

Designing a Race Car Rear Wing



Manmeet Singh (2114501)

ACMA6010 – Structural & Fluid Analysis

10/13/2023

Contents

Introduction	2
Importance and the Impact of the Reynolds Number and its Flow around the Wing and Estimation of the Reynolds Number of the fluid flowing around the wing.	2
Boundary Flow, Separation, and Turbulence around the Wing.....	4
Carbon Fiber Reinforced Composites (CFRP) and Historical, Current, and Future Developments of such Components.....	5
Historical Developments	6
Current Developments	6
Future Developments	6
Modeling Techniques for Simulation of Fluid Flow Around Wing.....	7
Advantages.....	7
Disadvantages.....	8
Wind Tunnel Testing.....	8
Analytical Methods	9
Test Setup and Instrumentation.....	10
Experimental Procedure.....	10
Analysis of Data and Comparison	11
Correlation and Iteration.....	11
Conclusion.....	12
References.....	13

Introduction

This study aims to discuss and analyze the fluid flow around the rear wing of a race car. This assignment involves various tasks which involve discussing the significance of the Reynolds number in fluid flow and its impact on the wing as well as the boundary flow around the wing and its relationship with separation and turbulence. The purpose is to highlight the aerodynamic principles and considerations that impact the shape and features of wings.

It is essential to investigate the implications of constructing the wing from Carbon Fiber Reinforced Plastic (CFRP). The performance implications are discussed using CFRP and discusses historical, current, and future developments of these elements. Finally, the report explains the modeling techniques for the simulation and analysis of the fluid flow around the wing.

Importance and the Impact of the Reynolds Number and its Flow around the Wing and Estimation of the Reynolds Number of the fluid flowing around the wing.

The Reynolds number (Re) is a dimensionless number for the characterization of the flow regime of a fluid (Graaff et al., 2000). It is the ratio of inertial forces to viscous forces and it's formula is:

$$Re = (\rho V L) / \mu$$

- ρ is the fluid density
- V is the fluid velocity of the object
- L is a characteristic length of the object (such as the wing chord length),
- μ is the fluid dynamic viscosity

The Reynolds number is essential as it determines the flow type around an object. It aids in predicting whether the flow is laminar (smooth and ordered) or turbulent (chaotic and irregular).

Regarding the flow around the wing, if a Reynolds number is low, the flow is laminar flow, and if a Reynolds number is high, the flow will be turbulent flow (Hosseini et al., 2016).

When the Reynolds Number is than 2100 it is considered a Laminar Flow. The flow is orderly and smooth for low Reynolds. The particles of air move in parallel layers without considerable mixing or turbulence. Streamlined flow lines indicate laminar flow and it will have low drag. However, it is also subjected to stall and separation, which can lessen lift and drag enhances.

When the number is higher than 2100 the flow gets chaotic and turbulent. Turbulent flow possesses swirling eddies and air particles mixing. It could increase the momentum mixing and reduce flow separation, leading to increased drag and downforce. Generally, turbulent flow is more needed for wing performance in motorsports applications, because it aids in maintaining better stability and control.

To project the Reynolds number of fluid flowing around the wing, the specific values for the density and velocity of the fluid, and the characteristic length of the wing need to be specified.

The density of air = 1.225 kg/m^3 at standard conditions.

Dynamic viscosity of air = $1.81 \times 10^{-5} \text{ kg/(m/s)}$

The velocity of air = 80 meters per second (aproximately 290 km/h)

The length of the rear wing of a race car is 1 meters for this calculation.

$$Re = 1.225 \times 80 \text{ m/s} \times 1 / 1.81 \times 10^{-5} = 5414364.641$$

It shows that the flow is well within the turbulent regime for these conditions.

According to (Duxbury, 2023) Formula 1 cars can reach up to 340km/h, so for tolerance purposes the equation above has been completed by entering a value of 80 m/s which equates to approximately 290 km/h.

According to (Abdalla, 2010) the rear wing chord length is 1m, so for this calculation this source has been used as a valid parameter.

Boundary Flow, Separation, and Turbulence around the Wing

The boundary flow surrounding the wing is the fluid behavior in nearby proximity to the surface of the wing. The flow could be impacted by certain factors such as wing shape and angle, surface roughness, and thickness of the boundary layer.

Separation happens when the flow of air surrounding the wing gets detached from the surface and does not follow the wing contour. It may lead to downforce loss and reduction in drag. Separation is unwanted because it lessens the entire wing's performance. The wing design objective is to lessen separation by regulating the flow patterns and keeping the attached flow as much as possible.

Particular features are essential to design the wing (the curved shape, angle of attack, and the element's existence such as flaps or spoilers, management of the flow, and minimizing separation). These characteristics operate the boundary layer and will create favorable pressure gradients for keeping the airflow attached to the surface of the wing to delay or prevent separation (Kurec et al., 2019).

Turbulence plays an important role in the flow surrounding the wing. Turbulent flow aids in maintaining better control and stability of the wing by augmenting mixing and lessening separation. It helps better flow attachment and minimization of adverse pressure gradients.

The design of the rear wing of a race car considers the required performance features (maximization of downforce and minimization of drag). It is shown in Figure 1

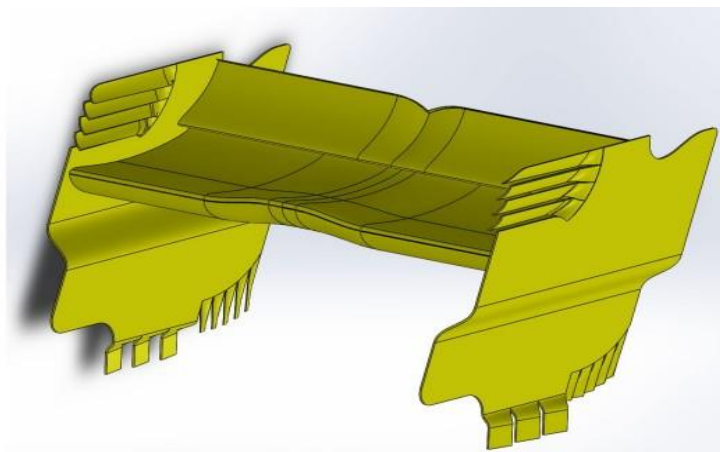


Figure 1: A rear wing of a race car

With the optimization of the shape of the wing and incorporation of elements boundary layer, the aim of an engineer is to create a balance between the production of downforce, reduction of drag, and increased stability. The design details and wing features depend on the requirements and constraints of the race car and its proposed application.

Carbon Fiber Reinforced Composites (CFRP) and Historical, Current, and Future Developments of such Components

There are several implications to constructing the wing from Carbon Fiber Reinforced Plastic (CFRP)

- **High Strength and Light Weight** - CFRP is called for its high strength-to-weight ratio. If CFRP is used, the wing could be constructed to be lightweight to maintain structural integrity and increase performance. It drastically reduces the weight of the race car which positively impacts by increasing acceleration, reducing braking distance, better handling, and more fuel efficient. Lighter wings also account for a better-balanced center of gravity which improves the overall dynamic of the vehicle (Rodriguez et al., 2019)
- **Structural Rigidity and Stiffness** - CFRP provides high stiffness and will provide excellent resistance to torsional and bending forces. This stiffness aids to maintain the shape of the wing under aerodynamic loads, avoiding deformation and preserving its effectiveness. A rigid wing progresses stability, reduces drag, and will control more precisely over the aerodynamic forces.
- **Aeroelasticity** - The CFRP stiffness also accounts for a process which is called aeroelasticity, where the stiffness of the wing associates with aerodynamic forces, causing variations in its performance and shape. This interaction is helpful and challenging for management. Appropriate design and analysis techniques are needed to make sure that aeroelastic effects will not compromise the performance of the wings or lead to instability (Khawaja et al., 2017).

Historical, current, and future developments of CFRP elements are comprehensively documented in industry standards and technical literature. The notable points are

Historical Developments

- **Introduction and Early Applications** - CFRP composites have been used in aerospace applications since the 1960s because of their exceptional weight features and strength. The experience and knowledge from aerospace applications have impacted the development of CFRP elements in various industries including motorsports.

Current Developments

- **Advanced Manufacturing Techniques** - Advancements in manufacturing techniques (automated fiber placement and resin transfer molding) have improved the quality and efficiency of production. These techniques cause the fabrication of complex shapes and optimize the performance of the wing.
- **Material Enhancements** - Ongoing research focuses on the development of new carbon fiber reinforcements, hybrid composites, and resin matrices to enhance the mechanical properties and adapt them for applications. The aim of these developments is to improve the performance of CFRP components.

Future Developments

- **Recyclability and Sustainability** - There is an increase in emphasis on the sustainability of CFRP components that includes race car wings. Efforts are conducted to create recyclable and bio-based materials to lessen the environmental influence of CFRP production and end-of-life disposal.
- **Integration of Smart Materials** - The incorporation of smart materials like piezoelectric materials and shape memory into CFRP structures could make adaptive and morphing wing designs. These wings modify their shape and aerodynamic features in action to variation in flow conditions, to further elevate the performance.

Industry standards and race regulations have an essential role in the application of CFRP components such as race car wings. Racing governing bodies (Fédération Internationale de l'Automobile (FIA) in Formula 1) establish regulations and guidelines to make sure safety, technological developments, and fair competition, while restraining

various parameters to keep a level playing field. These regulations generally involve limitations on materials, design characteristics, and dimensions to promote safety, control of costs, and prevention of excessive performance benefits.

Modeling Techniques for Simulation of Fluid Flow Around Wing

For the simulation of fluid flow around the wing, several modeling techniques could be applicable. Every technique has its advantages and disadvantages, and the selection of the modeling tool is impacted by factors such as the flow complexity, available computational resources, and desired accuracy level (Blocken et al., 2016). The three commonly used modeling techniques are:

- Computational Fluid Dynamics (CFD)
- Wind Tunnel Testing
- Analytical Methods.

Computational Fluid Dynamics (CFD) - is a numerical approach in which the governing equations of fluid flow are solved using computational algorithms. It provides a detailed and comprehensive analysis of the flow field around the wing.

Advantages

- a. Flexibility - CFD simulates different flow conditions, geometries of the wing, and operating parameters. It gives the freedom to adjust and iterate designs without the need for physical prototypes.
- b. Detailed Flow Visualization - CFD gives visual representations of the flow field that includes velocity vectors, pressure contours, and streamlines, to allow improved understanding of the complex flow phenomena.
- c. Quantitative Data - CFD provides quantitative data such as drag, lift, and pressure coefficients, which are important to assess the performance of the wing.
- d. Time and Cost Efficiency - In comparison to physical testing, CFD simulations are more economical and faster. It could save resources and time as engineers are allowed to analyze manifold design iterations and scenarios virtually.

Disadvantages

- a. Computational Resources - CFD simulations are computationally intensive, and need high-performance computing resources, particularly for complex geometries and fine mesh resolutions. Large-scale simulations could require considerable computational cost and time.
- b. Accuracy and Assumptions - CFD simulations include different assumptions (turbulence modeling that could introduce indecisions and will impact the accuracy results). Careful calibration and validation of experimental data are essential to achieve accurate predictions.
- c. Complexity - CFD needs expertise in computational procedures such as meshing, turbulence modeling, and post-processing methods. It could require training, skills, and knowledge to establish and evaluate CFD simulations efficiently.

Wind Tunnel Testing

Wind tunnel testing is done by building a scaled-down model of the wing or a complete race car, which is exposed to a airflow in a controlled environment (Szoke et al., 2021).

Advantages

- a. Physical Replication - Wind tunnel testing gives a physical picture of the flow field, permitting direct observation of the wing behavior and its interaction with the airflow.
- b. Realistic Boundary Conditions - Wind tunnels reproduce specific flow conditions (airflow velocity and temperature) which is critical to study the aerodynamic performance of the wing.
- c. Validation Data - Wind tunnel testing provides benchmark data to validate and calibrate CFD simulations to enhance their reliability and accuracy.

Disadvantages

- a. Cost and Time - Wind tunnel testing could be an expensive technique that needs the construction and instrumentation of physical models. It could take a longer time to obtain results than CFD simulations.

- b. Scale Effects - Scaling down the wing presents scale effects that could impact the result's accuracy. Factors such as Reynolds number scaling and behavior of the boundary layer could vary from the full-scale conditions.
- c. Limited Flexibility - Physical models restrict the ability to discover an extensive range of design variations and flow conditions. Iterations and modifications could be costly and time-consuming.

Analytical Methods

Analytical methods (potential flow theory and panel methods) give simplified mathematical models to predict the aerodynamic behavior of the wing.

Advantages

- a. Quick Estimates - Analytical methods are computationally efficient and will give quick estimations of aerodynamic forces. They are helpful in preliminary design assessments and parameter studies.
- b. Insight into Physics - Analytical methods provide insight into the underlying physics of the flow around the wing and will aid engineers in knowing the factors impacting aerodynamic performance.

Disadvantages

- a. Simple Assumptions - Analytical methods generally make assumptions to simplify mathematical equations, such as the assumption of inviscid and irrotational flow. These assumptions cannot capture the full complexity of the flow physics and will lead to inaccurate results.
- b. Limited Scope - Analytical methods are generally applicable for flow conditions and simple geometries. They are not suitable for highly complex or turbulent flows with intricate wing shapes.
- c. Lack of Detailed Flow Visualization - Analytical techniques give limited visual information concerning the flow field. They cannot capture the separation of flow, boundary layer behavior, or another complex flow process in an accurate way.

When performing laboratory testing to know the fluid flow around the wing prototype and to correlate it with the simulation results, various considerations need to be considered. The following outlines a general approach to performing experiments and obtaining and comparing the data.

Test Setup and Instrumentation

- a. Wind Tunnel - a wind tunnel facility is utilized to offer controlled airflow around the wing prototype. The wind tunnel needs to be capable of generating the required flow velocities and possesses a feasible test section size for wing accommodation.
- b. Orientation and Mounting - Appropriately mount the wing prototype in the wind tunnel to imitate the required operating situations and orientation.
- c. Instrumentation - Installation of proper measurement devices on the wing prototype and in the wind tunnel to get associated data. It may involve hot-wiring anemometers, pressure taps, force sensors, and surface visualization techniques (such as flow visualization paints or tufts).

Experimental Procedure

- a. Calibration - Prior to performing tests, calibrate the measurement devices to make sure accuracy and reliability.
- b. Test Matrix - Design a test matrix for a wide range of appropriate operating conditions, such as flow velocities or different angles of attack, allowing for a comprehensive understanding of the performance of the wing.
- c. Acquisition of Data - Conduct the tests based on the test matrix, gathering data simultaneously from all associated measurement devices. It could involve changing the angle of attack, modifying flow velocities, or other required parameters.
- d. Repeatability - Make sure that the test is repeatable by following standardized procedures and keeping constant operating conditions.

Analysis of Data and Comparison

- a. Data Processing - The processing of experimental data to get key parameters such as drag, lift, and pressure distributions. It involves filtering, averaging, and examining the data to obtain important results.
- b. Comparison Metrics - Define suitable metrics to compare experimental and simulated data. These metrics are lift and drag coefficients, profiles of pressure distribution profiles, or flow visualization images.
- c. Quantitative Analysis - Comparison of numerical values obtained from the experiment and simulation, analyzing the level of agreement or discrepancy. Statistical techniques such as error analysis are applicable for the quantification of differences.
- d. Visualization and Qualitative Analysis - Visualization and comparison of the flow trends and behavior seen in the experimental and simulated results. It helps to identify areas of agreement or discrepancies (flow separation points or turbulence regions).

Correlation and Iteration

- a. Iterative Approach - If discrepancies between the experimental and simulated data are found, iterate the simulation according to the understanding obtained from experimental results. It may involve adjustment of input parameters, refining of mesh resolution, or modification of boundary conditions.
- b. Sensitivity Analysis - Perform sensitivity analyses for the identification of parameters or assumptions having a major influence on the association between simulation and experiment. It helps to refine the simulation model and will get better the agreement with experimental data (Pang et al., 2020).
- c. Documentation - Documentation of the whole experimental procedure such as test setup, measurement procedures, acquired data, and comparison results. This documentation clarifies the correlation and will provide a reference for future work or validation studies.

It is essential to consider the uncertainties and limitations of the experimental and simulation methods throughout the whole process. The uncertainty sources are the accuracy of the instrument, measurement errors, limitations of a wind tunnel, and assumptions in the simulation. Correctly considering these uncertainties and conducting sensitivity analyses improve the reliability and confidence of the correlation between experiment and simulation.

Conclusion

It has also been seen that utilizing CFRP for the race car wing provides advantages such as high strength, lightweight construction, stiffness, and improved aerodynamic performance. Historical developments in aerospace and existing progressions in manufacturing techniques and enhancements of materials have surfaced the path for the application of CFRP in motorsports. Future development's objective is to promote sustainability, the introduction of smart materials, and the optimization of wing designs for improved performance.

Different modeling techniques have advantages and disadvantages to simulate the fluid flow around the wing

- CFD offers flexibility, quantitative data, and detailed flow visualization. However, it needs computational resources, involves assumptions, and will need expertise.
- Wind tunnel testing provides physical replication, validation data, and realistic boundary conditions. However, it could be costly, time-consuming, and prone to scale effects.
- Analytical methods give quick estimates and vision into flow physics. However, they depend on simplified assumptions and could have a limited scope and visual picture.

To get an accurate and comprehensive understanding of the flow around the wing, these modeling techniques are combined and then applied. Wind tunnel testing provides validation data for CFD simulations. However, analytical methods help in preliminary design assessments. The selection of a modeling tool defines the goals, obtainable resources, and required accuracy level for a particular application.

References

- Blocken, B., Stathopoulos, T. and Van Beeck, J.P.A.J., 2016. Pedestrian-level wind conditions around buildings: Review of wind-tunnel and CFD techniques and their accuracy for wind comfort assessment. *Building and Environment*, 100, pp.50-81.
- De Graaff, D.B. and Eaton, J.K., 2000. Reynolds-number scaling of the flat-plate turbulent boundary layer. *Journal of Fluid Mechanics*, 422, pp.319-346.
- Gürgen, S. and Sofuoğlu, M.A., 2019. Experimental investigation on vibration characteristics of shear thickening fluid filled CFRP tubes. *Composite Structures*, 226, p.111236.
- Hosseini, S.M., Vinuesa, R., Schlatter, P., Hanifi, A. and Henningson, D.S., 2016. Direct numerical simulation of the flow around a wing section at moderate Reynolds number. *International Journal of Heat and Fluid Flow*, 61, pp.117-128.
- Khawaja, H.A., Messahel, R., Souli, M., Al-Bahkali, E. and Moatamedi, M., 2017. Fluid solid interaction simulation of CFRP shell structure.
- Kurec, K., Remer, M., Mayer, T., Tudruj, S. and Piechna, J., 2019. Flow control for a car-mounted rear wing. *International Journal of Mechanical Sciences*, 152, pp.384-399.
- Pang, Z., O'Neill, Z., Li, Y. and Niu, F., 2020. The role of sensitivity analysis in the building performance analysis: A critical review. *Energy and Buildings*, 209, p.109659.
- Rodriguez, R.L., Lopes, J.C., Mancini, S.D., de Ângelo Sanchez, L.E., de Almeida Varasquim, F.M.F., Volpato, R.S., de Mello, H.J., de Aguiar, P.R. and Bianchi, E.C., 2019. Contribution for minimization the usage of cutting fluids in CFRP grinding. *The International Journal of Advanced Manufacturing Technology*, 103, pp.487-497.

Szőke, M., Devenport, W.J., Borgoltz, A., Roy, C.J. and Lowe, T., 2021. Advanced boundary simulations of an aeroacoustic and aerodynamic wind tunnel.

Convention: Green text: Comments and Explanations (non-regulatory). (n.d.). Available at:

https://www.fia.com/sites/default/files/fia_2026_formula_1_technical_regulations_public_issue_1_-_2022-08-16.pdf.

Abdalla, M. (2010). Aeroelastic tailoring using lamination parameters. *Structural and Multidisciplinary Optimization*, 637-646.

Duxbury, A. (2023). How fast is an F1 car? Top speeds of F1, IndyCar, MotoGP and more. *Autosport*.