

University of Wales Trinity Saint David

School of Engineering

Designing a Race Car Rear Wing

Structural and Fluid Analysis – ACMA6010

Name: Adam Japp

Student No: 2011764

Date: 02/07/2023

**Word count (Excluding cover page, contents page, Table of
Figures, Figures and Reference list) – 3,406.**

Contents

Table of Figures	3
1. Introduction	4
2. Reynolds Number	4
2.1. What is Reynolds Number	4
2.2. Reynolds Number Importance	4
2.3. Reynolds Number Calculation	6
3. Boundary Layer	9
3.1. Boundary Flow Discussion.....	9
3.2. Boundary Flow, Separation and Turbulence Around a Wing	9
3.3. Relationship to Separation and Turbulence Around Wing	11
3.3.1. Separation of Turbulent Boundary Layers	11
3.3.2. Separation Bubble	12
4. Car Wing Design	13
4.1. Implications of Using CFRP (Carbon Fiber Reinforced Plastic).....	13
4.2. Historical Developments	14
4.3. Current Developments	15
4.3.1. New Design Elements	15
4.4. Future Developments.....	16
5. Modelling Techniques.....	17
5.1. CFD (Computational Fluid Dynamics) and FEA (Finite Element Analysis)	17
5.1.1. Advantages of CFD.....	18
5.1.2. Disadvantages of CFD.....	19
6. Experiment Application.....	19
Conclusion	21
References.....	22

Table of Figures

Figure 1 Osborne Reynolds Flow Graph (Power, 2023).....	5
Figure 2 External Reynolds Number Example (Djojodihardjo, 2017)	6
Figure 3 Reynolds Number Calculation over Flat Plate	6
Figure 4 Laminar to Turbulent flow diagram (Engineering, 2016)	7
Figure 5 Reynolds Number Examples (Coffman, 2019)	8
Figure 6 Tested Aerofoil Geometries (Coffman, 2019)	8
Figure 7 Velocity Profile Diagram (Tec-Science, 2020)	9
Figure 8 Hydrodynamic Boundary Layer and Wing Profile (Tec-Science, 2020).....	10
Figure 9 Boundary layer separation on a blunt body (Tec-Science, 2020)	11
Figure 10 Velocity profile in a laminar and turbulent boundary layer (Tec-Science, 2020) ...	11
Figure 11 Separation Bubble (Tec-Science, 2020)	12
Figure 12 Laminar separation bubble on an aircraft wing (Tec-Science, 2020).....	13
Figure 13 2021 Rear Wing vs 2022 Rear Wing (Reynolds, 2021)	15
Figure 14 Variations of Current Rear Wings (motorsportmagazine, 2023)	16
Figure 15 New sliding scale on aerodynamic testing in F1 (Hughes, 2022)	17
Figure 16 Rear Car Wing With Use Of CFD Analysis (Pangerl, 2018)	18
Figure 17 Wind Tunnel Car Testing (Hughes, 2022)	20

1. Introduction

Car wings have seen change; however, the general principle remains unchanged. Downward force ensures vehicle remains in contact with the ground to maintain friction/grip.

This report reviews fluid flow around an F1 rear wing and discusses past, present, and future concepts. Discussion around Reynolds Number and the effects on the boundary layer, such as boundary flow, separation, and what can be done to improve Reynolds Number to achieve desired flow. Discussion around CFRP (Carbon Fibre Reinforced Plastic), followed by CFD (Computational Fluid Dynamics) Analysis discussion and how this is used within the aerodynamic industry. Advantages and disadvantages of CFD, finishing with experimental work being performed, discussing how all these techniques can be implemented into designing a car wing.

2. Reynolds Number

2.1. What is Reynolds Number

Reynolds number was discovered in 1883 by Osborne Reynolds. Reynolds number is the ratio of inertial forces to viscous forces. It is a dimensionless number used to categorise the fluids systems in which the effect of viscosity is important in controlling the velocities or flow pattern of a fluid. All parameters are given in the same units, meaning then when multiplied together they cancel each other out, resulting in Re being unitless.

Reynolds number determines whether a fluid has laminar, or turbulent flow. Based on the API 13D recommendations, it is assumed that a Reynolds number less than or equal to 2100 indicates laminar flow, and greater than 2100 indicates turbulent flow. (Rehm, 2008)

2.2. Reynolds Number Importance

Reynolds number is useful when trying to determine a type of flow for a specific application, for example. When flushing out pipework, turbulent flow is required, usually requiring a Reynolds number at or greater than 4,000 to ensure correct flushing, reducing contamination within a system. **Figure 1** highlights this.

Another example would be designing of a car rear wing. The boundary layer is trying to maintain for as long as possible, maintaining maximum downforce while the vehicle is in motion. Laminar flow is the requirement.

Laminar flow - If Reynolds number is **less than 2000**, it is laminar, in internal scenarios, this value is much greater in external flows (**500,000** or more), highlighted in **figure 2**. The accepted transition Reynolds number for flow in a circular pipe is $Re_{d,crit} = 2300$.

Transitional flow - **between 2000 and 4000**, flow is unstable due to onset of turbulence. Sometimes referred to as transitional flows.

Turbulent flow - If Reynolds number is **greater than 3500**, the flow is turbulent. Most fluid systems in nuclear facilities operate with turbulent flow. (Power, 2023)

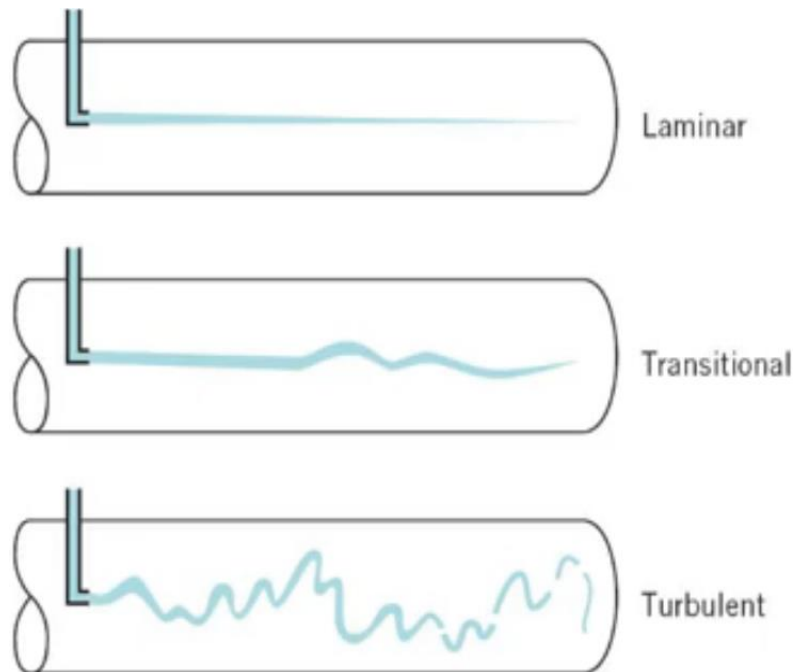
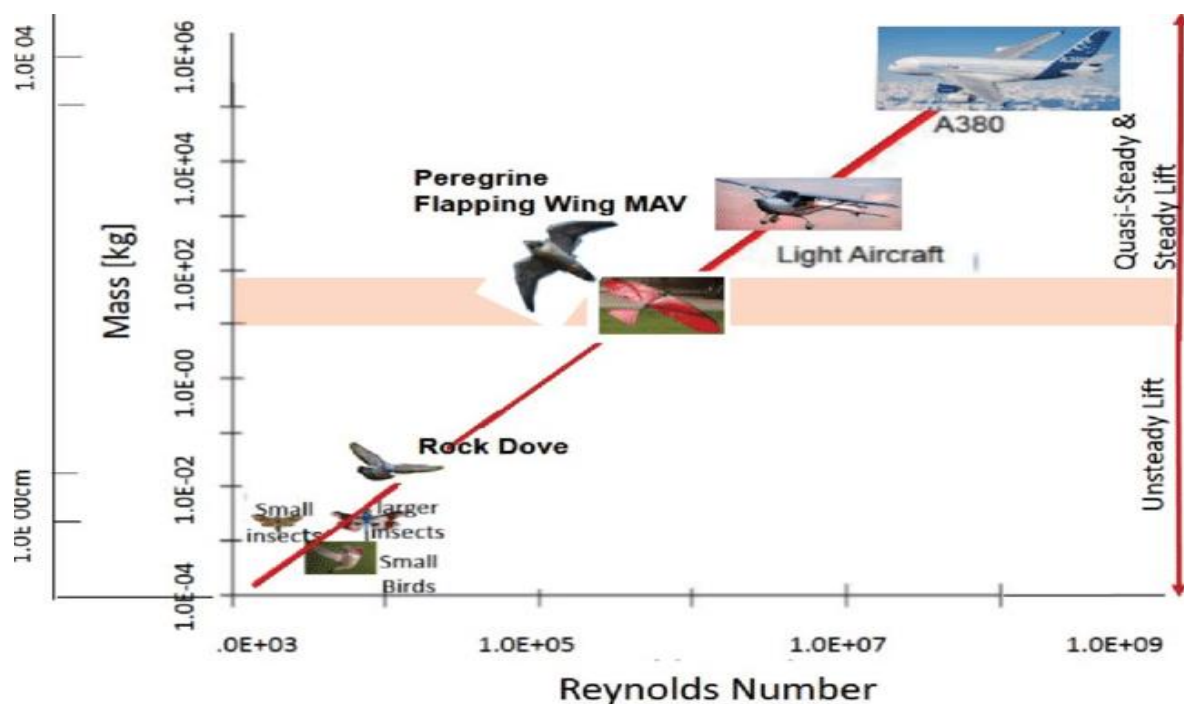


Figure 1 Osborne Reynolds Flow Graph (Power, 2023)

Flow around the wing can be heavily influenced by variables mentioned, by how pressure varies along the direction of flow. Other factors are speed, density, and viscosity (the stickiness) of the air. (Barnard, 2009). Aerofoils on an aircraft could have Re Numbers as high



as 500,000 (**Figure 2**). Dependant on application, type of aircraft and velocities involved etc, this figure also applies to F1. (Cadence, 2023)

Figure 2 External Reynolds Number Example (Djojodihardjo, 2017)

2.3. Reynolds Number Calculation

$$\text{Reynolds Number} = \frac{\text{Density} \times \text{Speed} \times \text{Length}}{\text{Viscosity}}$$

Calculation for Reynolds Number over a flat plate. Although a car wing is not too dissimilar, it does have the addition of some bends, this can influence Reynolds number as mentioned later in this section. It is being used to determine the flow present on the wing, laminar or turbulent flow.

$$Re = \frac{\rho v l}{\mu}$$

Equation for calculating Re Number over a flat plate.

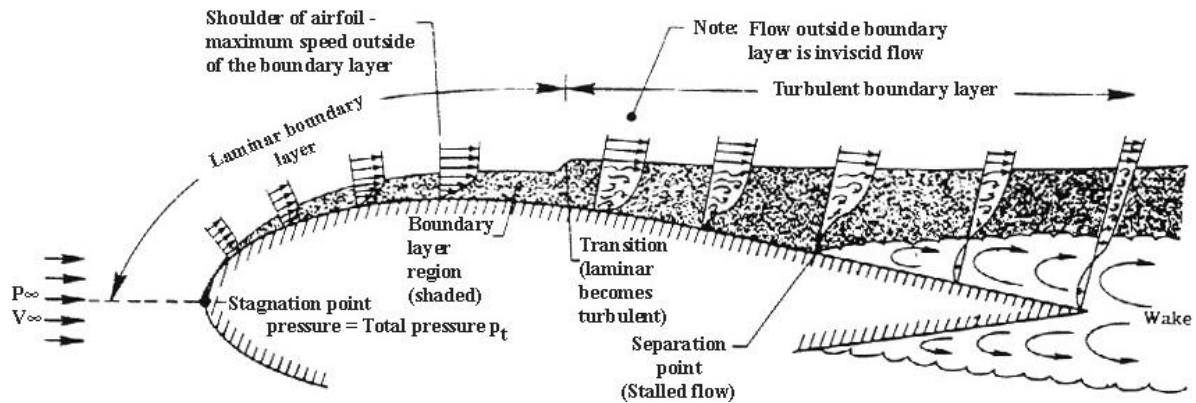
Two examples of **Re** over a flat plate, same velocities, different ambient temperature (**Figure 3**)

$\rho = 1.246 \text{ kg/m}^3$ at 10°C	(Density)	$\rho = 1.184 \text{ kg/m}^3$ at 25°C
$v = 100 \text{ m/s}$ at 10°C	(Velocity)	$v = 100 \text{ m/s}$ at 25°C
$l = 0.22\text{m}$ at 10°C	(Chord Length)	$l = 0.22\text{m}$ at 25°C
$\mu = 1.778 \times 10^{-5}$ at 10°C	(Dynamic Viscosity Of Air)	$\mu = 1.849 \times 10^{-5}$ at 25°C
$Re = \frac{1.246 \times 100 \times 0.22}{1.778 \times 10^{-5}} = 15.4 \times 10^{-5}$		10°C
$Re = \frac{1.184 \times 100 \times 0.22}{1.849 \times 10^{-5}} = 14.1 \times 10^{-5}$		25°C

Figure 3 Reynolds Number Calculation over Flat Plate

Both show different turbulent flow based simply on temperature. This figure is at the end of the plate, so will result in a turbulent flow. When measuring the flow at 0.05m from the front at 10°C., 3.5×10^{-5} is achieved, this is laminar flow, meaning velocity and position are key

factors in result. This is an effective use of **Re** as it allows a view to when/at what point along the plate/wing flow transitions from laminar to turbulent, shown in **figure 4**. Mentioned at the



beginning of **section 2.3**. Assumptions can be made that figures will have minimal change with the different variables considered.

Figure 4 Laminar to Turbulent flow diagram (Engineering, 2016)

Research suggests Reynolds Number ranges anywhere from $3 \times 10^6 - 3.2 \times 10^6$, at 43.82 m/s. *A R S Azmi et al 2017 IOP Conf. Ser.: Mater. Sci. Eng. 243 012030*. It can be seen however in **Figure 5** that the variable change is minimal. Geometry of tested wings can be seen in **Figure 6**. Based on the configurations that have been tested, NACA 0015 is the model that most closely represents the rear wing used on an F1 vehicle.

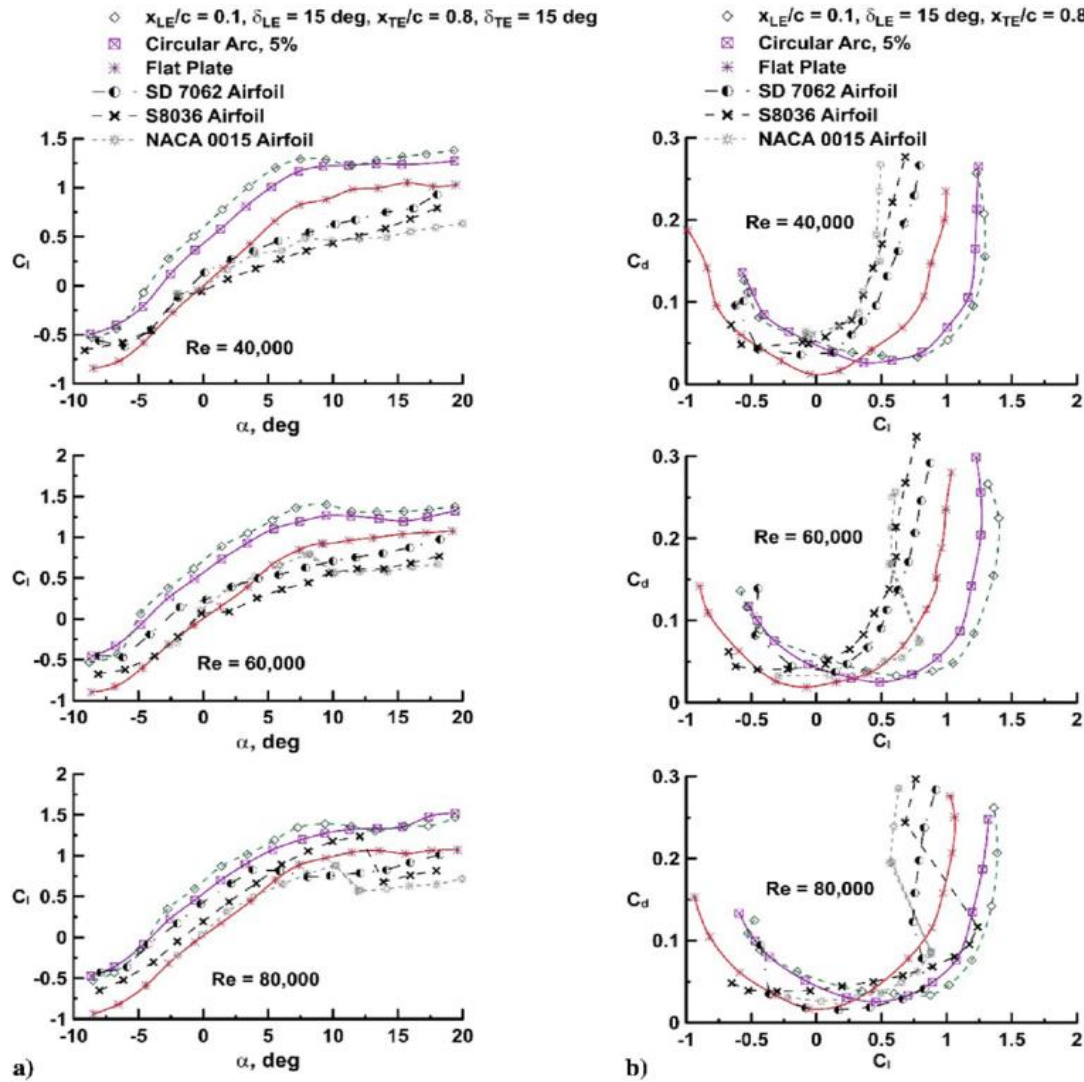


Figure 5 Reynolds Number Examples (Coffman, 2019)

Leading and trailing edge flap nomenclature (0.8% thick) (freestream left to right)

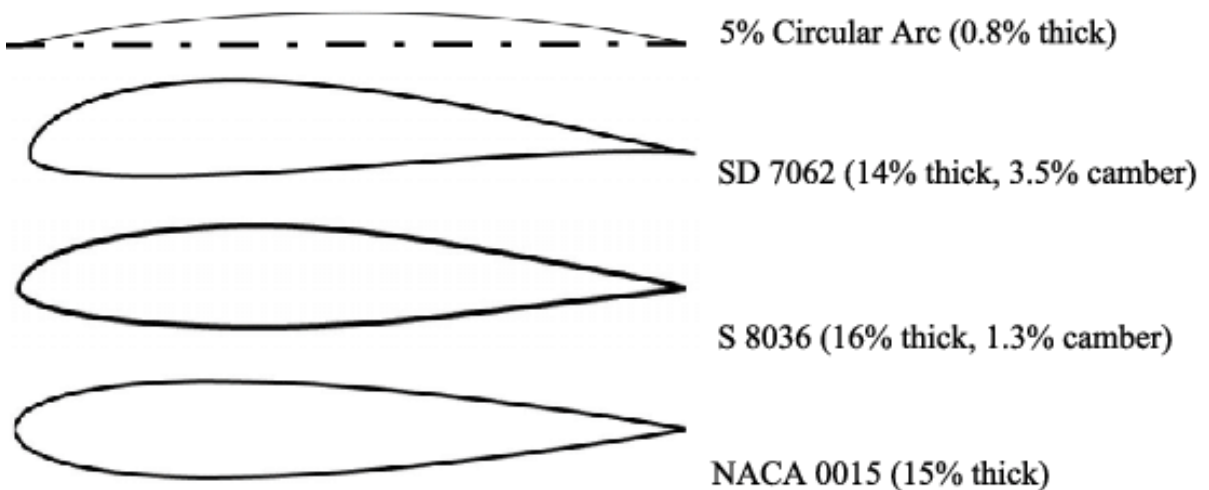


Figure 6 Tested Aerofoil Geometries (Coffman, 2019)

3. Boundary Layer

3.1. Boundary Flow Discussion

It is important to understand principles and concept. An important feature of flow past a vehicle is that air **appears** to stick to the surface. Right next to the surface is no measurable relative motion. Individual molecules do not physically stick, they move around randomly, but their average velocity component parallel to the surface is zero. (Barnard, 2009)

The relative velocity of air flow increases rapidly with distance away from the surface, as shown in **figure 7**, and only a thin layer is slowed down by the presence of the surface. This layer is what is known as the boundary layer. (Barnard, 2009)

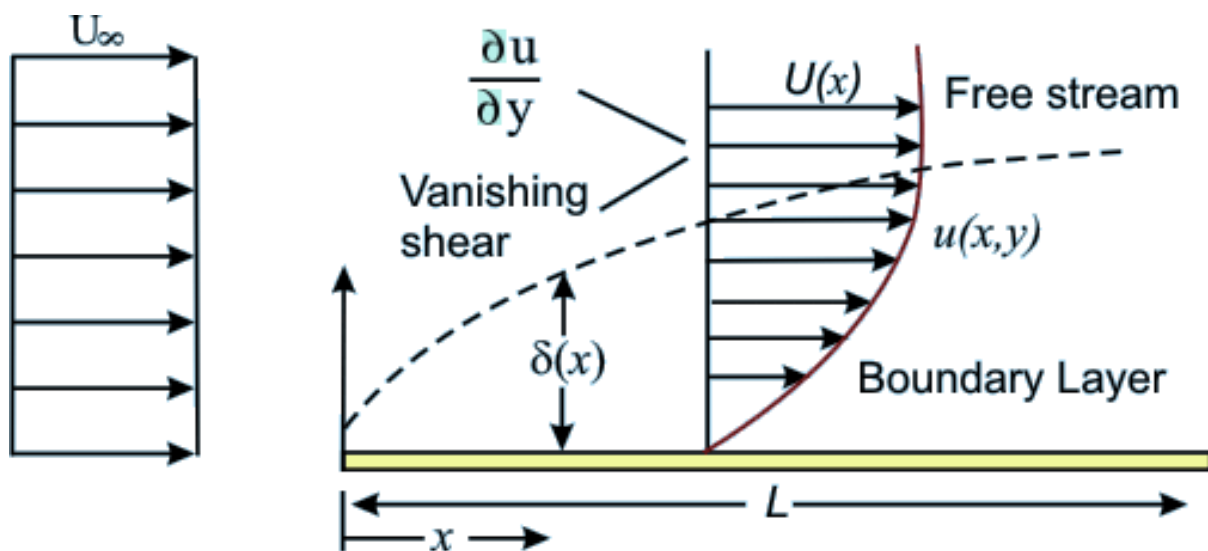


Figure 7 Velocity Profile Diagram (Tec-Science, 2020)

3.2. Boundary Flow, Separation and Turbulence Around a Wing

Requirement is required to maintain boundary layer for as long as possible while the vehicle is in motion to maintain grip. The design of the wing (multiple smooth edges) helps to ensure that boundary layer can remain for longer, this is achieved by minimising sharp edges. Having the wing designed like this ensures a longer, smoother boundary layer. (Tec-Science, 2020). Shown in **Figure 8**. Ensuring a smooth boundary layer is maintained to stop stall, this results in separation, resulting in traction loss, and creating turbulent airflow from the wing. This affects traction and driveability for the driver and affects drivers trying to overtake due to the wake turbulence/dirty air exiting the wing. All these advances were implemented in 2009 with

a reworked aerodynamic package designed to improve overtaking. These were built around a set of regulations by the OWG (Overtaking Working Group).

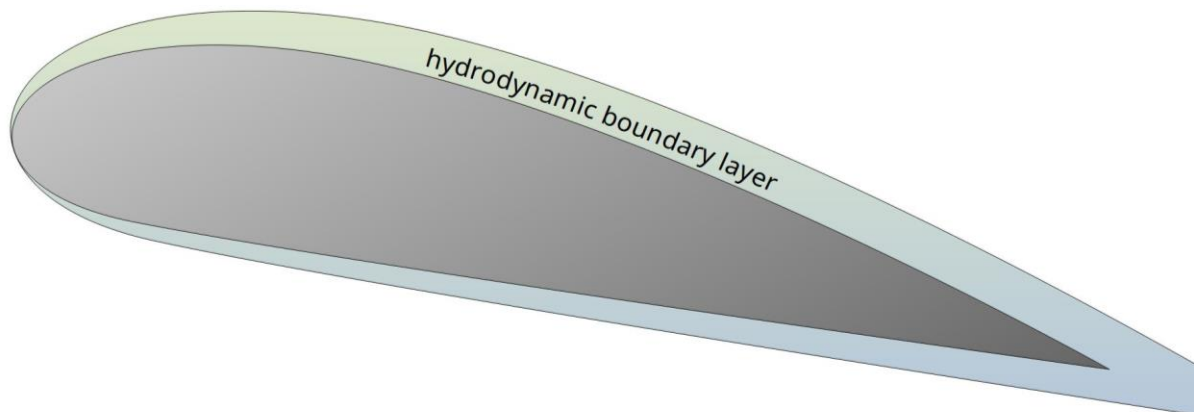


Figure 8 Hydrodynamic Boundary Layer and Wing Profile (Tec-Science, 2020)

Assuming the contour of a body has smooth transitions and the internal cohesion of the fluid (viscosity) is large enough to resist the inertial forces, flow can follow the contour. A typical hydrodynamic boundary layer develops around the body, with thickness being largely influenced by the viscosity.

With sharp transitions or when flowing around blunt bodies, the fluid is habitually no longer able to follow the profile. The boundary layer or flow begins to detach itself from the surface. Commonly referred to as a **boundary layer separation or flow separation**. Downstream of the separation point, vortices form, resulting in turbulent flow. (Tec-Science, 2020). Flow separation is particularly dangerous on a car wing, as this can result in lift. Such a dangerous flow separation is called stall. Description in **Figure 9**.

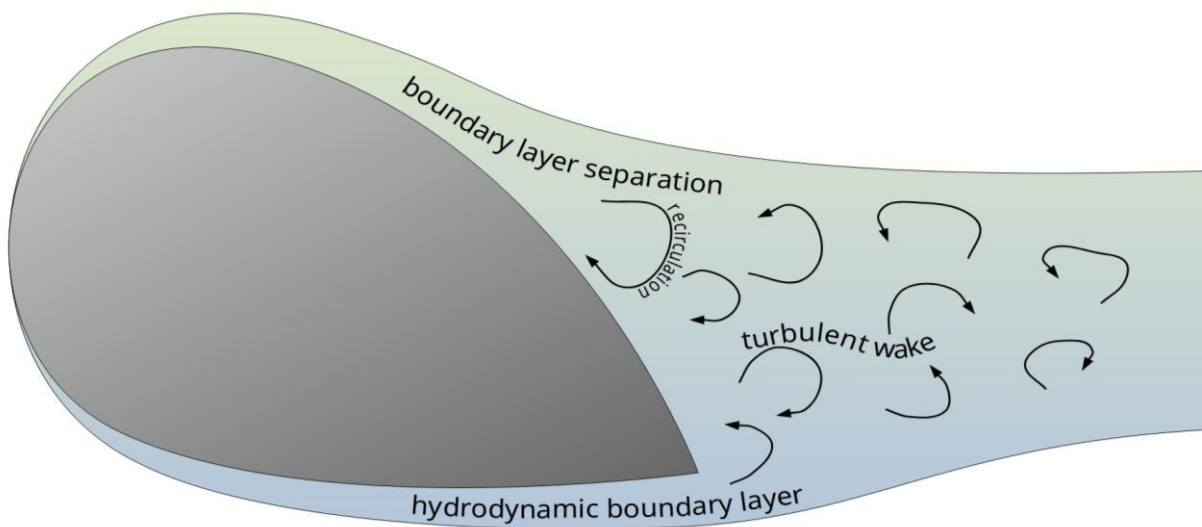


Figure 9 Boundary layer separation on a blunt body (Tec-Science, 2020)

3.3. Relationship to Separation and Turbulence Around Wing

3.3.1. Separation of Turbulent Boundary Layers

Flow around a body can be laminar and turbulent, dependant on Reynolds number. Laminar flow should be aimed for since this reduces flow losses to a minimum. However, this is only the case if no boundary layer separation occurs. If, this cannot be ensured, then turbulent boundary layers should be aimed for. This may sound counterintuitive, but a turbulent boundary layer can usually follow the profile longer than a laminar flow. The reason for this is increased transport of momentum between the fluid layers, which leads to a steeper velocity profile within the boundary layer. (Tec-Science, 2020). **Figure 10.**

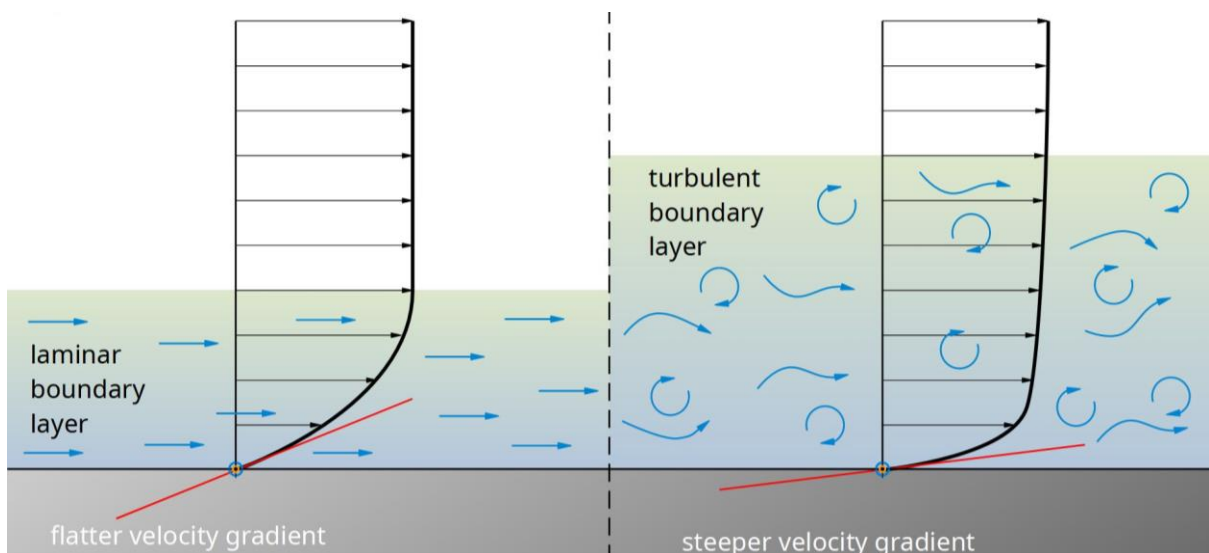


Figure 10 Velocity profile in a laminar and turbulent boundary layer (Tec-Science, 2020)

With a turbulent boundary layer, the velocity in the y-direction increases faster than with a laminar boundary layer. Meaning a larger velocity gradient near the wall, thus higher velocity components in the boundary layer. The resulting higher kinetic energy of the boundary layer can counteract the adverse pressure gradient more.

In the case of turbulent boundary layer, the separation point shifts downstream. The turbulent wake becomes narrower with the shifting of the separation point. This reduces flow losses and pressure drag, which results in a reduction of overall drag. A prime example of this is in golf ball designs.

Turbulent boundary layers adhere longer to the surfaces of the body around which flow occurs. The (pressure) drag is significantly lower. If there is a risk of flow separation, turbulent boundary layers should be aimed for. (Tec-Science, 2020)

3.3.2. Separation Bubble

Depending on the body being streamed around, flow that has detached can reattach. **Figure 11** shows laminar flow around a vehicle. At point A, flow separates and forms a type of circulating air cushion, this results in flow being able to reattach at point B. This is called the separation bubble.

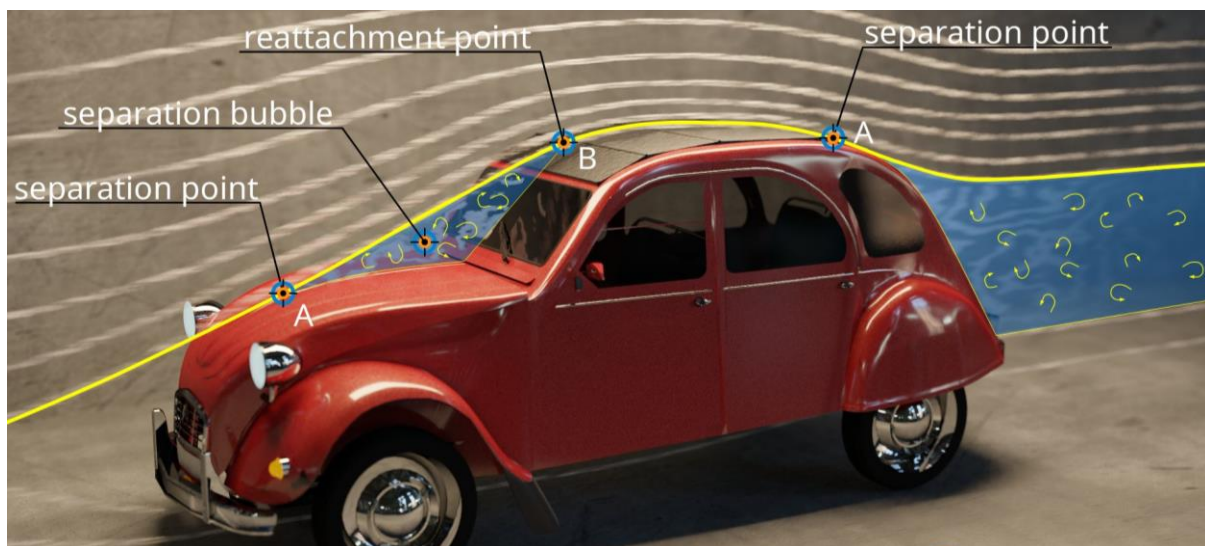


Figure 11 Separation Bubble (Tec-Science, 2020)

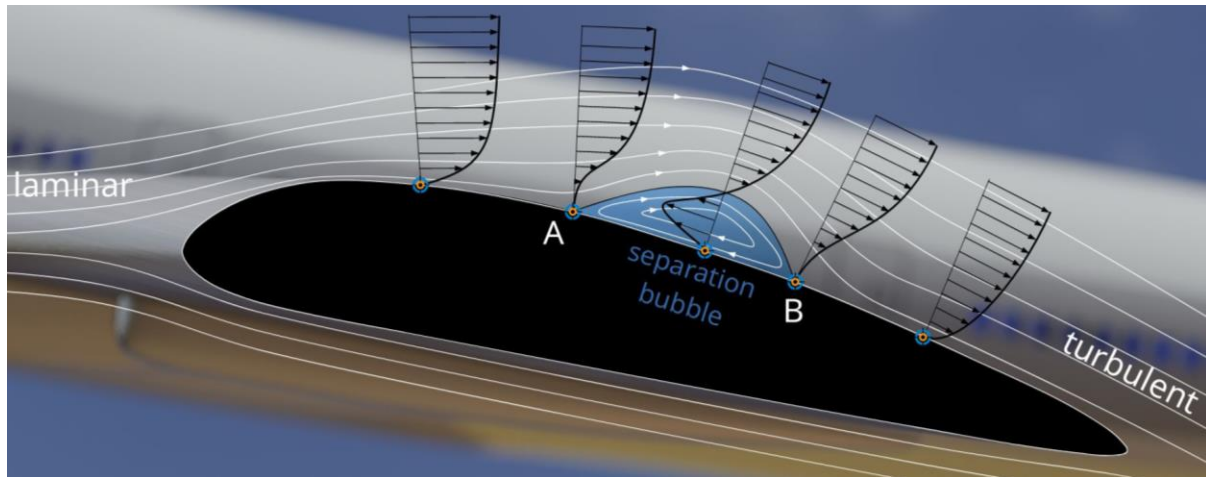


Figure 12 Laminar separation bubble on an aircraft wing (Tec-Science, 2020)

A separation bubble can form when air flows around airfoils. Laminar flow separates at one point and reattaches itself downstream, provided the angle of attack is not too high. In the area between the laminar separation point and point of turbulent reattachment, a laminar separation bubble forms (laminar because flow before the separation point is laminar). **Figure 12** shows an aircraft wing example.

The wings design has smooth edges. Meaning a hydrodynamic boundary layer is achievable. Indicating the boundary layer being sustained for a longer period of time, resulting in longer contact time, achieving higher grip efficiency from the wing, especially at higher velocities.

4. Car Wing Design

4.1. Implications of Using CFRP (Carbon Fiber Reinforced Plastic)

CFRP (Carbon Fiber Reinforced Plastic) is used in many current processes with numerous advantages, including. Lightweight, high fatigue resistance, higher tensile strength, and survives most severe environmental conditions. Additionally, disadvantages are present, which can play a decisive role in F1.

CFRP components are generally manufactured via moulding process in an autoclave (high pressure oven) at temperatures between 120°C and 180°C. Although different mould material options are available, PET-G-Filament is the most highly recommended due to its good release properties.

Two disadvantages of CFRP are. One of the most expensive materials to use, high conductor of heat and electricity, this is concerning, as high heat is present throughout the vehicles structure.

Being a good conductor of electricity can create a corrosion “cell” with steel structures in contact with carbon composite. This results in corrosion of other components coming into contact with the wing, this phenomenon, known as “Galvanic Corrosion,” means higher processing costs, labour hours, and structural issues throughout the vehicle.

Additionally, it fails under load, there are few to no warning indications and such failures can resemble explosions. (Even hydrotests done under water show levels of destruction upon failure.) This is a potential issue during a race, the fact a failure of any component could occur randomly, resulting in danger to the vehicle, driver, and other drivers.

Non-destructive periodic inspection of critical-service items is difficult, expensive, and time consuming. This associates to extremely prohibitive costs, being difficult to perform, and very labour intensive, both financially and physically. Forms of NDT (Non-Destructive Testing) include ultrasonic testing, X-ray, and Structural Health Monitoring (SHM). These methods of testing allow for testing to be carried during application.

Ultrasonic testing is extremely advantageous, as this can be done by having access to just one side of the component and can also identify internal defects while in conjunction sizing the defects. This kind of defect is highly probable with forces constantly being exerted on the wing, with high downforce and lift forces acting on the wing, could initiate stress cracks.

These are a few issues that are potentially present when using CFRP. There are however advantages to using CFRP, some have been highlighted at the beginning of **section 4.1**.

4.2. Historical Developments

The use of Carbon Fiber in Formula 1 cars had its roots in racing since the early 1980s. Before then, Formula 1 cars were made out of the same materials that the road cars were made from, such as steel or aluminium. At the end of the 1970s, some of the best engineers from Formula 1 came together to revolutionize the racing industry. Within a couple of years, they introduced the first cars to use carbon composite in their chassis. Today, we use Carbon Fiber more in the racing industry than ever before to make everything from chassis components, aero parts and even wheels. (RWCcarbon, 2019). The McLaren F1 Team introduced this in 1981, carbon fibre composites are now used widely in F1 components - almost 85% of a typical F1 car is made up of them.

4.3. Current Developments

Current rear wing designs have been manufactured with safety in mind, while ensuring that downforce is present and maintaining grip, however, current wings do produce less downforce than wings used earlier in the 21st century, examples are shown.



Figure 13 2021 Rear Wing vs 2022 Rear Wing (Reynolds, 2021)

Figures 13/14 show two types of wings used during races, dependant on requirements on race day. Design change could be influenced by temperature of environment, resulting in change of air pressure, it could be as of a result in tarmac being softer or harder, again influencing change.

4.3.1. New Design Elements

The highly distinctive shape is created by the elements, endplates and beam wing forming one continuous line with no sharp corners. This makes elements which rely on pressure difference between the underside and upper surface to work less effective in creating downforce but, greatly reduces the vortices of spinning air spilling off the wing corners where two separate airflows mix. Reducing these vortices makes the wake much less turbulent, resulting in a safer environment for the car behind. (Hughes, 2022)

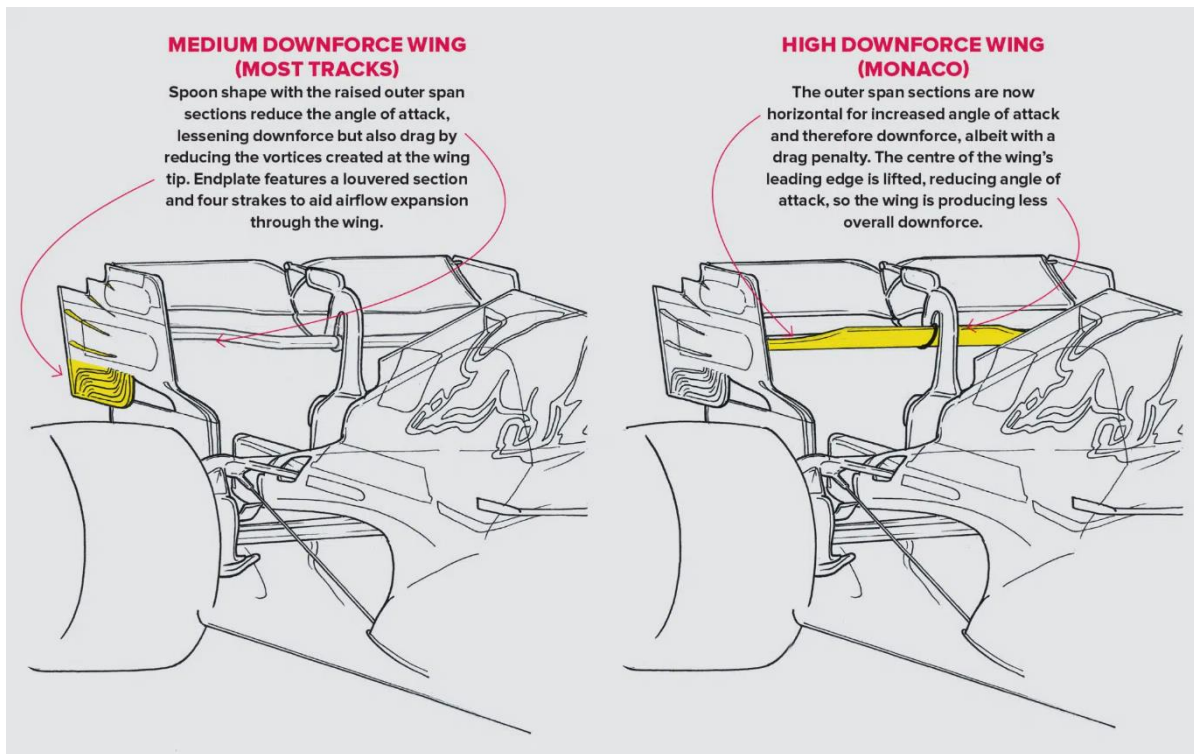


Figure 14 Variations of Current Rear Wings (motorsportmagazine, 2023)

4.4. Future Developments

F1 teams are always looking to the future, anticipating where improvements will come from next. However, it is important to state that end goals are changing for all competitors.

Where performance was priority, now teams are having to juggle a host of competing factors. Some include.

The cost caps were introduced in 2021, to ensure that teams are on a more level playing field, meaning all teams have a fair chance. This is something the bigger teams will need to consider. (FIA, 2022)

Savings on processing costs of materials, labour costs, and design costs are something all teams will now have to be weary of moving forward.

5. Modelling Techniques

5.1. CFD (Computational Fluid Dynamics) and FEA (Finite Element Analysis)

Presently, CFD and FEA are essential for the design and development of racing cars, working cohesively with wind tunnel testing. In fact, it is possible to run simulations of the car and its components prior to any wind tunnel testing, to pre-evaluate various configurations and submit to test only the most promising designs.

Teams' previous season's championship position	Percentage of current aero testing allowed for 2021	Percentage of current aero testing allowed for 2022-25
1	90%	70%
2	92.5%	75%
3	95%	80%
4	97.5%	85%
5	100%	90%
6	102.5%	95%
7	105%	100%
8	107.5%	105%
9	110%	110%
10+ or new team	112.5%	115%

Figure 15 New sliding scale on aerodynamic testing in F1 (Hughes, 2022)

Limitations, until recently have meant that a maximum of 65 test runs are allowed per week. This changed in 2021 in conjunction with the cost cap discussed in **section 4.4**. This means teams are now limited to 40 runs per week, a 30% reduction to previous years.

Additionally, for the very first time, each team's allowance of wind tunnel and CFD testing time will be defined by on-track performance. **Figure 15** shows an example of the new system being used, this is to create a more even playing field and give all teams an improved chance of podium finishes.

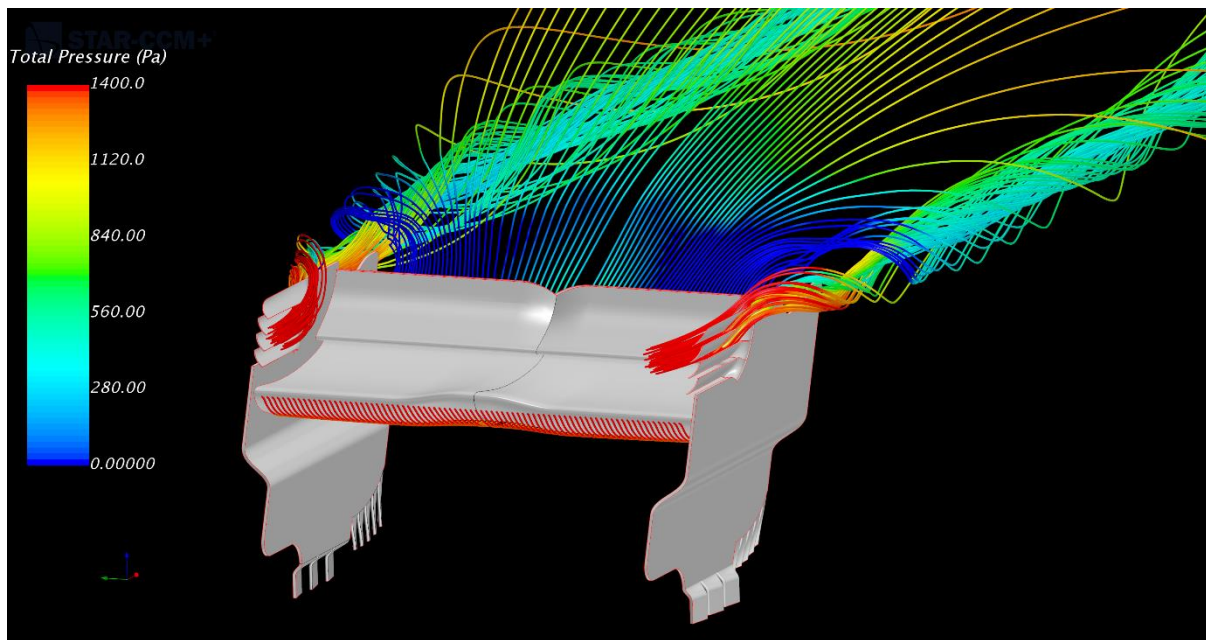


Figure 16 Rear Car Wing With Use Of CFD Analysis (Pangerl, 2018)

5.1.1. Advantages of CFD

Advantages are extensive, some have been discussed in **Section 5.1**. Some further advantages include:

- **Insight** - If you have a device and/or system design which is difficult or expensive to prototype or test through experimentation, CFD analysis enables you to virtually crawl inside the design and see how it performs. Additive Manufacturing is now working alongside CFD, allowing for improved designs to save on weight, and reduce process and manufacturing costs. (3D printing capabilities). Along with the ability to develop accurate scale models.
- **Foresight** - CFD can predict what will happen under particular circumstances, immediately answering many 'what if?' questions. You provide a set of boundary conditions, and the software gives you outcomes. Simply, you can predict how your design will perform, and test many variations until arriving at an optimal result. All this can be done before physical prototyping and testing.
- **Efficiency** - The foresight you gain helps design better and faster, save money, meet Regulations, and ensure FIA rules compliance. CFD analysis leads to shorter design

cycles and teams can get their car to track faster. Additionally, equipment improvements are built and installed with minimal downtime. CFD is a tool for compressing the design and development cycle allowing for rapid prototyping. (Dictionary, 2005)

5.1.2. Disadvantages of CFD

Disadvantages are minimal, some of the disadvantages are:

- **Uncertainty** – Uncertainty and error, can arise from factors such as mesh size, boundary conditions and turbulence models.
- **Model Limitations** – Based on mathematical models that are approximations of real-world systems, these models may not accurately represent all physical phenomena present in a system.
- **Computational Resources** – Can be computationally intensive, requiring significant resources, including powerful computers and high-performance computing systems. (Engineering, 2009)

CFD has not replaced Wind Tunnels but merged to create an ultimate means to an end, allowing creation of a model, followed by physical testing of said model produced with CFD.

6. Experiment Application

Discussed in **Section 5**. Ultimately commence design of car wing with CFD. This allows for computerised models to be produced without having to produce a prototype for testing. This means saved time, saved labour hours, and saved costs. Being able to produce a simulation for inspection of things like flow transition from Laminar to Turbulent.

Once simulations have been completed, and design has been finalised, the wing can be produced and tested via wind tunnel. It is suggested that a closed-return system is used, some reasons as follows:

- Saving in operational costs compared to Open-return systems, due to air recycling capabilities (retain the kinetic energy).
- Vehicle runs on a rolling road to get true indication of airflow (around wheels with centrifugal forces in play etc).
- Ride height simulation. This is done by using a Carbon Bar fixed to the car, allowing for accurate changes in ride height of 0.01mm.

- Laminar air flow being produced, compared to dirty air from open-return process.
- Can pressurise system to create more stable and accurate Reynolds Number.

This is the wind tunnel of choice for most aerodynamic designing. However, air is heated up in closed-return systems, which can affect air density, meaning change to Reynolds Number. It must be noted when using a scale model, velocities must be increased to achieve required Reynolds number, for example. If using a 1:10 scale, air speed must be 10 times speeds in real world application (10 mph in real world, 100 mph for scale model). Meaning 1:1 scale testing is more desirable.

Wing should be tested when attached to vehicle to simulate real world applications, additional downforces from car itself, tyres, front wing etc, again influencing rear wings design. **Figure 17.**



Figure 17 Wind Tunnel Car Testing (Hughes, 2022)

Regulations have had an influence on current and future designs as mentioned in sections **4.3, 4.3.1 and 4.4.**

Conclusion

Definition of Reynolds number, how it can affect the performance of components, specifically a rear wing. Reynolds number calculation produced, showing Reynolds number in laminar and turbulent flow. How changes in air density, temperatures etc can affect Reynolds Number outcome.

Boundary layer discussion, what it is, how it can affect the performance of a vehicle, and how to achieve a sustained boundary layer to increase performance and grip. Separation of Turbulence, along with separation bubble, and how we can maintain a boundary layer by creating a turbulent flow at a certain position, resulting in a re-attachment, maintaining the boundary layer for longer.

Followed up with a car wing design, past, present, and future developments, correlating these to some F1 regulations currently present.

CFD discussion, what it is, how it is used within the industry, its pros, and cons, how this has allowed for more advanced designing, allowing designs to be created and simulated, without the need to physically produce a prototype. Experimental work carried out can be correlated to CFD analysis that has been performed.

An experimental application discussion. Design, and how it would be created. Discussing why closed-return wind tunnels are advantageous over open-return systems. Finally discussing the pros of using CFD and wind tunnels, and discussing how scale models can affect Reynolds number, velocities, pressure changes etc.

References

- Barnard, R. H., 2009. *Road Vehicle Aerodynamic Design*. 3rd ed. Hertfordshire: MechAero Publishing .
- Cadence, 2023. *Cadence System Analysis*. [Online]
Available at: <https://resources.system-analysis.cadence.com/blog/msa2022-the-importance-of-laminar-flow-over-airfoils-in-aerodynamics>
[Accessed 28 July 2023].
- Coffman, C., 2019. *Efficient Low-Reynolds-Number Airfoils*. [Online]
Available at: https://www.researchgate.net/publication/334324225_Efficient_Low-Reynolds-Number_Airfoils/citation/download
[Accessed 5 August 2023].
- Dictionary, T. F., 2005. *CFD Computational Fluid Dynamics*. [Online]
Available at: <https://www.formula1-dictionary.net/cfd.html>
[Accessed 15 July 2023].
- Djojodihardjo, H., 2017. *Numerical Modelling of Flapping Wing Kinematics and Aerodynamics*. [Online]
Available at:
https://www.researchgate.net/publication/317637451_Numerical_Modelling_of_Flapping_Wing_Kinematics_and_Aerodynamics/citation/download
[Accessed 4 August 2023].
- Engineering, A., 2016. *Aerospace Engineering*. [Online]
Available at: <https://aerospaceengineeringblog.com/boundary-layer-separation-and-pressure-drag/>
[Accessed 8 August 2023].
- Engineering, Q., 2009. *Advantages and Disadvantages of Computational Fluid Dynamics*. [Online]
Available at: <https://www.quadco.engineering/en/know-how/cfd-advantages-and-disadvantages.htm>
[Accessed 15 July 2023].
- FIA, 2022. *FORMULA 1 FINANCIAL REGULATIONS*. [Online]
Available at:
https://www.fia.com/sites/default/files/fia_formula_1_financial_regulations_iss.12.pdf
[Accessed 29 July 2023].
- Groote, S. D., 2003. *F1Technical*. [Online]
Available at: <https://www.f1technical.net/articles/47>
[Accessed 14 July 2023].
- Guo, D. B., 2005. *Reynolds Number*. [Online]
Available at: <https://www.sciencedirect.com/topics/engineering/reynolds-number>
- Hughes, M., 2022. *Formula1*. [Online]
Available at: <https://www.formula1.com/en/latest/article.tech-tuesday-how-the-rear-wing-of-the-2022-car-has-been-designed-to-be-an-f1.70dkHlNyP8DiGDDnMf3tzU.html>
[Accessed 13 July 2023].

motorsportmagazine, 2023. *F1 rear wing designs: Feeling the force*. [Online]
Available at: <https://www.motorsportmagazine.com/archive/article/august-2021/36/f1-rear-wing-designs-feeling-the-force/>
[Accessed 28 July 2023].

Noble, J., 2022. *How carbonfibre technology is shaping F1's new era*. [Online]
Available at: <https://www.autosport.com/f1/news/how-carbon-fibre-technology-is-shaping-f1s-new-era/10386270/>
[Accessed 12 July 2023].

Pangerl, J., 2018. GRABCADCOMMUNITY. [Online]
Available at: <https://grabcad.com/library/f1-rear-wing-cfd-study-1>
[Accessed 29 July 2023].

Power, N., 2023. *Reynolds Number for Turbulent Flow*. [Online]
Available at: <https://www.nuclear-power.com/nuclear-engineering/fluid-dynamics/reynolds-number/reynolds-number-for-turbulent-flow/>
[Accessed 9 June 2023].

Rehm, B., 2008. *Reynolds' Number*. [Online]
Available at: <https://www.sciencedirect.com/topics/engineering/reynolds-number>
[Accessed 9 June 2023].

Reynolds, J., 2021. *ANALYSIS: Comparing the key differences between the 2021 and 2022 F1 car designs*. [Online]
Available at: <https://www.formula1.com/en/latest/article.analysis-comparing-the-key-differences-between-the-2021-and-2022-f1-car.4xYDhtOjDee4cEQ3P4RsK9.html>
[Accessed 29 July 2023].

RWCarbon, 2019. *Carbon Fiber & its History in Formula One Racing*. [Online]
Available at: <https://blog.rwcarbon.com/2014/08/12/carbon-fiber-its-history-in-formula-1-racing/>
[Accessed 12 July 2023].

Tec-Science, 2020. *Flow Separation (Boundary Layer Separation)*. [Online]
Available at: <https://www.tec-science.com/mechanics/gases-and-liquids/flow-separation-boundary-layer-separation/>
[Accessed 10 July 2023].