

# Structural & Fluid Analysis

## Designing a race car rear wing: Structural Design

*Coursework 2 of 2*



*Academic Coursework submitted in partial fulfilment of the requirement of*

*University of Wales Trinity Saint David for the award of the*

**BACHELOR OF ENGINEERING**

in

**MECHANICAL & MANUFACTURING ENGINEERING**

Submitted by

**JOSHUA J ASHCROFT**

(2114692)

Under the guidance of

**KELVIN LAKE**



PRIFYSGOL CYMRU

**Y Drindod Dewi Sant**

UNIVERSITY OF WALES

**Trinity Saint David**

SWANSEA - ABERTAWE

### Task Description – Coursework Brief

In class we have been discussing optimisation techniques that can be used within FEA. This assignment consolidates and builds upon this. Figure 1 shows the rear wing used in assignment 1. In this exercise you will optimise the composite construction of the wing to ensure that it does not flex excessively so that it is not considered to be a moveable structure. You must reference any sources you use, including AI (eg. ChatGPT).

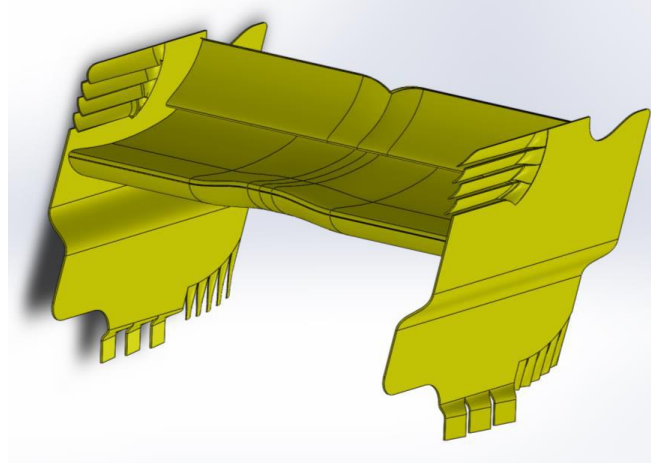


Figure 1: The rear wing

Figure 2 shows an FE model of the simplification of the wing. It is proposed that the wing is to be made from a unidirectional (UD) carbon fibre reinforced plastic (CFRP). The ends of the wing connect to the side supports such that these can be assumed to be fixed in 6 degrees of freedom.

Your task is to design the lay-up of the composite material such that when the forces seen on the wing as considered in assignment 1 are applied, the wing is still considered to be 'fixed' (i.e. the deformation is not large enough so that the wing is a movable structure). This wing needs use as little material as possible to reduce the weight of the wing.

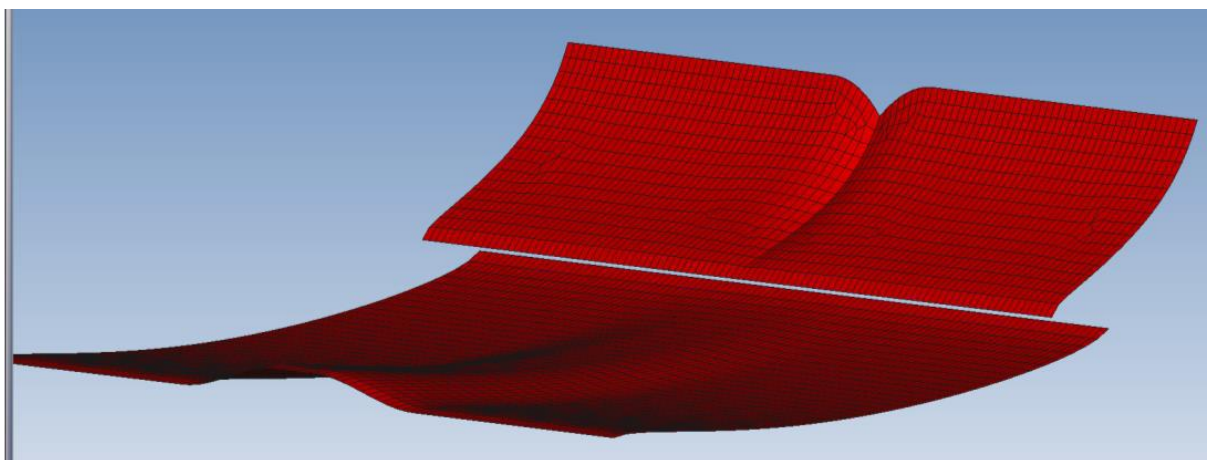


Figure 2: An FE model of the simplification of wing.

The unidirectional CFRP has the following material properties and each ply is 0.1mm thick. Assume that zero degrees runs 'fore to aft' along the length of the vehicle.

|                                                          |                        |
|----------------------------------------------------------|------------------------|
| Young's Modulus in the longitudinal direction ( $E_1$ ): | 132 GPa                |
| Young's Modulus in the lateral direction ( $E_2$ ):      | 10.8 GPa               |
| Poissons Ratio ( $\nu_{12}$ ):                           | 0.24                   |
| Inplane Shear Modulus ( $G_{12}$ ):                      | 5.65 GPa               |
| Transverse Shear Modulus (1-Z plane)( $G_{1z}$ ):        | 5.65 GPa               |
| Transverse Shear Modulus (2-Z plane)( $G_{2z}$ ):        | 5.65 GPa               |
| Mass Density ( $\rho$ ):                                 | 1540 kg/m <sup>3</sup> |
| Allowable longitudinal stress in tension ( $X_t$ ):      | 1513 MPa               |
| Allowable longitudinal stress in compression ( $X_c$ ):  | 1000 MPa               |
| Allowable transverse stress in tension ( $Y_t$ ):        | 43.4 MPa               |
| Allowable transverse stress in compression ( $Y_c$ ):    | 43.4 MPa               |
| Allowable inplane shear stress ( $S$ ):                  | 43.4 MPa               |

Given the geometry of the assembly, and a NASTRAN deck of the simplified wing (with steel material properties) generated in Solidworks, you are required to carry out the following tasks;

1. Discuss FEA optimisation techniques and by referencing appropriate sources, show how they have developed historically and discuss how you think they may develop in the future. Discuss how they improve the design process.
2. Create a model of the wing with the correct material properties in Design Studio. Lay this up with 25 plies of the UD. 5 at -90 degrees, 5 plies at -45 degrees, 5 plies at 0 degrees, 5 plies at 45 degrees and 5 plies at 90 degrees.
3. Optimise the structure to minimise the mass of the wing whilst meeting the displacement constraint. Your variables are the number of plies at each orientation. These plies can be placed at -90 degrees, -45 degrees, zero degrees, 45 degrees and 90 degrees.
4. During the development of this component, it is proposed that a prototype is produced and tested in a laboratory to understand the deformation of the wing when subjected to the forces seen and to correlate it to the simulation carried out in task 3. Discuss how you think this experiment should be carried out and how the acquisition of data should be obtained and compared to the simulation.
5. Relate your design to any technical literature and standards that cover this component.

You are required submit a report equivalent to 3000 words which discusses the above tasks.

## Contents

|                                                        |    |
|--------------------------------------------------------|----|
| Task Description – Coursework Brief .....              | 2  |
| Contents.....                                          | 4  |
| List of Figures & Tables .....                         | 5  |
| Introduction .....                                     | 6  |
| FEA Optimization Techniques .....                      | 7  |
| History of Finite Element Analysis.....                | 7  |
| Sizing Optimization .....                              | 8  |
| Shape Optimization.....                                | 8  |
| Topology Optimization .....                            | 8  |
| Future Developments in FEA Optimization .....          | 8  |
| Design Studio for GENESIS Model - Rear Wing Model..... | 9  |
| Minimising the Mass.....                               | 11 |
| Constraints .....                                      | 12 |
| Adding Forces.....                                     | 13 |
| Single Analysis.....                                   | 14 |
| Optimisation .....                                     | 16 |
| Ply Thickness – Real Life Numbers .....                | 18 |
| Stiffness to Weight – Initial CFRP .....               | 18 |
| Stiffness to Weight – Steel .....                      | 18 |
| Stiffness to Weight – Optimised CFRP .....             | 19 |
| Stiffness to Weight – Realistic Optimised CFRP.....    | 19 |
| Comparison of Stiffness-to-Weight Ratios .....         | 20 |
| Prototype Testing.....                                 | 21 |
| Conclusion .....                                       | 23 |
| Bibliography.....                                      | 24 |

## List of Figures & Tables

|                                                           |    |
|-----------------------------------------------------------|----|
| Figure 1: The rear wing .....                             | 2  |
| Figure 2: An FE model of the simplification of wing. .... | 2  |
| Figure 3 - CFRP Panels (IMG, 2024).....                   | 6  |
| Figure 4 - FEA Topology Optimisation .....                | 7  |
| Figure 5.....                                             | 9  |
| Figure 6 - Material Specification .....                   | 9  |
| Figure 7 - CFRP Layers & Angles.....                      | 10 |
| Figure 8 - Objectives For Optimisation .....              | 11 |
| Figure 9 - Flexion Constraints.....                       | 11 |
| Figure 10 - Flexion Constraints.....                      | 11 |
| Figure 11 - Adding Constraints.....                       | 12 |
| Figure 12 - Forces X- Z- .....                            | 13 |
| Figure 13 - Force Direction & Magnitude .....             | 13 |
| Figure 14 - Gensis Single Analysis .....                  | 14 |
| Figure 15 .....                                           | 14 |
| Figure 16 - Single Analysis Report.....                   | 14 |
| Figure 17 - Displacement of Initial Model.....            | 15 |
| Figure 18 - Optimised Colour Mesh .....                   | 16 |
| Figure 19 - Component Thickness.....                      | 17 |
| Figure 20 - ( ) wind testing model .....                  | 21 |
| <br>                                                      |    |
| Table 1 .....                                             | 15 |
| Table 2 .....                                             | 18 |
| Table 3 .....                                             | 20 |

## Introduction

Carbon Fibre Reinforced Plastics (CFRP) are increasingly utilized across various manufacturing sectors, including aerospace for components such as fuselages and wings, automotive for chassis and body panels, and even civil engineering. The primary advantage of CFRP is its impressive strength-to-weight ratio, which surpasses traditional materials like aluminium often used in automotive applications. A significant reduction in vehicle weight, approximately 10%, can lead to about a 7% improvement in fuel efficiency (Inma 2017). This efficiency gain has made CFRP an attractive material in automotive design.

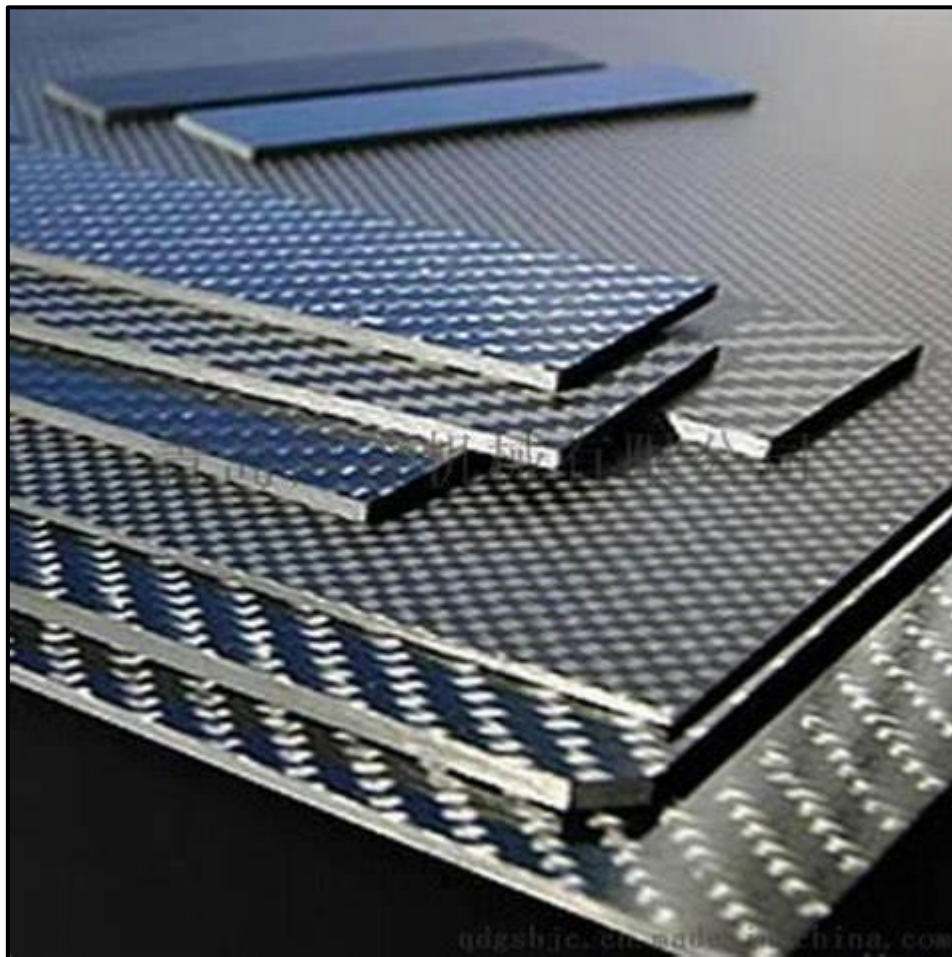


*Figure 3 - CFRP Panels (IMG, 2024)*

This report explores Finite Element Analysis (FEA) optimization techniques, assessing various methods used in component testing and data acquisition critical for design engineers.

## FEA Optimization Techniques

### History of Finite Element Analysis

The origins of FEA can be traced back to the late 19th century when John William Strutt Rayleigh developed a method for predicting the natural frequency of simple structures. By the 1940s, numerical techniques had advanced to predict the performance of more complex structures, marking the beginning of digital FEA with the advent of room-sized computers used by the US Military during World War II for ballistics calculations (Anon. 2018). In the 1960s, NASA contracted MSC to develop a general-purpose FEA software, now known as NASTRAN.

Finite Element Analysis (FEA) optimization is integral to refining and enhancing design performance by using simulations to meet specific user-defined objectives. These objectives might include minimizing mass, volume, stress, strain, or displacement, among other factors, while adhering to predefined constraints. In the automotive sector, the optimization goals can vary significantly between commercial vehicle manufacturers and race car designers.

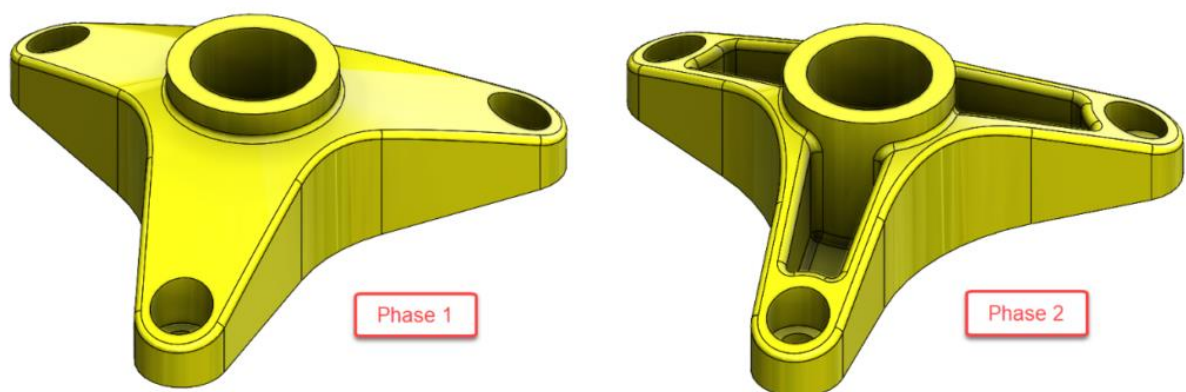


Figure 4 - FEA Topology Optimisation

### Sizing Optimization

Sizing optimization involves determining the optimal material thickness and dimensions for structural components to meet performance criteria while minimizing material usage. This method ensures that each part of the model meets the required strength and stiffness without excess material.

### Shape Optimization

Shape optimization focuses on refining the geometry of a model to improve performance under specified constraints and loading conditions. This involves iteratively adjusting the model's shape to achieve the best possible performance.

### Topology Optimization

Topology optimization is employed during the initial design phase to ensure efficient material distribution within a model, considering applied loads and spatial constraints (Anon. 2020). This method can be applied to entire components or specific design spaces within a model, aiming to enhance overall structural efficiency.

### Future Developments in FEA Optimization

Advancements in FEA optimization software continues to enhance the capabilities of design engineers, enabling more efficient, safer, and innovative designs while reducing costs. Future developments will likely focus on integrating more complex simulations and real-time data analysis.

## Design Studio for GENESIS Model - Rear Wing Model

A rear wing model has been developed in Solidworks, serving as the initial baseline for the project. The model is assumed to be infinitely rigid at both ends. It is proposed that the wing be fabricated from unidirectional carbon fibre reinforced plastic (CFRP).

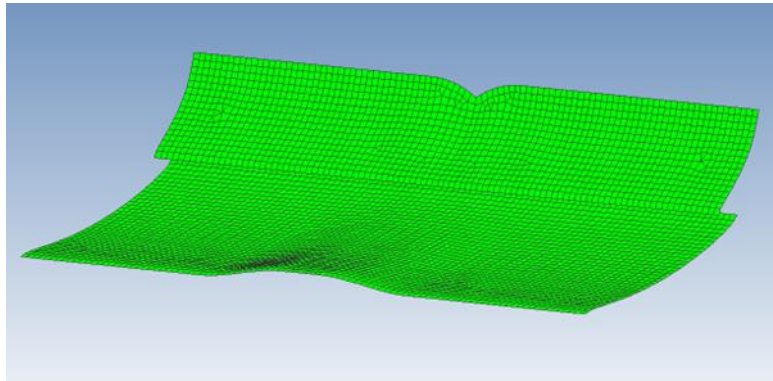


Figure 5

The Theta values were adjusted in the model to ensure that all the plies align correctly with the specified Theta direction for each grid.

| Edit a Material : 2-D Orthotropic Properties |          |
|----------------------------------------------|----------|
| E_1:                                         | 1.32E+11 |
| E_2:                                         | 1.08E+10 |
| Nu_12:                                       | 0.24     |
| G_12:                                        | 5.65E+9  |
| G_13:                                        | 5.65E+9  |
| G_23:                                        | 5.65E+9  |
| Rho:                                         | 1540.0   |
| A_1:                                         | 1.513E+9 |
| A_2:                                         | 1.E+9    |
| Tref:                                        | 4.34E+7  |
| GE:                                          | 4.34E+7  |
| F12:                                         | 4.34E+7  |

Figure 6 - Material Specification

The material properties were incorporated into the model according to the CFRP specifications. Carbon fibre composites offer excellent potential for lightweight designs due to their high stiffness-to-weight and strength-to-weight ratios. CFRP's density is approximately 20% that of steel and 57% that of aluminium (Dynexa, n.d.), meaning steel has a density of around 7700 kg/m<sup>3</sup>.

The group properties of the model were configured, including the selected material (CFRP), the thickness of each ply (1 mm), the total number of plies (5), and the orientation angle of each ply relative to the vehicle's front-to-rear direction.

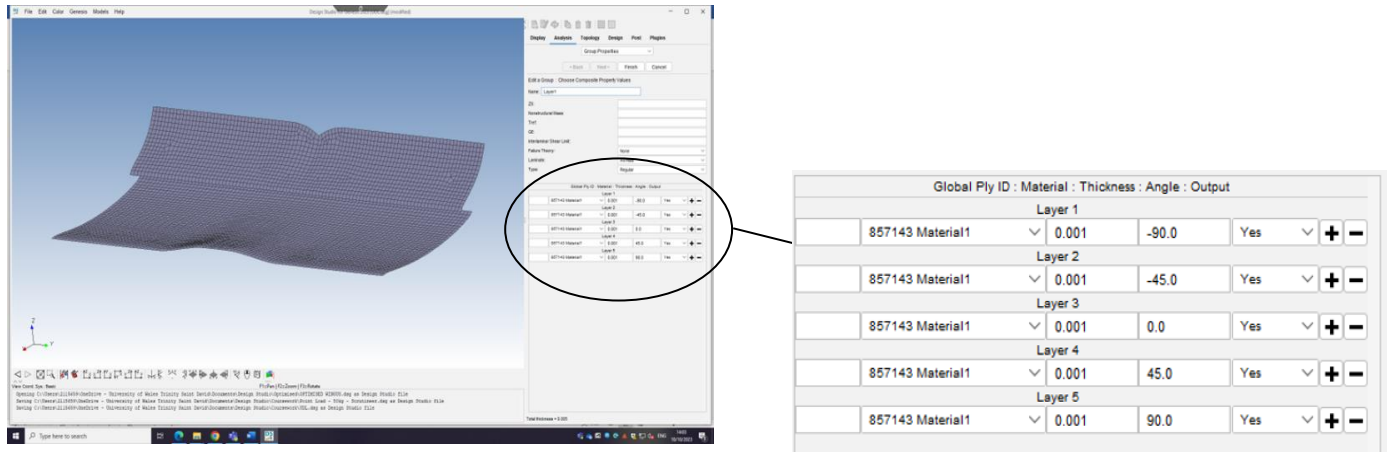


Figure 7 - CFRP Layers & Angles

The primary objectives for the FEA (Finite Element Analysis) were then considered, focusing on:

## Minimising the Mass

The primary objective in the design tab was to minimize the component's mass while ensuring it met all performance criteria. This was achieved by setting appropriate constraints and using optimization algorithms to refine the design.

Minimising the mass of the component is essential for various reasons, in the specific case of a rear wing for a Formula One car, a lightweight design is highly desirable *'allowing for higher speeds and greater fuel efficiency'* (3M, 2023). A lighter structure often also provides better aerodynamic characteristics which allow for increased acceleration and manoeuvrability.

The objective was set in the 'Design' tab to ensure that upon optimisation of the component, the key objective would be to minimise the mass.

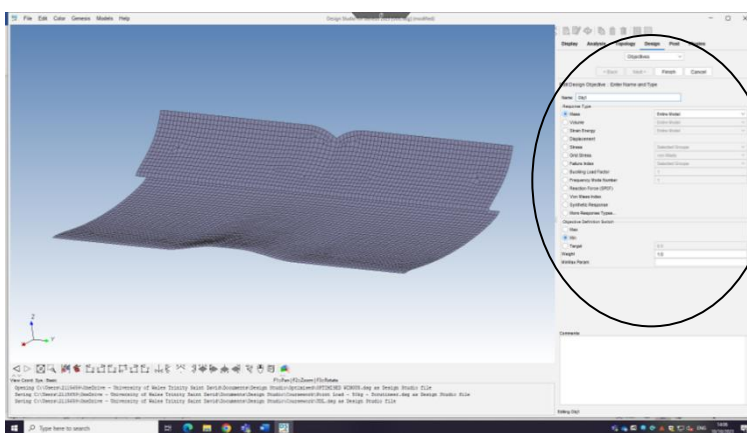


Figure 8 - Objectives For Optimisation

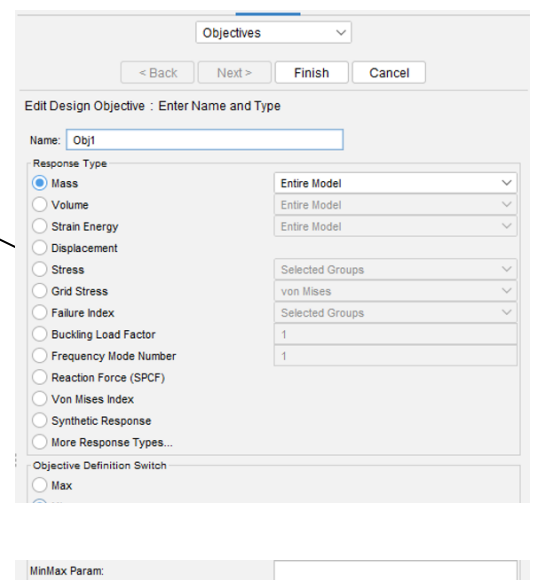


Figure 9 - Flexion Constraints



Figure 10 - Flexion Constraints

## Constraints

Constraints play a crucial role in Finite Element Analysis (FEA) optimization by ensuring that the generated designs do not compromise the safety or durability of the components. By imposing constraints such as maximum stress or deflection, the process ensures that any design produced by the software adheres to specified criteria. For this particular model, the primary constraint was the allowable deflection across the wing. According to Article 3.2.2 of the Fédération Internationale de l'Automobile (FIA) regulations (2022):

*"All aerodynamic components or bodywork influencing the car's aerodynamic performance must be rigidly secured and immobile with respect to their frame of reference"*

The dimensions determining the allowed amount of flexion were derived from Giuliana (2021), who states:

*"At the extremes of the air foil, a horizontal rearward force of 75kg will be applied, compared to 50kg. Overall, the wing must not exceed 1mm of flexion, while the previously allowed tolerance was a maximum of 3mm."*

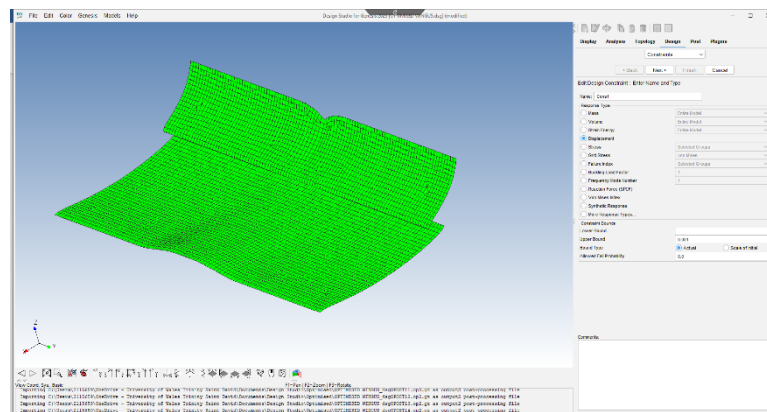


Figure 11 - Adding Constraints

In this context, the constraints for the model were set to a displacement limit of 0.001m across the component. This means that the maximum allowed displacement when running the single analysis is 1mm.

## Adding Forces

Forces were applied to the model based on the specifications provided by Giuliana (2021), including horizontal and vertical forces to simulate real-world conditions. The forces were distributed across the model using a uniform distribution load (UDL) method.

The forces to be added to the component are defined also by (Giuliana, 2021) stating that a horizontal force of 75kg and a 100kg force vertically should be applied to the component for testing.

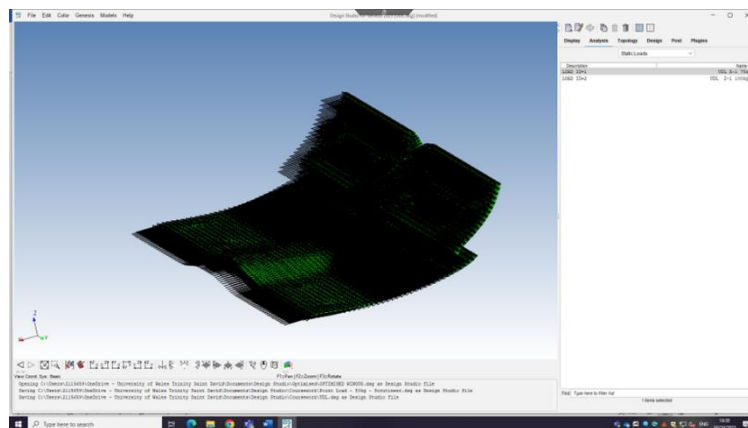


Figure 12 - Forces X- Z-

The forces added were in the Z- and X- directions in relation to the model, these directions are the front to rear directions of the car and also the downforce vertically.

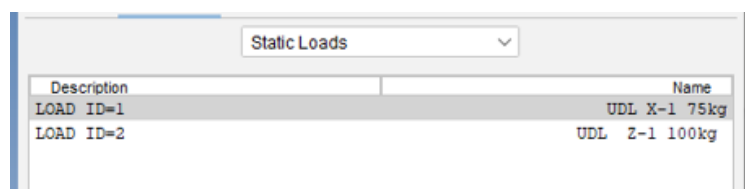


Figure 13 - Force Direction & Magnitude

When applying the magnitude of the forces acting upon the component it is important to remember that they are measured in Newtons, and also that the magnitude input by the user will add that magnitude of force to each vector, so to calculate the force per vector, the user must divide the total force by the number of vectors selected, in this case all 5263 vectors were selected, for a UDL, the calculation would be as follows:

$$100\text{KG} = 980.665 / 5263 = 0.18633194 \text{ N per vector}$$

$$75\text{KG} = 735.49875 / 5263 = 0.13974895 \text{ N per vector}$$

## Single Analysis

A single analysis was conducted using GENESIS software, generating a report that compared the actual performance of the model against the set constraints. The initial analysis revealed a constraint violation of 147.5%, indicating the need for further optimization to meet FIA standards (Fédération Internationale de l'Automobile 2022).

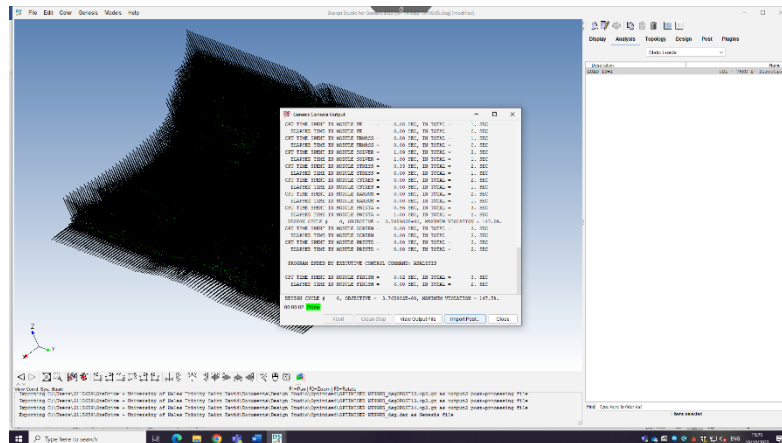


Figure 14 - Gensis Single Analysis

MAXIMUM VIOLATION = 147.5%.

Figure 15

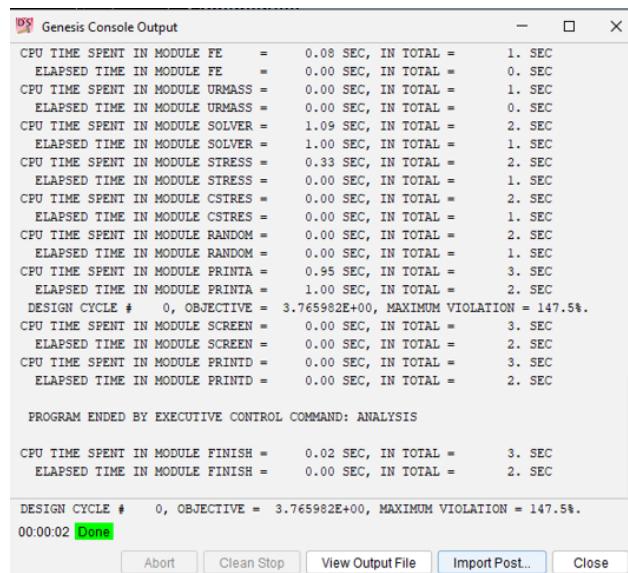


Figure 16 - Single Analysis Report

This constraint violation is not acceptable to the desired dimension. The (Fédération Internationale de l'Automobile, 2022) states that:

*'The car with the non-compliant rear wing may be disqualified from the race or event.'*

The analysis model shows the areas effected most by the loading exerted on the component, and the quantitative values of the flexion that occurred during the testing.

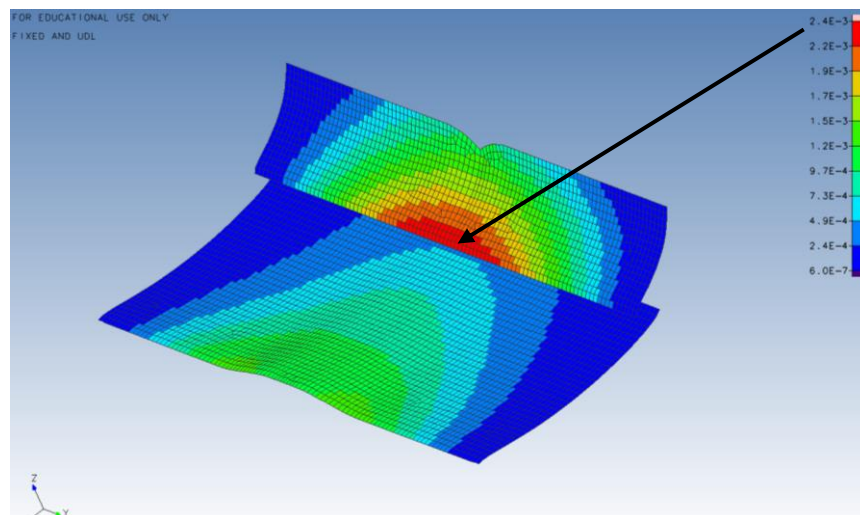


Figure 17 - Displacement of Initial Model

The darkest shaded areas of red indicate places on the component that has displaced the most. In the case of this analysis, the highest displacement was in the centre area on the top section which displaced by 2.4mm (1.4mm more than the specified constraint).

The baseline sample's HIS report also generated a reading for the mass of the component labelled as the objective, as this was previously set to be minimum. The layer thickness for each of the plies is also shown which was set previously in the design variables.

Table 1

| DESIGN CYCLE #0,<br>OBJECTIVE = 3.765982E+00,<br>MAXIMUM VIOLATION = 147.5%. |     |              |       |
|------------------------------------------------------------------------------|-----|--------------|-------|
| DESIGN VARIABLE #                                                            | 1 = | 1.000000E-03 | L1THK |
| DESIGN VARIABLE #                                                            | 2 = | 1.000000E-03 | L2THK |
| DESIGN VARIABLE #                                                            | 3 = | 1.000000E-03 | L3THK |
| DESIGN VARIABLE #                                                            | 4 = | 1.000000E-03 | L4THK |
| DESIGN VARIABLE #                                                            | 5 = | 1.000000E-03 | L5THK |

Before any optimisation, the mass of the component was 3.76kg with a layer thickness of 1mm for each of the plies, however this resulted in the 147.5% violation of the constraint

## Optimisation

It is now possible to run an optimisation cycle within the GENESIS software, this run will take into consideration the restraints and design variables previously set by the user and optimise the component to conform / converge to these parameters and constraints.

The design mesh/colour mesh DISP model analysis now shows the displacement across the component is  $9.8\text{E-}4\text{m}$  equating to  $0.98\text{mm}$ , which is within the tolerance range set.

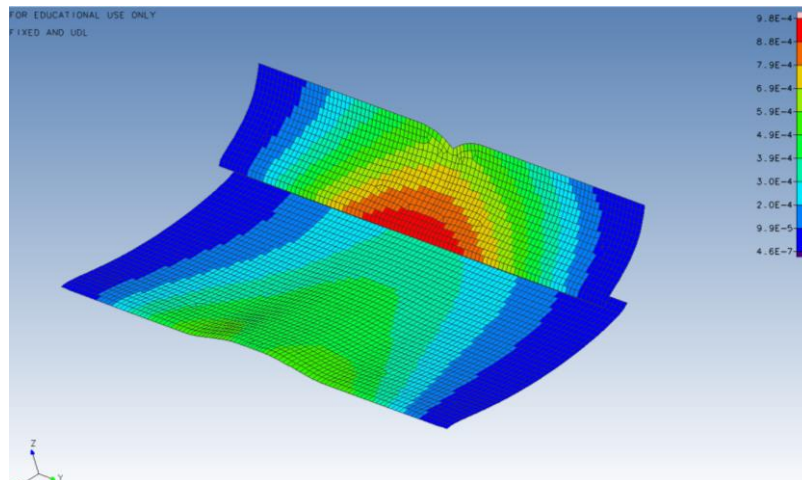


Figure 18 - Optimised Colour Mesh

The optimisation process ran a total of 13 cycles before hard convergence within the parameters.

| DESIGN CYCLE #13,<br>OBJECTIVE = 4.788959E+00,<br>MAXIMUM VIOLATION = 0.0%. |     |              |       |
|-----------------------------------------------------------------------------|-----|--------------|-------|
| DESIGN VARIABLE #                                                           | 1 = | 3.072919E-04 | L1THK |
| DESIGN VARIABLE #                                                           | 2 = | 1.353106E-03 | L2THK |
| DESIGN VARIABLE #                                                           | 3 = | 2.998070E-03 | L3THK |
| DESIGN VARIABLE #                                                           | 4 = | 1.489704E-03 | L4THK |
| DESIGN VARIABLE #                                                           | 5 = | 2.100075E-04 | L5THK |

The table above shows the HIS report for the final cycle, the mass of the component is now  $4.78\text{kg}$ , which is a  $1.02\text{kg}$  increase from the original model, however, the maximum violation of constraint is now  $0\%$ . A notable design variable change is the in the distribution and subsequently the adaptation to the overall thickness of the component.

The outer most plies of the component have reduced significantly in thickness, whereas the inner layers have increased. This may be because the most stresses and strains occur at

these points, so they have been thickened to reduce the deflection to within our parameters.

The overall thickness of the component is now 0.00635m (6.35mm), this is an increase of 1.35mm from the baseline model.

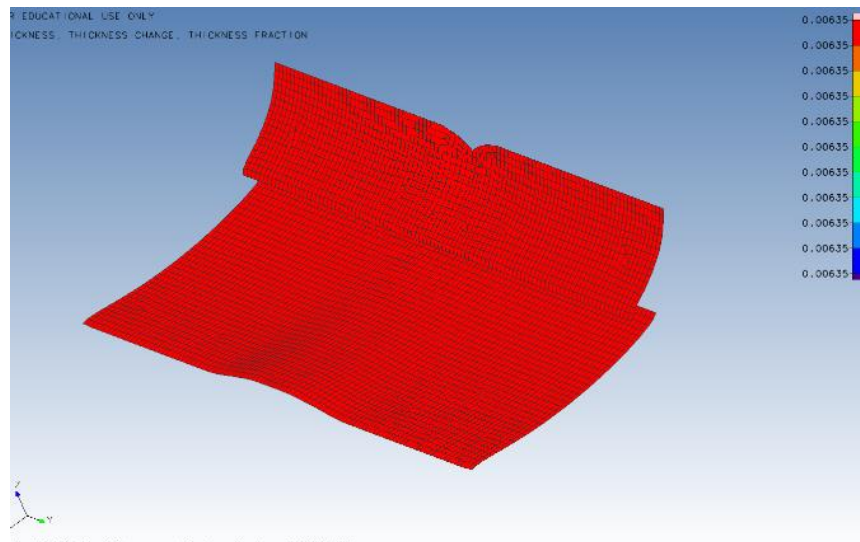


Figure 19 - Component Thickness

### Ply Thickness – Real Life Numbers

The above-generated report highlights the optimal solution for the component tested, however, the ply thicknesses generated are unachievable in a real-life manufacturing scenario, so to counter this, a discrete design variable was added to only allow ply thicknesses of a manufacturable standard.

After another optimisation cycle using the new design variable parameters, the ply thicknesses are given only increments of 0.1mm.

Table 2

| DESIGN CYCLE #17,<br>OBJECTIVE = 4.895777E+00,<br>MAXIMUM VIOLATION = 0.0%. |     |              |       |
|-----------------------------------------------------------------------------|-----|--------------|-------|
| DESIGN VARIABLE #                                                           | 1 = | 3.000000E-04 | L1THK |
| DESIGN VARIABLE #                                                           | 2 = | 1.400000E-03 | L2THK |
| DESIGN VARIABLE #                                                           | 3 = | 3.000000E-03 | L3THK |
| DESIGN VARIABLE #                                                           | 4 = | 1.500000E-03 | L4THK |
| DESIGN VARIABLE #                                                           | 5 = | 3.000000E-04 | L5THK |

### Stiffness to Weight – Initial CFRP

$$\text{Stiffness (K)} = \frac{F}{x} \quad K = \frac{1716.16375}{2.4} = 715.0682292 \text{ N/mm}$$

$$\text{Stiffness/weight} = \frac{715.0682292}{3765.982 \times 9.81} = 0.0193553 \text{ N}^2/\text{mm}$$

### Stiffness to Weight – Steel

$$\text{Stiffness (K)} = \frac{F}{x} \quad K = \frac{1716.16375}{2.3} = 746.158152 \text{ N/mm}$$

$$\text{Stiffness/weight} = \frac{746.158152}{7531.76 \times 9.81} = 0.0100986 \text{ N}^2/\text{mm}$$

## Stiffness to Weight – Optimised CFRP

$$\text{Stiffness (K)} = \frac{F}{x} \quad K = \frac{1716.16375}{0.98} = 1751.1875 \text{ N/mm}$$

$$\text{Stiffness/weight} = \frac{1751.1875}{4788.959 \times 9.81} = 0.0372754 \text{ N}^2/\text{mm}$$

## Stiffness to Weight – Realistic Optimised CFRP

$$\text{Stiffness (K)} = \frac{F}{x} \quad K = \frac{1716.16375}{0.99} = 1733.498737 \text{ N/mm}$$

$$\text{Stiffness/weight} = \frac{1733.498737}{4857.976 \times 9.81} = 0.0363746 \text{ N}^2/\text{mm}$$

### Comparison of Stiffness-to-Weight Ratios

From the testing of the different materials and optimisations, it is evident that the initial un-optimised CFRP component and the steel component have almost the same displacement under the same loading, however, the steel component is double the mass of the CFRP resulting in stiffness to weight ratio being around half of the CFRP component.

The stiffness-to-weight ratio can be further improved through optimisation of the component, this almost doubles the ratio, however, this would not be achievable due to manufacturing constraints as previously mentioned.

Table 3

|                 | Displacement<br>(mm) | Mass (kg) | Stiffness-to-weight Ratio<br>(N <sup>2</sup> /mm) |
|-----------------|----------------------|-----------|---------------------------------------------------|
| Initial CFRP    | 2.4                  | 3.76598   | 0.0193553                                         |
| Steel           | 2.3                  | 7.53176   | 0.0100986                                         |
| Optimised CFRP  | 0.98                 | 4.78895   | 0.0362754                                         |
| Realistic Model | 0.99                 | 4.85797   | 0.0363746                                         |

## Prototype Testing

The development of a Formula One car starts with a team creating intricate 3D CAD models for each component, encompassing transmission parts, electronics, mechanical designs, and critical aerodynamic components.

Aerodynamics is a crucial aspect of the design process, with teams dedicating significant time to perfect these components. Computational Fluid Dynamics (CFD) simulations are essential in this phase, enabling design engineers to evaluate various wing designs and configurations under simulated forces.



*Figure 20 - (i) wind testing model*

Once the design engineers are satisfied with the performance in CFD simulations, the next step is physical prototyping. Typically, a prototype model is created and tested in a wind tunnel. Formula One teams often possess their own wind tunnel facilities, allowing them to meticulously control airflow and other testing conditions. These wind tunnels are equipped with powerful fans to replicate real-world conditions the components may face.

Components are usually manufactured using rapid prototyping and 3D printing techniques.

According to Motorsport.com (2022):

*"In the past, some teams actually ran full-scale cars in wind tunnels but now the maximum is set at 60 per cent to reduce costs".*

The wind tunnel simulates track conditions, including airspeed and angle of attack. However, as Motorsport.com (2022) notes:

*"The speed is limited to a maximum of 180 km/h, which means they cannot fully test all aspects of the car's performance".*

This implies that despite the detailed analysis of the collected data, complete validation only occurs through in-application testing. Nonetheless, results from wind tunnel tests can be compared with CFD analysis, allowing for design adjustments.

Data from wind tunnel testing can be acquired through various methods depending on the test's specific goals and parameters. Common techniques include:

- **Pressure Sensors:** These sensors are strategically placed to measure air pressure at specific points, helping to determine the pressure distribution over the model and crucial for comparing with CFD simulations.
- **Visualisation Techniques:** Unlike sensors, these techniques offer a qualitative approach, illustrating the physical distribution of airflow around the component.
- **Force and Load Sensors:** These sensors convert forces and loads into digital data, which can then be analyzed by engineers.

## Conclusion

In conclusion, analysis of Finite Element Analysis (FEA) data suggests that the CFRP component would be significantly lighter than the steel component. To confirm these results, it would be essential to create a scale model within the allowed dimensions and conduct wind tunnel testing. The results of this testing could be compared to CFD results, enabling further optimization using FEA software like Design Studio.

It is clear that FEA optimization software is an invaluable tool for design engineers, enhancing efficiency, safety, and innovation while reducing costs.

## Bibliography

model, Wind testing,

[https://th.bing.com/th/id/R.e6b9a0b24d1cc1e83676d847c097c3f8?rik=TFJenyABxDW0Yw&riu=http%3a%2f%2fdebutart-static-v1.s3.amazonaws.com%2fprojectitem%2f7%2f8%2ff2f7e88ce79035271d2acaf37f81aa84%2f5687\\_full-retina.jpg%3f1458748227&ehk=urgoFgxYqGSt9eWV5NQVbDKMN](https://th.bing.com/th/id/R.e6b9a0b24d1cc1e83676d847c097c3f8?rik=TFJenyABxDW0Yw&riu=http%3a%2f%2fdebutart-static-v1.s3.amazonaws.com%2fprojectitem%2f7%2f8%2ff2f7e88ce79035271d2acaf37f81aa84%2f5687_full-retina.jpg%3f1458748227&ehk=urgoFgxYqGSt9eWV5NQVbDKMN).

<https://th.bing.com>. [Online]

Available at:

[https://th.bing.com/th/id/R.e6b9a0b24d1cc1e83676d847c097c3f8?rik=TFJenyABxDW0Yw&riu=http%3a%2f%2fdebutart-static-v1.s3.amazonaws.com%2fprojectitem%2f7%2f8%2ff2f7e88ce79035271d2acaf37f81aa84%2f5687\\_full-retina.jpg%3f1458748227&ehk=urgoFgxYqGSt9eWV5NQVbDKMN](https://th.bing.com/th/id/R.e6b9a0b24d1cc1e83676d847c097c3f8?rik=TFJenyABxDW0Yw&riu=http%3a%2f%2fdebutart-static-v1.s3.amazonaws.com%2fprojectitem%2f7%2f8%2ff2f7e88ce79035271d2acaf37f81aa84%2f5687_full-retina.jpg%3f1458748227&ehk=urgoFgxYqGSt9eWV5NQVbDKMN)

[Accessed 20 06 2024].

3M, 2023. *Lessons to learn from Formula 1 lightweighting design*. [Online]

Available at: [https://www.3m.co.uk/3M/en\\_GB/transportation-uk/commercial-speciality-vehicle/stories-and-infographics/full-story/?storyid=0059b019-e3e5-4eb6-9b7d-8b63374dd412#:~:text=For%20decades%2C%20lightweighting%20has%20been%20a%20crucial%20element,are%20manyfold%3](https://www.3m.co.uk/3M/en_GB/transportation-uk/commercial-speciality-vehicle/stories-and-infographics/full-story/?storyid=0059b019-e3e5-4eb6-9b7d-8b63374dd412#:~:text=For%20decades%2C%20lightweighting%20has%20been%20a%20crucial%20element,are%20manyfold%3)

[Accessed 20 06 2024].

Dynexa, n.d. *Dynexa*. [Online]

Available at: <https://www.dynexa.de/en/about-cfrp/advantages-cfrp-vs-stainless-steel/#:~:text=Carbon%20fiber%20composite%20materials%20score%20with%20excellent%20lightweight,and%2057%20%25%20of%20the%20density%20of%20aluminum>.

[Accessed 20 06 2024].

Fédération Internationale de l'Automobile, 2022. *Formula 1 Technical Regulations*, s.l.: s.n.

Giorgio Piola, M. S., 2021. *F1 braced for flexi-wing showdown in Baku*. [Online]

Available at: <https://www.motorsport.com/f1/news/f1-braced-for-flexi-wing-showdown-in-baku/6529890/>

[Accessed 12 October 2023].

Available at: <https://fractory.com/topology-optimisation/#:~:text=Topology%20optimisation%20is%20a%20mathematical%20method%20used%20at,load%2C%20and%20space%20limitations%20to%20create%20a%20design.>

[Accessed 20 06 2024].

Anon., 2020. *SDC Verifier*. [Online]

Available at: <https://sdcverifier.com/articles/why-do-you-need-to-optimize-your-fea-model/#:~:text=Types%20of%20optimization%20in%20FEA%201%20Topology%20optimization,respecting%20some%20constraints.%20...%203%20Sizing%20Optimization%20>

[Accessed 21 06 2024].

Anon., 2018. *O'Donnell Consulting Engineers*. [Online]

Available at: <https://www.odonnellconsulting.com/what-we-do/expert-witness-forensic-engineering/forensic-engineering-expert-witness/history-of-fea/>

[Accessed 21 06 2024].

Giuliana, R., 2021. *Motorsport Week*. [Online]

Available at: <https://www.motorsportweek.com/2021/06/13/technical-insight-how-the-fia-will-test-flexible-wings-going-forward/>

[Accessed 21 06 2024].

Motorsport.com, 2022. *Motorsport.com*. [Online]

Available at: <https://www.motorsport.com/f1/news/how-is-an-f1-car-made/7626323/>

[Accessed 25 06 2024].

Velling, A., 2020. *Fractory*. [Online]

Inma, 2017. *Reinforced Plastics*. [Online]

Available at: <https://www.reinforcedplastics.com/content/features/the-carbon-fiber-makes-its-way-in-theautomotive-industry>

[Accessed 27 06 2023].

Jonathan Viquerat, J. R. A. K. H. G. A. L. E. H., 2021. *Direct shape optimization through deep reinforcement learning*. [Online]

Available at: <https://www.sciencedirect.com/science/article/pii/S0021999120308548>

[Accessed 27 06 2024]