

University of Wales Trinity Saint David

Wales Institute of Art and Science



Application of Computer Aided Engineering Report

Structural Design and Fluid Analysis – ACHA6010

Jake Lawrence

1450966

12/2023

Contents

Tables and Figures.....	i
1 Introduction.....	1
2 Validation	2
2.1 Cantilever with Uniform Load.....	2
2.2 Cantilever with Axial Load	4
2.3 Bimetallic Strip with Thermal Load	6
2.4 Beam with Gravity Loading.....	7
2.5 Validation Conclusion	9
3 Topology	9
4 Linked Study	11
4.1 Rigid Dynamics Study	12
4.2 Steady State Thermal Study.....	14
4.3 Static Structural Linked Study.....	16
5 Conclusion	19
6 References	20

Tables and Figures

FIGURE 1 - CANTILEVER WITH UNIFORM LOAD TESTING PARAMETERS AND ANALYTICAL SOLUTION (DLUBAI SOFTWARE, 2016A)	2
FIGURE 2 - CANTILEVER WITH UNIFORM LOAD 5MM MESH.....	2
FIGURE 3 - CANTILEVER WITH UNIFORM LOAD 2MM MESH.....	3
FIGURE 4 - CANTILEVER WITH AXIAL LOAD TESTING PARAMETERS (DLUBAI SOFTWARE, 2015)	4
FIGURE 5 - CANTILEVER WITH AXIAL LOAD 4MM MESH	4
FIGURE 6 - CANTILEVER WITH AXIAL LOAD 1MM MESH	4
FIGURE 7 - BIMETALLIC STRIP WITH THERMAL LOAD TESTING PARAMETERS (DLUBAI SOFTWARE, 2018)	6

FIGURE 8 - BIMETALLIC STRIP WITH THERMAL LOAD 0.5MM MESH	6
FIGURE 9 - BIMETALLIC STRIP WITH THERMAL LOAD 0.1MM MESH	6
FIGURE 10 - BEAM WITH GRAVITY LOADING TESTING PARAMETERS (DLUBAI SOFTWARE, 2016b)	7
FIGURE 11 - BEAM WITH GRAVITY LOADING 10MM MESH.....	8
FIGURE 12 - BEAM WITH GRAVITY LOADING 2.5MM MESH.....	8
FIGURE 13 - TOPOLOGY STUDY EXAMPLE PLATE LOADINGS	9
FIGURE 14 - TOPOLOGY STUDY EXAMPLE 80% MASS REDUCTION.....	10
FIGURE 15 - TOPOLOGY STUDY EXAMPLES - 10% MASS REDUCTION (1), 30% MASS REDUCTION (2), AND 30% MASS REDUCTION HALF MESH SIZE (3).....	10
FIGURE 16 - ANSYS LINKED STUDY.....	11
FIGURE 17 - EXAMPLE OF THE RIGID DYNAMICS STUDY.....	12
FIGURE 18 - RIGID DYNAMIC STUDY SETUP	12
FIGURE 19 - RIGID DYNAMICS RESULTS STANDARD PISTON	13
FIGURE 20 - RIGID DYNAMICS RESULTS STEEL PISTON	13
FIGURE 21 - 6061-T6 ALUMINIUM MATERIAL SETUP.....	14
FIGURE 22 - THERMAL PROFILE STANDARD PISTON.....	15
FIGURE 23 - THERMAL PROFILE ALTERNATE MATERIAL.....	15
FIGURE 24 - THERMAL PROFILE ALTERNATE GEOMETRY.....	15
FIGURE 25 - PISTON CROWN MESH SIZING	16
FIGURE 26 - TOTAL DEFORMATION STANDARD PISTON.....	17
FIGURE 27 - TOTAL DEFORMATION ALTERNATE MATERIAL.....	17
FIGURE 28 - TOTAL DEFORMATION ALTERNATE GEOMETRY.....	17
FIGURE 29 - STRESS DISTRIBUTION STANDARD PISTON	18
FIGURE 30 - STRESS DISTRIBUTION ALTERNATE MATERIAL	18
FIGURE 31 - STRESS DISTRIBUTION ALTERNATE GEOMETRY	18
TABLE 1 - CANTILEVER WITH UNIFORM LOAD MESH SIZE COMPARISON RESULTS.....	3
TABLE 2 - CANTILEVER WITH AXIAL LOAD DEFORMATION MESH SIZE COMPARISON RESULTS Z AXIS.....	5
TABLE 3 - CANTILEVER WITH AXIAL LOAD DEFORMATION MESH SIZE COMPARISON RESULTS X AXIS	5
TABLE 4 - BIMETALLIC STRIP WITH THERMAL LOAD MESH SIZE COMPARISON RESULTS	7
TABLE 5 - BEAM WITH GRAVITY LOAD RESULTS MESH SIZE COMPARISON	8
TABLE 6 - TEMPERATURE BOUNDARY CONDITIONS (UWSTD, 2023b).....	14

1 Introduction

Computer-aided Engineering (CAE) has become an essential tool in the engineering industry, playing a pivotal role in Research and Development (R&D), testing of designs and Computer-aided manufacturing. However, while CAE programs can give significant advantages, they also have limitations. As described by Jones *et al.* (2017) the use of CAE can increase productivity. CAE can be utilised to limit the resources required for iterative design. Where previously, components of varying designs and materials needed to be prototyped and physically tested. Using CAE, a range of components can be tested and improved from design to design by incorporating knowledge acquired from previous designs at a much-reduced cost of time and resources. This has enabled engineers to gain an understanding of the impact of design changes on a component before any manufacturing or tooling is needed as well as facilitating a wider range of design avenues that were previously limited by cost and time constraints. For example, Ferguson and Wakeman (2004) describe how iterative design in CAE can enable engineers to create a first set of prototype hardware for testing that is closer to an optimised solution than would be available without CAE.

This report investigates the validity of the process of design and refinement of components using CAE and the potential uses and limitations of this software. Specifically, the software used for this report is Ansys Workbench. The report includes four validation studies, these include:

- A cantilever with a uniform load to assess accuracy of the deformation response of a material when a pressure is applied.
- A cantilever with an axial force applied to assess the deformation response to an applied force.
- A Bimetallic strip with a thermal load applied to assess the deformation response caused by thermal expansion of different materials.
- A beam with a gravity loading to assess the accuracy of an acceleration on the deformation response of a material.

Each Validation study will be conducted with a range of three mesh sizes in order to assess the impact of the mesh on the accuracy of the results.

An example of the use of a Topology study will be discussed.

The report will also include a series of linked studies of an internal combustion engine assembly to demonstrate the potential uses of CAE on the design process. The assembly will focus on the piston design and will include a crankshaft, connecting rod (con-rod), gudgeon pin and the piston. These linked studies will include: a kinematics simulation of loadings placed on the piston from the gudgeon pin due to acceleration forces caused by the piston inertia as the crankshaft rotates, a thermal simulation of the effect of thermal degradation of the material due to heat and a structural analysis of the deformation and stresses on the piston during simulated operation comparing a standard material, an alternate material, and an alternate design.

2 Validation

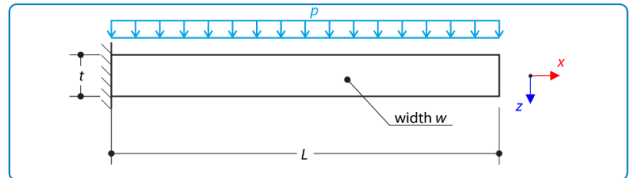
All testing parameters for the validation studies are provided by Dlubai Software and are designed for use in RFEM 5. RFEM 5 is widely used by structural engineers and is primarily used for Finite Element Analysis for example, Rohini and Shanmukha (2017) and (Boned Ferrer, 2023). Its proven use will provide an accurate test for the validity of the Ansys analysis. Material properties along with relevant pressures, forces, acceleration, and thermal inputs were applied for the Ansys Workbench tests as shown in the relevant figures.

2.1 Cantilever with Uniform Load

The Figure (Figure 1) below shows the parameters for the test, equations required, and nominal results for the test. Examples of the test results from the 5mm and 2mm mesh can be seen in Figure 2 and 3.

Material	Elastic-Plastic	Modulus of Elasticity	E	210000.000	MPa
		Poisson's Ratio	ν	0.000	
		Shear Modulus	G	105000.000	MPa
Geometry	Plate	Length	L	1.000	m
		Width	w	0.050	m
		Thickness	t	0.005	m
Load	Plate	Pressure	p	2750.000	Pa

Small deformations are considered and the self-weight is neglected in this example. Determine the maximum deflection $u_{z,max}$. The aim of this example is to show that a surface of the surface stiffness type *Without Tension* behaves linearly under bending.



Analytical Solution

The bending moment M for the plate under continuous loading $q = pw$ is defined by

$$M = -\frac{q(L-x)^2}{2} \quad (16-1)$$

Recalling the elementary equation for the beam deflection using the Euler-Bernoulli theory

$$EI_y \frac{d^2 u_z}{dx^2} = M \quad (16-2)$$

where I_y is the quadratic moment of a cross-section with respect to the y -axis and it is given by

$$I_y = \frac{wt^3}{12} \quad (16-3)$$

We integrate the equation (16-2) from 0 to L twice. Upon applying the boundary conditions for cantilever beam $u_z(0) = u_z'(0) = 0$, the solution takes the following form

$$u_z(x) = \frac{qx^2(6L^2 - 4Lx + x^2)}{24EI_y} \quad (16-4)$$

The maximum displacement occurs at the point $x = L$:

$$u_{z,max} = \frac{qL^4}{8EI_y} = \frac{3pL^4}{2Ew^2t} = 157.143 \text{ mm} \quad (16-5)$$

Figure 1 - Cantilever with Uniform Load Testing Parameters and Analytical Solution (Dlubai Software, 2016a)

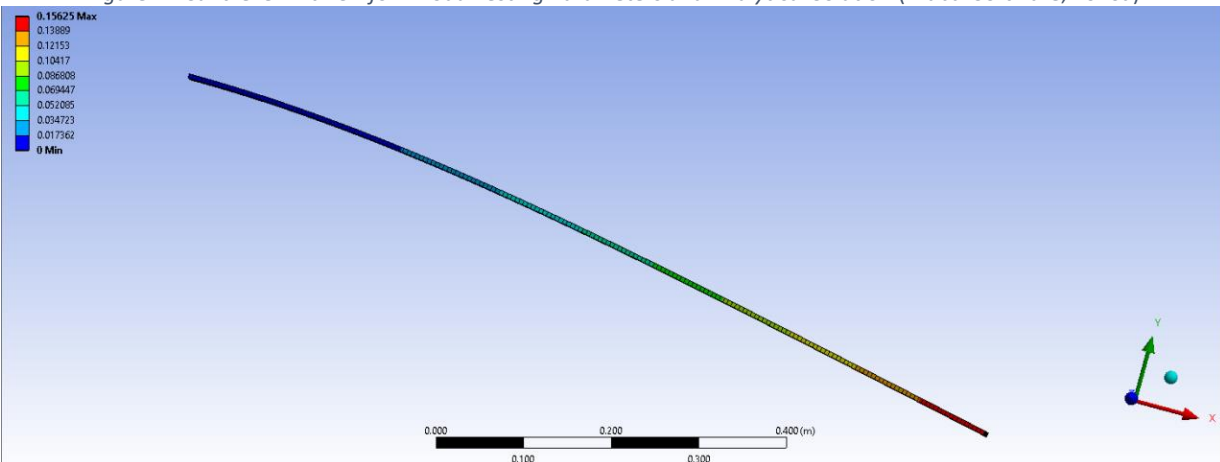


Figure 2 - Cantilever with Uniform Load 5mm Mesh

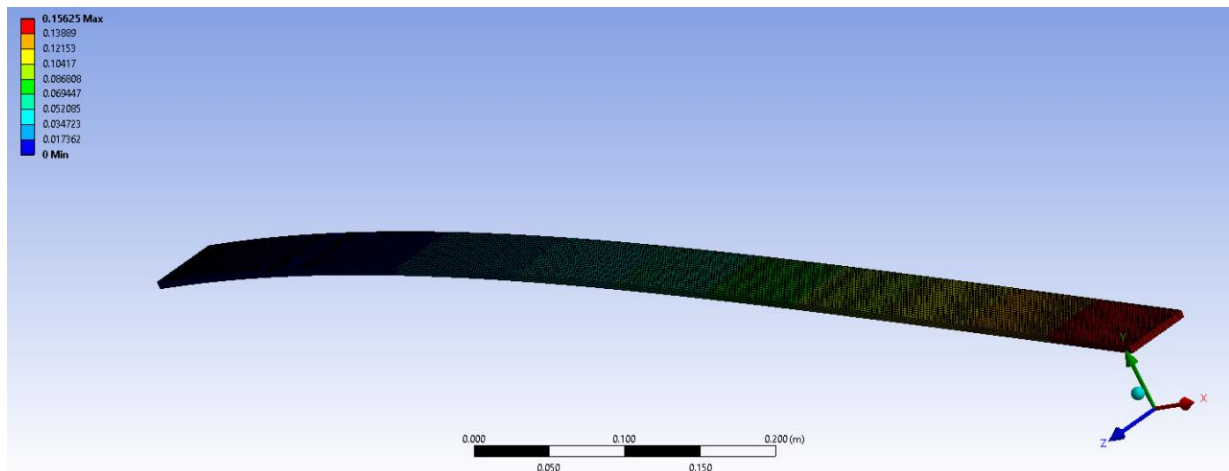


Figure 3 - Cantilever with Uniform Load 2mm Mesh

This test was conducted with a fixed support at one end as a boundary condition and a uniform pressure of 2750pa across the beam. From Figures 2 and 3 it can be observed that CAE software can be used to provide a visual representation of deformation of a material due to pressure. This visual representation shows a colour scale representing when deformation is occurring, this can be used by engineers to determine areas of a component that can be improved.

Table 1 - Cantilever with Uniform Load Mesh Size Comparison Results

Mesh Size	Nominal Deflection (mm)	Ansys Deflection (mm)	% Error
5mm	157.143	156.25	0.568
2.5mm	157.143	156.25	0.568
2mm	157.143	156.25	0.568

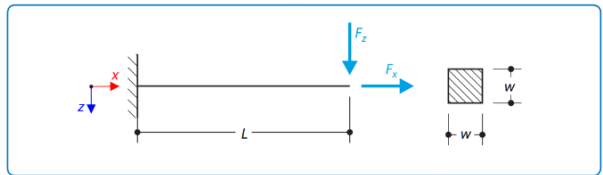
From the table above it can be observed that the mesh size had no impact on the results of this test for the mesh sizes tested. This is not expected, as explored by Jumaa and Yousif (2019) it is expected that accuracy should increase as mesh size reduces due the increase of the fidelity of model simulation. It can also be seen that Ansys is under predicting the nominal result. If a study similar to this was used in the design of a component, then this would need to be considered in the factor of safety considerations of any design.

2.2 Cantilever with Axial Load

Figure 4 below shows the parameters for the test, equations required, and nominal results for the test. The test was conducted with a fixed support at one end with a transverse force of 7650N and an axial force of 1600N. Examples of the test results from the 4mm and 1mm mesh can be seen in Figure 2 and 3.

Material	Steel	Modulus of Elasticity	E	210000.000	MPa
		Poisson's Ratio	ν	0.300	—
Geometry	Cantilever	Length	L	10.000	m
		Cross-section width	w	0.080	m
Load		Transversal Force	F_z	7.650	kN
		Axial Force	F_x	1.600	kN

The self-weight is neglected in this example. Determine the maximum deformations of the structure $u_{z,max}$ and $u_{x,max}$ by means of the geometrically linear analysis, second order analysis and large deformation analysis.



Analytical Solution

Linear Analysis

Desired deformation can be found by means of Geometrically linear analysis very simply. The transversal force and the axial force act independently and the force equilibrium is determined from the original geometry, because the expected deformations are small. The axial force causes only the axial deformation. Both deformations are defined by well-known formulae.

$$u_{x,max} = \frac{F_x L}{EA} = 0.012 \text{ mm}$$

(42 – 1)

Verification Example: 0042 – Bending Cantilever with Axial Force

$$u_{z,max} = \frac{F_z L^3}{3EI_y} = 3557.478 \text{ mm}$$

(42 – 2)

Where A is the cross-section area and I_y is the quadratic moment of the cross-section to the y-axis¹.

Figure 4 - Cantilever with Axial Load Testing Parameters (Dlubai Software, 2015)

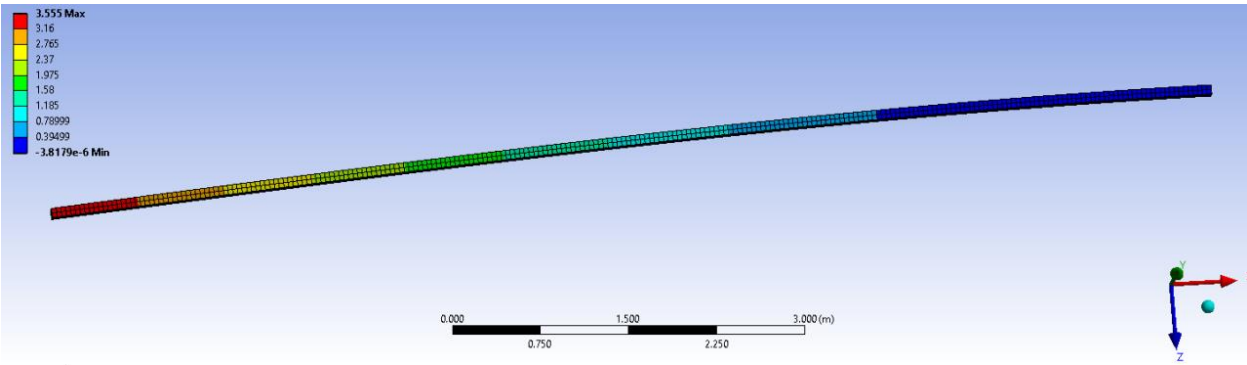


Figure 5 - Cantilever with Axial Load 4mm Mesh

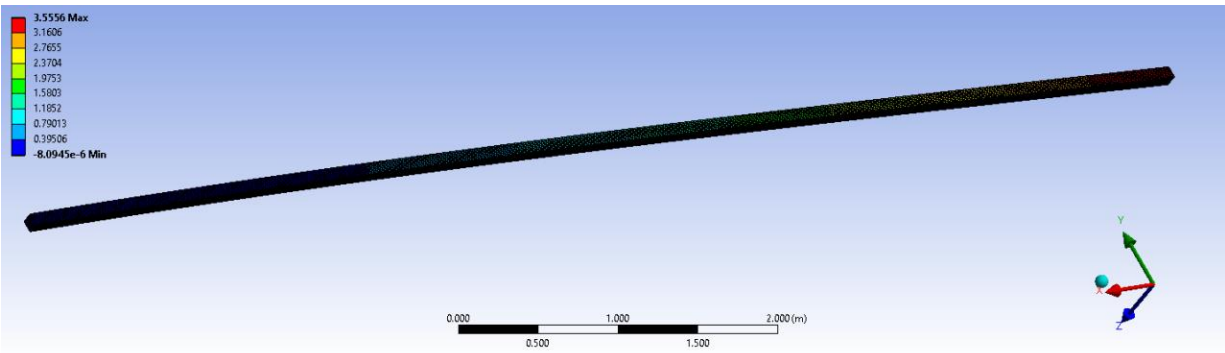


Figure 6 - Cantilever with Axial Load 1mm Mesh

Table 2 - Cantilever with Axial Load Deformation Mesh Size Comparison Results Z Axis

Mesh Size	Nominal Deformation (mm)	Ansys Deformation (mm)	% Error
4mm	3555.478	3555	0.013
2mm	3555.478	3555.4	0.002
1mm	3555.478	3555.6	0.003

Table 3 - Cantilever with Axial Load Deformation Mesh Size Comparison Results X Axis

Mesh Size	Nominal Deformation (mm)	Ansys Deformation (mm)	% Error
4mm	0.012	21.324	177600
2mm	0.012	21.326	177616
1mm	0.012	21.327	177625

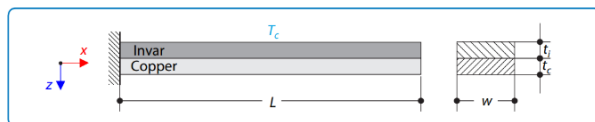
From the table (Table 2) it can be observed that the testing is accurate to around 0.01%. It can also be observed that the mesh size does have an impact on the results, but this is insignificant in this test. It can be noted that the 4mm mesh and 2mm mesh under-predicts while the 1mm mesh over-predicts, suggesting there is a 'sweet spot' for mesh size for a given simulation. Different tests requiring a different ideal mesh size is further explored by Jumaa and Yousif (2019).

It can be seen from Table 3 that the result does not correlate for deformation in the X axis, the X axis represent length deformation. Due to the magnitude of the error this may be due to a difference in how deformation is mapped to the axis' between RFEM 5 and Ansys. This error could also be due to errors in test setup, a retest would be needed to confirm results.

2.3 Bimetallic Strip with Thermal Load

The test was set up with two equally sized strips of metal, copper, and invar, bonded together with a fixed support at one end. Figure 7 below shows the parameters for the test, equations required, and nominal results for the test. A thermal load of 100°C was applied equally to the assembly.

Material	Invar	Modulus of Elasticity	E_i	137000.000	MPa
		Poisson's Ratio	ν_i	0.280	—
		Coefficient of Thermal Expansion	α_i	1.200×10^{-6}	$^{\circ}\text{C}^{-1}$
	Copper	Modulus of Elasticity	E_c	130000.000	MPa
		Poisson's Ratio	ν_c	0.354	—
		Coefficient of Thermal Expansion	α_c	2.000×10^{-5}	$^{\circ}\text{C}^{-1}$
Geometry		Cross-section Width	w	5.000	mm
		Layer Thickness	$t_i = t_c = t$	1.000	mm
		Length	L	100.000	mm
Load		Thermal Loading	T_c	100.000	$^{\circ}\text{C}$



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

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

Figure 7 - Bimetallic Strip with Thermal Load Testing Parameters (Dlubai Software, 2018)

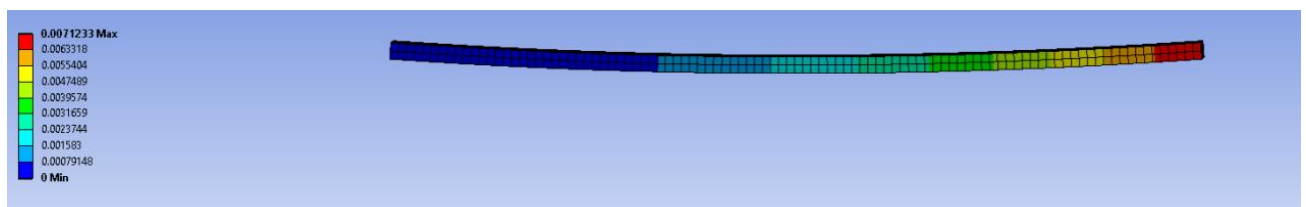


Figure 8 - Bimetallic Strip with Thermal Load 0.5mm Mesh

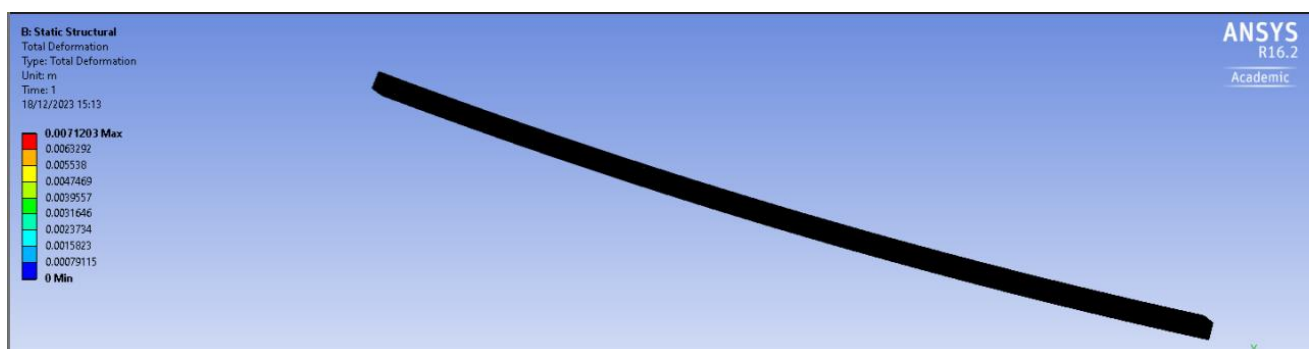


Figure 9 - Bimetallic Strip with Thermal Load 0.1mm Mesh

Table 4 - Bimetallic Strip with Thermal Load Mesh Size Comparison Results

Mesh Size	Nominal Result (mm)	Ansys Result (mm)	% Error
0.5mm	7.049	7.1233	1.054
0.25mm	7.049	7.123	1.050
0.1mm	7.049	7.1203	1.011

From Table 4 it can be observed that the test is accurate to around 1%, it can be noted that the Ansys result is over predicting when compared to the nominal result. It can also be observed that the accuracy of the test increases as the mesh size decreases however this increase in accuracy is small when compared to the increase of time required for the testing. For example, the 0.1mm mesh took around five times longer to calculate than the 0.5mm mesh for a 0.043% increase in accuracy. Therefore, the time cost against accuracy increase must be considered when designing components in industry.

2.4 Beam with Gravity Loading

This test was setup with a fixed support at one end of the beam and an acceleration applied evenly to the body to simulate the effect of gravity. The value of this acceleration will be 10 m/s² to match the nominal results. Figure 10 below shows the parameters for the test, equations required, and nominal results for the test.

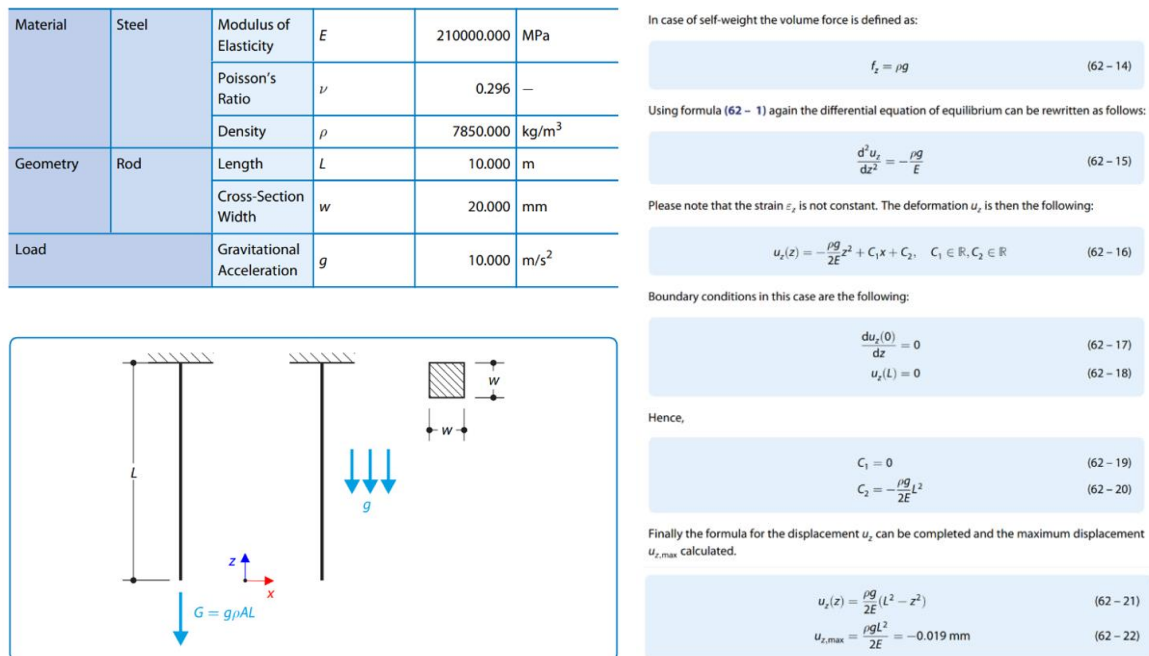


Figure 10 - Beam with Gravity Loading Testing Parameters (Dlubai Software, 2016b)

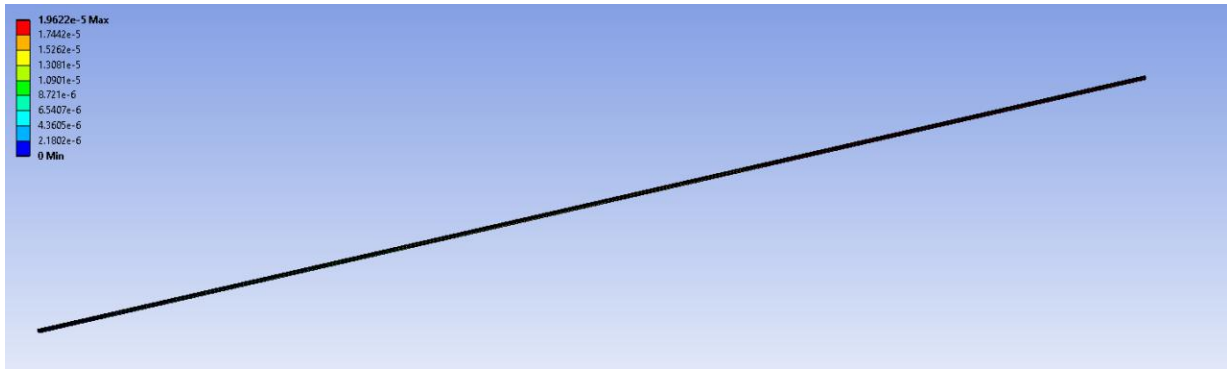


Figure 11 - Beam with Gravity Loading 10mm Mesh

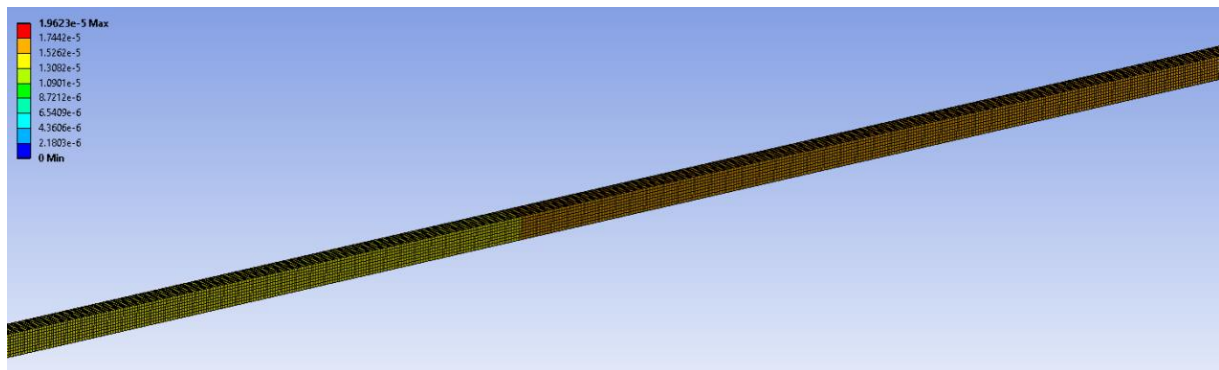


Figure 12 - Beam with Gravity Loading 2.5mm Mesh

Table 5 - Beam with Gravity Load Results Mesh Size Comparison

Mesh Size	Nominal Result (mm)	Ansys Result (mm)	% Error
10mm	0.019	0.019622	3.274
5mm	0.019	0.019622	3.274
2.5mm	0.019	0.019623	3.279

From Table 5 it can be observed that while the test results are accurate to the nominal results, the error percentage is the highest of the successful validation tests. This may be due to the fact that the beam simulated has a large length (10m) when compared to the deformation being simulated (0.019mm), additionally the deformation seen is of a much smaller magnitude of the mesh size suggesting a finer mesh may increase fine detail accuracy. This test highlights some limitations of CAE where fine details in large objects can require a large amount of time to simulate accurately. It can also be seen from the table that the mesh size has little impact on the result error however larger changes in the magnitude of the mesh size may yield different results, more testing would be needed.

2.5 Validation Conclusion

In conclusion for the validation of the use of Ansys, it can be seen from testing that CAE can be used to accurately assess a materials response to various inputs. It can also be seen that mesh size can have an impact on the accuracy of the results in some cases. It should be noted that the limitations of CAE testing would need to be considered for any specific use-case when being used to gain information on new component designs.

3 Topology

Topology describes the use of CAE software in order to optimise a material layout for a given criteria, for example mass or volume as further explored by Zhu, Zhang and Xia (2016). These simulations involve an iterative process where a design is simulated and then modified, the simulations are then compared and a third is created from the results. This process continues in order to refine a design. This gives the process a time cost to accuracy assessment that needs to be made given as the more iterations that are completed the more accurate the simulation will be, however it would cost an increasing amount of time. A common limitation of this process is that the software does not make considerations for the manufacturing needed to produce its design, as discussed by W. Zhang *et al.* (2016), leading to designs that would be either expensive and time consuming to manufacture or require new machines and tooling. However, this process can be useful for engineers to assess where optimisations can be made and inspire new designs.

This section shows an example of the use of a Topology Study in Ansys to refine the design of a plate as shown in Figure 13 below. The plate was set up with a fixed mounting and two bearing loads (500N) applied in opposite directions.

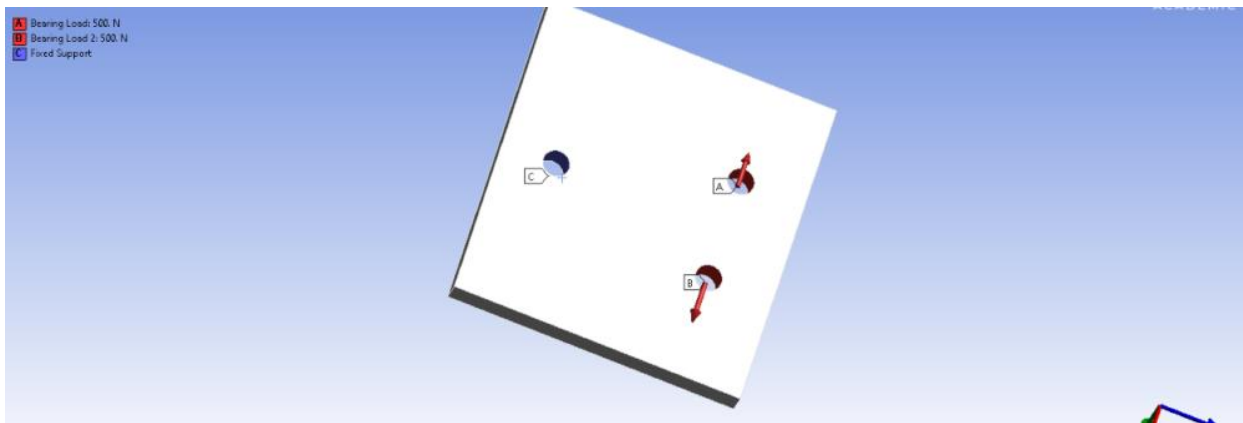


Figure 13 - Topology Study Example Plate Loadings

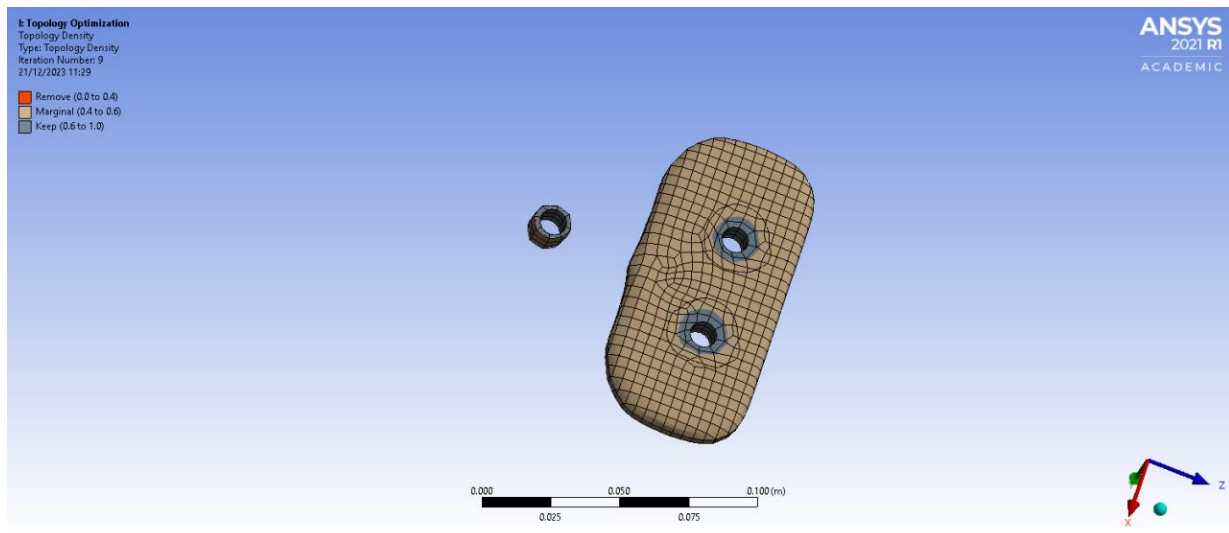


Figure 14 - Topology Study Example 80% Mass Reduction

The component shown in Figure 14 was created to highlight some of the flaws of a Topology study discussed previously. As can be seen the program has reduced the mass by 80% but has rendered the component unusable as the plate no longer consists of one component. This is due to the fact that the simulation is only limited by pre-defined boundary conditions of the user and therefore these conditions must be considered carefully while using this method of analysis.

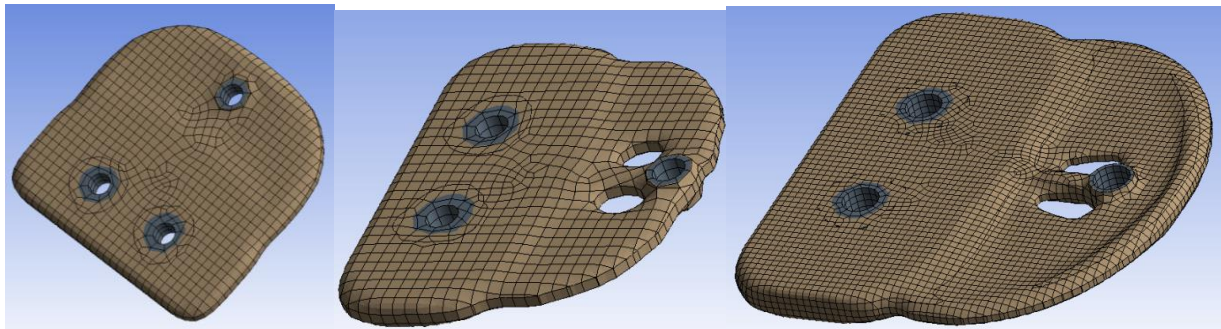


Figure 15 - Topology Study Examples - 10% Mass Reduction (1), 30% Mass Reduction (2), and 30% Mass Reduction Half Mesh Size (3)

From the examples in Figure 15 it can be observed that the study can be used to find areas to reduce mass as shown in (1). However as shown in (2), where the simulation has opted to hollow the areas under the fixed support, it can sometime take a less optimal route to mass reduction than could be available for example further thinning and rounding of the plate overall. It can also be observed when comparing (2) to (3) that the mesh size has a significant impact on the fine detail of the result produced.

4 Linked Study

The linked study was set up using the following studies.

- A ridged dynamics study to export motion loads placed on the joint between the piston and the gudgeon pin due to rotation of the crank shaft.
- A steady state thermal study of the piston to export data on structural degradation due to heat.
- A Static Structural study of the piston to assess deformation and stress concentrations of the piston.

Shown below is the linked study in Ansys.

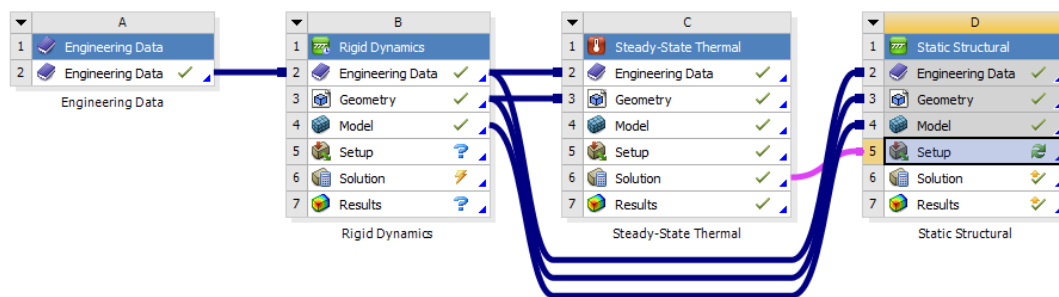


Figure 16 - Ansys Linked Study

This section will conduct three separate studies to demonstrate the way Ansys and other CAE systems can be used in the design process. These studies involve a change of piston design parameters between tests to replicate potential use cases. The first piston design is referred to as the standard design and is provided, along with its material data (6061-T6 Aluminium), were provided by UWSTD (2023). The second, referred to as the alternate material design, used Ansys' standard material for structural steel and is a replication of an engineer wanting to change a designs material without prior knowledge of its effects. This example could represent an engine design team tasked with a change of fuel, for example petrol to diesel, necessitating a change of piston material. The third is an alternate design of the piston using the original material with sections of the piston removed, this is to replicate the use-case in which an engineer has a view to reduce mass and can use CAE to help predict the effect of changes made to the design. This example could represent a motorsport team looking for areas to reduce mass. The design differences of the third piston can be seen in Figures 22-24 where the piston has had material removed from the crown and holes drilled into the underside, these are simple changes that could easily be machined with basic tooling.

4.1 Rigid Dynamics Study

Shown below is an example of the Rigid Dynamics study.

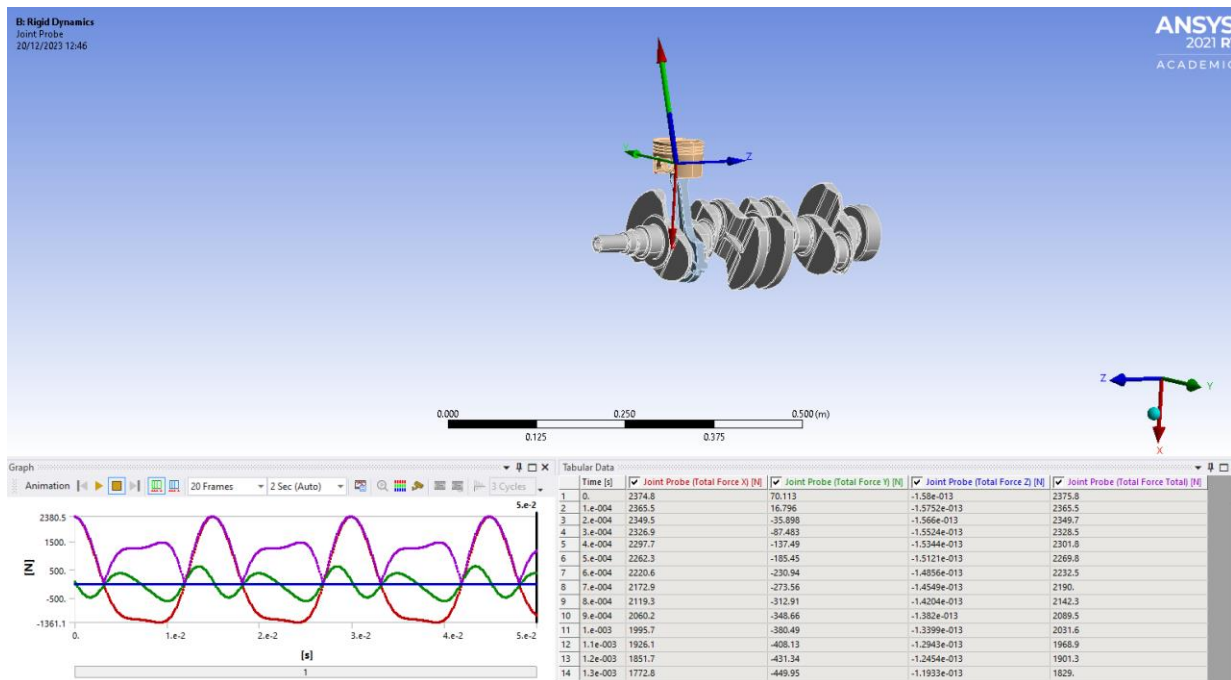


Figure 17 - Example of the Rigid Dynamics Study

The test was set up with relevant joints allowing the assembly to rotate to replicate the motion of a real crank assembly. A rotational velocity of 4000 RPM was applied to the crankshaft and a joint probe was placed on the joint between the gudgeon pin and the piston. Testing setup for joints and time stepping is shown below (Figure 18). The maximum load was exported for use in the Static Structural study for each piston design as this would need to be factored in when designing for overall strength. If considering fatigue life when designing components, it would be useful to export minimum loadings as well to create a load profile for testing.

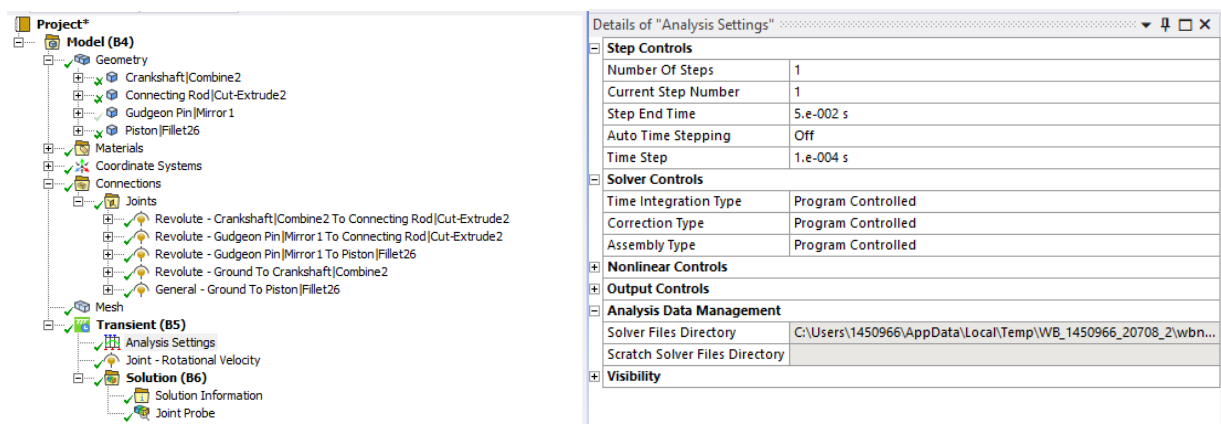


Figure 18 - Rigid Dynamic Study Setup

It is important to test each design as the varying mass caused by differing material density or design volume will affect the inertial load placed on the piston due to its acceleration. An example of this can be seen below (Figure 19 and 20), where the Steel Piston creates nearly three times as much inertia force than the standard design (6906.8N to 2380.5N). This is due to the densities of the materials, 2.7 g/cm³ for 6061-T6 aluminium (Aerospace Specification Metals Inc., 2023) and 7.85 g/cm³ for Steel (Value taken from Ansys).

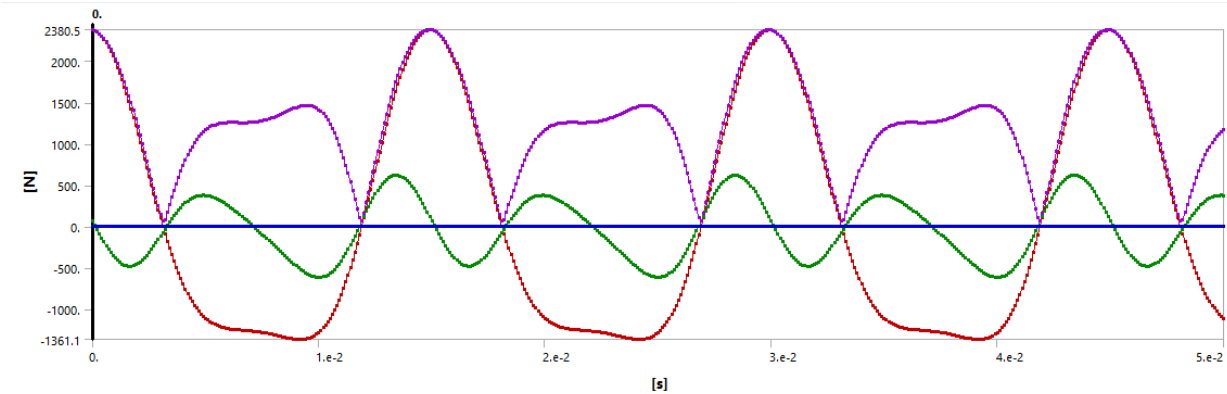


Figure 19 - Rigid Dynamics Results Standard Piston

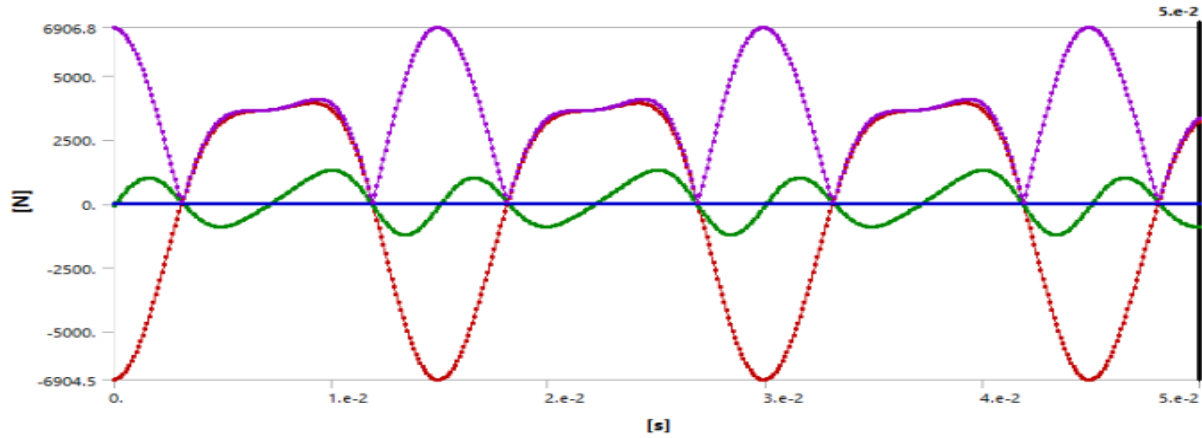


Figure 20 - Rigid Dynamics Results Steel Piston

4.2 Steady State Thermal Study

The steady state thermal study was setup using film coefficients and applied heat loadings provided by UWSTD shown below. This was applied to the model as a series of convection temperature inputs. The 6061-T6 was setup with a temperature dependant Young's Modulus as shown below, it should be noted that the material properties shown in Figure 21 were used through all testing with 6061-T6 aluminium. All temperatures are °C.

Table 6 - Temperature Boundary Conditions (UWSTD, 2023b)

Piston Area	Heat Transfer Coefficient	Temperature
Crown	497.5	928.5
Crown Side	447.5	425
Top Groove	437.5	190
Middle Groove	437.5	190
Oil Groove	338.75	163.75
Underside	675	91.25
Skirt	462.5	107.5

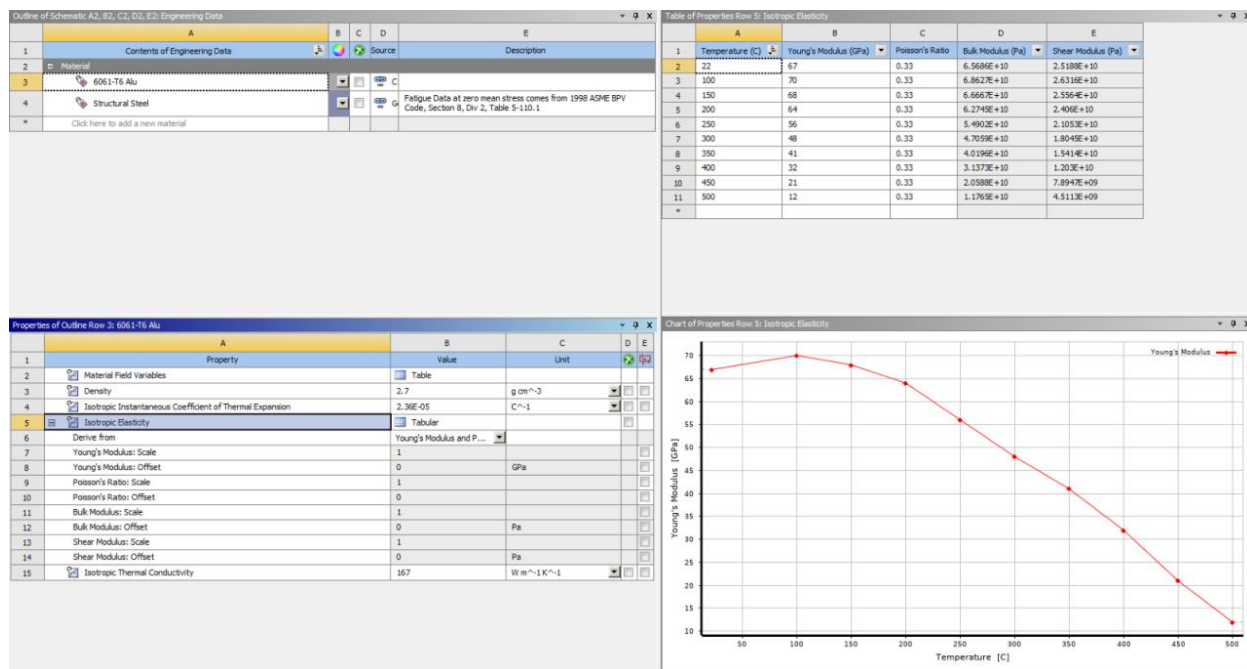


Figure 21 - 6061-T6 Aluminium Material Setup

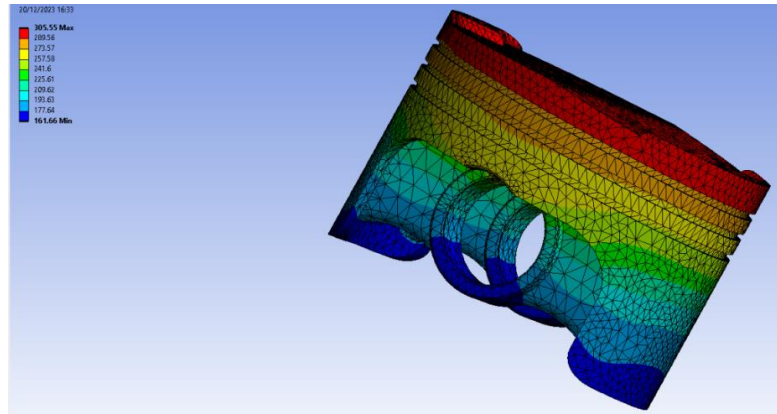


Figure 22 - Thermal Profile Standard Piston

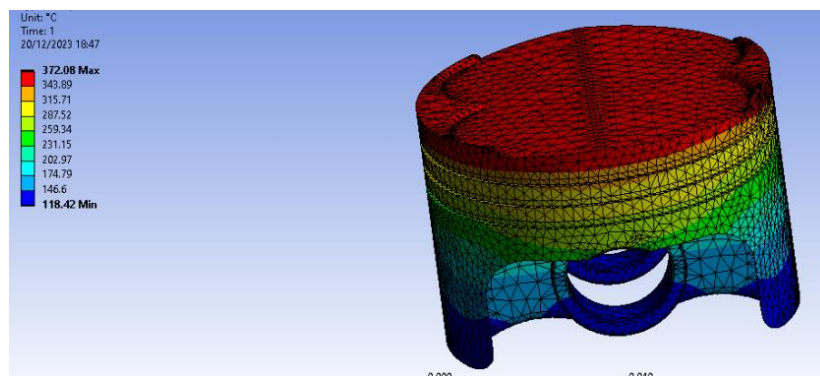


Figure 23 - Thermal Profile Alternate Material

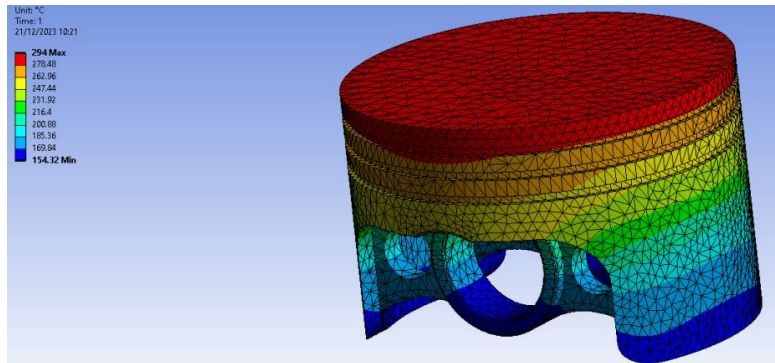


Figure 24 - Thermal Profile Alternate Geometry

From the Figures above (22-23) it can be observed that the difference in material has led to a higher peak temperature (305.55 to 372.08 °C) and lower minimum temperature (161.66 to 118.42 °C) between the designs for the alternate material. This suggests that the original piston material is conducting more heat away from the ‘hotspot’ at the crown. This is expected when comparing the materials thermal conductivity, 167W/mK for aluminium (UWTSD, 2023b) and 60.5 w/mK for Structural Steel in Ansys. It can be observed when comparing Figures 22 and 24 that the maximum and minimum temperatures are lower for the alternate geometry piston, 305.55 to 294.4 °C and 161.66 to 154.32 °C respectively. This may be due to the alternate piston having less surface area that the ‘Crown heat transfer’ coefficient and temperature condition is applied to, highlighting the limitations of testing using the same boundary conditions. This could also be a result of having more surface area on the underside to dissipate heat due to the holes.

4.3 Static Structural Linked Study

The static structural study was set up importing motion loads and thermal profiles as shown in Figure 16. The test was setup using 'Soft Springs' to keep the model in place during testing otherwise the resultant force from the motion loads would mean the model would move away from the 0,0,0 coordinate causing problem for the software. This highlights a common problem associated with CAE where a program's understanding of 3D space is limited to the information provide to it by programmers and users. Results for Total Deformation and Stress were calculated for each design. For this test, the mesh for the model was finer at the crown of the piston for better fidelity of results as shown in Figure 25 below the piston body has a mesh size of 2mm with a mesh size on the crown of 1mm. This technique of using finer meshes around key area of a model can increase simulation accuracy while limiting time needed when compared to using the finer mesh for the whole model. The Figure below shows an example of the mesh sizing-tool with only using a finer mesh on the raised areas of the piston crown and on using a finer mesh on the whole piston crown as used for testing on all models.

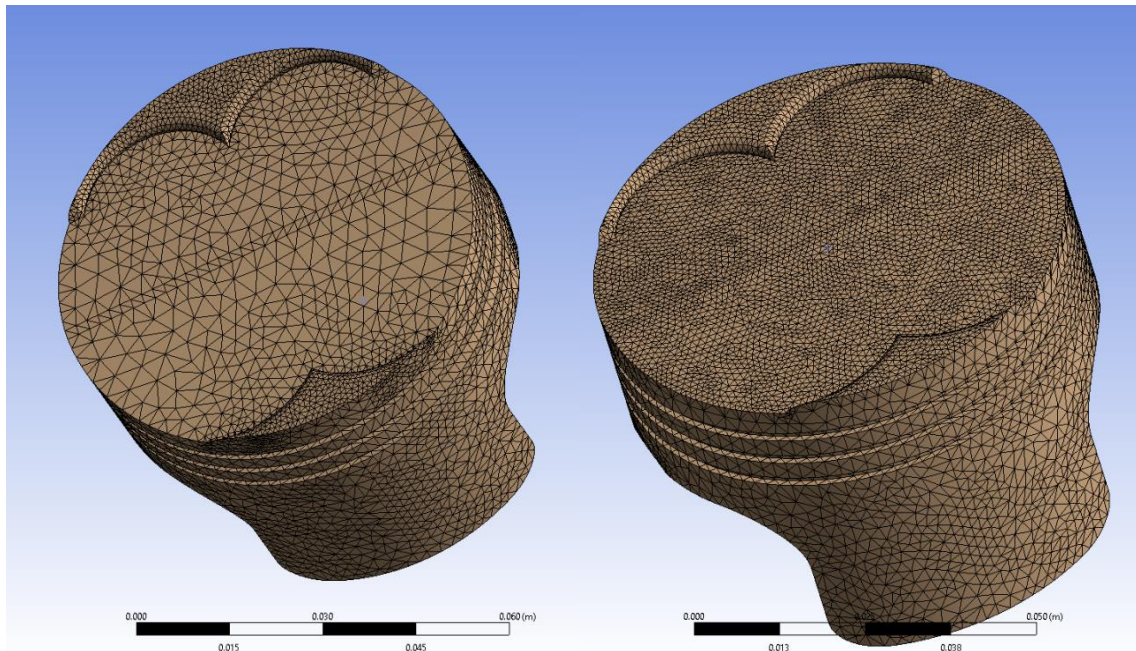


Figure 25 - Piston Crown Mesh Sizing

The Figures on the next page show a comparison of the total deformation of the designs.

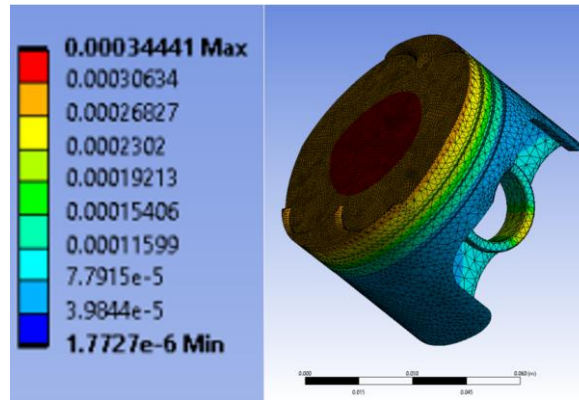


Figure 26 - Total Deformation Standard Piston

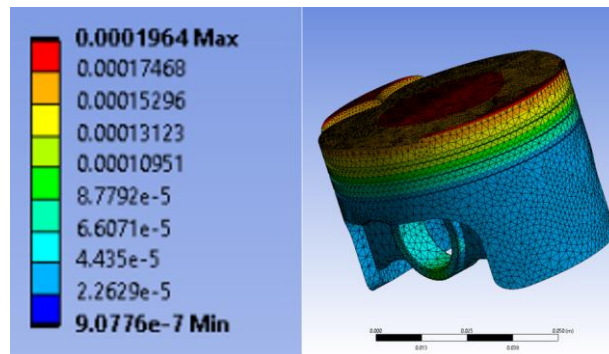


Figure 27 - Total Deformation Alternate Material

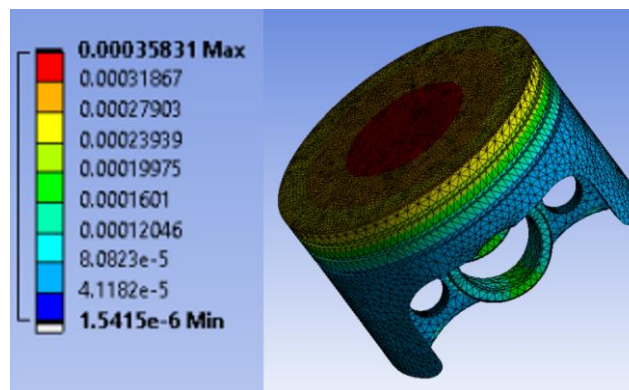


Figure 28 - Total Deformation Alternate Geometry

From Figures 26 and 27 it can be observed that the alternate material has less deformation, this is expected due to the higher strength of the steel. However, this result may be misleading as the standard structural steel in Ansys does not have a temperature dependant Young's Modules like the 6061-T6 aluminium tested. This highlights the limitations of CAE testing as the accuracy of the test are dependant on user defined parameters. When comparing Figures 26 and 28 it can be observed that the alternate geometry piston has a higher maximum deformation due to its reduction in strength from the lost material. However, the deformation on the joint between the piston and the gudgeon pin has reduced, likely due to the reduction in mass reducing the motion loads. This highlights the importance of importing the motion loads for each piston to create an accurate simulation of real-world loads but also further highlights that the real-world relevance of testing conducted using CAE is highly dependent on the accuracy of information provided by the user.

The figures below show the stress distribution of each design.

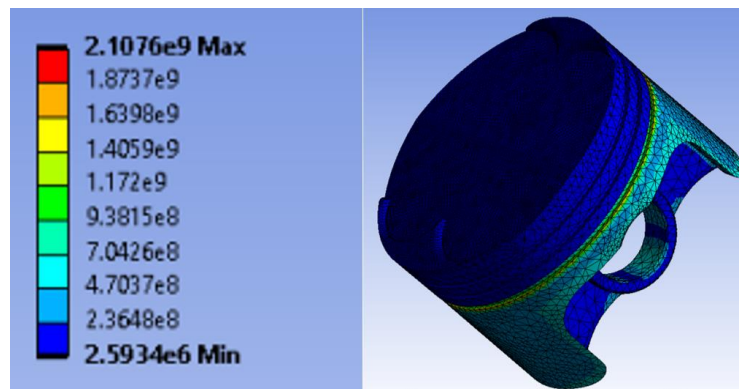


Figure 29 - Stress Distribution Standard Piston

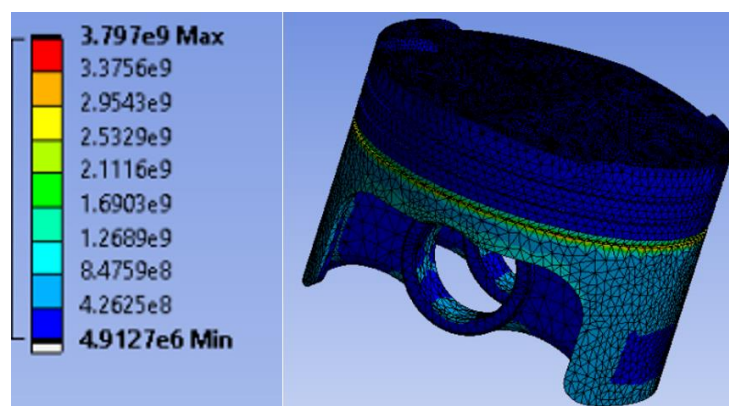


Figure 30 - Stress Distribution Alternate Material

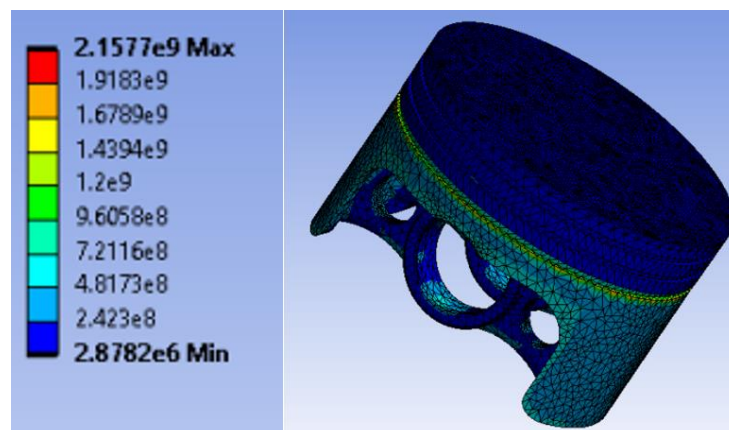


Figure 31 - Stress Distribution Alternate Geometry

When comparing Figures 29 and 30 it can be observed that the overall stress on the piston has increased likely due to the increase in mass of the piston with the alternate material due to its density. When comparing Figures 29 and 31 it can be observed that the stress has increased for the alternate geometry, however this increase is not significant. When comparing Figures 29 -31 it can be observed that despite the geometry changes this stress concentrations across the piston have not changed significantly. This section highlights how CAE can be useful in providing information on the effect of changes to design parameters.

5 Conclusion

In conclusion, CAE can be a useful tool to engineers of all disciplines and provide a method of testing and gaining understanding while limiting resources and funding needed for such testing. It has been highlighted that this kind of testing can be useful to an iterative design process for example when starting out with an existing design (standard piston), CAE can be used to investigate avenues of development like a switch to diesel fuel necessitating change to a steel piston or a low-budget motorsports team looking to reduce mass with limited tooling. The alternate parameters of the alternate material piston and alternate geometry piston can be tested, and this information can be used to inform a further design for testing thus creating an iterative workflow. It is important however to be considerate of the limitation of testing when analysing results, these limitations being a need for user understanding of the problem in order to establish relevant boundary conditions for the test. The effect of mesh size and the problem of 'accuracy against time' that it causes needs to be understood before accurate testing can be conducted. In the example of mass reduction, topology can also be useful however it is important to assess the limitations of this as well. For example, as shown previously, the designs produced using this software can lead to an increased difficulty in machining so are best used to inform about areas of a design from which mass can be reduced. This can still provide a benefit as it reduces time needed for engineers to build separate models and test them individually.

Overall, CAE is useful in aiding engineers in working efficiently however the limitations must be understood to ensure accurate results.

6 References

Aerospace Specification Metals Inc. (2023) *Aluminum 6061-T6*. Available at: <https://asm.matweb.com/search/SpecificMaterial.asp?bassnum=ma6061t6> (Accessed: 20 December 2023).

Boned Ferrer, S. (2023) *Parametric design of pipe bridges: a case of automation and optimization in structural design*. UPC BarcelonaTech. Available at: <http://hdl.handle.net/2117/397634>.

Dlubai Software (2015) *0042 - Bending Cantilever with Axial Force*. Available at: <https://www.dlubal.com/en/downloads-and-information/examples-and-tutorials/verification-examples/000042>.

Dlubai Software (2016a) *0016 - Elastic Bending - Continuous Load*. Dlubai Software. Available at: <https://www.dlubal.com/-/media/769CFB1370D742D9BEE0D37876B3E637.ashx>.

Dlubai Software (2016b) *0062 - Gravity Loading*.

Dlubai Software (2018) *0078 - Bimetallic Strip*. Available at: <https://www.dlubal.com/-/media/B437DA0115FB496C9C7BA58F08960A5E.ashx>.

Ferguson, B. and Wakeman, R. (2004) 'Developing race engines using advanced CAE tools', *SAE Technical Papers* [Preprint]. Available at: <https://doi.org/10.4271/2004-01-3510>.

Jones, R. *et al.* (2017) 'The Ecosystem for Engineering Design Applied to Formula SAE', *SAE Technical Papers*, 2017-March(March). Available at: <https://doi.org/10.4271/2017-01-1324>.

Jumaa, G.B. and Yousif, A.R. (2019) 'Numerical modeling of size effect in shear strength of FRP-reinforced concrete beams', *Structures*, 20, pp. 237–254. Available at: <https://doi.org/10.1016/j.istruc.2019.04.008>.

Rohini, B. and Shanmukha, C. (2017) 'Modelling and Analysis of a Multi-Storey Building using RFEM', *International Journal of Science Technology & Engineering*, 3(09). Available at: https://d1wqtxts1xzle7.cloudfront.net/53593566/IJSTEVS3I9033-libre.pdf?1497961754=&response-content-disposition=inline%3B+filename%3DModelling_and_Analysis_of_a_Multi_Storey.pdf&Expires=1703276832&Signature=dkR~4hlBeimZ0oJCmb1XmTA07AEtWKwv07SXcxIS2LRtHb327.

UWSTD (2023a) *Piston Assembly*.

UWSTD (2023b) *Piston Thermal Study Data*.

W. Zhang *et al.* (2016) 'A new topology optimization approach based on moving morphable components (MMC) and the ersatz material model', *Structural and Multidisciplinary Optimization* [Preprint].

Zhu, J.H., Zhang, W.H. and Xia, L. (2016) 'Topology Optimization in Aircraft and Aerospace Structures Design', *Archives of Computational Methods in Engineering*, 23(4), pp. 595–622. Available at: <https://doi.org/10.1007/s11831-015-9151-2>.