107	4.42	1.719	3	0.210407	0.210407
9	4.4105	4.42	1.469	4	0.214932	0.214932
10	4.4102	4.42	1.203	3	0.221719	0.221719
15	4.4088	4.42	1.031	3	0.253394	0.253394
20	4.4072	4.42	0.906	4	0.289593	0.289593
25	4.4057	4.42	0.719	3	0.323529	0.323529
30	4.4039	4.42	0.688	2	0.364253	0.364253
35	4.4017	4.42	0.562	2	0.414027	0.414027
40	4.3992	4.42	0.719	2	0.470588	0.470588
45	4.3976	4.42	0.688	2	0.506787	0.506787
49.5	4.3957	4.42	0.641	2	0.549774	0.549774

Table 4 showing the results for the transient study

6.2 Material Properties

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	7850	kg m ⁻³		
3	Isotropic Secant Coefficient of Thermal Expansion				
4	Coefficient of Thermal Expansion	1.2E-05	C ⁻¹		
5	Isotropic Elasticity				
6	Derive from	Young's Modulus and Poisson...			
7	Young's Modulus	2.1E+11	Pa		
8	Poisson's Ratio	0.29			
9	Bulk Modulus	1.6667E+11	Pa		
10	Shear Modulus	8.1395E+10	Pa		
11	Tensile Yield Strength	3.29E+08	Pa		
12	Tensile Ultimate Strength	4.4E+08	Pa		
13	Isotropic Thermal Conductivity	52	J m ⁻¹ s ⁻¹ C ⁻¹		
14	Specific Heat, C _p	485	J kg ⁻¹ C ⁻¹		
15	Isotropic Resistivity	1.7E-07	ohm m		

Figure 17 showing the material properties for the low carbon steel

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	1800	kg m ⁻³		
3	Isotropic Secant Coefficient of Thermal Expansion				
4	Coefficient of Thermal Expansion	2.58E-05	C ⁻¹		
5	Isotropic Elasticity				
6	Derive from	Young's Modulus and Poisson...			
7	Young's Modulus	46100	MPa		
8	Poisson's Ratio	0.357			
9	Bulk Modulus	5.373E+10	Pa		
10	Shear Modulus	1.6986E+10	Pa		
11	Tensile Yield Strength	1.44E+08	Pa		
12	Tensile Ultimate Strength	2.74E+08	Pa		
13	Isotropic Thermal Conductivity	73.9	J m ⁻¹ s ⁻¹ C ⁻¹		
14	Specific Heat, C _p	1070	J kg ⁻¹ C ⁻¹		
15	Isotropic Resistivity	1.4E-07	ohm m		

Figure 18 showing the material properties for the magnesium alloy

	A	B	C	D	E
1	Property	Value	Unit		
2	Density	2680	kg m ⁻³		
3	Isotropic Secant Coefficient of Thermal Expansion				
4	Coefficient of Thermal Expansion	2.15E-05	C ⁻¹		
5	Isotropic Elasticity				
6	Derive from	Young's Modulus and Poisson...			
7	Young's Modulus	7.3E+10	Pa		
8	Poisson's Ratio	0.33			
9	Bulk Modulus	7.1569E+10	Pa		
10	Shear Modulus	2.744E+10	Pa		
11	Tensile Yield Strength	2.05E+08	Pa		
12	Tensile Ultimate Strength	2.54E+08	Pa		
13	Isotropic Thermal Conductivity	152	J m ⁻¹ s ⁻¹ C ⁻¹		
14	Specific Heat, C _p	963	J kg ⁻¹ C ⁻¹		
15	Isotropic Resistivity	4.4E-08	ohm m		

Figure 19 showing the material properties for the aluminium alloy

6.3 Results

6.3.1 Magnesium Results

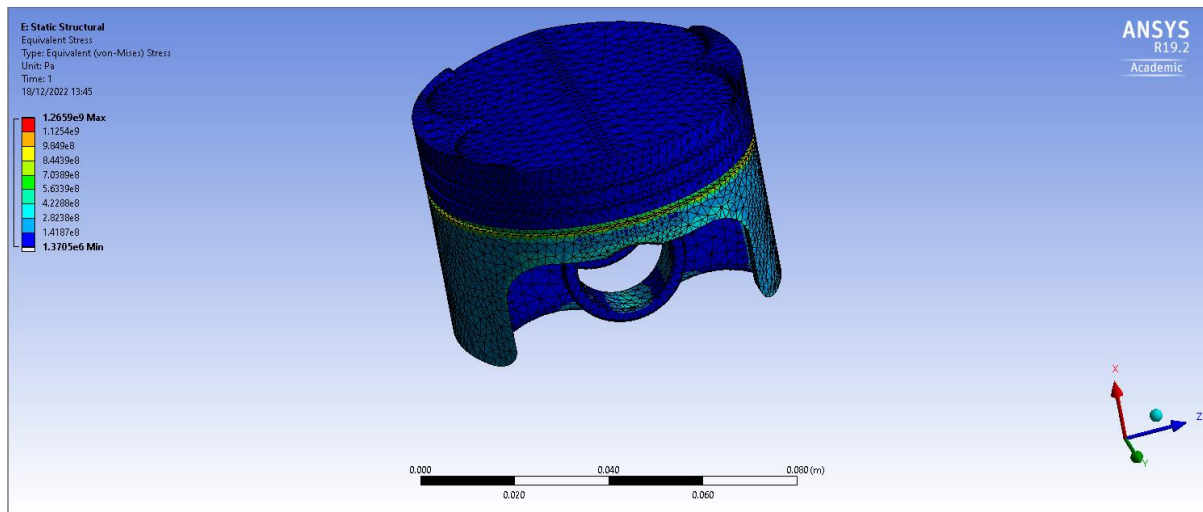
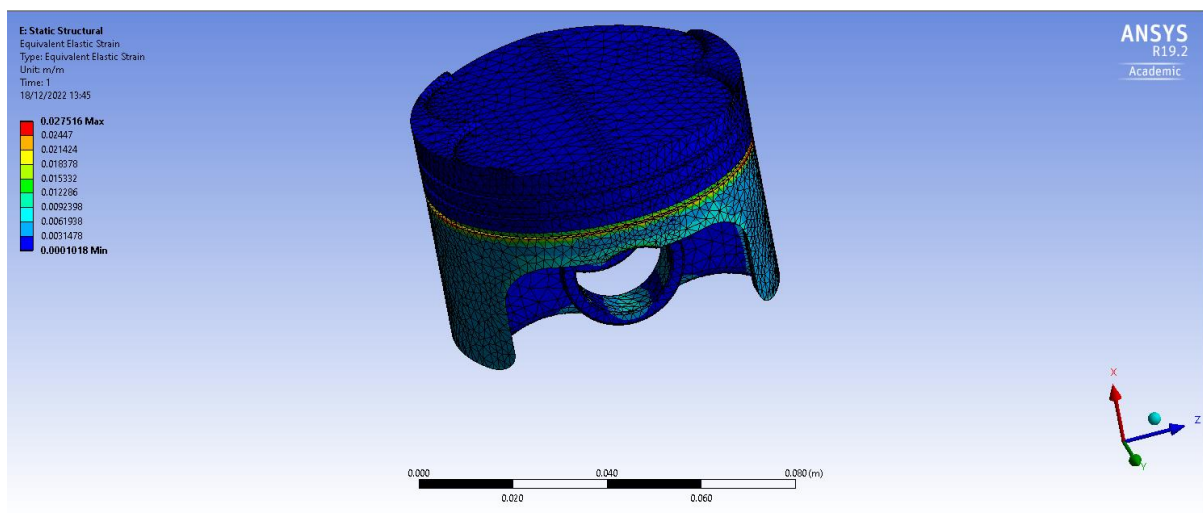


Figure 20 showing the equivalent stress on the magnesium piston



.Figure 21 showing the equivalent strain on the magnesium piston

6.3.2 Aluminium Piston Results

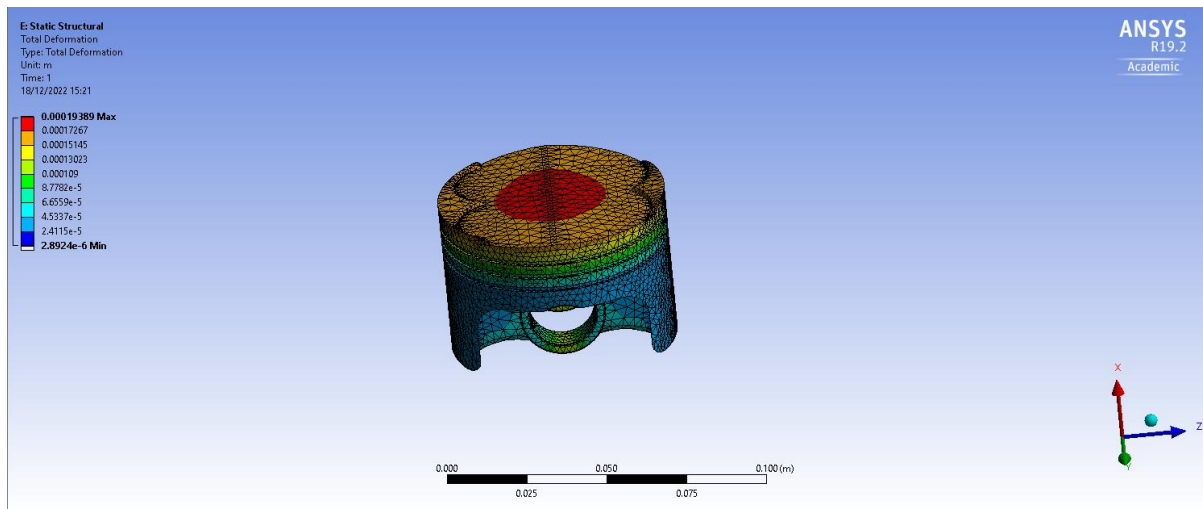


Figure 22 showing the displacement on the aluminium piston

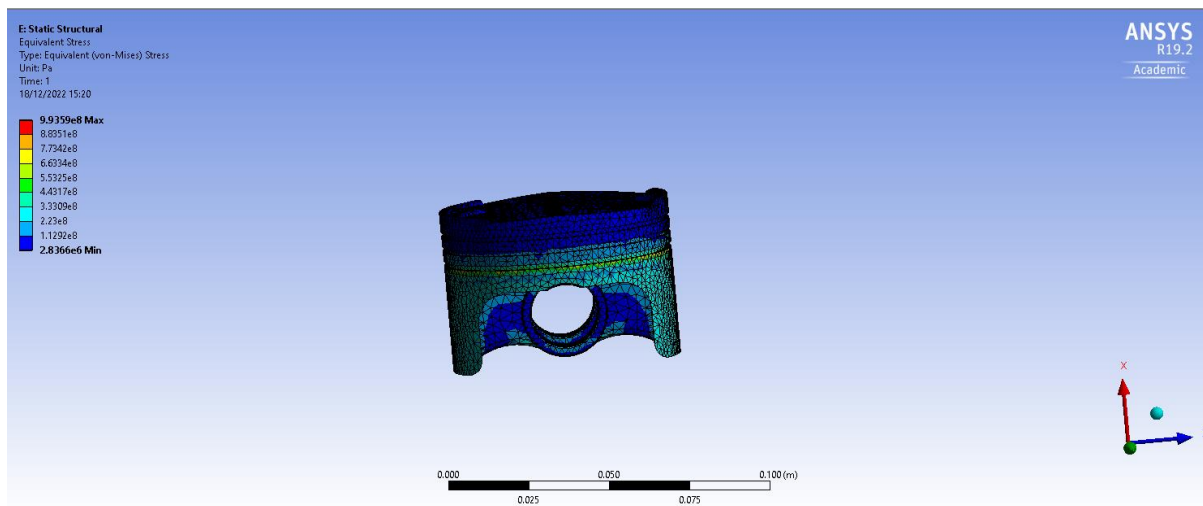


Figure 23 showing the equivalent stress on the aluminium piston

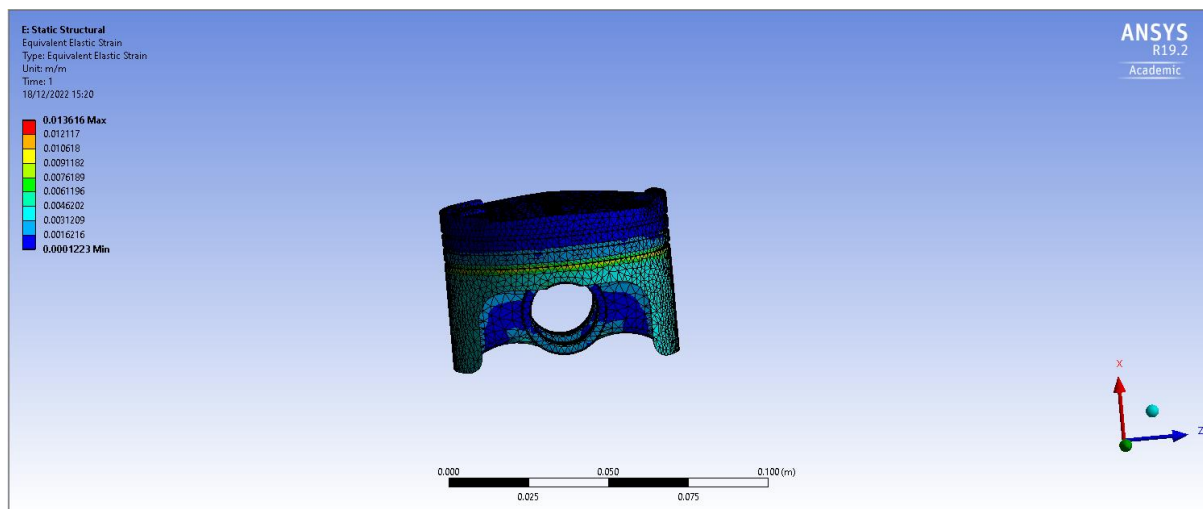


Figure 24 showing the equivalent strain on the aluminium piston

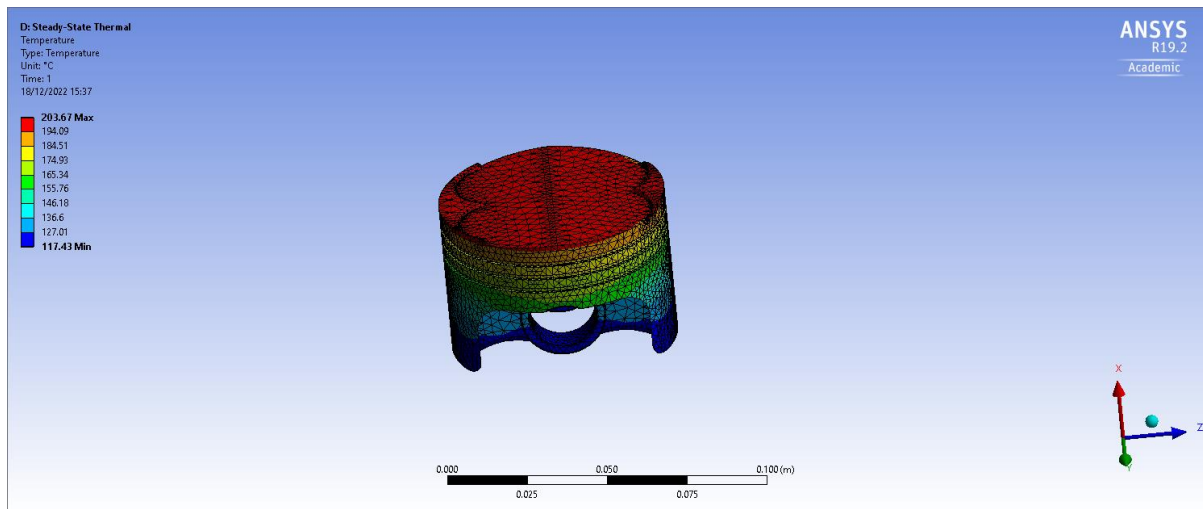


Figure 25 showing the temperature profile for the aluminium piston

6.3.3 Low Carbon Steel Piston

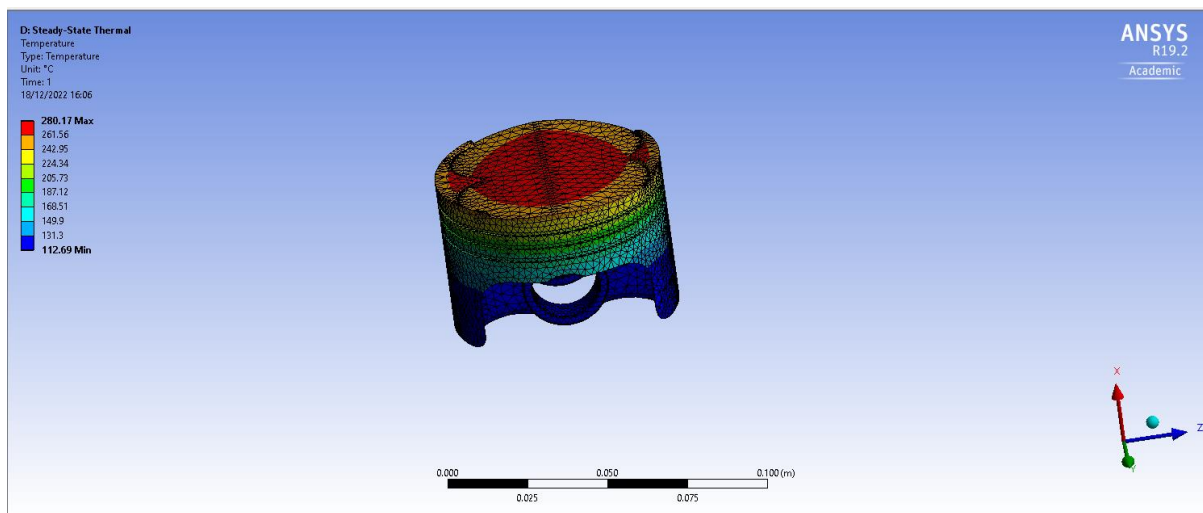


Figure 26 showing the temperature profile for the low carbon steel piston

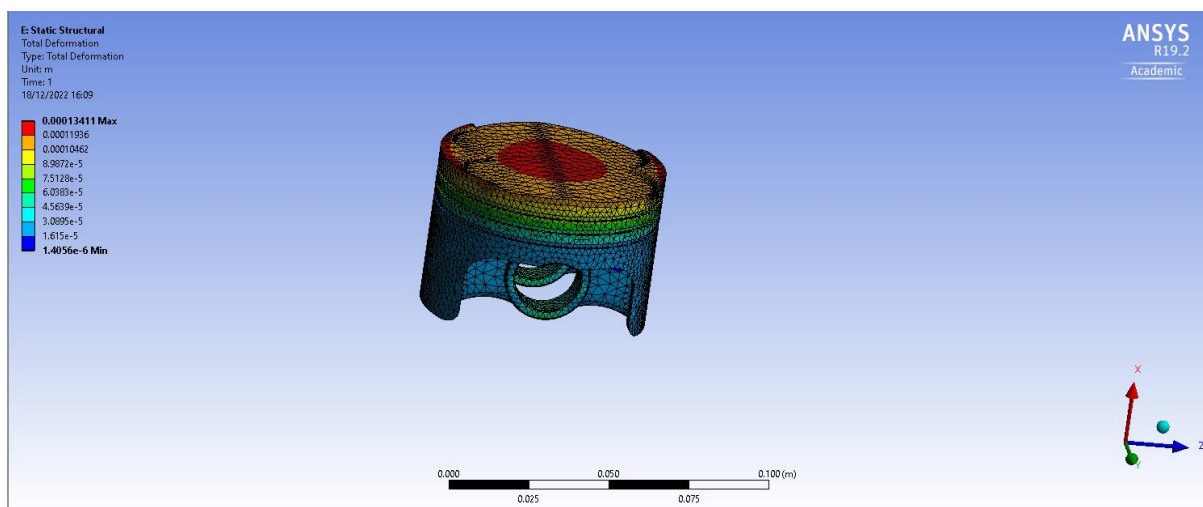


Figure 27 showing the deformation on the low carbon steel piston

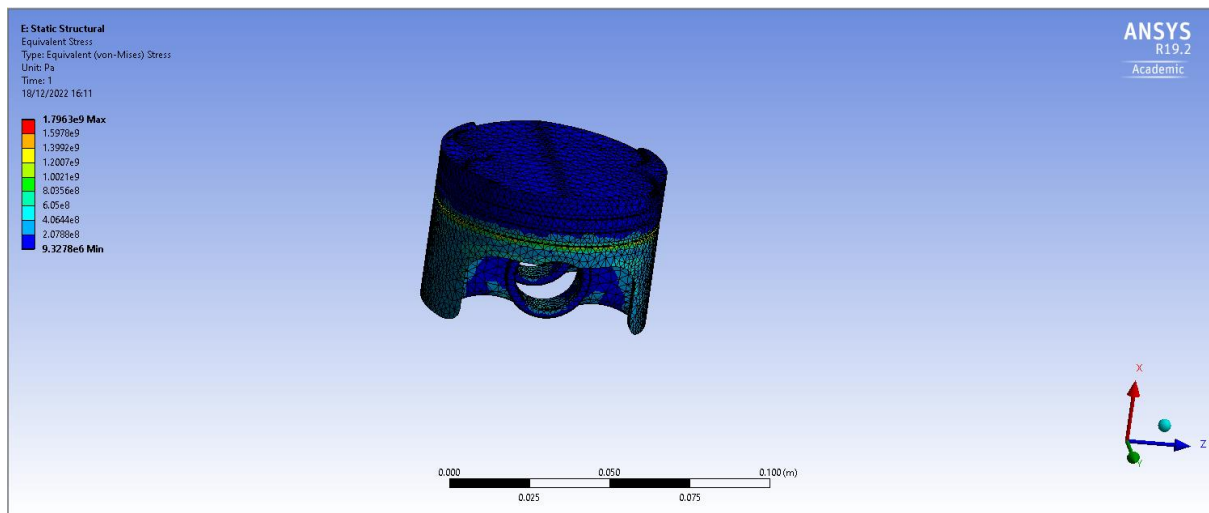


Figure 28 showing the equivalent stress on the low carbon steel piston

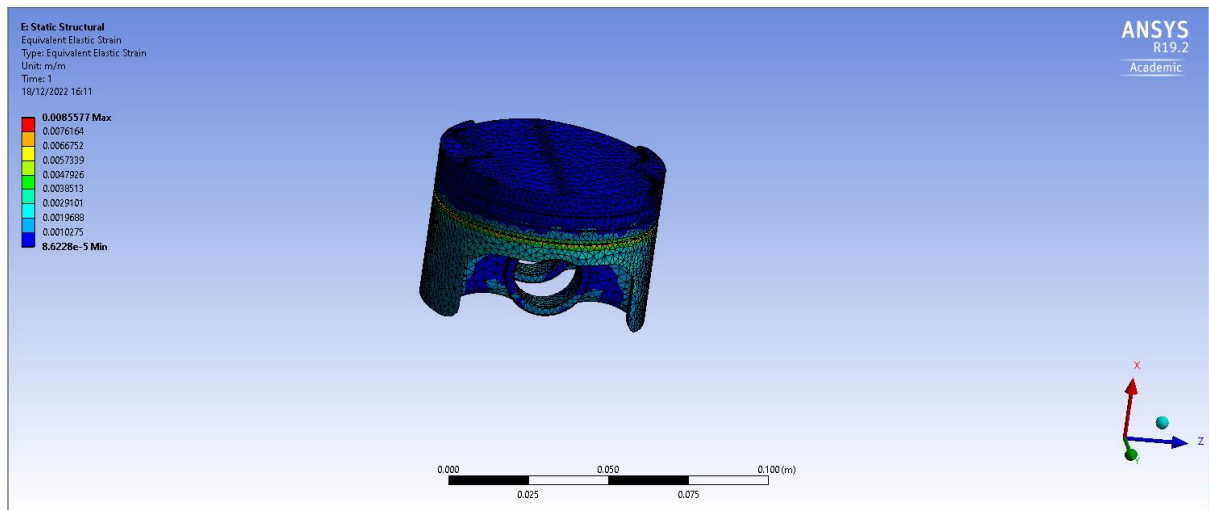


Figure 29 showing the equivalent strain on the low carbon steel piston

6.4 Results Graphs

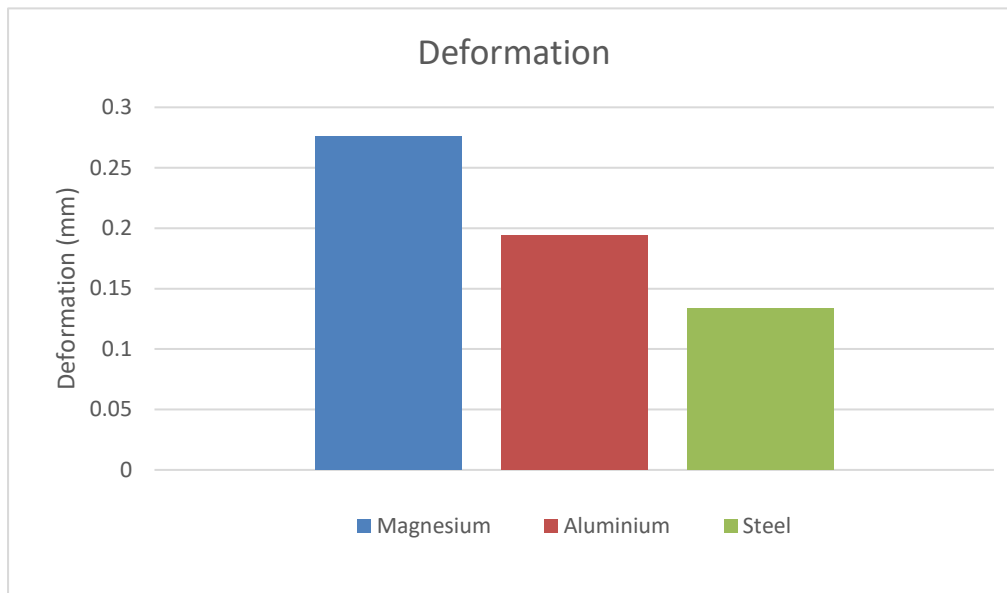


Figure 30 showing a comparison of deformation

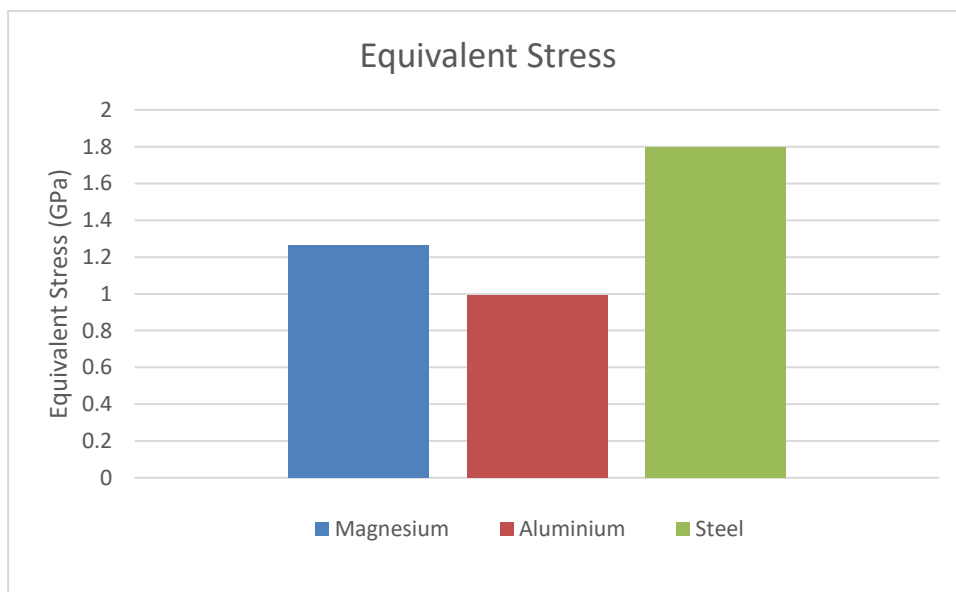


Figure 31 showing a comparison in equivalent stress

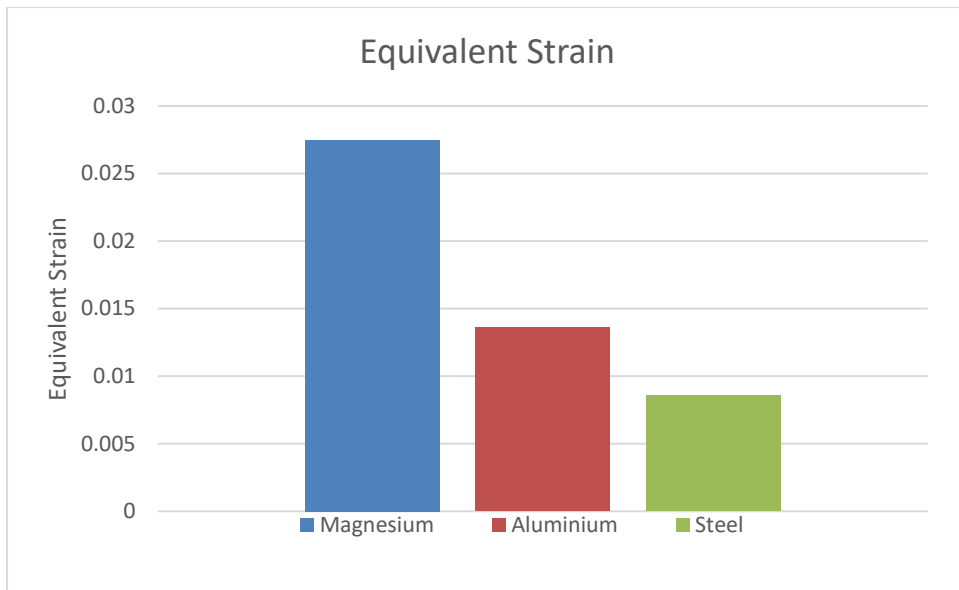


Figure 32 showing a comparison in equivalent strain

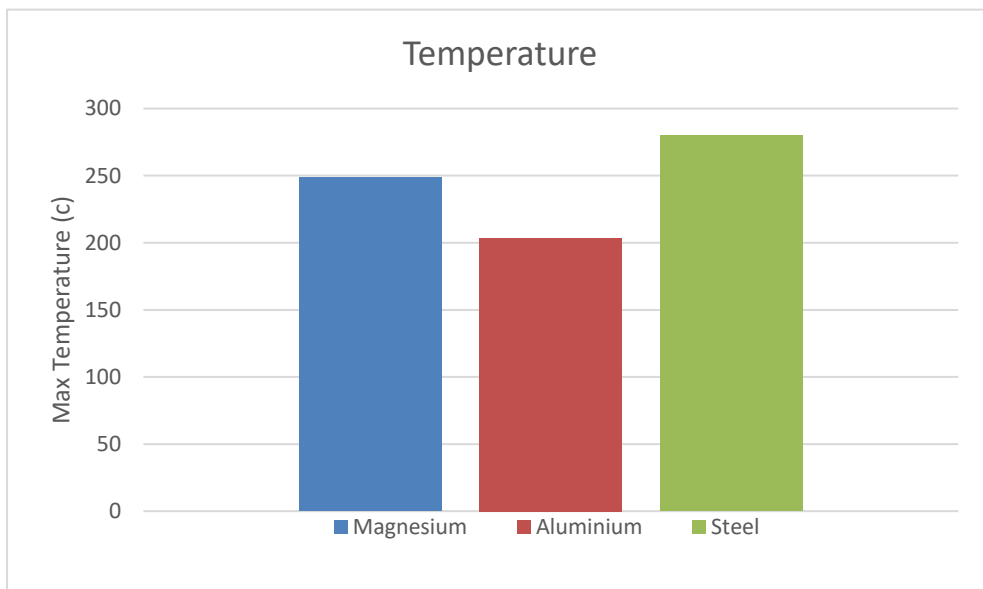


Figure 33 showing a comparison in max temperature