

DESIGNING A RACE CAR REAR WING: STRUCTURAL DESIGN

ACMA 6010: Structural & Fluid Analysis

Andrew Burgess (2115107)
2115107@student.uwtsd.ac.uk

Contents

Introduction 2

Finite Element Analysis Techniques..... 2

General Analysis..... 4

 Lay-up/Design Optimisation 4

 Initial Results 5

 Optimisation..... 8

Physical Testing/Verification..... 10

Conclusion..... 11

References 12

Appendices..... 15

 Appendix A – Base Model 15

 Appendix B – Constraint Placement 15

 Appendix C – Force Placement 16

 Appendix D – Vector Orientation..... 17

 Appendix E – Initial Deformation of Main Element Skin..... 17

 Appendix F – Initial Deformation of Rear Element Skin..... 18

 Appendix G – Optimisation Process and Results 19

 Appendix H – The Optimised Displacement of the Main and Rear Element..... 20

Introduction

This report looked into the design, optimisation and verification of a rear wing design. More specifically, this investigated the upper surfaces of the two main elements of the wing. It was proposed that the skin of the rear wing is produced using traditional carbon fibre reinforced plastic (CFRP) lay-up using a number of layers within a series of orientations. The strength of the resulting part is dependent on the number of layers, its thickness and the orientations in which each layer is achieved.

Using a number of sources, the Finite Element Analysis technique was explored. This included how they are carried out, any benefits or drawbacks and how they may improve upon the design of the final product. The development of such techniques was discussed as well as predictions on how they may be further developed to improve on the methods already in place.

Finite Element Analysis Techniques

Finite Element Analysis (FEA) is used to establish the results of applied loads through the structure of a complex body and used where the geometry is too complex to obtain results through traditional mathematical methods or to achieve results quickly (Rimini, et al., 2022). The results of which include the distribution of stress, deformation and strain (Baccouch, 2021). This is performed by dividing the geometry into multiple shapes to form a mesh.

At the nodes of each element, where adjacent elements meet, the calculations are performed (Sifakis and Barbič, 2016). These nodes are connected with lines which form the boundaries of each element. Smaller elements result in more calculations but a higher accuracy (Rimini, et al., 2022). This compromise is summarised by Convergence. The convergence describes the point at which the mesh, and more specifically the number of elements, achieves the most accurate result (SimScale, 2023). Increasing the number of nodes, increases the total degrees of freedom and increases the result accuracy (Sifakis and Barbič, 2016). Shown in figure 1, increasing degrees of freedom produces a larger number of values predicting the overall effect and is seen when the quantity plateaus. Increasing the Degrees of freedom beyond this point will not alter the result, only increasing solve time (Thelin, et al., 2021).

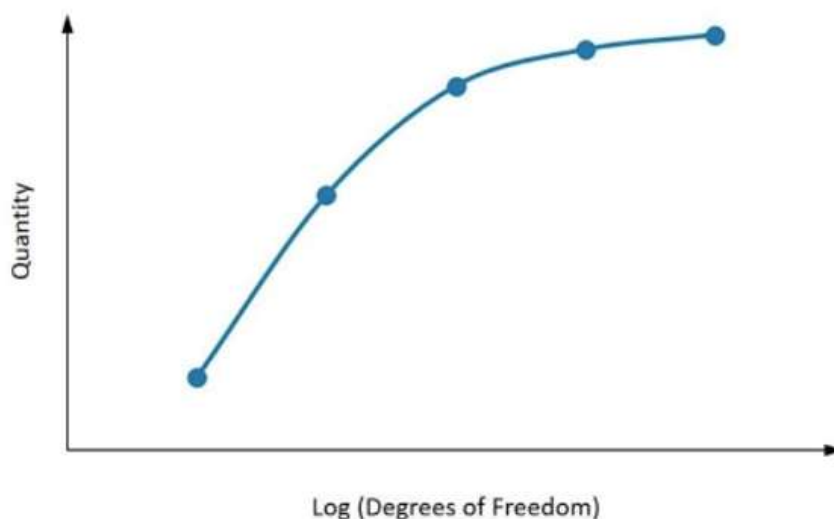


Figure 1: The convergence of a quantity with an increasing degree of freedom (SimScale, 2023)

The principle of investigating the behaviour of a structure using a finite number of points dates back to the 3rd century in China (Lay-Yong and Ang, 1986). The theoretical principle of Finite Element Analysis was developed in 1955 for use within aviation to maintain the required strength, whilst minimising mass (Yue, 2010). This principle states that as the input (force, pressure, etc) is applied, a number of possible outcomes are achieved that express the different ways the body reacts. The result with the lowest total energy is selected as the final outcome. (Baccouch, 2021)

These results are used to aid in the design process. A common method of optimisation using FEA is topology optimisation. This technique directs material into areas to provide greater integrity (Sigmund and Maute, 2013). The process begins with a CAD (Computer Aided Design) model, providing the geometry to be analysed. Features like holes

are protected to ensure they are not consumed (Broekaart, 2015). The Objectives and constraints are specified which includes strength, factor of safety, etc. This method benefits the design process as fewer design iterations are required as the problem is solved with the desired conditions in a single process. With the development taking less time, prototypes and concepts are produced quicker (Sigmund and Maute, 2013).

Despite these benefits, it can only direct material and not refine the shape to minimise stress. Shape optimisation is commonly used following topology optimisation to develop the design into a practical model. Areas are identified where the measured outcome is beyond an acceptable limit and making changes to bring the design closer to the objective (Duan and Sheppard, 2002). The final outcome will differ from the optimised version as practical issues like mass, size and the manufacturability in processes like machining or composite lay-up (Chang, 2015).

Composites are generally anisotropic and rotated in consecutive layers to provide strength and stiffness in different axis. Alternating layers evenly provide properties consistent in all axes but is inefficient in applications like rear wings where the forces are high but along fewer axes. To establish the layer thickness and their orientation, composite optimisation is used. Composite optimisation requires the user to objectives like mass, constraints like the deflection and design variables like layer thickness (Macquart, et al., 2018). The results can be discretised to provide values useful for the design and lay-up of the composite part. This can almost eliminate the need for iterative design through the generation of practical results (Sifakis and Barbić, 2016). Other physical factors that can affect the performance of a composite structure includes the fibre density, fibre thickness and weave patterns. Composite optimisation can refine these parameters (Almeida and Awruch, 2009). Recent developments resulted in solving methods using genetic algorithms, which finds a solution through a number of iterations. Unlike traditional methods, variables are both altered randomly and inherit beneficial variables from previous iterations until an optimum solution is found (MathWorks, 2023). This method has been used extensively to solve problems that incorporates multiple design variables that would otherwise be difficult to solve without experimental data (Brighenti, 2005). This development from the classic algorithm whereby each iteration is completed through the progression of the design variable, with the result complying or violating the constraints (MathWorks, 2023).

General Analysis

The rear element skin design will be developed using the composite optimisation technique, with the skin composed of unidirectional CFRP. The lay-up procedure uses multiple layers, bonded with a polymer resin. For the initial optimisation the thickness of each CFRP ply is set at 0.1mm with a total of 25 unidirectional plies, separated into groups based on their orientation from the zero-vector shown in Appendix D. The number of plies within each orientation is as follows.

Layer No.	Layer Orientation	Main Element Skin (mm)	Rear Element Skin (mm)
Layer 1	-90°	0.5	0.5
Layer 2	-45°	0.5	0.5
Layer 3	0°	0.5	0.5
Layer 4	45°	0.5	0.5
Layer 5	90°	0.5	0.5
Total Thickness (mm)		2.5	2.5

Figure 2: The Unidirectional ply orientation and quantity to begin the analysis.

The purpose of this analysis was to produce a structure intended to comply with the FIA rules over maximum skin deflection. This rule stipulates the maximum deflection for the skin to be no more than 2mm when a force of 60N is applied to its surface (Fédération Internationale de l'Automobile, 2022). For this model, the force was applied perpendicular to the skin as this would best reflect the process used during scrutineering as well as the downforce during racing.

Lay-up/Design Optimisation

It was required to optimise the design to minimise the mass of the rear wing but to have acceptable displacement. The process begins with the FEA model of the element skins. Shown in Appendix 1, these are surface models, or shells, with an infinitesimally small thickness.

The elements are fixed at each end in six degrees of freedom. This is intended to reflect the connection of the element to the end plates as shown in Appendix B. By fixing the ends in this way, only deformation of the skin is present within the results and not the rotation or translation at each end. The assembled wing may exhibit deformation through the end plates and wing structure during scrutineering but given that this deformation is unknown and the skin deflection test covers the properties of the skin only, the fixed constraint was applied.

This is resembled in Appendix C. In the software, the equal distribution of the force is shown by the test force distributed over the number of elements on each skin. The properties of CFRP were created along with their layer orientations. The base orientation of the elements were established (shown in Appendix D) the plane of 0° was set along the theoretical length of the car, with the unidirectional orientations measured from this plane.

Initial Results

Once the solver had modelled the results of this analysis, the stiffness and the stiffness-to-weight ratio were calculated as shown in the formulae below. This was used to compare the performance of the element skins at each stage of the optimisation process.

$$F = kx$$

Where,

F = the applied force on the element skin (N)

k = The stiffness of the element skin (N/mm)

x = The deflection of the element skin (mm)

Using this stiffness value, the stiffness to weight ratio was used as another method of comparison with the optimised elements. The formula for this ratio as follows.

$$\text{Stiffness to Weight ratio} = \frac{k}{mg}$$

Where,

K = The stiffness of the element skin (N/mm)

m = The mass of the element skin (kg)

g = Acceleration due to Gravity (m/s^2)

To reduce the mass of the element skins, the number of plies, overall thickness and subsequently the stiffness of the skin is reduced, and the deflection will increase. The composite optimisation provided balance between the mass and deflection.

Main Element

The displacement of the main element skin is seen in Appendix E and Figure 3. The displacement on this section was a maximum of 0.49mm, located where the cup was applied. This initial lay-up produced a mass of 1.29353kg, which was to be minimised in the proceeding optimisation.

Using the formula for the applied force, the stiffness of the main element was found.

$$k = \frac{F}{x}$$

$$k = \frac{60}{0.49}$$

$$k = 122.449 \text{ N/mm}$$

Therefore, the stiffness to weight ratio of the main element was calculated as follows.

$$\text{Stiffness to Weight ratio} = \frac{k}{mg}$$

$$\text{Stiffness to Weight ratio} = \frac{122.449}{1.29353 \times 9.81}$$

$$\text{Stiffness to Weight ratio} = 9.6496 \text{ N}^2/\text{mm}$$

Layer No.	Layer Orientation	Main Element Skin (mm)
Layer 1	-90°	0.5
Layer 2	-45°	0.5
Layer 3	0°	0.5
Layer 4	45°	0.5
Layer 5	90°	0.5
Total Thickness (mm)		2.5
Displacement (mm)		0.49
Mass (kg)		1.29353
Stiffness (N/mm)		122.449
Stiffness to Weight Ratio (N ² /mm)		9.6496

Figure 3: The initial lay-up and results of the analysis on the main element

Rear Element

Deformation on the rear element was larger with a maximum of 1.9872mm of deformation, as shown in Appendix F. These displacement results suggested the initial design had a superfluous amount of material. For the purpose of comparison, the stiffness of the skin was calculated. This was completed using the following formula.

The stiffness of the rear element was found.

$$k = \frac{F}{x}$$

$$k = \frac{60}{2}$$

$$k = 30 \text{ N/mm}$$

The Stiffness to weight ratio was calculated as follows.

$$\text{Stiffness to Weight ratio} = \frac{k}{mg}$$

$$\text{Stiffness to Weight ratio} = \frac{30}{0.589412 \times 9.81}$$

$$\text{Stiffness to Weight ratio} = 5.1884 \text{ N}^2/\text{mm}$$

Shown below the stiffness and stiffness to weight ratio of the main and rear element differed to the main element. Although the test specifies a force of 60N, there is no consideration of the area. The pressure over the surface on the rear element was higher. The profile of the element would contribute to this difference, though it is unknown by what proportion. Yet, the displacement of the rear element meant optimisation would be limited due to its close proximity to the limit of 2mm.

Layer No.	Layer Orientation	Rear Element Skin (mm)
Layer 1	-90°	0.5
Layer 2	-45°	0.5
Layer 3	0°	0.5
Layer 4	45°	0.5
Layer 5	90°	0.5
Total Thickness (mm)		2.5
Displacement (mm)		1.9872
Mass (kg)		0.589412
Stiffness (N/mm)		30
Stiffness to Weight Ratio (N ² /mm)		5.1884

Figure 4: The initial lay-up and results of the analysis on the rear element

Optimisation

The structure was optimised using Design Studio for Genesis. The objective of the optimisation was to minimise the overall mass of the element skins. The thickness of each layer was where this objective was achieved, known as the variables. The limiting constraint was the 2mm displacement of each skin. Given that deflection can provide an aerodynamic benefit (Autosport, 2020) and that the FIA mandates restricted skin displacement, it was considered worthwhile maximising this value to be as competitive as possible (Fédération Internationale de l’Automobile, 2022). This operation was applied to both the main and rear element skins, it was anticipated that the rear element would be thicker to accommodate to the smaller surface area.

Optimisation Process

The process began with the existing model. This used the same parameters such as the applied forces and constraints on movement. The process was carried out separately on each element to ensure complicity on both elements. A minimum thickness of each layer was set to 0.1mm whilst the maximum thickness was set to 2.5mm. The thickness of each ply was 0.1mm, hence the minimum thickness of each layer was at least one ply thick. In the interest of weight-saving it was decided to set the maximum thickness of each layer to 2.5mm. This meant the element could also increase in thickness if deemed necessary by the solver.

Shown in Appendix H, the maximum displacement of the main and rear element was less than 2mm at the front of each element. The main element optimisation was completed within eight cycles and with hard convergence and the rear element skin was completed within three cycles, also with hard convergence. The results for each element can be seen in Figure 5 below.

Layer No.	Layer Orientation	Main Element Skin (mm)	Rear Element Skin (mm)
Layer 1	-90°	0.252	0.48
Layer 2	-45°	0.156	0.52
Layer 3	0°	0.127	0.529
Layer 4	45°	0.11	0.519
Layer 5	90°	0.268	0.442
Total Thickness (mm)		0.913	2.49
Mass (kg)		0.4746	0.5871
Maximum Displacement (mm)		0.1993	0.1993
Stiffness (N/mm)		30	
Stiffness to Weight Ratio (N ² /mm)		6.4435	5.2088

Figure 5: The optimised layer thicknesses and deformation properties of both element skins.

Within the main element skin, layer one (-90°) was the thickest at 0.272mm, followed by layer five (90°) at 0.25mm. These layer orientations travel perpendicular to the length of the car, terminating at the end plates. This structure would be expected for CFRP, where it is strongest when the deflection is perpendicular to the ply orientation (Mohamed and Abdelbary, 2023).

Within the rear element, the thicknesses of layers one and five were lower than all other layers. This arrangement was found during its optimisation process that met the set objective and constraint but only when the plies of a particular orientation were stacked continuously in a single layer and not staggered. Similarly, it is possible that the layer position within the skin would impact the displacement due to the complex geometry. However, the nature in which this analysis was constructed prevented the plies being placed sparsely in differing orientations and potentially failing to achieve a better solution (Altair, 2021).

Discretisation of the Results

The most significant issue with these results was the calculated layer thickness. Using the standard 0.1mm thick CFRP ply, the resulting layers above do not comprise of wholes plies. For instance, layer one of the main element would require 2.52 plies to construct and would be impractical to produce. If a proposed manufacturing tolerance of 0.25mm by Performance Composites (2023)is considered, parts manufactured through manual lay-up may deflect beyond the 2mm limit. By discretising the results, the mass would increase to comply with the 2mm constraint.

The following layer thicknesses, along with the results of the previous analyses, were calculated as follows.

		Optimised-Discretised Element Skins		Optimised Element Skins		Base Model Element Skins	
Layer No.	Layer Orientation	Main Element Skin (mm)	Rear Element Skin (mm)	Main Element Skin (mm)	Rear Element Skin (mm)	Main Element Skin (mm)	Rear Element Skin (mm)
Layer 1	-90°	0.3	0.5	0.252	0.48	0.5	0.5
Layer 2	-45°	0.2	0.6	0.156	0.52	0.5	0.5
Layer 3	0°	0.2	0.6	0.127	0.529	0.5	0.5
Layer 4	45°	0.2	0.6	0.11	0.519	0.5	0.5
Layer 5	90°	0.3	0.5	0.268	0.442	0.5	0.5
Total Thickness (mm)		1.2	2.8	0.913	2.49	2.5	2.5
Displacement (mm)		1.506	1.51	2.0	2.0	0.49	2.0
Mass (kg)		0.6209	1.4488	0.4746	0.5871	1.29353	0.589412
Stiffness (N/mm)		39.8406	39.7351	30	30	122.449	30
Stiffness to Weight Ratio (N ² /mm)		6.5409	2.7957	6.4435	5.2088	9.6496	5.1884

Figure 6: The results of the base analysis, optimised and optimised-discretised results.

Shown in figure 6, this process had resulted in a thicker main and rear element skin. Therefore, the displacement had been reduced further below the 2mm constraint. Compared to the initial results, the displacement had increased with the removal of material. The largest weight-saving was seen with the main element skin with 52% reduction in mass. The rear element experienced an increase in mass, which was attributed to the discretisation of the layer thicknesses. The stiffness-to-weight ratio had increased for the main element but decreased for the rear element. This can be attributed to the substantial increase in thickness but suggests that a better compromise could be achieved through discretisation, rather than rounding up each layer thickness. Additionally, it suggests a different material may achieve similar displacement but with a lower mass. However, the displacement was considered far enough below the 2mm limit to be confident that, despite the manufacturing tolerances, would remain compliant. Given the optimised-only results were outside of the 0.25mm tolerance, it can be reasoned the physical element skins will not exceed 2mm of displacement.

Physical Testing/Verification

The purpose of physical testing was to provide verification of the analysis that had been completed previously; to ensure it has a sufficiently low margin for error. In this instance, it was recommended that the idea of the verification pyramid was to be used. This principle starts at the base of a pyramid where coupon tests are performed. This is intended to investigate the material properties on standardised samples, using both physical testing and simulation. If the results of the test and simulation corroborate each other, the verification moves to the next level. If the results do not agree, the simulation is amended until confirming results are achieved (Bruyneel, et al., 2014). Each new level grows in complexity until the full physical system is reached. This is intended to develop a product with predictable properties; simultaneously reducing development costs compared to iterative design techniques (McCaslin, 2021).

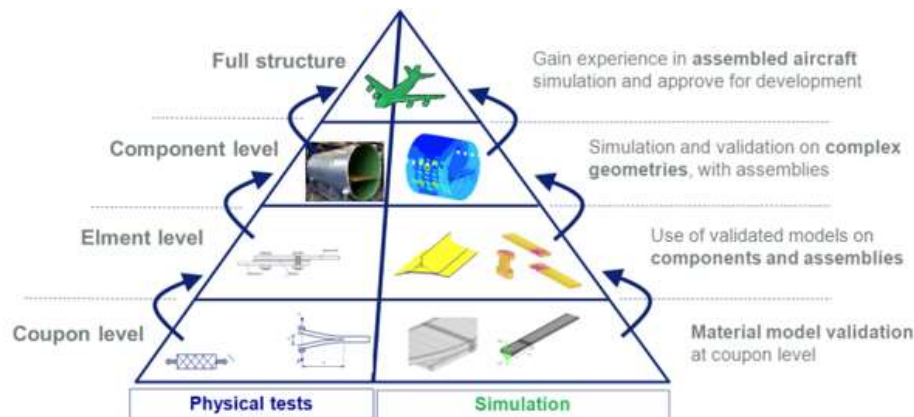


Figure 7: The verification triangle showing the increasingly complex simulation and testing stages that run simultaneously (Bruyneel, et al., 2014).

For the coupon testing smaller tests measuring basic properties like flexural tests are used. For these tests, the sample is placed between two fixed points and a load is applied using one or more moving points. Loads are normally applied using rounded or pointed tools which allowed loads to be simulated along with the end conditions. This is expressed in material standards like ATM D790 (Thackeray, 2017). The purpose of this standard is to test the flexural strength and subsequently the flexural modulus of the material (Park and Seo, 2011). To simulate the physical results a digital copy of the standardised sample is created, using the optimised lay-up pattern. Applying a known force to the model, the stress, strain and deflection can be obtained. These results are compared directly to the results of the physical test, establishing whether the analysis was correctly modelled and to determine the model accuracy (Bachhav, et al., 2022).

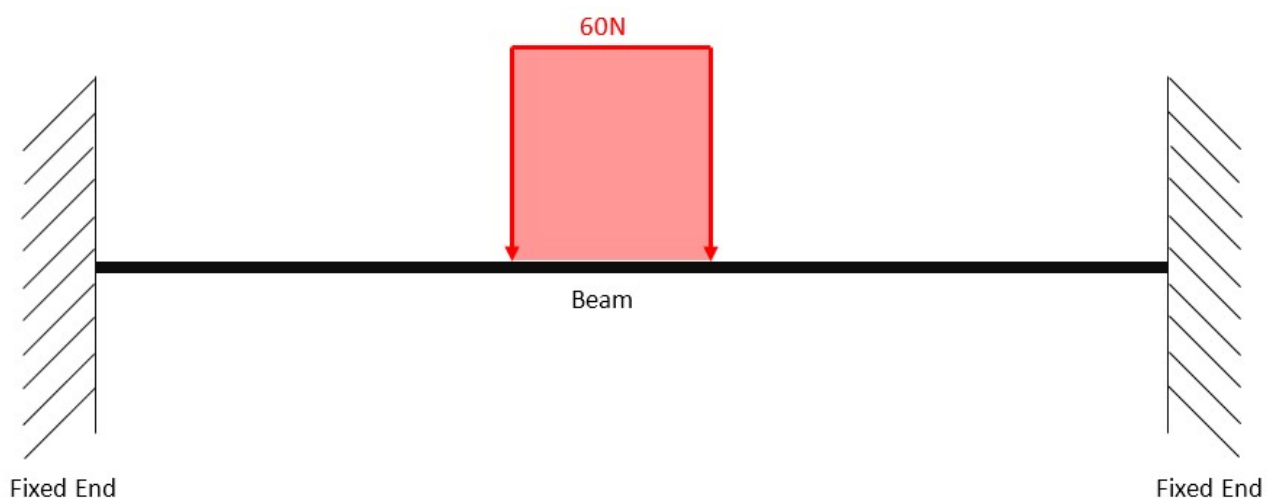


Figure 8: The rear wing element system expressed as a simply supported beam.

The physical sample and digitally modelled body are prepared into a standard form, typically rectangular in shape. The size of which will vary according to the standard of the test. The sample is flat and manufactured within the limits of the specification (Thackeray, 2017). This will inevitably have an impact on the displacement and cannot be compared directly to the displacement of the element due its simplicity (Abe, et al., 2018).

An additional coupon test for physical samples is tensile testing. This would require another specially prepared sample in a profile compatible with the machine to be used (Kumar, et al., 2021). The machine would apply tensile load and measure factors such as tensile strength and stiffness. The sample would be manufactured to orient the Unidirectional fibres to simulate the structure of the element skin, determining the strength of the sample in both the X And Y orientations (Mohamed and Abdelbary, 2022). A digital copy of this test would coincide with this to verify the FEA analysis.

The method established by the Fédération Internationale de l'Automobile (2022) for testing the skin displacement involved the use of a vacuum cup that adheres to the skin. The outer diameter of the vacuum cup measures 50mm and it is through the cup that the 60N of force is applied. The deflection is measured from the outer diameter, taking the original position as the datum. This deflection would be measured at the centre of the element, where it is at a maximum. This procedure does not specify conditions like temperature or moisture content (Bascom and Drzal, 1987). Given the simplicity of this model, this could be considered an element and would follow the coupon testing (Bruyneel, et al., 2014). A physical sample would be produced to the same scale and structure to verify the analysis completed previously.

For full structure testing, the analysis would also take place on a fully assembled rear element. This would include multiple components like spars and ribs which would be analysed individually (Castro and Zeeshan, 2020). This would provide a closer representation of the assembled wing and the simulation would be more complex accordingly (UNNÞÓRSSON, et al., 2014). The same FIA skin deflection test would be used to compare to the simulated model.

The results of these tests would be compared directly with that of the analysis to verify the accuracy of the FEA analysis. For any physical test it would be important to prepare the samples, ensuring there are no imperfections. Porosity, impurities, delamination and damaged fibres can weaken the sample or produce stress intensifiers (UNNÞÓRSSON, et al., 2014). A number of samples would identify internal defects and to potentially test the affect of manufacturing tolerances on the strength of the element skin (McCaslin, 2021).

Conclusion

It was found during the initial stages of this analysis that there was an unnecessary amount material within the initial lay-up. Therefore, the mass had to be minimised. Within the optimisation process, the mass was reduced to its lowest, along with the stiffness and stiffness-to-weight ratio. Subsequently, the displacement was at its highest of 2mm. Discretisation of the layer thicknesses was carried out to produce a manufacturable element skin and provide a tolerance in the calculated displacement to ensure the skin is compliant despite manufacturing tolerances. To physically verify the results of the FEA analysis it was proposed performed FEA and physical tests in increasing complexity. The likeness of these results would indicate the original optimisation was accurate.

References

- Fédération Internationale de l'Automobile (2022) *2023 Formula 1 Technical Regulations*, Available at: https://www.fia.com/sites/default/files/fia_2023_formula_1_technical_regulations_-_issue_4_-_2022-12-07.pdf (Accessed: 1 August 2023).
- Abe, Y., Fujisawa, Y., Murai, Y. and Mori, K.-i. (2018) 'Thickening Process of Concave Edge for Increasing Stiffness and Fatigue Strength of Ultra-High Strength Steel Sheets.' *Toyohashi, Procedia Manufacturing*, 15(2018), pp. 605-611
- Almeida, F. S. and Awruch, A. M. (2009) 'Design optimization of composite laminated structures using genetic algorithms and finite element analysis.' *Composite Structures*, 88(2009), pp. 443-454.
- Altair (2021) *Optimization of Composite Structures*.
Available at: https://2021.help.altair.com/2021/hwsolvers/os/topics/solvers/os/opt_composite_structures_intro_c.html (Accessed 02 September 2023).
- Autosport (2020) *How does a Formula 1 car work? Wings, diffusers and more explained*.
Available at: <https://www.autosport.com/f1/news/how-does-a-formula-1-car-work-wings-diffusers-and-more-explained-4982275/4982275> (Accessed 18 June 2023).
- Baccouch, M. (2021) *A Brief Summary of the Finite Element Method for Differential Equations*. London: intechOpen.
- Bachhav, C. Y., Sonawwanay, P. D., Naik, M. and Thakur, D. G. (2022) *Experimental and FEA analysis of flexural properties of 3D printed parts*. Singapore: Elsevier.
- Bascom, W. D. and Drzal, L. T. (1987) *The Surface Properties of Carbon Fibers and Their Adhesion to Organic Polymers* Washington: NASA.
- Brighenti, R. (2005) Fibre distribution optimisation in fibre-reinforced composites by a genetic algorithm. *Composite Structures*, 71(1), pp. 1-15.
- Broekaart, D. (2015) *What does it take to optimize your designs with the use of FEA?*
Available at: <https://www.linkedin.com/pulse/what-does-take-optimize-your-designs-use-fea-dolf-broekaart> (Accessed 15 July 2023).
- Bruyneel, M. et al. (2014) *Damage Modelling of Laminated Composites: Validation of the Intra_Laminar Law of SAMCEF at the Coupon Level for UD Plies*. Seville: European Society for Composite Materials.

Castro, X. and Zeeshan, R. A. (2020) 'Aerodynamic and Structural Design of a 2022 Formula One Front Wing Assembly', *Fluids*, 5(4), pp. 237-260.

Chang, K.-H. (2015) *e-Design Computer-Aided Engineering Design*. Cambridge: Academic Press.

Duan, X. and Sheppard, T. (2002) 'Shape Optimisation Using FEA Software: A V-shaped Anvil as an Example', *Journal of Materials Processing Technology*, 120(2002), pp. 426-431.

Kumar, R., Mikkelsen, L. P., Lilholt, H. and Madsen, . B. (2021) *Experimental Method for Tensile Testing of Unidirectional Carbon Fibre Composites Using Improved Specimen Type and Data Analysis*, Basel: MDPI.

Lay-Yong, L. and Ang, T.S. (1986) 'Circle measurements in ancient China' *Historia Mathematica*, 13(4), pp. 325-340.

Macquart, T. et al. (2018) 'Optimisation of Composite Structures – Enforcing the Feasibility of Lamination Parameter Constraints with Computationally Efficient Maps' *Composite Structures*, 192(2018), pp. 605-615.

MathWorks (2023) *What Is a Genetic Algorithm?*

Available at: <https://uk.mathworks.com/discovery/genetic-algorithm.html>

(Accessed 06 October 2023).

McCaslin, S. (2021) *Validation and Verification in Finite Element Analysis (FEA)*

Available at: <https://control.com/technical-articles/validation-and-verification-in-finite-element-analysis-fea>

(Accessed 08 October 2023).

Mohamed, Y. S. and Abdelbary, A. (2022) 'Theoretical and experimental study on the influence of fiber orientation on the tensile properties of unidirectional carbon fiber/epoxy composite' *Alexandria Engineering Journal*, 67(2023), pp. 693-705.

Park, S.-J. and Seo, M.-K. (2011) *Interface Science and Technology*. Amsterdam: Academic Press.

Performance Composites (2023) *Carbon Fiber Composite Design Guide*

Available at: <https://www.performancecomposites.com/about-composites-technical-info/124-designing-with-carbon-fiber.html>

(Accessed 01 September 2023).

Rimini, S., Vincent, J. and Wake, N. (2022) 'Computer-Aided Design Principles for Anatomic Modeling' *3D Printing for the Radiologist*. Amsterdam: Elsevier, pp. 45-59.

Sifakis, E. and Barbić, J. (2016) *Finite Element Method Simulation of 3D Deformable Solids*. San Rafael: Morgan & Claypool.

Sigmund, O. and Maute, K. (2013) 'Topology optimization approaches' *Structural and Multidisciplinary Optimization*, 48(2013), pp. 1031-1055.

SimScale (2023) *What Is FEA | Finite Element Analysis?*

Available at: <https://www.simscale.com/docs/simwiki/fea-finite-element-analysis/what-is-fea-finite-element-analysis/>

(Accessed 10 July 2023).

Thackeray, K. (2017) *The Definitive Guide to ASTM D790 Flexure Testing of Plastics*

Available at: <https://www.instron.com/en-gb/testing-solutions/astm-standards/the-definitive-guide-to-astm-d790>

(Accessed 03 September 2023).

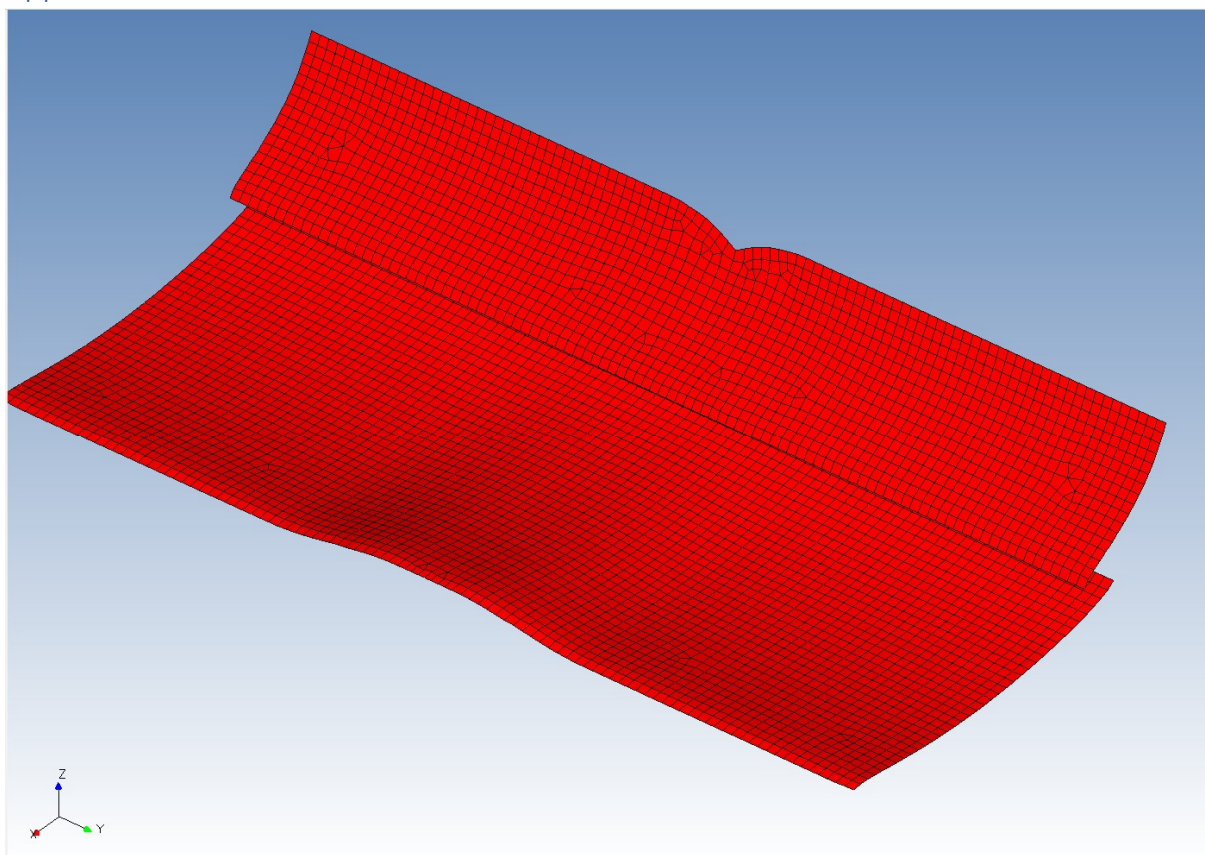
Thelin, C. et al. (2021) 'Spatially defined optimization of FEA using nodal surrogate models' *Structural and Multidisciplinary Optimization*, 64(2021), pp. 813-828.

Unnþorsson, R., Jonsson, M. J. and Runarsson, T. P. (2014) *NDT Methods for Evaluating Carbon Fiber Composites*, Reykjavik: Icelandic Research Council.

Yue, X. (2010) 'Finite Element Analysis on Strains of Viscoelastic Human Skull and Duramater' *Finite Element Analysis*. London: IntechOpen, pp. 1-44.

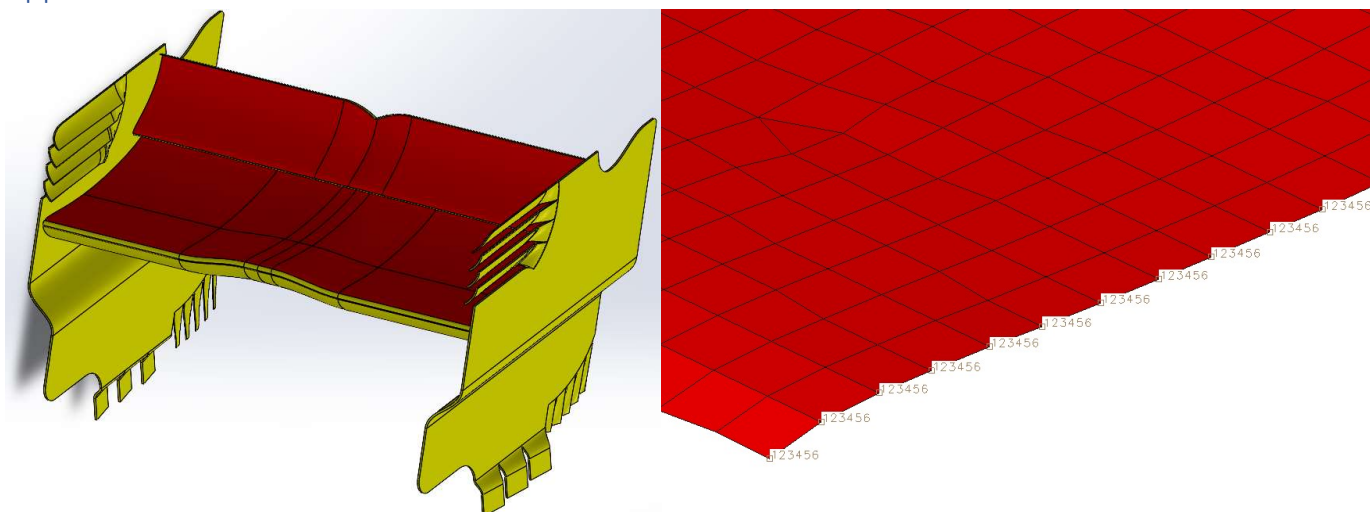
Appendices

Appendix A – Base Model



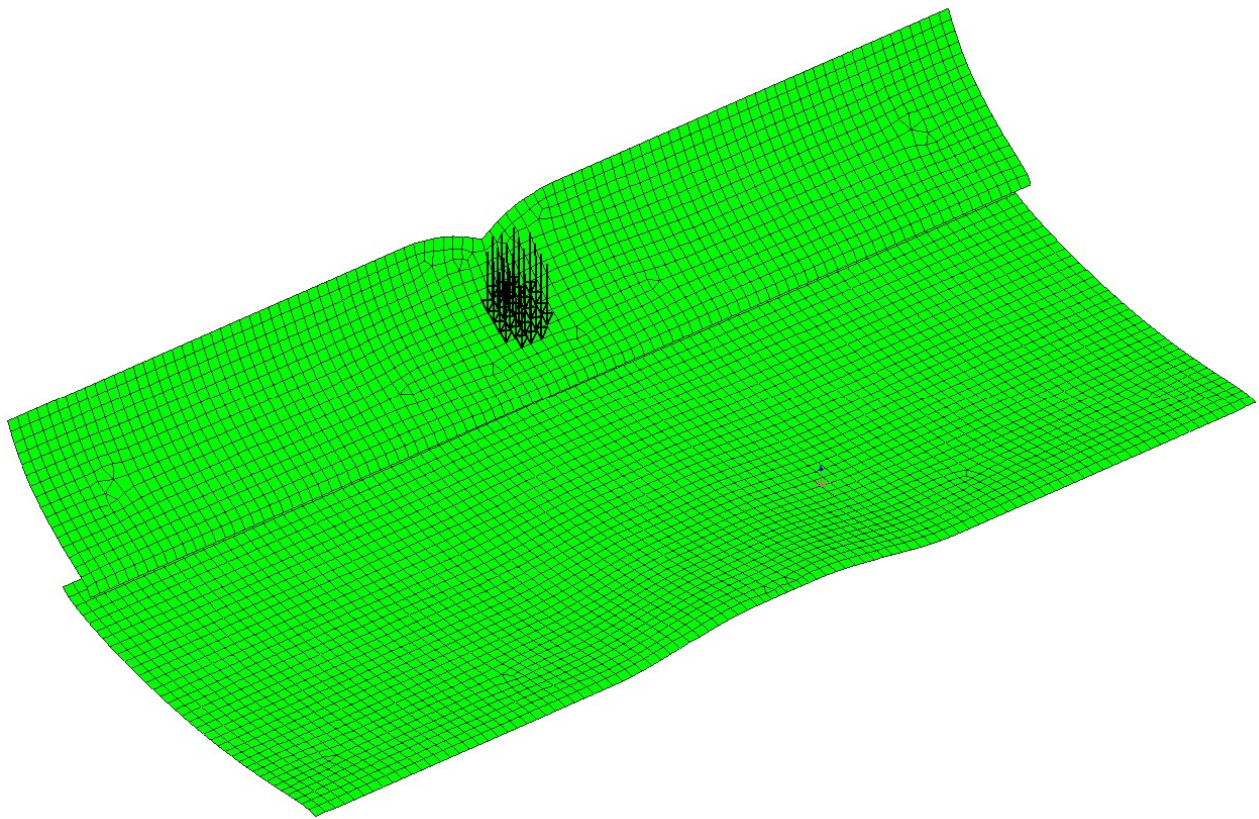
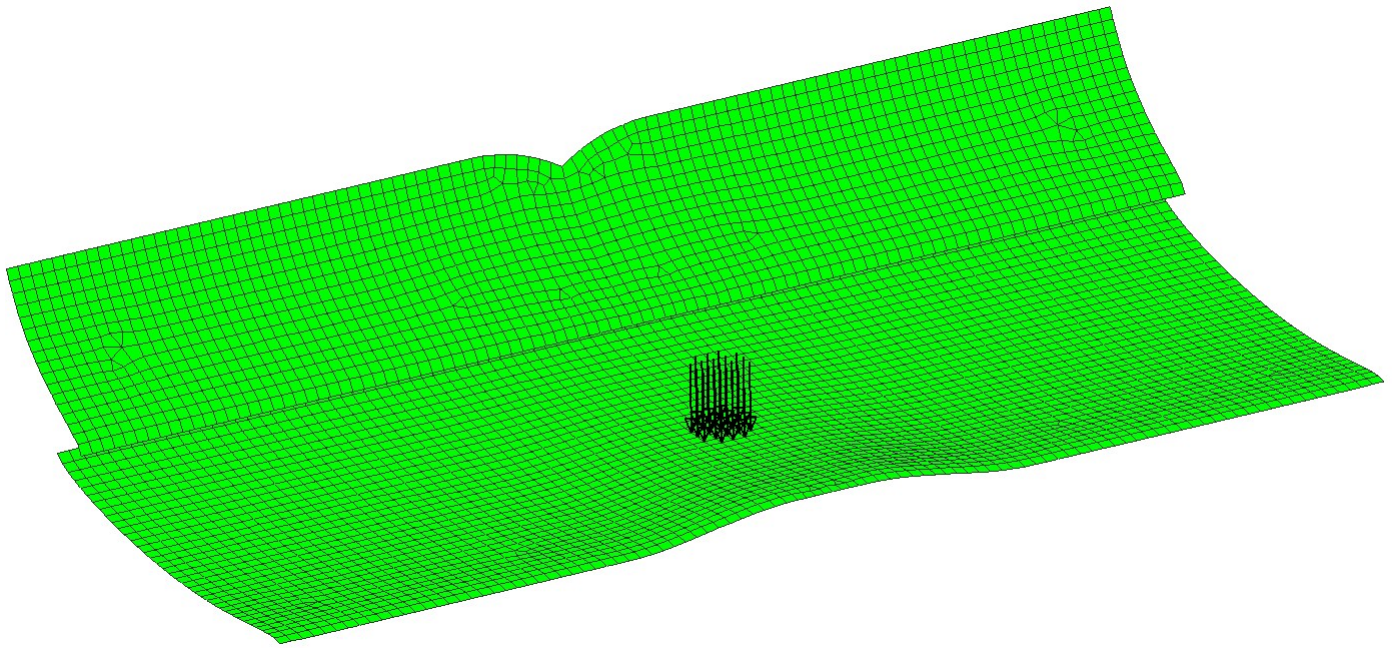
Appendix A: The base main and rear element skins. The are surface models, and do not have a thickness attributed them, not does it have any constraints or loads.

Appendix B – Constraint Placement



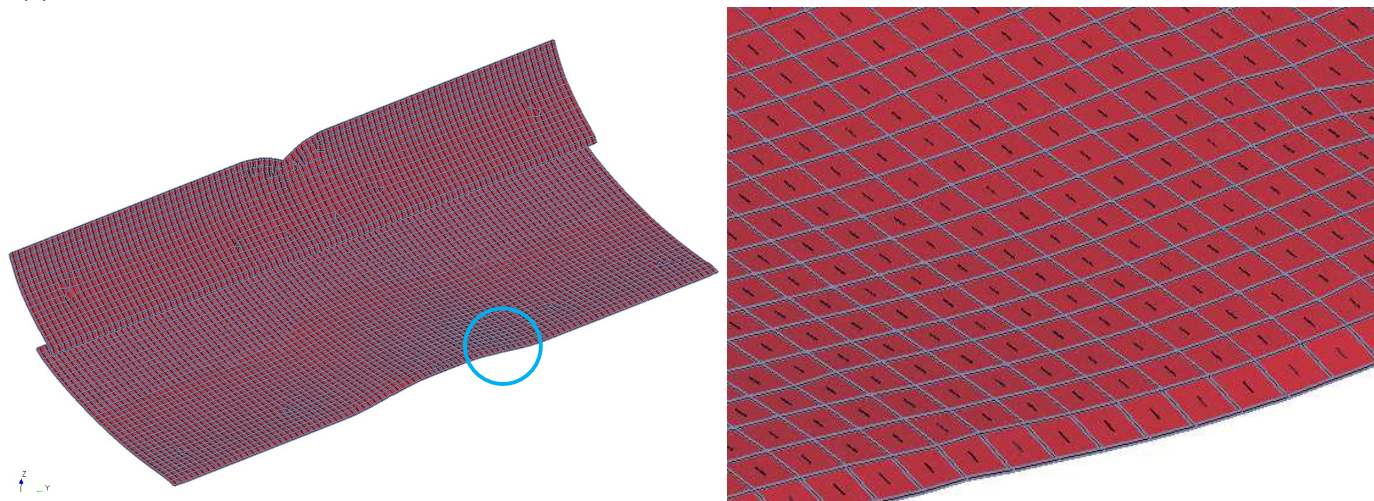
Appendix B: Simulation of the end plates through fixed constraints. Though it is not accurate to what would be experienced during scrutineering, this constraint allows the stiffness of the material to be assessed without rotation or translation of each end.

Appendix C – Force Placement



Appendix C: Uniformly Distributed Load (UDL) applied over the surfaces of the main and rear element. The magnitude was set to 60N according to FIA testing procedure.

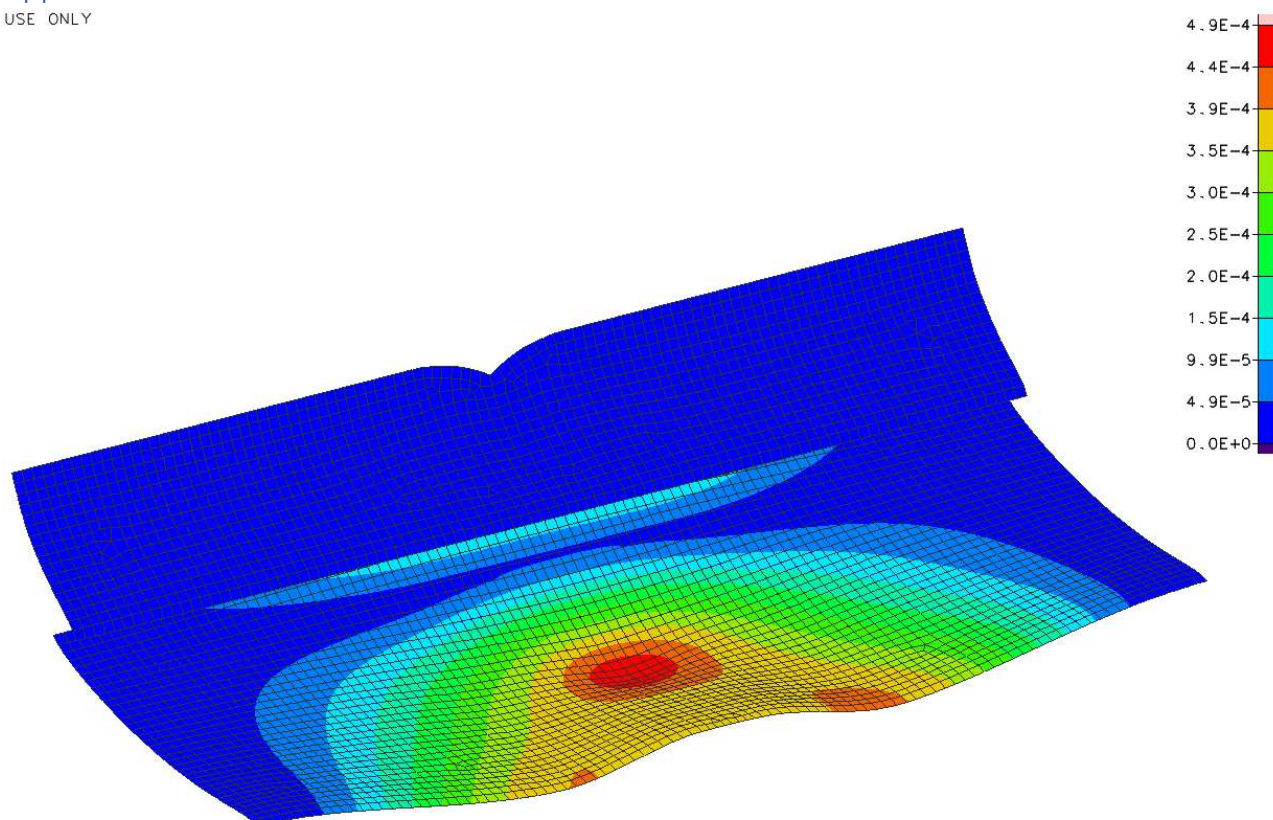
Appendix D – Vector Orientation



Appendix D: Orientation of the fibre datum. This simulates the 0° fibre orientation that run from fore-to-aft of the car.

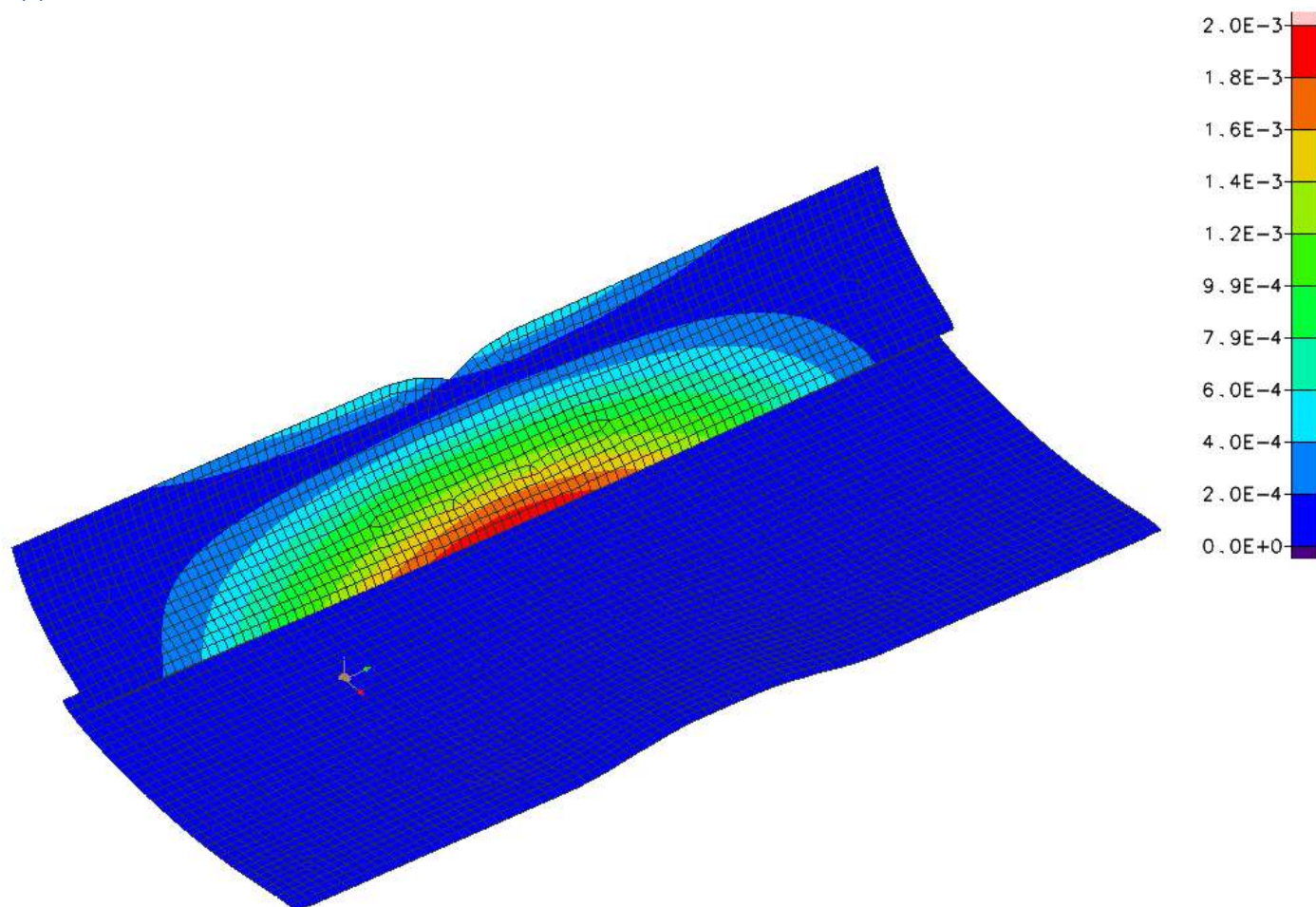
Appendix E – Initial Deformation of Main Element Skin

USE ONLY



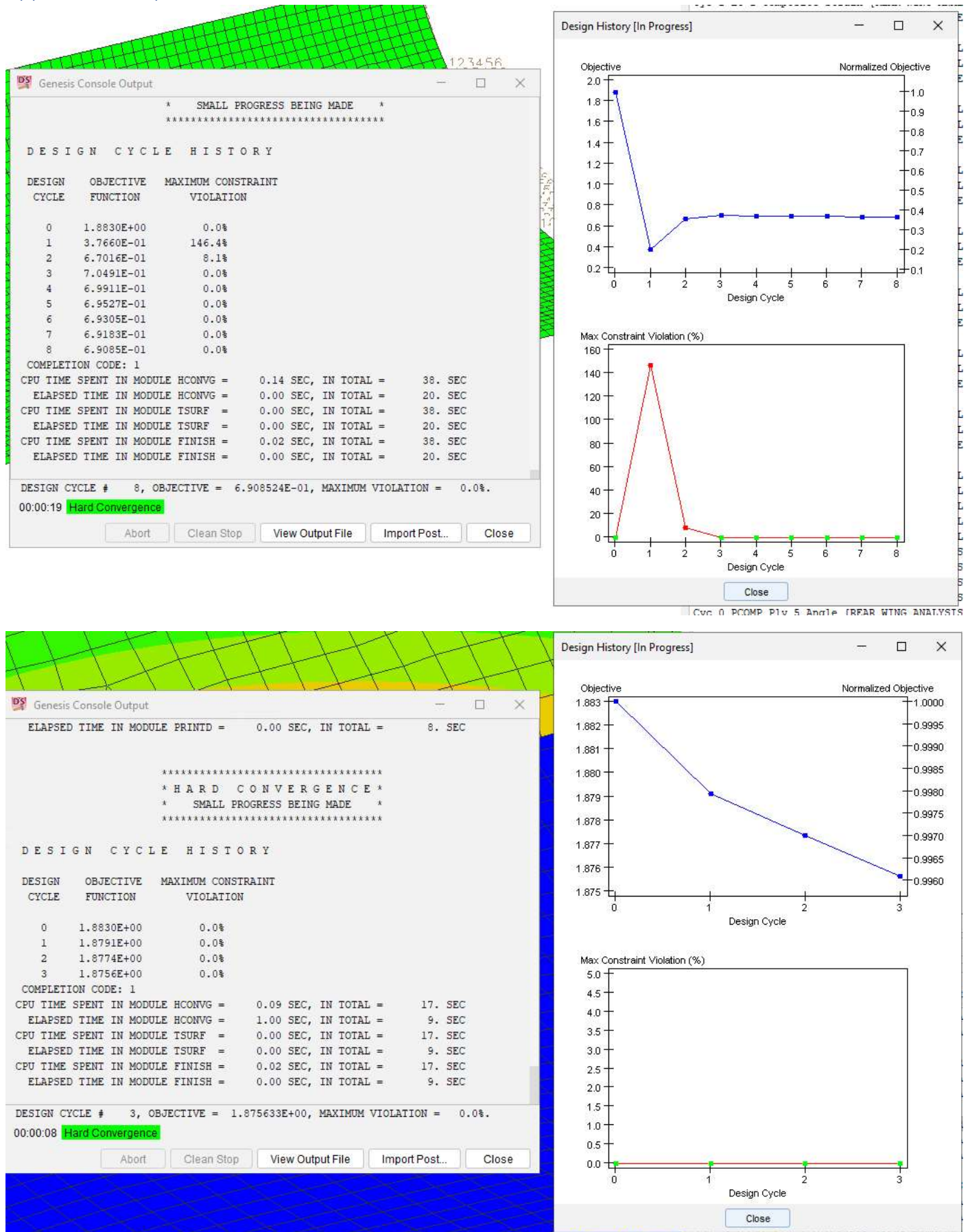
Appendix E: The initial deformation of the main element skin. The displacement of the element is shown in the scale (right) expressed in metres. This used the initial layup, given a total thickness of 2.5mm and has the smallest displacement as a result.

Appendix F – Initial Deformation of Rear Element Skin



Appendix F: The initial deformation of the rear element. The displacement of the element is shown in the scale (right), expressed in metres. This used the initial layup and had the lowest displacement of all the rear element iterations. Despite this, the lower surface area means it has a larger displacement than the main element.

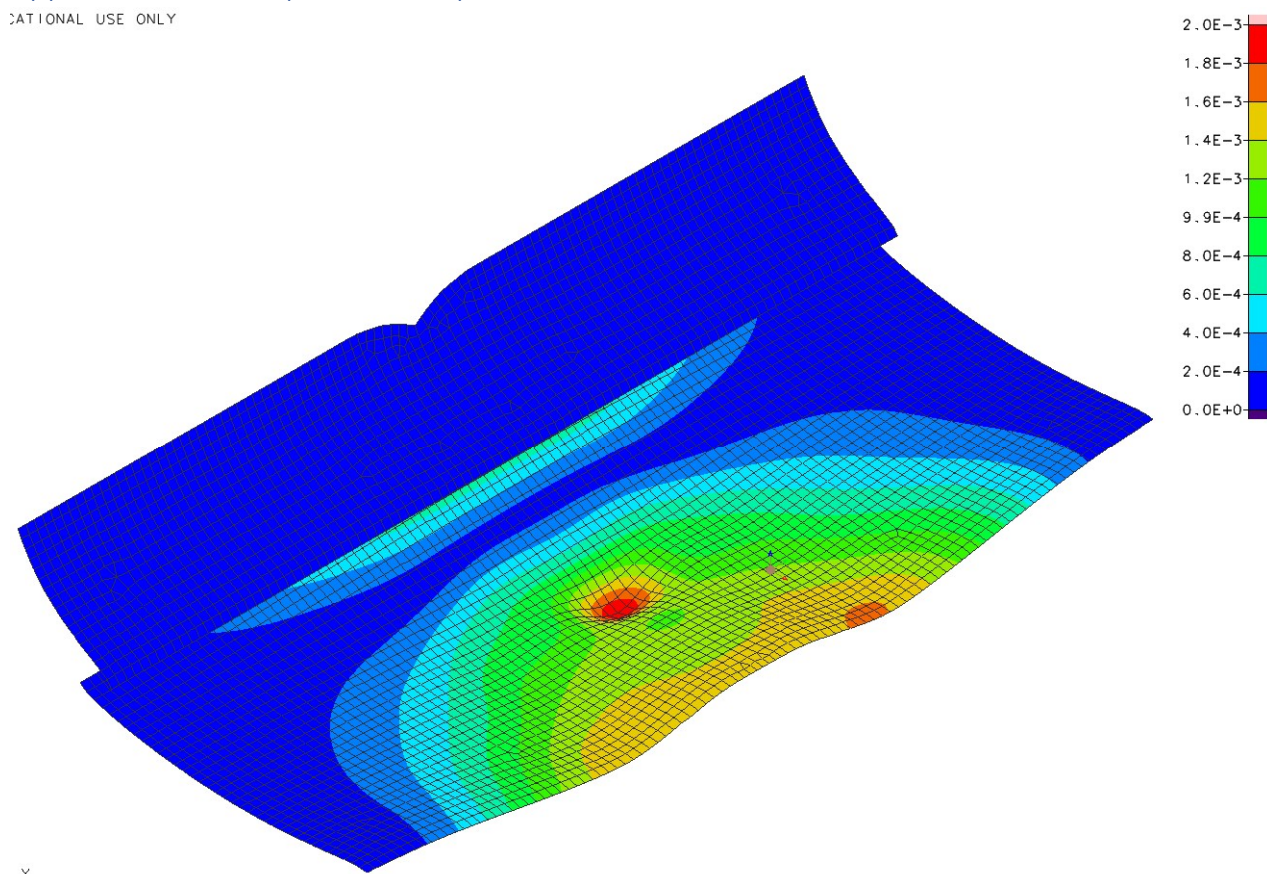
Appendix G – Optimisation Process and Results



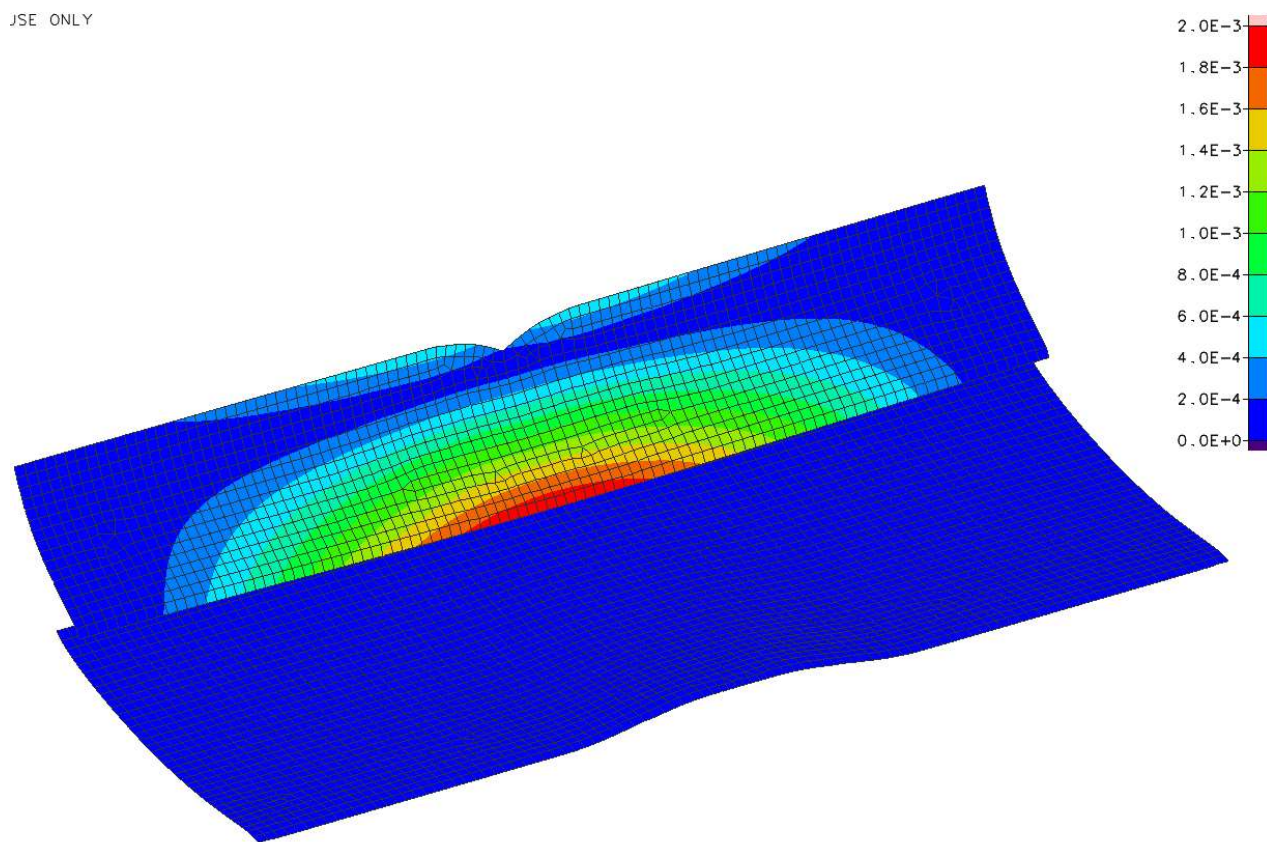
Appendix G: The optimisation process of the main element (top) and the rear element (bottom), showing the convergence within a number of design cycles. Both processed were solved with hard convergence. A solution was found within eight cycles for the main element whilst on the rear element, a compliant solution was found in three cycles. The mass obtained at each cycle is shown under objective function column.

Appendix H – The Optimised Displacement of the Main and Rear Element

ATIONAL USE ONLY



JSE ONLY



Appendix H: The optimised displacement of the main element and rear element. The displacement of the element is shown in the scales (right), expressed in metres. The displacement occurred mostly at the centre of the element, at the edges where there is the least amount of support offered by the constraints at each end.