

University of Wales Trinity Saint David
School of Engineering

Designing A Race Car Rear Wing: Fluid
Flow Assessment
Structural & Fluid Analysis
ACMA6010

Lewis Owen Nekrews
2008701
02/07/2023

Summary

The objective of this assignment was to examine the airflow around the rear wing of a race car, discuss the analytical techniques required to determine the wing's behaviour when force is acting on it, address the suggested composite materials to manufacture the wing and the prototype methodology needed to create the rear feature.

The rear wing, tasked to evaluate, was identified as a formula one type design. The importance of the feature was evident in its impact on the performance of a race car and the airflow's behaviour around it through drag and lift forces.

The rear wing was of a dual-element aerofoil which, through research, was considered to be more effective than a single element due to it being able to create more downforce and limited flow separation. Flow separation is the team's primary challenge, and engineers would gladly take the penalty of more drag if it meant delayed laminar flow. However, the teams must abide by regulations as there are limitations to the aerofoil design and the number of elements the rear wing can include.

The task description included the Reynolds number to be estimated with the current geometry given—the wing's cross-section using modelling software. Then after measuring the chord length the Reynolds number was calculated using a chosen velocity. The results were then compared to other race car aerofoils to assess their values and prove that only velocity can be changed to suit different geometry within a limit.

The boundary layer and its importance to the airflow around the wing were proven next. The study suggested that the boundary layer would increase in thickness by the distance between the free-flowing velocity and the surface with zero velocity due to the air viscosity. The geometry of the wing and its angle of attack impact the thickness and determine if the boundary layer is laminar or turbulent, suggesting that a thin boundary layer means less flow separation.

The coursework stated that the material for the prototype was carbon fibre and required the report to discuss the influence this has on the rear wing. The findings suggested that the stiffness-to-weight ratio of carbon fibre makes it a suitable and popular choice for the feature. Its impact on the aerodynamic forces can give a correctly fabricated carbon fibre wing optimum flex to handle cycled loading without too much drag penalty.

While exploring computation fluid dynamics, great potential was found for future development, if implemented correctly. This method could be utilised with the plans of modelling the entire vehicle and not just the feature of interest, a guide to executing the flow simulation provided with the forces acting on the wing discussed. The flow simulation procedure also provided examples of the resulting forces and pressure the airflow has on the wing.

Scaled prototyping was the last area to cover in the coursework. Schematics were designed for the wind tunnel apparatus, which aided in describing the airflow behaviour by including pitot tubes, which were placed behind the model, to read static pressure. The procedure of calculating drag and lift forces using an apparatus was also demonstrated.

The study concluded that through reliable research and methodology, essential data could determine the behaviour of the wing when impacted by airflow. Recent computation methods and experimental technology, with the backing of theoretical data also, can prove effective in developing a new rear wing for a race car. The research showed the importance of rear wing design and how minor variations can significantly impact the race car's performance.

Table of Contents

Summary	1
Table of Contents	3
Introduction	4
Reynolds Number	5
Reynolds Number Results	6
Boundary Layer	7
Carbon Fibre	8
Computational Fluid Analysis	9
Wind Tunnel Testing	11
Testing Wind Tunnel Methods	12
Conclusion	13
Further Work	14
References	15

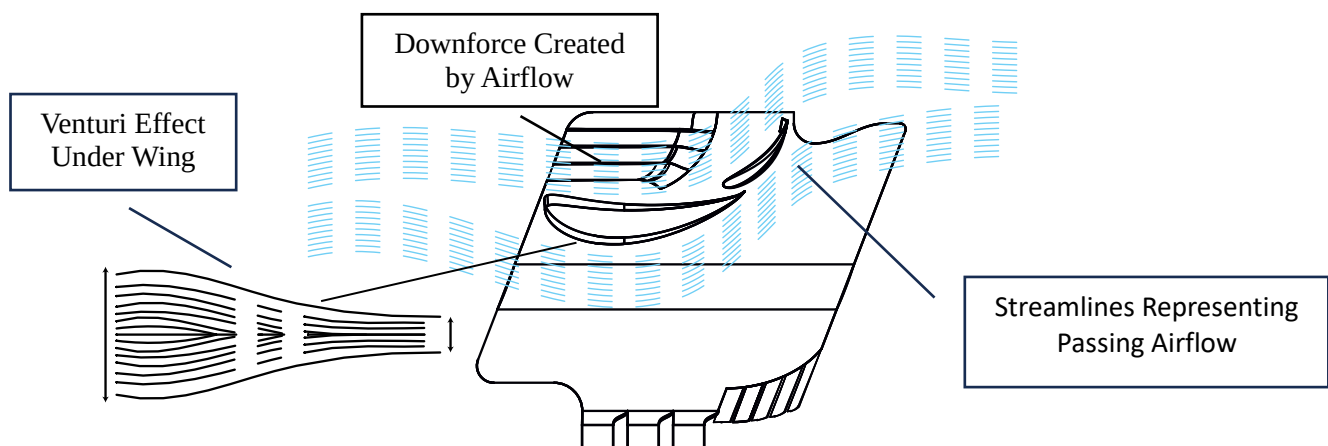
Introduction

A major challenge which faced any automotive racing company is that of how to control the aerodynamic forces around a vehicle. Teams have invested millions in the sports racing industry in order to become competitive, and much of that investment has gone into the study of aerodynamic features. This investment certainly included the study of fluid flow around the rear wing, and with the given model, the problem was how to critically analyse the rear wing structure and fluid flow around it.

As F1chronicle (2021) explains, the rear wing is an inverted aerofoil that can include elements and endplates to give optimum lift and downforce ratio. As the air streamlines hit the aerofoil, it deflects to the top, where the streamlines are smooth, and the bottom airflow squeezes into a venturi effect, as Opong (2022) mentioned. Where the opposite effect occurs, as velocity increases at the bottom side of the wing, the pressure decreases. The pressure at the top of the wing is greater than the bottom, which is how the wing creates downforce. The problem facing teams, when designing a rear wing, is how to balance the cycled loading and create enough downforce. This impacts on the lap time as explained by Smith (1984) as corner speeds are more important than the highest speed of the vehicle travelling straight.

Because there is a downforce, there is also a drag force created. The drag occurs because the air predominantly opposes the very end of the aerofoil, creating resistance and vortexes at the top of the feature. To counter this, the model given in study includes end plates with vaned sections. The added side features lower the pressure by mixing the inside pressure with the outside, decreasing the drag. This dual element wing has thirty-three per cent of an F1 car's downforce with a thousand kilograms produced as stated by Temayne (2004) though more recent regulations would impact on this statistic.

The study's goal was to understand the airflow around the rear wing, and furthermore assume its behaviour when forces act upon its geometrical profile, with backing from trusted sources and previous experiments. The objective was also to increase knowledge of fluid flow characteristics and the theory behind their significance in motorsport.



Reynolds Number

With the given geometry three-dimensional model, the coursework required to examine the theory involving the Reynolds number and why it proved valuable when testing the performance of aerodynamic components. The study was to estimate the value of the turbulent and laminar flow and the location of the transition when air velocity hits the feature. A further requirement was to identify the influence of the dual element with gurney flap attachment as both impact downforce, as long as the angle of attack is sufficient. McBeath (2006) mentioned that eight degrees or larger are required.

According to Barnard (1996), the Reynolds number is a ratio, and its fixed values are air density and viscosity, while the number is entirely independent of its size and speed alone in car aerodynamics. He continues to state that if the Reynolds number is to increase by its speed, then the boundary layer becomes thinner and is used to determine the flow the wing experiences.

It determined that the rear wing was to be a prototype and must be able to perform aerodynamically for the purpose. To save time and cost, scaling down the model can be considered, as ideally, the air flow would behave the same according to the Reynolds number formula. If the Reynolds number is the same for both full-scale and half-scale, the value would remain the same if the velocity value is adjusted to compensate. However, increasing the velocity would have a set limit. Schultz (2016) explained an alternative for scaled wind tunnel experiments, when tackling this issue by reducing the velocity instead and using a different flow medium or utilising the dependency of kinematic viscosity.

The goal for the race team engineers is to reduce car instability while also creating a sufficient drag / lift ratio. According to Yakkundi (2011), the Reynolds number should be laminar, in streamlined format, and turbulence must be limited as it produces more drag force.

The calculations estimated were on the rear wings model given and other vehicle aerofoils with sufficient data researched. The subject of study was a dual-element wing profile with a slotted side plate. For the study, chord length and angle of attack were calculated, as Benzing (2012) studied in his book, *Ali Wings*.

The results from the study were then compared to other Reynolds numbers recorded by McBeath (2006), who mentioned the value being greater for dual-element wings than for single. Further research compared Katz (2006) data with the prototyped cars and, also the original aerofoil from the Ferrari F2002 model drawn initially by Majmader (2019) on GrabCad (2019). These values were evaluated to show the different geometry's impact and prove that velocity can be changed even to create the same value Reynolds number.

Reynolds Number Results

Reynolds Number Results at 44.7 m/s, 15°C						
Air foil Name	Element Amount	Angle of Attack(deg)	Chord Length (m)	Calculations	Result	Test No.
Airfoil Given in Coursework	Dual	20.53	0.446914	$\frac{1.225 \times 44.7 \times 0.44691}{1.4776 \times 10^{-5}}$	1656177	1
Ferrari F2002 by Majimader	Dual	26.16	0.31097	$\frac{1.225 \times 44.7 \times 0.31}{1.4776 \times 10^{-5}}$	1148811	2
Toyoat Egle analysed by Katz	Dual	20.44	0.51054	$\frac{1.225 \times 44.7 \times 0.51}{1.4776 \times 10^{-5}}$	1889979	3
NCA 4412 by Mcbeath	Single	12	0.4	$\frac{1.225 \times 44.7 \times 0.4}{1.4776 \times 10^{-5}}$	1482336	4

Table 1 Reynolds Value Comparison Speed & Temperature

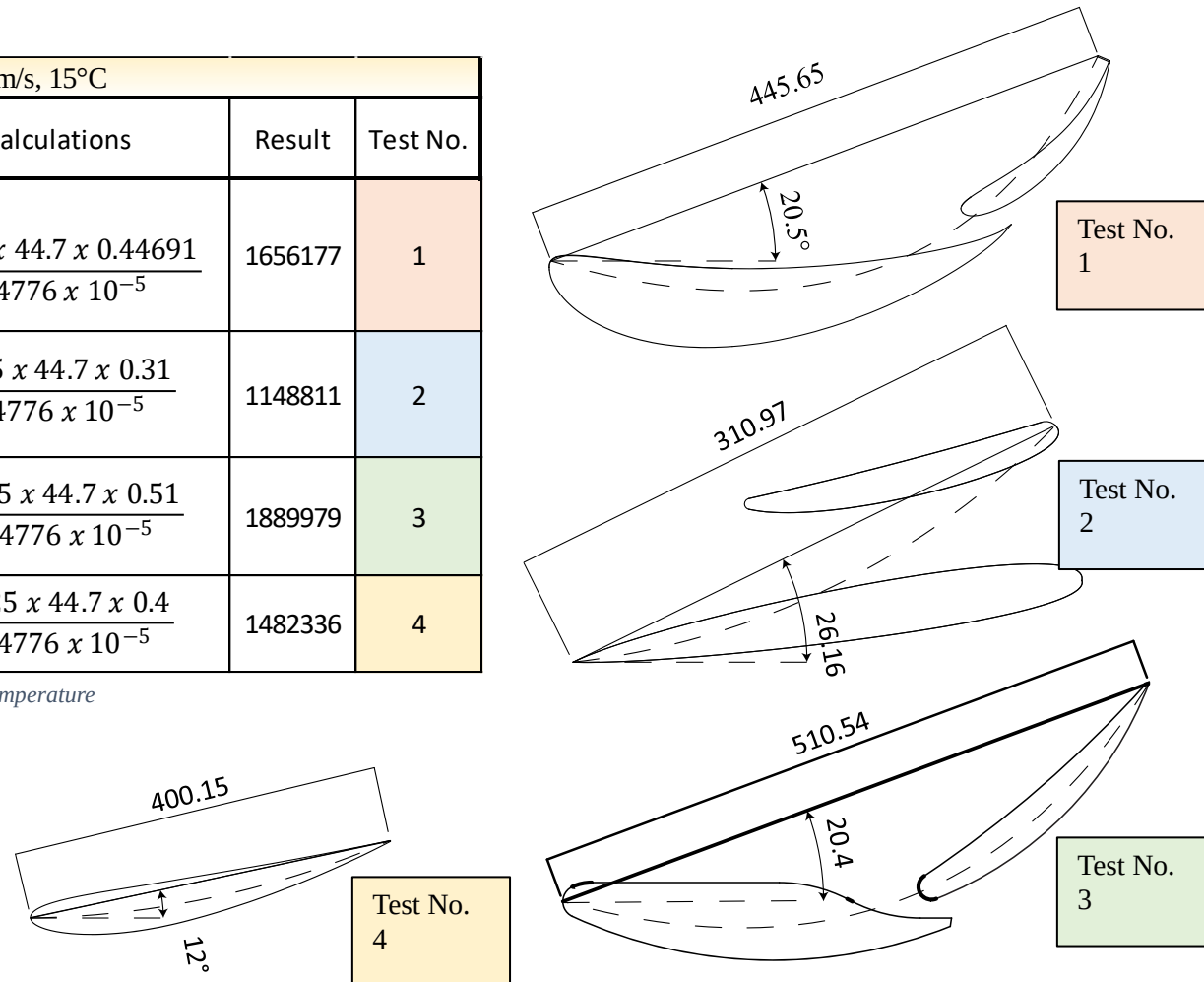


Figure 2 Airfoils of Study and Chord lengths Displayed

Boundary Layer

The boundary layer and its relevance to the rear wing were examined. An important consideration is that the velocity moving over the wing is not all equal value because of the fluid viscosity. The velocity on the wings surface is zero, called the no slip condition. As the velocity gradually increases, the further its distance away from the surface where it is called the free stream velocity. This change is the rear wings boundary layer. In the book *Aerodynamics in Race Cars*, Joseph Katz mentioned that the impact of the boundary layer being too thick for rear wings could even result in additional drag and loss in downforce. Katz (2006)

The layer can be either laminar flow or turbulent flow. Initially flow would be laminar, and if the air were to move in another direction, it would become turbulent. The region of change from laminar to turbulent is called the transition stage, where the energy loss in the boundary layer is more significant due to friction. Katz (2006) again explained this as being named the skin friction coefficient and is much greater in turbulent boundary layers. It is important to note that it is better to accept the frictional drag penalty in race aerodynamics rather than have flow separation, which would lead to a loss in downforce.

Any wing geometric properties can suggest an increase in the boundary layer thickness and must be controlled to delay the increased laminar flow for minimum separation. Dual elements, such as the one of interest, are efficient at keeping the pressure at the top of the wing for the downforce to be created and reducing the flow separation, as racecarengineering.com (2023) explained. The boundary layer thickness is also dependent on the angle of attack. As it increases, the flow direction is slower, especially near the wall and can even reverse the direction. Hucho (1986).

Race teams value the importance of studying the thickness of the boundary layer as it directly influences surface friction and flow separation around the wing. The turbulence that high wind speed creates influences the layer thickness. Barnard (1996) explained that it moves the transition between laminar and turbulent forward, impacting wing performance.

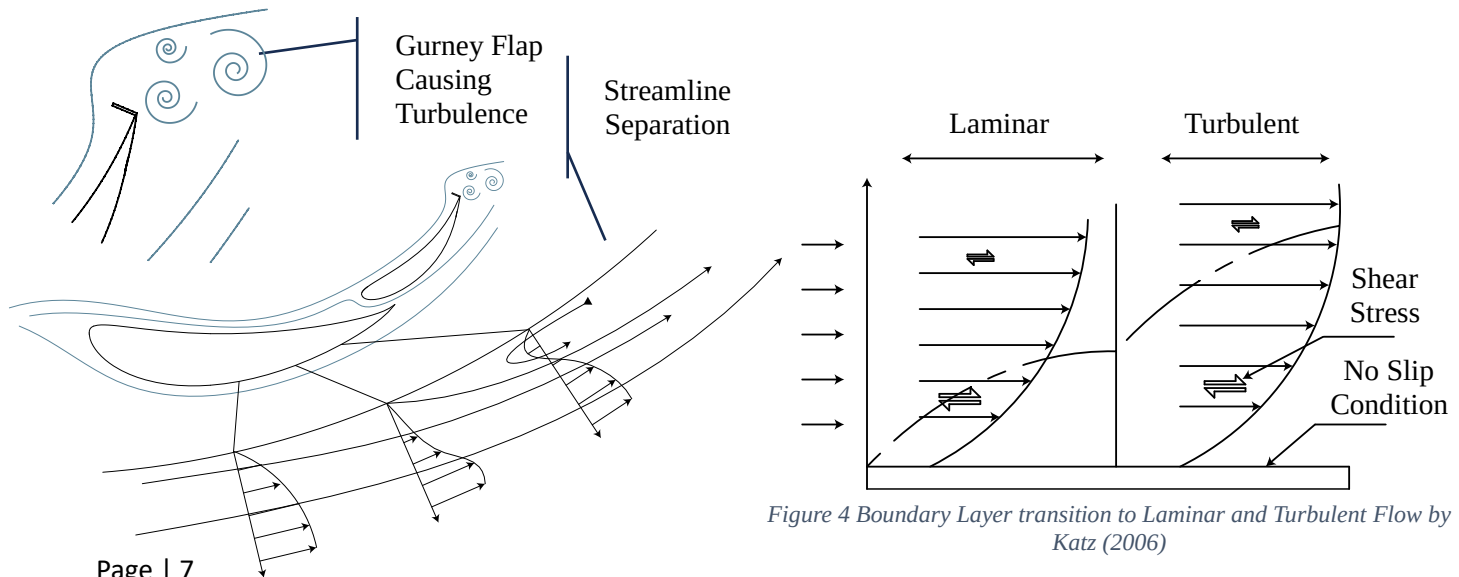


Figure 3 Flow Separation under rear wing of study as Described by Hucho (1987)

Figure 4 Boundary Layer transition to Laminar and Turbulent Flow by Katz (2006)

Carbon Fibre

In the task description it specified that carbon fibre was to be used for the rear wing. Utilising this material would have been an obvious choice considering it is favoured by most motorsports organizations, such as formula one, according to Mallick (2007). It is well known that the high modulus-to-density ratio of carbon fibre gives teams the benefits of not adding weight to the car whilst still being able to deal with high loading resistance caused by aerodynamic forces.

The carbon fibre process involves spinning the fibre and weaving it into shape. The most common pattern weaved is a 1x1 pattern and, according to the Estcarbon.com (2023) is the tightest pattern and most straightforward to manufacture. Another material which can be combined with carbon fibre is the aluminium honeycomb mesh, for Mcbeath (2016) mentions that the side plates are made from this to hold the structure in place. He also discusses that a degree of flex in the wing's materials is required for aerodynamic performance.

Once the layers of fibres are created, they are impregnated with resin, placed on the required mould, prebuilt in the rear profile, and left to settle in a temperature control environment. Machined aluminium inserts are commonly included at the stress point of the rear wing to support the carbon fibre, for example, in the Caterham CT05 f1 bolted wing assembly by Bebb (2014), which included similar inserts for supporting the wing's frame.

In formula one, regulations change annually, especially in the aerodynamic features of the race car. In 2019 major changes came to the rear wing to attempt to improve on-track spectacle, according to source Tyson (2018) whose article detailed that these changes may result in speed penalties. In regard to material regulation, Fia.com (2022) states that material must be commercially available, and any new composites must be proposed for approval.

Significant improvements have been developed over time regarding how the performance of the rear wing is observed to be affected by stiffness and weight. Jim Hall introduced the chaparral 2E with the aerofoil on high struts. Kimble (2016) explained that the innovation of the rear wing was still clearly underdeveloped and took time to understand the dynamic forces acting on the wing entirely.

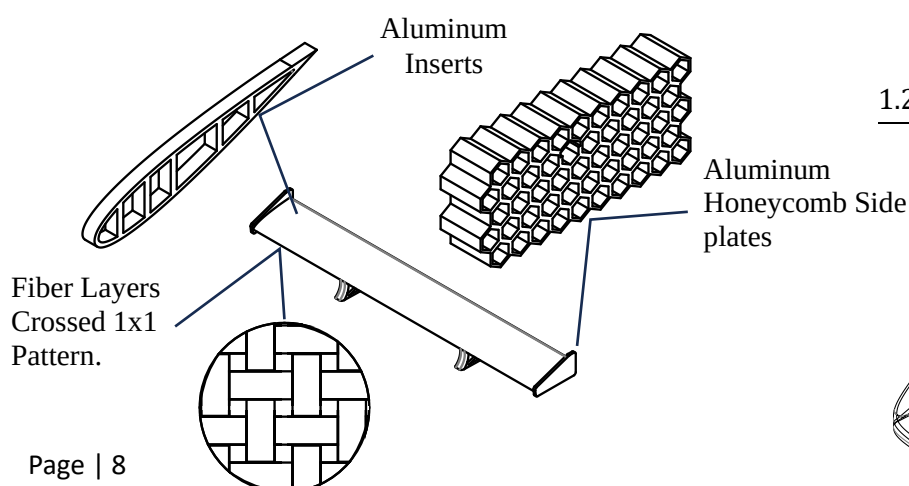


Figure 5 Material Components in rear Wing Design based on illustrations by wrcracing.com (2019)

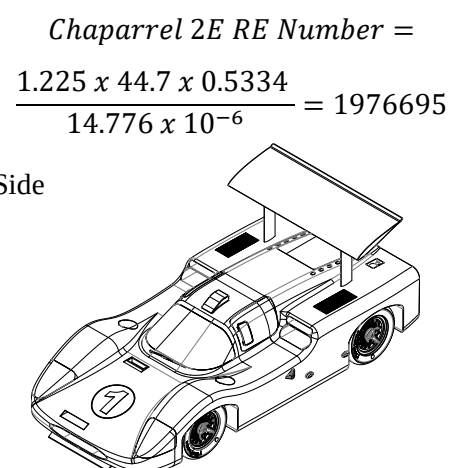


Figure 6 Aluminium Air Foil NACA 0009 12° Nose Down, Milliken (1995)

Computational Fluid Analysis

Analysing the flow behaviour through governing equations such as the continuity equation, which is the study of the differential volume of the flow and with Navier Stokes equations, can prove too complex a method of finding the flow behaviour around an object. They are non-linear, second-order derivative equations.

The basic concept of computation fluid dynamic or CFD directs the calculations into smaller, more manageable volume fields called a mesh. With this method, the conservation of mass and moment is written for each small volume. With the Navier-Stokes formula, we can get a discontinuous function of velocity and pressure, according to Nayler (2023). However, he also mentioned that a more refined volume could create the solution, which is the exact value of the Navier Stokes equations. These are then reformed into algebraic equations, which are solved simultaneously.

A preferable modelling and simulation method would be through Ansys flow simulation (Ansys 2023). This software is described as an industrial standard for flow simulation and is used by many race teams in motorsport. A full-scale model should include the vehicle to which the rear wing must be assembled. The car body was based on the Ferrari 2002, modelled by Majmader(2019) on Grabcad (2023), which had suitable geometry to attach the rear wing given.

The development of CFD over recent generations has made it a more reliable and valuable tool when comparing data further into the project. Even Barnard (1996) predicted this in the book Road Vehicle Aerodynamic Design before CFD had a positive reputation. He stated that CFD will become cheaper and more reliable but suggested that it is unlikely teams will substitute wind tunnel methods for CFD but instead use them to complement each other.

CFD gives engineers the freedom of experimentation with little significant cost, especially in the design area of development. Teams are always looking at new methods for improving features' aerodynamic behaviour, such as the rear wing, which has a rapid turnaround, as mentioned by Mueller (2015)

The accuracy of the results could be disputed due to possible numerical error. An example could be the grid created around a component and its shape. Barnard (1996) stated that one of the significant issues is the accuracy of grid generation, which may not consider the more minor details of the design, to make sense of the simulation constructed.

A procedure was designed to describe how a simulation technique would be addressed and which measurements of interest were to be included. A flow simulation on Solidworks (2023) was implemented to show what data can be extracted when the designed procedure is ongoing.

Velocity: 44.7 m/s Temperature: 15°C & Scale :1:1 Flow Type :Laminar & Turbulent Density: 1.2 kg/m³ & Pressure : 0.103MPa Vehicle : Ferrari F2002 with Attached Rear Wing	Simulation Procedure Based on Methods used by Air shaper (2023)	
Task	Task Description	Results
Average Forces & Friction Forces	Simulate the integration of air pushing and pulling in the perpendicular axis of the wing and show the constant forces evaluated due to the wing's position. Show the friction forces also acting across the surface of the wing. The two details would then be displayed and listed in plot points.	(N)
Plotting Surface Pressure	Identify where the has the highest pressure and determine whether the wing is correctly designed geometrically and has the correct angle of attack. Identify the stagnation points on the pressure simulation and where the most air contact is on top or bottom.	(Psi)
Visualisation: Plotting Lined Velocity Flow & friction Lined Flow	Horizontal displacement of velocity streams shows the flow pattern and identifies any turbulent flow or vortexes. Show Friction drag in lined format, and identity any detachment around the lng or anything that may cause turbulence.	m/s & (N)
Drag Coefficient and Aerodynamic Power	Calculate the two types of air drag from pressure and friction that display restricted again to the wing, and calculate the Power needed to maintain a constant speed with the inclusion of the rear wing to the race car.	Unitless / Watts
Assumptions in Study = Accurate mesh model including all external features, Constant Velocity, Temperature, Fluid Viscosity		

Table 2 Flow Simulation Procedure Based on Methods Researched from AirShaper.com (2023)

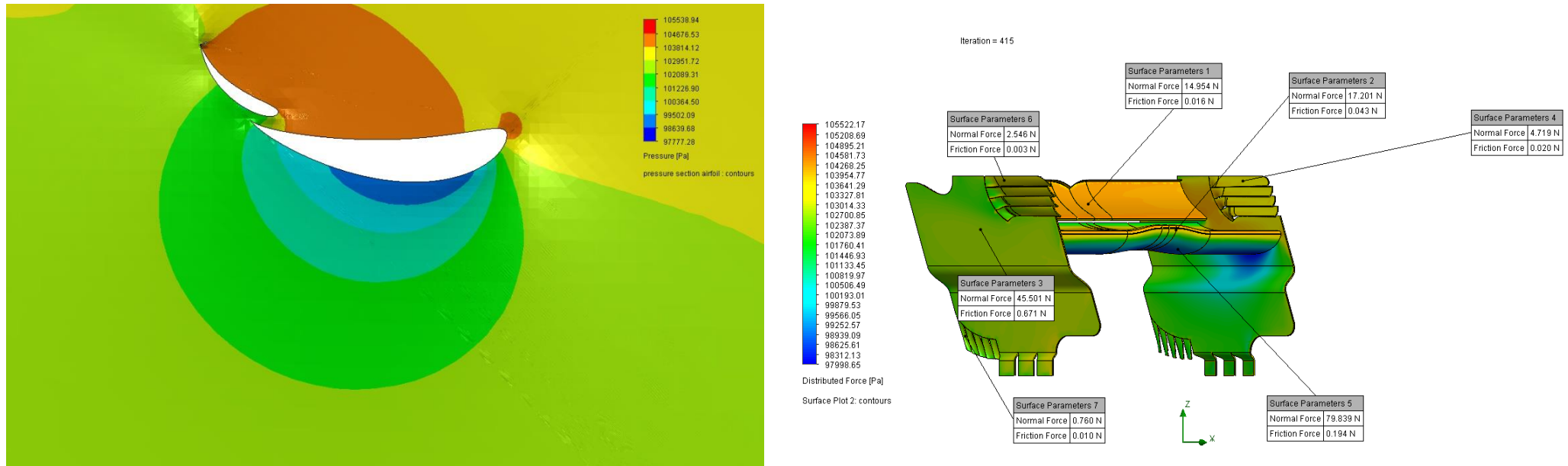


Figure 7 Flow Simulation Results Including Pressure Distribution & Normal Force and Frictional Force Data

Wind Tunnel Testing

Open return tunnel testing would be the next step in the study. It was based on methods of a similar concept Hiro Fujimori discussed in an interview about the design of the wind tunnel for the Toyota Eagle MkIII (mulsancorner.com). Classification of the wind tunnel design is a subsonic design, meaning its Mach number is <1 as velocity decrease and pressure increases.

The idea of using a whole car instead of just the rear wing was based on similar justification as the CFD analysis, that of being able to have realistic data when the vehicle is on track. Also, the model was scaled down to ten per cent for practicality purposes when fitting it onto an academic lab facility. However, Fujimori (2019) used twenty per cent scaled models for the Eagles development.

Evaluating the stagnated and static pressure was required at the rear end of the race vehicle. This was done using a pressure probe for the stagnating pressure and a pitot tube tapped behind the rear wing for the static pressure. A similar method used on the VDAS AFI600S, an acquisition kit by Tecquipment.com (2023), who create a Subsonic wind tunnel for academia. The fundamental equation for flow measurements in wind tunnels is stated by Colman (2011), who suggested Bernoulli's equation for the pressure at the same point. However, this led to our first assumption of neglecting viscosity.

Lift and drag are then examined through methods based on Michigan State University www.engin.umich.edu (2023). The method describes placing scales in the wind tunnel, putting the model on one scale and a weight / pulley on the other. The model and weight are attached and when the wind tunnel activates, the drag and lift forces are calculated. Finally, these would be validated by the lift and drag coefficients, where the area would have to consider the entire model. An assumption during this test is that the neglect of the friction forces acting on the wing and area would be estimated due to the complex geometry.

The datasheet in the TECequipment.com (2023) for the subsonic wind tunnel included a smoke-generating device. This method was used to observe the flow behaviour around the rear wing. This, and with a camera linked to the computer, would be able to view the flow from a side view around the model. This method of comparing methodological data and simulation data uses the same velocity, however, due to the scaling down method, calculating the Reynolds number would mean the velocity would have to be increased substantially to be accurate, as mentioned previously. Pressure could, however, be compared, as proven by Morden (2010), whose comparison study on a high-speed train showed a similar value in surface pressure despite the scaling.

The coefficient forces, however, showed to be different. This is because confining a scaled model in a smaller area would increase flow around the model and cause greater lift and drag readings according to Katz (2006). Another consideration would be that some components in a vehicle cannot be scaled down in size.

Testing Wind Tunnel Methods

Pressure Testing Procedure	
Task 1	Adjust the Model in a suitable position where the probe and pitot tube are above and behind,
Task 2	Calculate the density of the air inside the wind tunnel noble equation. $PV = RT$
Task 3	Use Instrumentation attached to the wind tunnel to calculate the difference in $p_0 - p$
Task 4	Solve for V using Bernoulli's equation $V = \sqrt{\frac{2(p_0 - p)}{\rho}}$

Table 3 Procedure For Pressure Testing Layout based on Colman & Libii, N, J (2011)

Lift And Draft Forces	
Task 1	Remove the Mounting from the pressure experiment and attach the string to the model.
Task 2	Place Drag and Lift Assembly into the wind tunnel, including scales, weight with still attached and free moving model
Task 3	Activate the wind tunnel to exert the same force as previously stated.
Task 4	Calculate the disputable force on the model using aquation data and compare it to the ceofficat equations $C_L = \frac{F_L}{\frac{1}{2}\rho V^2 A}$ $C_D = \frac{F_D}{\frac{1}{2}\rho V^2 A}$

Table 4 Procedure for Lift and Drag Layout based on Smith, W M & engin.umich.edu (2023)

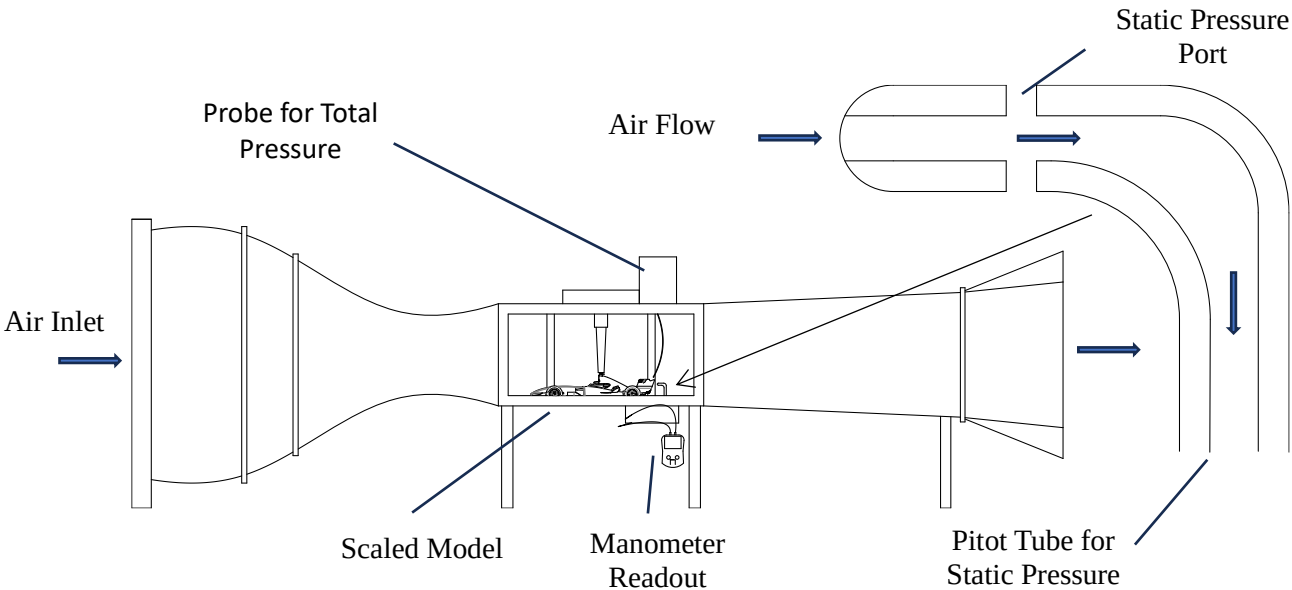


Figure 8 Pressure Testing Procedure with Open Loop Wind Tunnel Desing Based on tequipment.com (2023)

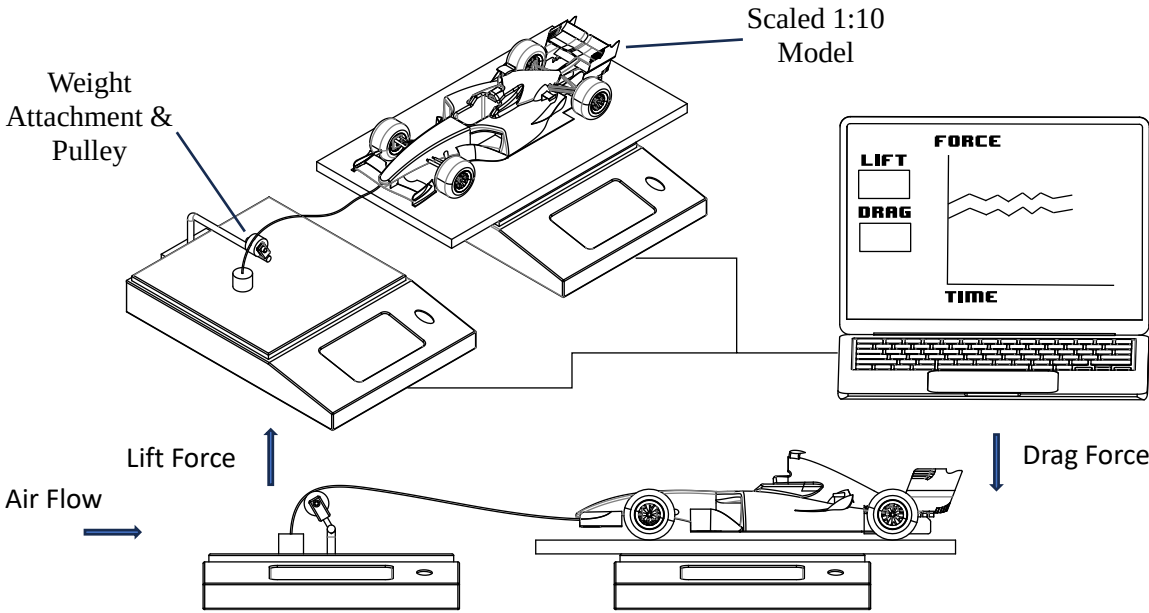


Figure 9 Lift and Drag Procedure with Open Loop Wind Tunnel base on methods by www.engin.umich.edu (2023)

Conclusion

The report discussed the design of a race car's rear wing, which could then be used to compete in a race car series. The task involved calculating the Reynolds number of the fluid flowing around the wing. The boundary layer was also discussed regarding how it relates to the turbulence and its experiences with forces against it.

Additionally, the report was to explore and discuss reasons why the wing was designed in the fashion illustrated and provide implications in performance for using a composite material like carbon fibre. Another requisite was to examine the historical differences between the material present and when the rear wing was first introduced to motorsport racing.

Study testing methods when analysing the airflow on a prototype model behaviour around the wing, included computation and experimental methods.

This study was important to gain further in-depth knowledge on the behaviour of fluid flow around an object and consider the influences that change the rear wing performance. An acknowledgement was made regarding the correct procedure and technique used in motorsport that assist in understanding fluid flow characteristics.

Research proved that the geometry and position of the wing on a high-performance vehicle need both analytical and experimental data for it to reach expectations competitively. The rear wing was designed like this to influence the fluid flow direction and manipulate it to increase its performance, like the implementation of the endplate and vanes.

In conclusion, it could be said that the rear wing of a race vehicle is an aerodynamic feature that is greatly affected by its geometric profile and environment. The fluid flow around the rear wing is influenced by mainly these two areas and, after research, require a great many considerations in order to achieve the desired results in performance.

Further Work

Further work would be to use more professional methods of experimental studies, such as full-scale modelling, providing there are no restrictions in the regulations. Full-scale wind tunnel testing, in a closed circuit, is predicted to give exceptional validation to CFD data. The idea behind a closed-circuit wind tunnel is that it has continuous air flow and can be considered more efficient.

The final stage of development is when road testing must be applied. However, research through known aerodynamic vehicle textbooks has fundamentally based most testing on computation and wind tunnel methodology, because more data is available.

However, dynamic pressure and lift and drag coefficient can be recorded during road testing. Teams usually use pitot tubes to measure the velocity acting on the vehicle. They are normally arranged in aero rakes, which the aerodynamic team can process, that come from the rakes of the rear wing when it is positioned behind it, according to source Smedley (2019), a consultant at F1.

Closed Return Wind Tunnel Design Based on Beazley (2018) Schematic from Science of Super Cars

1. Fan Blades
2. Fan Motor (electric)
3. Modified Ferrari 2002 With Installed Wing Studied
4. Vehicle Entrance Ramp
5. Rolling Road Test Section
6. Contraction 4:1
7. Turning Vanes

Table 4 Components Associated with Closed Wind Tunnels Listed Beazley (2018)

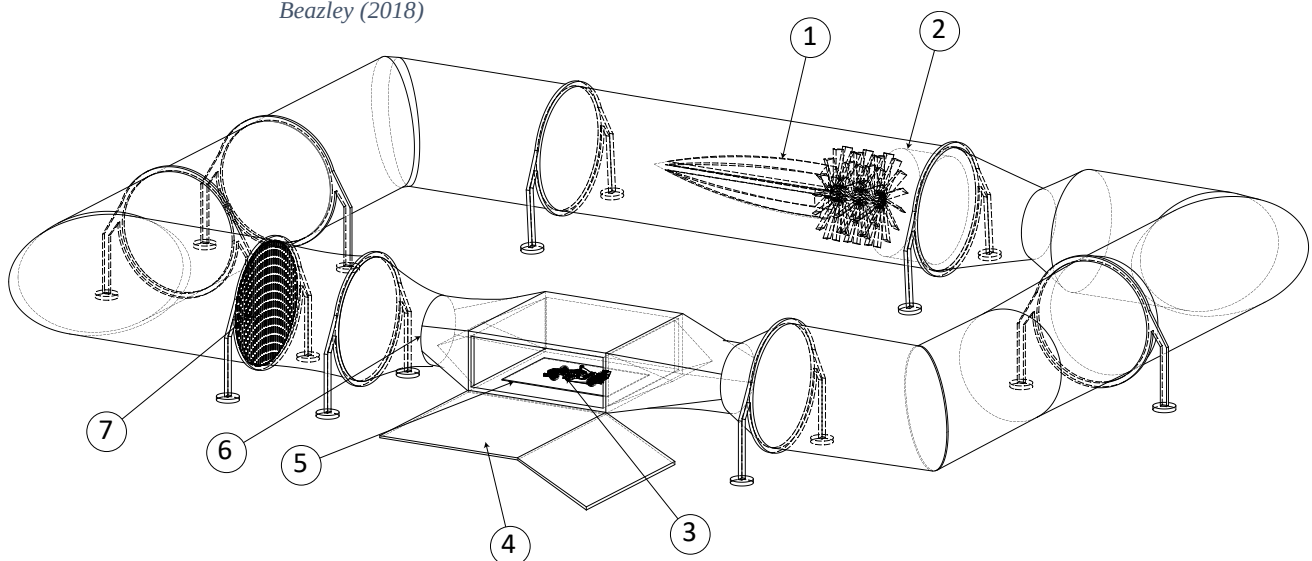


Figure 13 Full scale wind tunnel testing Design based on Previously Created by Beazley (2018)

References

- Airshaper (2023) Aerodynamic Processes for Simulation [Online] [AirShaper - Aerodynamics Made Easy](#)
- Ansys(2023) Simulation software for Aerodynamic Air Flow, available at : [Get in Touch | Ansys](#)
- Barnard, R, H (1996) Road Vehicle Aerodynamics Design, London, Longman Publishing
- Beazley, M (2018) The Science of Supercars, London, Octopus Publishing Group Ltd
- Benzing, E (2012) Ali Wings – Their Design and Applications to Racing Cars, Milan, Giorgio Nada Editore s.r.l
- Colman, J (2011) Wind Tunnels and Experimental Fluid Dynamics, Rejeka, ResearchIntechOpen
- easycomposites (2023) Nomex Aramid Honeycomb [Online]
<https://www.easycomposites.co.uk/nomex-aramid-honeycomb> Accessed on 21th June 2023
- easycomposites (2023) How to Make Carbon Fibre Rear Wing Mouldless Method, [Online] [How to Make a Carbon Fibre Rear Wing, Mouldless Method - Easy Composites](#) accessed on 21th June 2023
- engineeringart (2015) Caterham CT05 F1 Car RR Wing Bolted Assy by Charlie Bebb [Online]
[Caterham CT05 F1 Car Rear Wing Bolted Assembly \(2014\) \(engineeringart.uk.com\)](#) Accessed on 22nd June 2023
- estcarbon (2023) Carbon Fiber all Patterns Explained, [Online]
<https://www.bing.com/search?q=estcarbon+fibr+patterns&cvid=334c0b6672474d0cab0035fd1a45a001&aqs=edge..69i57.4591j0j4&FORM=ANAB01&PC=DCTS> Accessed on: 20th June 2023
- F1Technical (2023) Ferrari F2002 Specifications [Online] [Ferrari F2002 - F1technical.net](#) Accessed on 28th June 2023
- FIA, (2022) Formula 1 Technical Regulations, [Online] [2022 formula 1 technical regulations - iss 6 - 2021-09-13 0.pdf \(fia.com\)](#), Accessed June 16th 2023
- GrabCad (2019) Ferrari F2002 Full Scale Model by Fawaz Bukht Majmader [Online] [F1 Car CFD Simulation | 3D CAD Model Library | GrabCAD](#) Accessed on 12 June 2023
- Hucho, W, H (1987) Aerodynamics Of Road Vehicles, London, Butterworths Publishers
- Illuis555, (2023) Aerodynamics of the new Rally 3 Contenders, [Online] [The aerodynamics of the new Rally3 contenders – WRCWings](#), Accessed June 8th 2023
- Katz, J (2006) Race Car Aerodynamics – Designing for Speed, (Second Edition) Cambridge, Bentley Publishers
- Kimble. D (2016) The Chaparral 2E Was A guiding Light In The Race Car Aerodynamics [Online]
[The Chaparral 2E Was A Guiding Light in Race Car Aerodynamics \(motortrend.com\)](#), Accessed on 22nd June 2023
- Libii, N, J (2011) Wind Tunnel in Engineering Education by Indiana University- Purdue University For Wayne [Online] [47219455.pdf \(core.ac.uk\)](#) Accessed on 26th June 2023
- Majmader, F,B. (2023) Mechanical engineering Graduate – Model Designer For Ferrari F2002 [Online] [\(3\) Fawaz Bukht Majmader | LinkedIn](#)

Mallick, PK. (2007) Fiber-Reinforced Composites. (Third Edition), London, CRC Press

Media, (2021), How do Formula 1 Rear Wings Work ?, [Online] [How Do Formula 1 Rear Wings Work? | F1 Rear Wing Aerodynamics \(f1chronicle.com\)](#), Accessed 2023, June 1st

Mcbeath, S (2011) Competition Car Aerodynamics, (Second Edition), Yeovil, Haynes Publishing

Mcbeath, S (2016) Competition Car Composites, (Second Edition), Dorchester, Veloce Publishing

Milliken, W. F. (1995) Race Car Vehicle Dynamics, Warrendale, SAE International

Motorsport.com (2020) The Small Wing Detail That Shows Mercedes Is still Pushing [Online] [The small wing detail that shows Mercedes is still pushing \(motorsport.com\)](#), Accessed on 22nd June 2023

Mueller, J.D. (2015) Essentials of Computational Fluid Dynamics, London CRC Press

Mulsannercorner(2019) Interview with Hiro Fujimori on the Toyota Eagle MkIII, {Online] [Mulsanne's Corner: WFO; Ward, FUJIMORI, and others \(mulsannescorner.com\)](#)

Murad, J (2023) Computational Fluid Dynamics - Magnus Effect [Online] [Articles \(jousefmurad.com\)](#) Accessed on 25th June 2023

Naylor, D (2023) A General Introduction To Fluid Mechanics And Its Applications [Online] [Chapter 1 Intro to Fluid Mechanics - ONLINE FLUID MECHANICS, Professor D. Naylor \(drdavidnaylor.net\)](#) accessed on June 23rd 2023

Oberkampf, L, W (2002). Verification And Validation in Competition Fluid Dynamics – University of Southampton [Online] [version.2-23-02.part1 \(southampton.ac.uk\)](#) Accessed on 29th of June 2023

Opong, R. (2019), What Are Venturi Tunnels In F1? (Venturi Effect Explained), [Online] [What Are Venturi Tunnels in F1? \(Venturi Effect Explained\) - FLOW RACERS](#), Accessed 2023, June, 1st

Schuetz, T. (2016) Aerodynamic of Road Vehicles, (Fifth Edition), Warrendale, SAE International

Smith, C. (2010), Engineer to Win by Carroll Smith, London, Bloomsbury Publishing PLC

Smith, W, M.- University of Michigan College of Engineering Drag & Lift Experiment Lecture Demonstration Lab [Online] [Michigan Engineering - University of Michigan College of Engineering \(umich.edu\)](#) Accessed on 26th June 2023

Solidworks 2023, Flow Simulation, Available at : [SOLIDWORKS Flow Simulation | SOLIDWORKS](#)

Talukder, Md. M. M. (2019) Study and Analysis of Aerodynamic Properties of a Car Model, Saarbrücken, Lap Lambert Academic Publishing

Tequipment.Com (2023) Data Sheet Subsonic Wind Tunnel 600mm [Online] [AF1600S-Subsonic-Wind-Tunnel-Datasheet-0623.pdf](#) Accessed on 27th June 2023

Tequipment.com (2023) Smoke Generator (AFIA11) [Online] [SMOKE GENERATOR | TecQuipment](#) Accessed on 27th June 2023

Tequipment.com (2023) Subsonic Wind Tunnel 600mm (AF1600S) [Online] [SUBSONIC WIND TUNNEL 600mm | TecQuipment](#) Accessed on 27th June 2023

Tremayne, D. (2006), The Science Of Formula 1 Desing: Expert Analysis of the Anatomy of the Morden Grand Prix Car (Second Edition) , Yeovil, J H Haynes & Co Ltd

Tyson, W (2018) Technical: How Brawn's F1 2019 Rule Changes Aim to Spice Up Racing [Online] <https://www.crash.net/f1/feature/911642/1/how-f1s-2019-rule-changes-aim-spice-racing> , Accessed on 22nd June 2023

Weingart, R (2015) on Track Testing Validation Method of Computation Fluid Dynamic Simulation of A formula SAE Vehicle [Online] [Weingart_ku_0099M_13860_DATA_1.pdf](#) Accessed on 28th June 2023

Yakkundi, V (2011) Aerodynamics of Cars – And Experimental Investigations of the Synergy of wind tunnels and CFD, Saarbrücken, Lap Lamper Academic Publishing