

University of Trinity Saint David School of
Engineering



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

Designing a Kiteboard Hydrofoil Wing: Structural Design
Structures & Fluid Analysis ACMA6010

Sean Mee

2218461

24/10/25

Table of Contents

1	Introduction.....	1
2	Discussion - FEA Optimisation Technique Literature.....	2
2.1	FEA History & Limitations	2
2.2	FEA Current & Future Applications	3
3	FEA Optimisation	4
3.1	Finite Element Analysis of Blade Design.....	4
3.2	Fibre Orientation Optimisation.....	6
3.3	Laboratory Design Verification.....	9
3.3.1	Loading Conditions	10
3.3.2	Deflection Measurement Methodology.....	10
3.3.3	Strain Measurement Methodology (Secondary Data)	10
3.3.4	Validity.....	10
4	Discussion – Optimised Design	11
5	References.....	13
	Table 1 Thickness and Number of Layers in the Final Optimised Design	9
	Figure 1 Tacoma Bridge Collapse due to Simple Harmonic Motion (SHO) (Study.com, 2022)	3
	Figure 2 Image of Point Loading in FE Model.....	4
	Figure 3 FE Deflection of Steel Blade	5
	Figure 4 Von Mises Stress over Wing Structure for Initial Fibre Layup.....	5
	Figure 5 Layer Thickness During First Optimisation	6
	Figure 6 Component Total Mass During First Optimisation	7
	Figure 7 Layer Thickness During Second Optimisation.....	8
	Figure 8 Component Mass During Second Optimisation	8
	Figure 9 Von Mises Stress of Wing after the Second Optimisation	9
	Figure 10 Hydrofoil in use for a 'Jump' (24-7 Boardsports, 2010)	11

Abstract

Finite Element Analysis (FEA) and optimisation tools were used in this report to evaluate improvements to a hydrofoil wing design by reducing its mass while maintaining performance, as measured by material deformation of 8.6 mm across the component span under a 500 N load. Additionally, a factor of safety was assessed to ensure an acceptable tolerance of 40% before material failure. The original steel component had a mass of 0.465 kg and exhibited a maximum deflection of 8.6 mm under the simulated load.

Two sequential FE design optimisations were performed using iterative design software to vary five specific fibre layer orientations. The final optimised design achieved a component mass of 0.172 kg. This design exhibited a maximum Von Mises stress of 120 MPa and satisfied the required factor of safety. A laboratory verification methodology is also proposed to validate the accuracy of the FEA model. This process involves applying the 500 N load condition to a prototype and measuring deflection using a high-precision displacement sensor, such as an LVDT.

While the FEA successfully identified a design that met the stated requirements, the constraints applied are insufficient to confirm that the component would be fully fit for purpose. Additional considerations include the composite's resistance to cavitation and the limitations of modelling these forces as a point load. Addressing these factors would require more detailed FEA and an understanding of the typical fluid flow around the component.

1 Introduction

This report will utilise Finite Element Analysis (FEA) to assess and optimise the structural characteristics of a kiteboard hydrofoil wing. This design exercise aims to identify the optimal fibre layup design for the composite component, replacing a mild steel material with a fibre-reinforced plastic to achieve optimisation of the hydrofoil wing's mass while ensuring that the structural deflection matches that of the original steel baseline. The project will complete an analysis based on the reaction to a 500N point load to represent the stress experienced by the wing during normal operation applied at the geometric centre.

Secondly, this project will aim to assess how these results might be verified through testing and to discuss and assess the validity of the FE model through relevant technical literature.

2 Discussion - FEA Optimisation Technique Literature

2.1 FEA History & Limitations

Finite Element Analysis (FEA) has numerous applications, including biomechanical modelling, automotive engineering, and material science. It was initially developed for structural analysis in the 1950s and is now used in a diverse number of engineering fields.

The development of FEA first began with the work of Turner et al, (1956), when the term 'finite element method' (FEM) was introduced, defining a means to compute systems of complex stress by numerical means. The numerical FE method was further refined and expanded by various researchers, notably Zienkiewicz (1971), who extensively documented its applications and theoretical advancements. FEM was primarily used in structural analysis for aerospace and civil engineering fields, where complex structural systems required detailed, if time-consuming, evaluation.

The introduction of digital computation allowed FEM to be applied to an increasingly wide number of fields, creating Finite Element Analysis FEA. This also enabled more complex simulations that addressed dynamic behaviours, which are common in many modern engineering applications (Tai, 1999). From the beginning of the 1980s, FEM as a numerical analysis tool was refined through the development of software capable of increasingly complex analysis previously considered impractical (Liu, 2009). These CAD/CAE applications allow the saving of both time and physical resources otherwise used for potentially costly physical prototypes.

Yet, there are basic flaws with how FEA models work and the assumptions they make. In applications such as 'bulge-formed' steel parts, the non-linear deformation zones create accuracy problems for FEA modelling due to element distortion (Zhai et al, 2021). This is indicative broadly of the failures with most FEA systems to adequately model material defects, inclusions or other features which cannot be modelled. Additionally, FEA methods rely on accurate load conditions and geometry modelling. Unforeseen loading, vibrations or plastic deformation can alter the computed failure risks, and for load-bearing, this can make structures more susceptible to operational stresses (Liu et al., 2023). A common example is the induction of simple harmonic motion (SHO) into a structure, which a FEA model alone wouldn't be capable of predicting. The oscillation of a structure, shown in Figure 1 is an example of the Tacoma bridge, which can create compounding stresses which otherwise would be impossible for a part or structure to experience in the course of standard loading.



Figure 1 Tacoma Bridge Collapse due to Simple Harmonic Motion (SHO) (Study.com, 2022)

2.2 FEA Current & Future Applications

As FEA techniques have developed, the incorporation of topological optimisation has become more commonplace. Topological optimisation can use information from FEA models to explore optimal use of material to achieve key metrics (Zhu et al., 2019). Current models utilising this technique often result in highly organic structures which cannot be easily achieved by traditional methods, and so currently designs often compromise between manufacturability and design goals. With the progression of additive manufacturing into a more mainstream usage, along with the wide integration of CAM, we may see FEA design systems which can take some of these limitations into account based on manufacturing geometric limitations. Aignătoaie (2013) emphasises the importance of integrating CAD with FEA simulations in order to simplify a design process.

As 3D modelling is now industry standard, tools that enable three-dimensional FEA analysis directly from CAD permit evaluation of performance characteristics while a part is being designed (Choi, 2015); which is in contrast with a typical design cycle of engineering for load cases after an initial design has been moved forward. Iso-geometric analysis (IGA), which allows the integration of CAD and FEA processes, represents the future evolution of this principle. IGA addresses previous challenges associated with mesh generation by employing the same functions used in CAD modelling to simplify the transition from design to analysis by allowing the use of exact geometries rather than approximated discrete meshes (Agrawal & Gautam, 2018).

3 FEA Optimisation

3.1 Finite Element Analysis of Blade Design

Finite Element Analysis has been used in this report to assess the wing's reaction to a load of 500N at the geometric centre of the component. This load represents the maximum resultant force applied to the blade as a result of fluid flow across the aerofoil. As shown in Figure 2, the force acts as a point load in the centre of the blade.

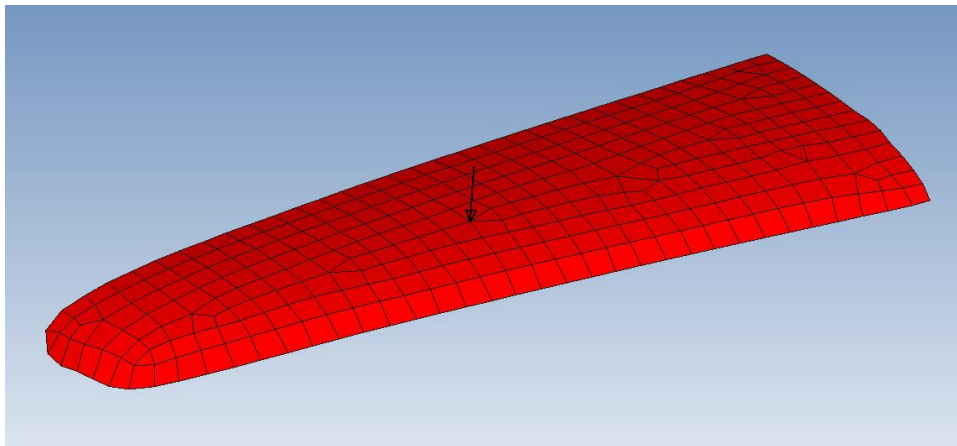


Figure 2 Image of Point Loading in FE Model

The objective of this exercise is to identify the optimal layup design, the deflection to equal that of the steel component, enabling optimisation of component mass. The blade is modelled as a single skin or 2D mesh. This simplification reduces mesh complexity and improves processing time by limiting the number of degrees of freedom. Since the component is made from thin sheet steel, this modelling method can be applied with minimal loss of accuracy.

The Steel component displayed a maximum deflection of 8.6mm, shown in Figure 3. This deflection sets the optimisation target to compare the deflection of a nominal layup of 25 epoxy resin-carbon fibre layup of 0.1mm layers at -90, -45, 0, 45 & 90 degrees, where 0 degrees is pointing down the length of the structure, which had 33mm of deflection.

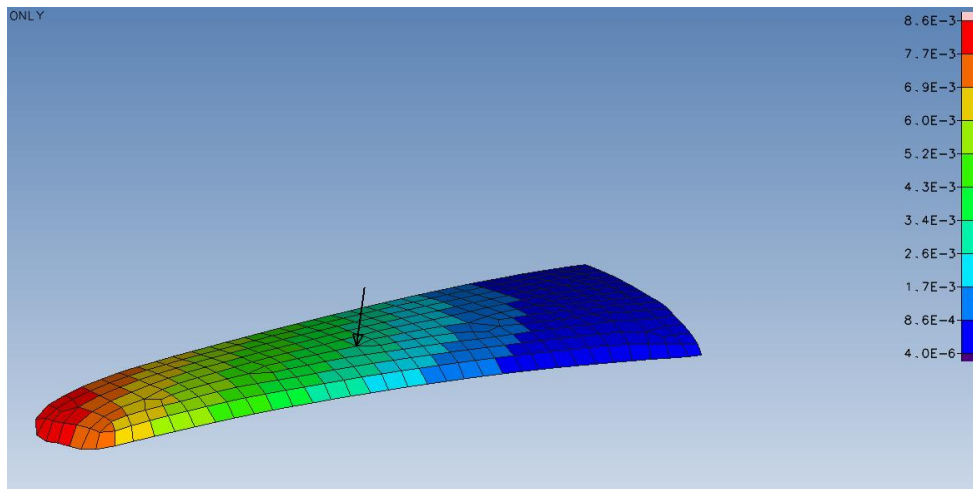


Figure 3 FE Deflection of Steel Blade

The initial layup, which had its fibres pointed in equally proportioned directions, had a poor result when compared to the steel baseline. However, the mass of this layup is 0.116kg, whereas the original steel component is 0.465kg. Despite the composite having a higher volume ($6.04 \times 10^{-5} \text{m}^3$ vs $7.55 \times 10^{-5} \text{m}^3$), we can see that the fibre composite can offer significant weight savings already in this initial stage.

The share of stress across the CFRP material is shown in Figure 4. The yield strength of epoxy resin, a common polymer used in CFRP materials, is conservatively 200 MPa (Li et al, 2021) but is often reported as higher than this. Under the Von Mises stress model, the component experienced 190MPa, and the Major Principal Stress was 160MPa, so the true value may be in this area. However, this mesh is comparatively low resolution, and calculated stresses tend upwards with higher fidelity modelling, as implied by Caliber Design (2020). As such, the true value could be a lot higher.

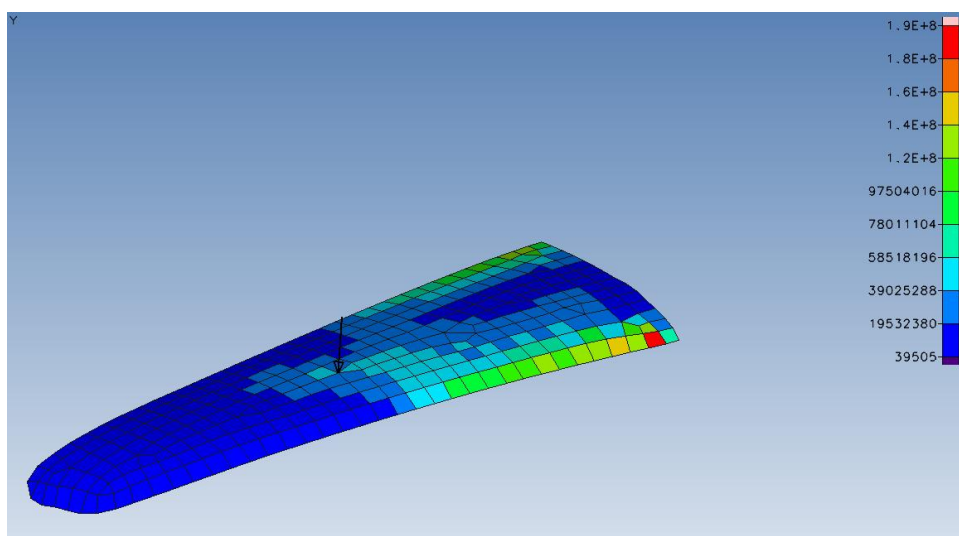


Figure 4 Von Mises Stress over Wing Structure for Initial Fibre Layup

Additionally, both Von Mises stress and the locations of Major Principal Stress looked similar to Figure 4, in which the maximum stress is concentrated in a single cell. We therefore do not know with much certainty the behaviour of this stress concentration. Due to these limitations, this particular layup pattern is in danger of exceeding the material's yield stress and failing, despite theoretically being within the upper stress limit.

3.2 Fibre Orientation Optimisation

Using FE design optimisation, the structure was optimised to minimise mass, while maintaining the same deflection as the original steel structure. Each Layer was limited to a 2mm thickness. The results of this initial optimisation are shown in Figures E and F. Each layer number represents as follows:

- Layer 1 = -90°
- Layer 2 = -45°
- Layer 3 = 0°
- Layer 4 = 45°
- Layer 5 = 90°

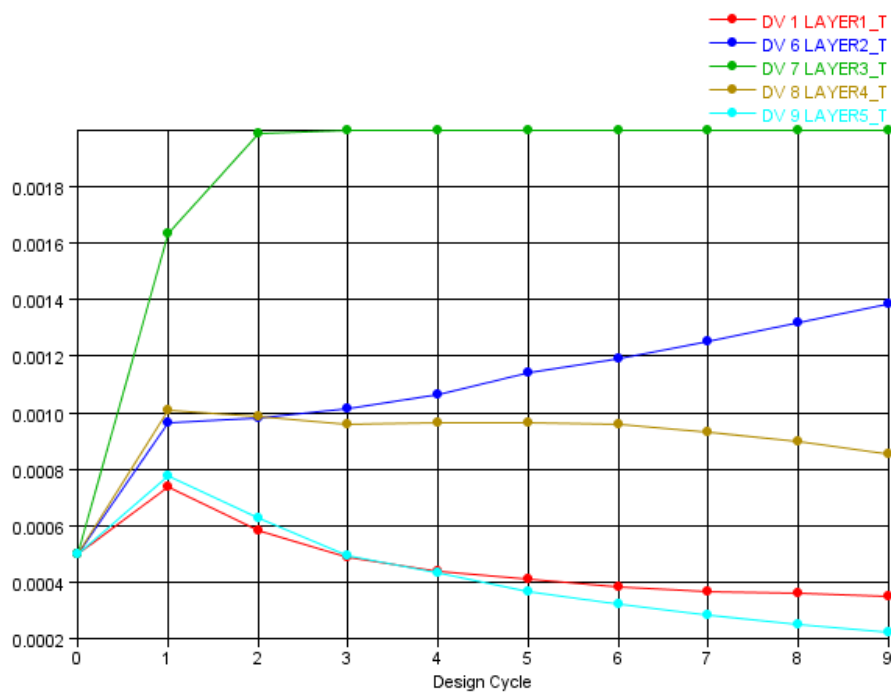


Figure 5 Layer Thickness During First Optimisation

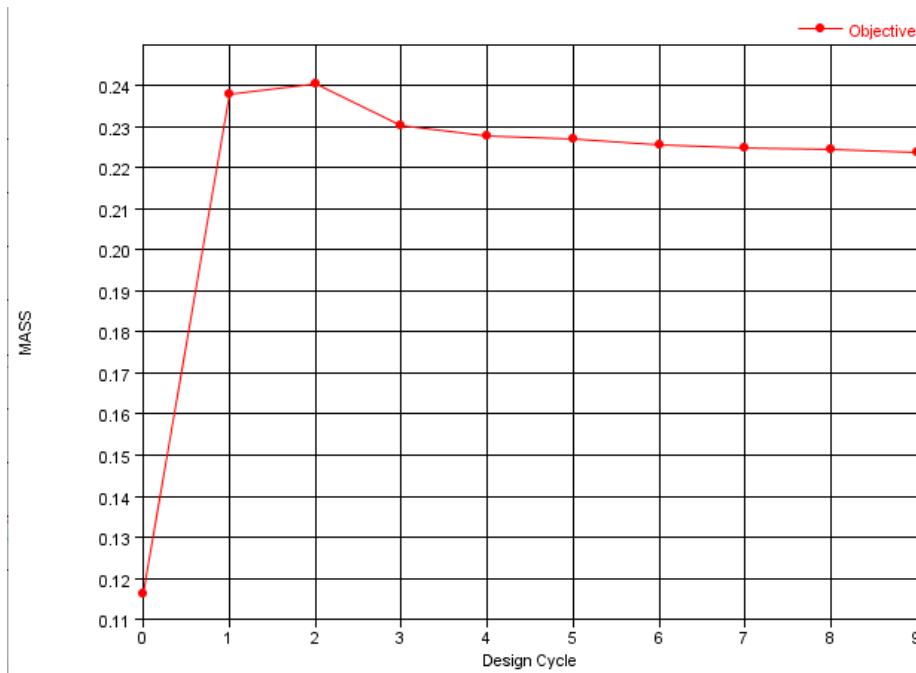


Figure 6 Component Total Mass During First Optimisation

Layer 3, which runs parallel with the length of the blade, quickly reached its maximum value by the 3rd iteration in Figure 5. The other layers also all jump up in value in the initial cycle, which is the programme correcting for deflection error. The remainder of the design cycles, we can see the programme try to optimise the structure with Layer 3 at a maximum thickness, resulting in a 0.234kg structure. This is a 100% improvement over the initial steel structure, but potentially not the optimal configuration due to the layer limitations.

A second optimisation was run with a maximum layer thickness of 3mm. In Figure 7, Layer 3 displays a natural plateau near 3mm, which is suggestive of an ideal convergence. Unexpectedly, layup directions in -90 & 90 (which are functionally the same direction, perpendicular to the blade length) are not absent, despite the blade deflection being perpendicular to their strongest an-isotropic direction.

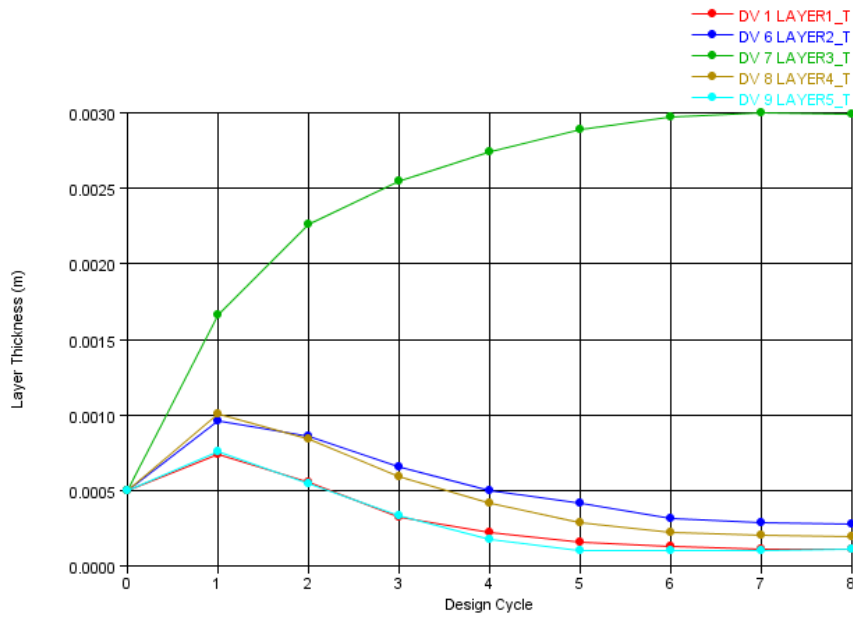


Figure 7 Layer Thickness During Second Optimisation

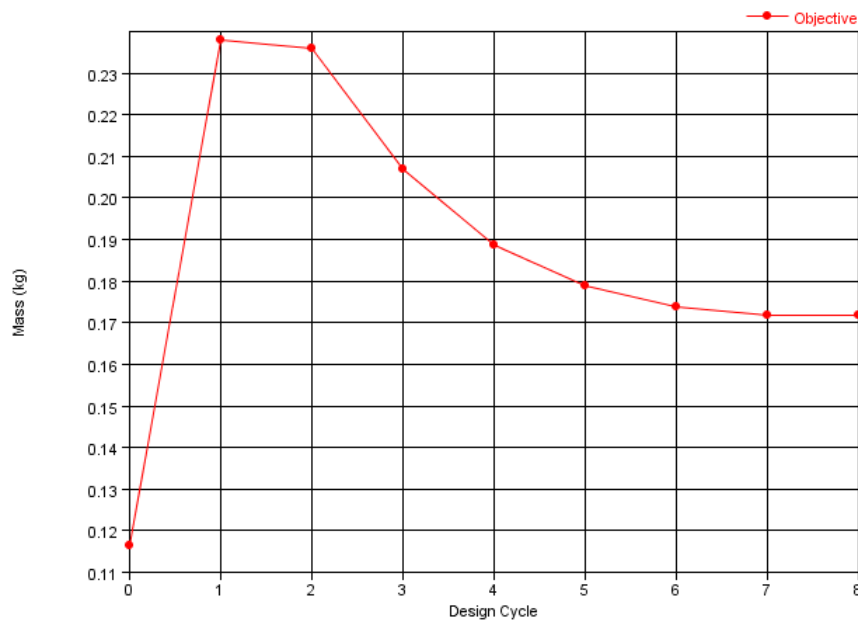


Figure 8 Component Mass During Second Optimisation

Figure 8 shows a similar initial peak to Figure 6, correcting for deflection error initially, but has a much deeper and gradual trough reflecting iterative design improvements performed within a more realistic constraint. A final mass of 0.172kg provides a 36% improvement over the initial optimisation. The maximum Von Mises stress in the second optimised blade is 120MPa. The stress is distributed more evenly across the part in Figure 9 than in Figure 4; the primary stress concentration remains in the same location, as expected. It is still evident that the mesh could be further refined to enhance the resolution of these stress concentrations. Nevertheless, the optimised component

demonstrates a theoretical factor of safety of 40%, based on Li et al.'s (2021) low-end yield strength of 200 MPa. A 40% factor of safety is considered an acceptable tolerance to account for potential inaccuracies in this finite element (FE) model, especially when compared to the original layup, which exhibited a factor of safety of only 5%.

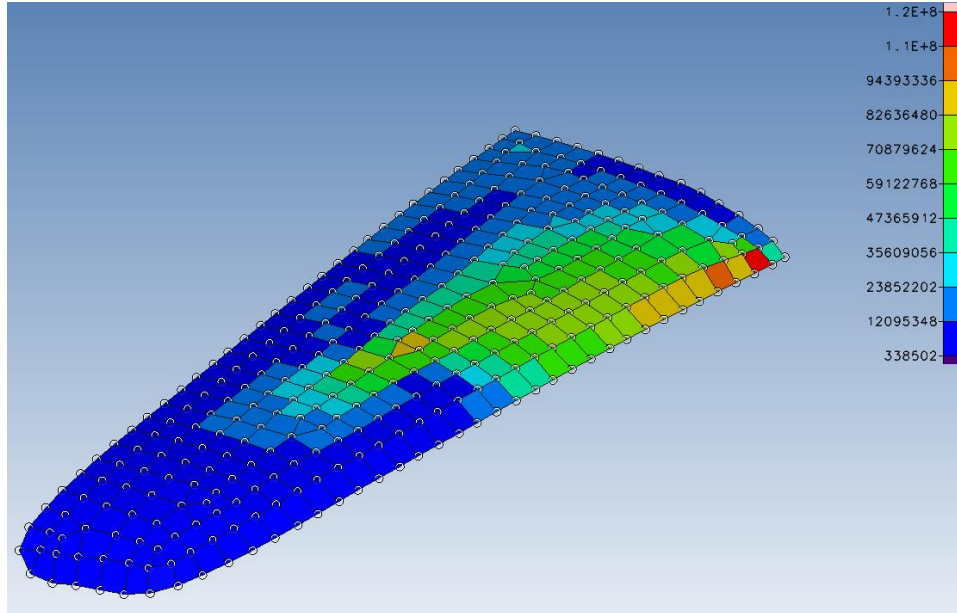


Figure 9 Von Mises Stress of Wing after the Second Optimisation

3.3 Laboratory Design Verification

A laboratory test should be conducted based on this final design, with the final layup configuration shown in Table 1. A laboratory experiment should aim to measure the deflection of the component, as direct material properties or stresses are difficult to measure, as discussed later. The displacement is the primary measurement and proxy for how close the FE model is to reality. Stress can be used to corroborate any conclusion.

	DIRECTION (DEGREES OFF NORMAL)	THICKNESS (mm)	No. LAYERS
LAYER 1	-90	0.2	2
LAYER 2	-45	0.4	4
LAYER 3	0	3	30
LAYER 4	45	0.3	3
LAYER 5	90	0.2	2

Table 1 Thickness and Number of Layers in the Final Optimised Design

3.3.1 Loading Conditions

The laboratory test must apply the load condition used in the simulation: a point load of 500N at the centre of the prototype, using a 51kg mass suspended from the prototype's geometric centre.

Additionally, the mounting constraints must replicate the fixed boundaries assumed in the FEA and be suspended from the edge of the prototype to achieve the same span. As such, the prototype should have an additional flange to the near 'fixed' side, by which it can be secured to a dedicated, rigid rig.

3.3.2 Deflection Measurement Methodology

A high-precision displacement sensor, Linear Variable Differential Transformer (LVDT) (Nguyen et al, 2023), should be positioned to measure the displacement at successive positions along the component, to capture a deformation curve for multiple sections along the length of the component. An alternative, faster and low-cost and low-resolution method would be to place a camera level with the component, and place a ruler or grid behind the component, reading the displacement by eye.

3.3.3 Strain Measurement Methodology (Secondary Data)

Strain gauges fixed to the component near where maximum stress concentrations are predicted could offer verification of the stresses predicted in the FE model. Using the relationship of $E = \text{stress}/\text{strain}$, a value could be determined for stresses in that area. The limitation of this is that each layer is anisotropic, and so has a different value for Young's Modulus based on the direction the strain gauge measures in.

3.3.4 Validity

If the experimental measurements closely align with the predicted deflection and stress results, the FEA model is valid for this load case. The percentage error is the primary way to compare the difference seen in deflection and stress from the model. We can expect any measured value to be higher than that found in the FE model for several reasons:

- The Young's modulus of the material used in the model is ideal, and could be compromised by a non-ideal manufacturing, due to air pockets, layer thickness tolerances etc.
- Assumptions made in the model may be faulty or not be reflective of reality.
- The model has inherent inaccuracies and tends towards real values as the mesh size increases (Zienkiewicz, 1971)

4 Discussion – Optimised Design

The re-design of this component replaces a mild steel material with a fibre-reinforced plastic, which is a common material used in hydrofoils for its excellent weight-to-strength ratio. The analysis is based on a point force representing the resultant pressure over the wing. However, there are other sources of stress on a hydrofoil which don't conform to a quasi-static load case. These commonly can include vibrations and cavitations caused by the nature of the fluid flow over the component (Lelong et al, 2017). These forces on the wing create unstable and dynamic load cases, the pressure on the hydrofoil and a given moment, potentially creating wear on the component.

Additionally, the complexity of fluid dynamics around hydrofoils necessitates more than a simplistic point load model. Wu et al (2014) present simulations which show variations in the forces experienced by a hydrofoil at the leading and trailing edges of the structure. Additionally, none of the forces exerted on such a structure actually work as a point force, but rather as a distributed load due to pressure differences on either side of the blade. A study by Gunnarson et al. emphasises that forces acting on hydrofoils exhibit non-linear characteristics. Point load models couldn't capture such characteristics. They also indicate that drag and wake effects influence the forces on a hydrofoil blade (Gunnarson et al, 2019).

While fibre reinforced resins have a preferable performance by reducing the component's weight, changing the material of a component comes with other considerations. Corrosion and wear through cavitation could be an important consideration. Cavitation is caused by flow separation over component geometry at the leading edge of a hydrofoil (Fan et al, 2024) and can damage the component regardless of the stresses it's placed under. Epoxy resins (a common choice for fibre-reinforced plastic matrices) have a much lower hardness value, 60HV (Srinivasaiah et al, 2024). CR4 mild steel, which is a common low-carbon steel, has upwards of 320HV in the Vickers Hardness scale (Manney, 2024). A significant drop in hardness could mean that the new material isn't able to withstand the cavitation caused by the design of the component's geometry.



Figure 10 Hydrofoil in use for a 'Jump' (24-7 Boardsports, 2010)

During kitesurfing, a hydrofoil may leave if a surfer performs a jump pictured in Figure 10. In this case, the hydrofoil is the first part of the board to impact the water, and will experience a set of forces unrelated to those analysed in this report. This simulation looks at unidirectional layups, and the final optimisation has many of these fibres placed in the same direction. An impact against the fibres in a side-on impact, which is likely given the orientation of the blades perpendicular to the direction of travel, may lead to the matrix material failing. (Abdellah et al., 2021).

5 References

- 24-7 Boardsports, 2010. Kitesurfing Foils.
https://24-7boardsports.com/kitesurfing-foils/?srsltid=AfmBOoqIP6lB9ySmKcARNmxd6y6YLxKcC_uZ425yAwpAON0ZOwJ-Yf9R
- Caliber Design, 2020. How to Interpret FEA Results.
<https://www.caliberdesign.co.nz/how-to-interpret-fea-results/>
- Manney, D., 2024. CR3 vs CR4 Steel: Showing Their Performance in Various Environments.
<https://www.schuettemetals.com/blog/performance-differences-between-cr3-vs-cr4>
- Study.com, 2022. Tacoma Narrows Bridge | Collapse & History.
<https://study.com/learn/lesson/tacoma-narrows-bridge-collapse-facts-causes-physics.html>
- Zienkiewicz, O. C., 1971. The finite element method in engineering science. London: McGraw-Hill.
- Turner, M., Clough, R. W., Martin, H. C., & TOPP, L. J. (1956). Stiffness and deflection analysis of complex structures. *Journal of the Aeronautical Sciences*, 23(9), 805-823.
<https://doi.org/10.2514/8.3664>
- Tai, K. (1999). Integrated design optimization and analysis using a spreadsheet application. *International Journal of Mechanical Engineering Education*, 27(1), 29-40.
- Liu, Y. (2009). Simulating twisting behavior of thin-walled beams using simplified finite element models. *International Journal for Computational Methods in Engineering Science and Mechanics*, 10(4), 259-265.
- Zhai, J., Sun, Y., Zhang, Q., & Qin, X. (2021). Forming analysis of bulge formed joints based on adaptive finite element method. *Mechanika*, 27(2), 175-181.
- Liu, Z., He, Z., Tu, L., Liu, X., Liu, H., & Liang, J. (2023). A fatigue reliability assessment approach for wind turbine blades based on continuous time bayesian network and fea. *Quality and Reliability Engineering International*, 39(5), 1603-1621.
- Zhu, L., Feng, R., Li, X., Xi, J., & Wei, X. (2019). Design of lightweight tree-shaped internal support structures for 3d printed shell models. *Rapid Prototyping Journal*, 25(9), 1552-1564.
- Aignătoaie, M. (2013). FEA study on the elastic deformation process of a u-shape bimetal actuator. *Advanced Materials Research*, 837, 305-309.
- Choi, J. (2015). Development of an automation tool for the three-dimensional finite element analysis of machine tool spindles. *Journal of the Korean Society of Manufacturing Technology Engineers*, 24(2), 166-171

Agrawal, V. and Gautam, S. (2018). Iga: a simplified introduction and implementation details for finite element users. *Journal of the Institution of Engineers (India) Series C*, 100(3), 561-585.

Li, X., Zhang, X., Chen, J., Huang, L., & Lv, Y. (2021). Uniaxial tensile creep behavior of epoxy-based polymer using molecular simulation. *Polymers*, 13(2), 261.

<https://www.mdpi.com/2073-4360/13/2/261>

Nguyen, T. C. N., Vu, L. V., Nguyen, H. Q., Huyền, Đ. T. T., Nguyen, D. B., & Quang, M. T. (2023). Development and application of linear variable differential transformer (lvdt) sensors for the structural health monitoring of an urban railway bridge in vietnam. *Engineering, Technology & Applied Science Research*, 13(5), 11622-11627.

Lelong, A., Guiffant, P., & Astolfi, J. (2017). An experimental analysis of the structural response of flexible lightweight hydrofoils in cavitating flow. *Journal of Fluids Engineering*, 140(2).

Wu, Q., Huang, B., & Wang, G. (2014). Numerical simulation of transient flows around a 3d pitching hydrofoil. *Advances in Mechanical Engineering*, 7(2).

Fan, M., Sun, Z., Ran, Y., & Li, Z. (2024). Numerical study of the evolution of unsteady cavitation flow around hydrofoils with leading-edge tubercles. *Journal of Applied Physics*, 135(4).

Srinivasaiah, A. M., Santhosh, N., Munipapanna, A., Gopalakrishna, S. B., Krishnappa, M., & Venkataramaiah, H. H. (2024). Influence of nano graphene filler on hardness, impact strength and density of glass epoxy composites. *Engineering Reports*, 6(12).

<https://onlinelibrary.wiley.com/doi/10.1002/eng2.12973>

Gunnarson, P., Zhong, Q., & Quinn, D. (2019). Comparing models of lateral station-keeping for pitching hydrofoils. *Biomimetics*, 4(3), 51.

Abdellah, M. Y., Hassan, M. K., Mohamed, A. F., & Backar, A. H. (2021). Cyclic relaxation, impact properties and fracture toughness of carbon and glass fiber reinforced composite laminates. *Materials*, 14(23), 7412.