UNIVERSITAT POLITÈCNICA DE VALÈNCIA

Escuela Técnica Superior de Ingeniería del Diseño

UNIVERSITAT POLITÈCNICA DE VALÈNCIA

School of Design Engineering

Study of the influence of over-wheel winglets on Formula prototypes

End of Degree Project

Bachelor's Degree in Aerospace Engineering

AUTHOR: Rubia Mongort, Paula de la

Tutor: Margot, Xandra Marcelle

ACADEMIC YEAR: 2022/2023

Bachelor's Thesis

Study of the Influence of Over-Wheel Winglets on Formula 2 Prototypes

Universitat Politècnica de València
Escuela Técnica Superior de Ingeniería del Diseño

Aerospace Engineering Degree

94 pages

Author
Paula De La Rubia Mongort

Tutor
Xandra Marcelle Margot Ratié

ACKNOWLEDGEMENTS

First of all, I would like to express my most sincere gratitude to my tutor, **Xandra Marcelle Margot Ratié**, who has welcomed me and given me invaluable help during all these months. For being one of those teachers who project much more than just lessons onto the students. Without you it would not have been possible.

Thanks also to the Valencian motor racing team **Campos Racing**, who did not hesitate to help me when I contacted them, thanks to whom it has been possible to bring the project as close as possible to the reality of the Formula 2 prototypes and thanks to whom I awoke my passion for motor racing.

To my friends, for always believing in me and pushing me to go one step further in my ideals, to be faithful to my objectives and for helping me to see situations with perspective.

To my colleagues from Air Nostrum, for giving me such enriching lessons day after day.

But most of all, to my parents and grandparents, who have always been there even in my worst moments when I couldn't even stand myself. For always taking an interest in my passion and for pushing me to be myself in my day to day life. For letting me make my decisions freely. And above all, for loving me as I am. Even if I don't say it too much, you can't get an idea of how much I love you and how essential you are to me on a daily basis.

ABSTRACT

The present project will involve aerodynamic study using Computational Fluid Dynamics (CFD) software on the influence of over-wheel winglets on Formula car prototypes, designed in a simple way using Computer-Aided Design. The effects caused by the wheels on the incident fluid, due to the movement of the wheels, will also be considered, concluding in a study of the aerodynamic forces and characteristics of the flow and recirculation in both cases, with and without winglets, and it is intended to study the benefit of the lamination of the flow produced by these elements.

All this will lead to different behaviour of air over the car, what will cause different forces on the single-seaters and thus, a different performance between the two geometries under consideration.

RESUMEN

El presente proyecto consistirá en el estudio aerodinámico mediante software de Dinámica de Fluidos Computacional (CFD) de la influencia de los alerones sobre las ruedas en prototipos de coches de Fórmula, diseñados de forma sencilla mediante Diseño Asistido por Ordenador. También se considerarán los efectos causados por las ruedas sobre el fluido incidente, debido al movimiento de las mismas, concluyendo en un estudio de las fuerzas aerodinámicas y características del flujo y recirculación en ambos casos, con y sin winglets, y se pretende estudiar el beneficio de la laminación del flujo producido por estos elementos.

Todo ello dará lugar a un comportamiento diferente del aire sobre el coche, lo que provocará fuerzas diferentes sobre los monoplazas y, por tanto, un rendimiento diferente entre las dos geometrías consideradas.

RESÚM

El present projecte consistirà en l'estudi aerodinàmic mitjançant l'ús de programes de Dinàmica de Fluids Computacional (CFD) de la influència dels alerons sobre les rodes en prototips de cotxes de Fórmula, dissenyats de manera senzilla mitjançant Disseny Assistit per Ordinador. També es consideraran els efectes causats per les rodes sobre el fluid incident, a causa del moviment d'aquestes, concloent en un estudi de les forces aerodinàmiques i característiques del flux i recirculació en tots dos casos, amb i sense winglets, i es pretén estudiar el benefici de la laminació del flux produït per aquests elements.

Tot això donarà lloc a un comportament diferent de l'aire sobre el cotxe, el qual provocarà forces diferents sobre els monoplaces i, per tant, un rendiment diferent entre les dues geometries considerades.

Contents

List of Figures

List of Tables

Nomenclature

”	Inches
μ	Dynamic viscosity of the fluid [kg/(m $\cdot s$)]
ν	Kinematic viscosity of the fluid [m^2/s]
ω	Angular speed [rad/s]
ρ	Fluid density [kg/m^3]
τ	Stress tensor
τ_{ij}	Stress tensor in a face with constant i and oriented in j direction
€	Euro
°	Degrees
a	Speed of sound in the fluid [m/s]
C_D	Drag coefficient
C_L	Lift coefficient
CAD	Computer-Aided Design
CFD	Computational Fluid Dynamics
cte	Constant
D	Drag force [N]
DRS	Drag Reduced System

E	Total energy [J]
$F1$	Formula 1
$F2$	Formula 2
Fb	Body forces
FIA	Federation Internationale de l'Automobile
g	Gravitational acceleration [m/s^2]
g_i	Gravitational acceleration in i direction [m/s^2]
GB	Gigabyte
h	hours
hp	Horse power
k	Thermal conductivity
L	Lift force [N]
l	Characteristic length of the object into study [m]
M	Mach number
m	Mass [kg]
MHz	Unit of measurement of frequency, Megahertz
p	Pressure [Pa]
q	Heat sources or skins
R	Radius [m]
$RANS$	Reynolds-Averaged Navier-Stokes
Re	Reynolds number
S	Surface [m^2]

SST Shear Stress Transport

T Temperature [K]

t Magnitude of time [s]

u Velocity in x direction [m/s]

V Flow velocity [m/s]

v Fluid velocity [m/s]

v Velocity in y direction [m/s]

w Velocity in z direction [m/s]

Chapter 1

Introduction and Objectives

1.1 Research Motivation and Objectives

Formula racing is a highly competitive motorsport that pushes the limits of each team's drivers, technology and engineering in order to create the best mix of these factors and thus, be the best in each category. Over the years, aerodynamics has become increasingly important, playing a crucial role in the performance of the prototype car, having several aerodynamic components such as wings, diffusers and sidepods, which have undergone significant improvements that have led to an increase in downforce while reducing drag.

In recent years, and with recent aerodynamic improvement and changes to the 2022 Formula 1 car models, new elements have been highlighted compared to previous versions of these cars. Amongst these elements, the upper winglets on the front wheels of F1 cars stand out. These winglets are small aerodynamic devices located above the front wheels of the cars, which are designed to manipulate the turbulent flow of the air around the wheels, and thus improve the whole performance of the car. These winglets can be seen in the 2023 F1 car shown in the Figure 1.1.

Figure 1.1: Image where upper wheel winglets can be easily seen on a F1 car [1]

Although the aerodynamic influence of these elements that form the formula cars has been studied, the aerodynamic influence of the upper front wheel winglets on the overall aerodynamic performance of the car as a whole, is still small and a unexplored area of study given its brief development in the short history of the car's concept. However, understanding the effects of these, can provide useful information for the optimisation and design of the cars in these categories, as well as their lap times, a highly important factor to be taken into account within the teams.

By carrying out a detailed study of the influence of these aerodynamic elements using Computational Fluid Dynamics software, it is intended to evaluate the effect over aerodynamic forces and the behaviour of the flow which is affected by this area and the influence of the rotation of the wheels.

Furthermore, given the large number of formula car categories, it is intended to study the effects of these elements on Formula 2 prototypes since it is an optimization that has not yet been implemented in this category (it has only been implemented in F1) and it is intended to study whether it would also be beneficial to implement in lower categories.

Ultimately, this study aims to contribute to the area of motorsport engineering by providing a study on these elements for which no real data is known, and therefore, leading to a better understanding of aerodynamics in this field where its optimization plays a fundamental role on the teams results.

1.2 Significance of the Study

In the field of automotive engineering, the study and knowledge of aerodynamics play a fundamental role in the behaviour of formula cars, and even more so since the latest update in their design. The results of this research have highly significant implications on different aspects to be taken into account in motorsport. Thus, these areas may be such as the following:

This study provides a deeper understanding of air distribution and pressures on overall car performance, as well as the influence of over-wheel winglets on downforce generation and drag coefficient reduction.

In addition, knowing how these elements influence the air particles that take up the area affected by them can stimulate technological innovation in race car engineering by attempting to optimize car designs through observation of their behavior. This information may encourage the engineers who design F2 cars to include these elements in their single-seaters in order to have races in which the driver is increasingly more important than the car. In addition, for categories in which each team builds its own car, it is of vital importance to know the behavior of each of its aerodynamic elements.

These elements could even be used differently for each Grand Prix, as is done with other surfaces, allowing slight differences in the overall design, in order to bring the air passing over the wheels to different areas of the car that are beneficial depending on the circuit where it is raced. In F2 cars, all the teams have the same single-seater supplied by the Italian brand Dallara, on which small changes in the setup or adjustments are allowed, but this element could be left among these small changes as it does not involve an outlandish aerodynamic change compared to the total of the car, and thus allow each team to have some flexibility in this field.

These elements also affect the downforce of the car, and therefore the grip of the single-seater, which can make a big difference in the speed at which corners can be taken, as it will be more difficult to lose control as more force is generated.

That is why this study can serve as a new resource to be taken into account by researchers and academics interested in the field of automotive engineering and aerodynamics since it is an element that has not been studied much before.

Overall, the importance of this study lies in the improvements in both the performance and safety of race cars, and could be an element of great study and implementation in more formula categories, benefiting teams and drivers to improve their lap times.

1.3 Scope and Limitations

As mentioned above, the study aims to reach the conclusions that the aerodynamic studies will give and to observe the supposed improvement offered by the new design with over-wheel winglets due to the reasons given above.

However, it is important to consider the limitations of the study carried out. These include numerous cases such as the following.

On the one hand, the motorsport field is an area of extreme secrecy, so the availability of specific data on car aerodynamics is confidential for each team. So much so that not only Formula 1 teams are deprived of the information they have about the influence of the aerodynamic elements under study, but also of the car's own aerodynamic data and aerodynamic coefficients among other racing categories. That is why the Campos Racing team has been called upon to provide more detailed images and measurements of the car as well as some aerodynamic data and racing knowledge. However, this lack of scientific data may affect the quantity and quality of the information.

On the other hand, the simulations that will be carried out will have certain variations with respect to the real cases given that both CAD and setup of the CFD, even though the real measurements of the Formula 2 cars will be used, will take simplifications in their parts in order to be able to carry out a "simple" study with a computer for own use instead of a professional computer that should be used to be able to execute the real car cases in a precise way. Later, when using the computational fluid mechanics program, certain simplifications will be made to the mesh, always looking for the most accurate value so that the case can be calculated safely in a calculation time that is not too long. This is why, although the comparisons will not be the same as the CFD studies of the real car, it will be possible to observe the trend within a considered margin.

That is to say, the study, given the complexity of studying Formula prototypes, will take simplifications to enable the study from private computers, but efforts will be made to make the result as realistic as possible.

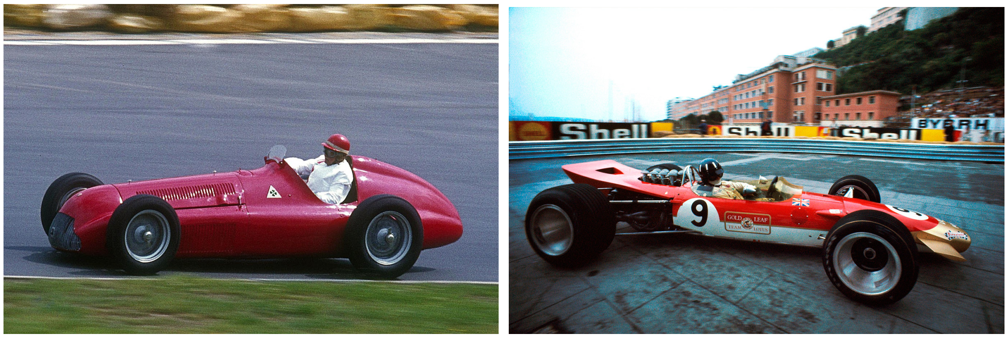
Chapter 2

Literature Review

2.1 History of Aerodynamics in Formula 1

Throughout the history of racing cars, aerodynamics has become increasingly important and has come to play a fundamental role in the performance of the teams. So much so that small aerodynamic differences on the same car can cause big differences in the safety and speed of the car, and in the position of each team in the corresponding classification. This is the reason why this field has been studied more and more in depth.

It was not until 1950 that the first Formula 1 drivers' world championship was held, where the cars (seen in Figure 2.1(a)) did not have any aerodynamic elements. Thus, in the 60's is when the study of aerodynamics began, with the aim of minimizing aerodynamic drag and maximizing speed on the straights, also introducing simple concepts such as ailerons which helped to improve stability, and chimney-shaped air intakes. These can be seen in the Figure 2.1(b) below.



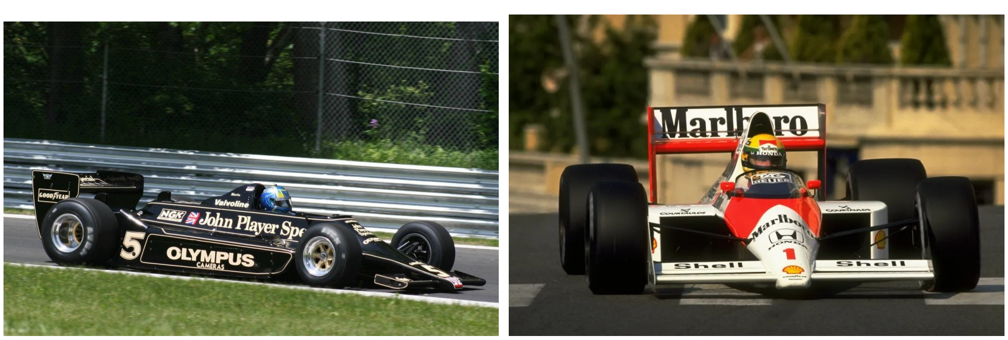
((a)) 1950 prototpype [2]

((b)) 1968 prototpype [3]

Figure 2.1: Cars at the beginning of Formula 1 history

But it was not until the late 1970s that aerodynamics became a fundamental aspect to be taken into account by the teams, when elements such as ailerons that can be seen in Figure 2.2(a) began to be added, which made it possible to generate greater downforce and thus increase the speed at which corners were taken due to this increased grip.In addition, new aerodynamic concepts such as Ground Effect was studied and taken into account for the car designs.

The real breakthrough came in the 1990s, when the aerodynamic load of the single-seaters began to be distributed differently for each track or even strategy. This was made possible by elements such as adjustable spoilers and diffusers. An image of these cars can be seen in 2.2(b), where a completely different concept from the rest pf the times can be observed.



((a)) 1979 prototpype [4]

((b)) 1993 prototpype [5]

Figure 2.2: Cars when aerodynamic begins to take on importance

In recent decades, aerodynamics in racing cars has remained in constant evolution thanks to technological and scientific advances that allow the study of this modality, reaching designs such as the ones shown in 2.3. In addition, the possibility of studies using wind tunnels and CFD optimizations provided a deep understanding of the behavior of air and made it possible to extrapolate this knowledge to optimize the performance of the cars.

Thus, in 2005 a new category called "GP2 Series" [6] emerged, which would later be renamed "FIA Formula 2 Championship" and whose cars will be the subject of study in this project. With this category, other categories were also created that served both to train pilots at different levels, and to race with different objectives.

Figure 2.3: Image of the Ferrari F1-75 [7]

A comparison between weight and speed of the cars presented in the previous images will be made in the following table 2.1 from which very enriching results can be obtained.

Car model	**Weight**	**Máximum speed**
Alfa Romeo 158 (1950) [2]	710 kg	75 m/s
Lotus 49 B (1968) [8]	520 kg	80.5 m/s
Lotus 79 (1979) [9]	575 kg	92.2 m/s
McLaren MP4/8 (1993) [10]	505 kg	94.4 m/s
Ferrari F1-75 (2022) [11] [12]	798 kg	96.4 m/s

Table 2.1: Comparison of car performance over the years

From these values, several conclusions can be drawn, such as the fact that the weight has been oscillating due to the changes in materials (such as the change to carbon fiber between 1979 and 1993) as well as the increase in both, aerodynamic elements and in weight due to the size of the single-seaters and the reinforcements that seek the safety of the drivers, since the current safety is nothing like that of previous decades.

In summary, it can be said that the history of aerodynamics in formula category racing cars has been a learning process, going from cars whose objective was to run as fast as possible with hardly any additional aerodynamic elements beyond the fairing of the car itself, to concepts that seek to optimize the performance of the car by maximizing downforce and minimizing drag to achieve the goal of balance between straight-line speed and cornering grip.

2.2 Formula Car Main Concepts

In this section, the main elements in formula cars will be studied, especially those that affect aerodynamics, since this is the main point of study of the project. In addition, to achieve good results, it is key to distribute the aerodynamic loads in a precise way so that the car as a whole generates as much aerodynamic load as possible but without slowing it down.

In recent decades, moreover, some aerodynamic elements have movable surfaces that allow the car to adapt to each racing situation.

2.2.1 Front Wing

The front wing is one of the most important aerodynamic parts of the cars since this one is used for both, to generate downforce at the front of the car and to deflect the air towards the rest of the car [13]. Due to the importance of this element, racing regulations are strictly rigorous with these.

There are different parts that make up the whole front wing, as will be seen in Figure 2.4 and explained below, from the inside to the outermost part of the wing.

- Nose: The nose is the part that joins the wing to the rest of the vehicle.
- Flaps: This part of the wing has different elements with an aerodynamic profile section that form a cascade, allowing the objectives that have been previously described.
- Endplates: These are the vertical pieces located at the outermost part of the aileron, that like winglets, intend to reduce induced drag.

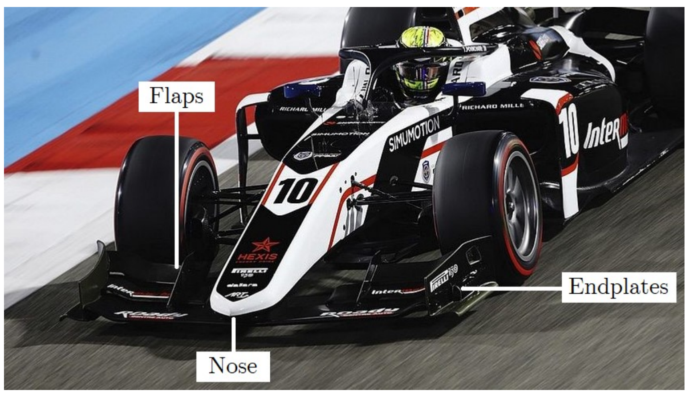


Figure 2.4: Dallara F2 front wing [14]

2.2.2 Rear Wing

The rear wing, as the front one, is one of the most important aerodynamic parts of the car since it is used to generate downforce at the rear part of the car with the air that has already interacted with the rest of the car. The air, when interacting with parts of the car, generates vortices and this makes difficult to know where the air will be moving to. This element also has a disadvantage, which is the dirty air that leaves behind it and that can negatively affect the cars positioned behind it [15].

The main different parts that make up the rear wing will be explained below and will be seen in Figure 2.5.

- Main plane: It is the main part of the wing and has an aerodynamic profile section.

- Flap: It works similarly to the main flap even though it is has smaller dimensions. This is the element that will be moved by the DRS (system that will be explained below) when activated.

- Endplates: As for the front wing, these are the vertical pieces located at the outermost part of the aileron, that like winglets, intend to reduce induced drag by preventing air located in high pressure areas to move to low pressures areas. In the rear wing it also allows the wing to attach to the rest of the car.

- Central attachment element: Vertical components that attach the central area of the wing to the car itself. Here is where the DRS actuator is located, moving the flap.

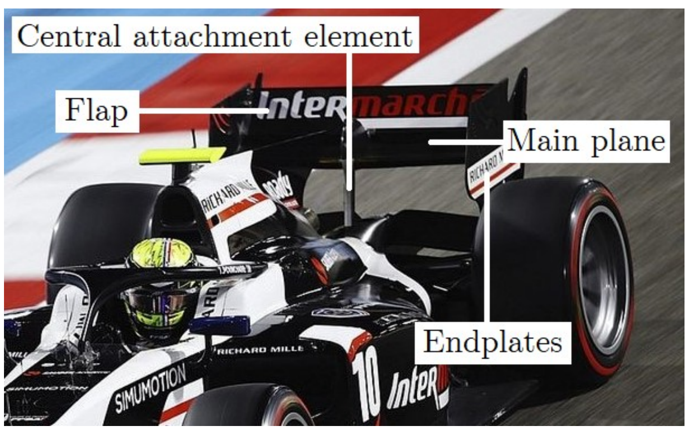


Figure 2.5: Dallara F2 rear wing [14]

The Drag Reduced System (DRS) is the mobile device that allows the upper flap to move in a parallel direction to the incoming airflow, thus, since the flap is an aerodynamic shaped section, both the drag and downforce generated will reduce with respect to its main position. This causes an increase in the speed of the car by a few extra kilometers per hour. But as said,it also reduces downforce, this is why it can only be used on certain straights of the circuit previously defined.

2.2.3 Flat Bottom and Diffuser

The flat bottom and diffuser are two of the most aerodynamically efficient elements, due to the ground effect, since a great downforce is generated. Despite generating this aerodynamic force, the drag they produce is much lower, so a great aerodynamic efficiency can be concluded. These can be seen in the Figure 2.6

This effect is due to the geometry, which is characterized by a Venturi effect (which will be explained in more detail in the next chapter), where, by Bernoulli's principle, as the velocity of a fluid increases, the pressure decreases, and since the air in the flat bottom has less pressure, downforce is generated. This flat bottom has different parts that allow the air to follow the trajectory needed to achieve a more efficient car, but also prevents the air surrounding the car from entering in this critical area in order to maintain the desired conditions inside this aerodynamic element.

However, at the rear of the flat surface, the diffuser is located, whose geometry changes so that it no longer remains close to the ground with a slight slope. This is because, unlike the flat bottom, this new geometry has the main purpose of expanding the air at the bottom of the car to the atmospheric pressure surrounding it.

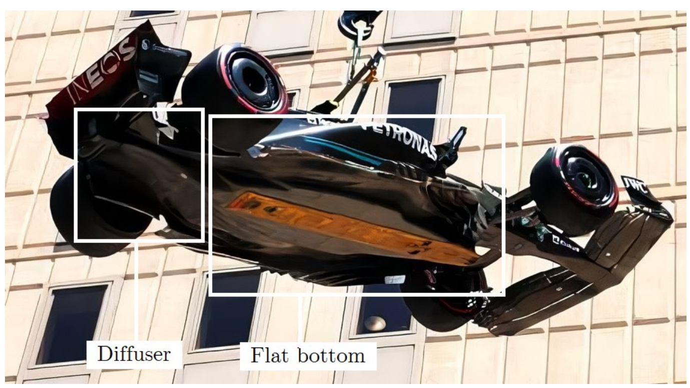


Figure 2.6: Flat bottom and diffuser of a 2023 F1 car [16]

2.2.4 Over-Wheel Winglets

As a brief description of what has been explained above, these elements have only been implemented in the Formula 1 category since the last regulation change for 2022, but the intention of these is to reduce the turbulent air generated by the wheels and divert it from reaching the rear wing.

Wheel spin, incident with air speed, disturbs the flow around it, so these new devices are intended to guide this air to the desired areas of the car to make the car as efficient as possible. These elements can be easily seen in the following Figure 2.7.

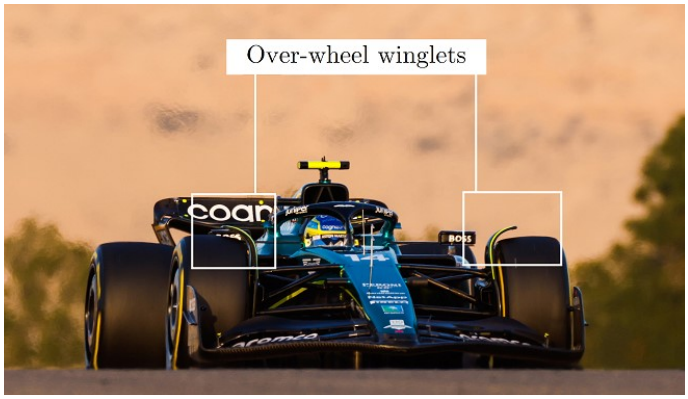


Figure 2.7: Image of a 2022 car where over-wheel winglets are easily seen [17]

Chapter 3

Fundamentals of Aerodynamics

Aerodynamics is the discipline of Fluid Mechanics that studies the effects that a fluid around a solid body causes on the solid body itself. In addition, it is important to note that the fluid being studied in aerodynamics is air, in gaseous state, because if it were others such as liquid, this branch would be called "hydrodynamics", or "gas dynamics" for other fluids.

3.1 Fluid Definition, Properties and Classification

Fluid dynamics is extremely important for many different applications, where the study of the flow is essential to understand its behaviour. This study is done by measuring the characteristics of the fluid taken into account, such as pressure, velocity, density and temperature.

Fluids can be characterized according to their properties into different groups.

3.1.1 Newtonian and Non-newtonian

Newtonian fluids are those that have a viscosity that may vary with temperature. That is, they are fluids whose stress due to viscous forces is linearly related to the strain rate by means of a proportionality coefficient (fluid viscosity) which is a property of the fluid itself.

In contrast, Non-Newtonian fluids do not have a unique viscosity, but its viscosity is dependent on the temperature and force to which the fluid is subjected.

3.1.2 Steady/Unsteady

Stationary flow is a flow that is not dependent on time 3.1, this is, a fluid whose properties are constant over time at a fixed point in space in the same situation. In addition, steady flow is beneficial to study as it is less complex and quicker to calculate a stationary.

This is, being "P" a given property of the fluid:

$$\frac{\partial P}{\partial t} = 0 \tag{3.1}$$

Conversely, non-stationary flow is flow that is not constant over time 3.2, in other words, it is time-dependent. In this type of flow, the streamlines are not constant and therefore, its particles can change their trajectory.

$$\frac{\partial P}{\partial t} \neq 0 \tag{3.2}$$

3.1.3 Compressible and Incompressible

Compressible fluids are those whose density changes significantly when subjected to pressure variations, so the volume of the fluid will be affected to this extent.

However, it should be noted that there are fluids whose molecules are at a very close distance from each other, so that despite the application of considerable pressure on this medium, its density remains practically the same as before applying it. These flows are called "incompressible", and it is important to know that their density does not necessarily remain exactly the same at the initial instant, but that their volume variation in the event of very high changes in pressure is insignificant.

For gases, to determine whether the fluid under study is compressible or incompressible, its velocity is evaluated with respect to the speed of sound. This is done by means of the Mach number (M) 3.3, which is a dimensionless measure and is calculated by dividing the velocity of the air (or of the moving object, depending on the reference system used) by the speed of sound at the conditions of study, since it varies with temperature and altitude with respect to sea level.

- M: Mach number
- V: Flow velocity
- a: Speed of sound in the fluid

$$M = \frac{V}{a} \tag{3.3}$$

This number is of vital importance for aerodynamics as it defines certain flow regimes:

- Subsonic regime: M≤0.8. Shock waves do not appear.
- Transonic regime: 0.8<M≤1.2. Shock waves appear and there are no analytical theories to calculate it. This is an area that separates subsonic and supersonic regimes.
- Supersonic regime: 1<M≤5. Shock waves appear but there are no subsonic areas.
- Hypersonic regime: M>5. At extremely high velocities, the flow induces significant heating in the layers adjacent to the flow boundary layer, leading to molecular dissociation and other chemical phenomena.

In addition, compressible effects for Mach values below 0.3 can be neglected, so that gases are considered compressible from this value onwards.

3.1.4 Laminar and Turbulent

Laminar flow is characterised by a smooth and orderly movement of the fluid, whereby the particles move in layers, following orderly and therefore predictable trajectories.

On the other hand, turbulent flow is characterised by chaotic and therefore unpredictable movements, generating recirculations, vortices and irregular eddies. This type of fluid is normally observed in situations with high velocities or when the fluid encounters elements that hinder its passage. These can be seen in Figure 3.1.

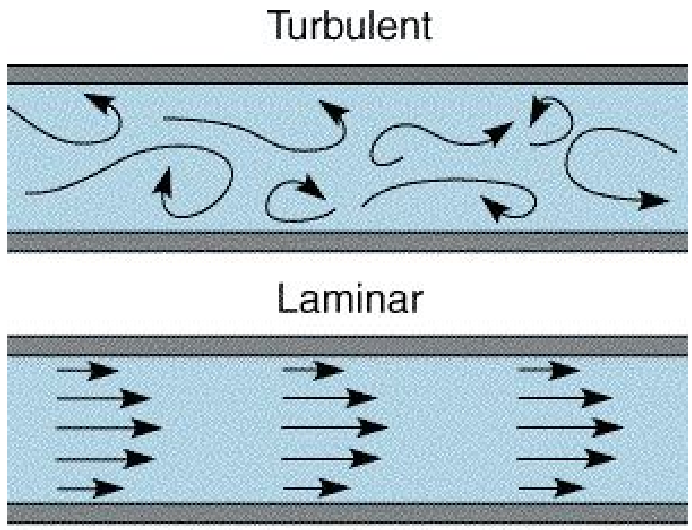


Figure 3.1: Turbulent and laminar fluxes respectively [18]

This characterisation is classified by a dimensionless number, which makes it possible to separate flows in different regimes and indicates the relationship between inertial and viscous stresses. This will be the Reynolds number:

- ρ: Fluid density.
- v: Fluid velocity.
- L: Characteristic length of the object into study.
- ν: Kinematic viscosity of the fluid.
- μ : Dynamic viscosity of the fluid.

$$Re = \frac{\rho \cdot L \cdot v}{\mu} = \frac{v \cdot L}{\nu} \tag{3.4}$$

For low Reynolds, the flow is said to be laminar, while for high Reynolds numbers, the study flow will be turbulent, with a transition regime in between, which may be gradual.

3.1.5 Viscous and Inviscid Flow

As described above, the Reynold number relates the inertial effects to the viscous ones, so it can be said that, at a reynold number lower than 1, the viscous forces are greater than the inertial ones, while when Re > 1, the opposite happens. Thus, when the Reynolds number is high, of the order of 10^6, viscous effects may be neglected, except in the called boundary layer, seen in section 3.1.5.1, where viscous effects need to be considered.

A viscous fluid is one in which viscosity takes on great importance, i.e. the layers of the fluid generate friction and resistance when moving, thus generating energy and momentum between the layers, which causes a damping effect in the fluid and a loss of energy. These fluids deform with a smooth and gradual behaviour, leading to reversible ultralaminar fluids for very low Reynolds numbers.

On the other hand, non-viscous fluids are those in which the viscosity is negligible and therefore, there is no friction or internal resistance between the layers of the fluid as it moves, leading to a behaviour that allows the flow to move in a quick way.

3.1.5.1 Boundary Layer

A boundary layer is a thin region next to a solid surface. In this region, the fluid properties, such as velocity and viscosity, undergo significant changes from the surface to the external flow. In addition, the boundary layer can have two main states: laminar or turbulent, as seen in Figure 3.2.

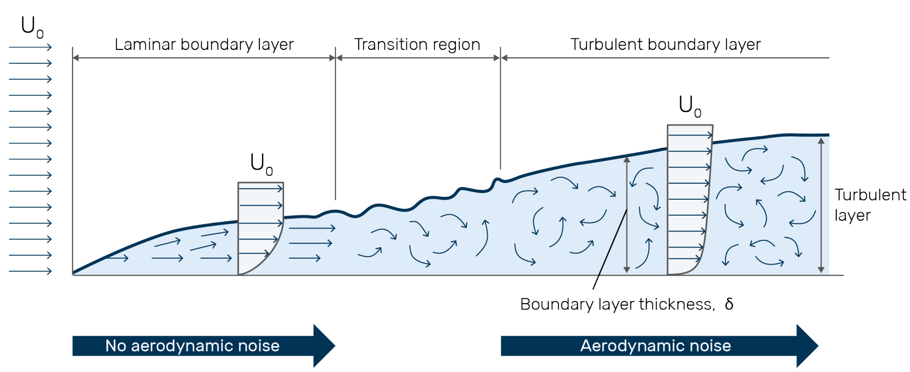


Figure 3.2: Turbulent and laminar boundary layer [19]

The formation of the boundary layer occurs due to the interaction between the fluid and the solid surface. For example, when a fluid flows over a solid surface, the fluid in direct contact with the surface moves more slowly due to friction, causing a no-slip condition, while the fluid in the outer region flows at a higher velocity. This velocity difference between the boundary layer and the external flow generates velocity gradients and causes changes in other fluid properties.

3.2 Governing Equations of the Flow

There are several main equations that describe the fundamental principles that govern the behaviour of fluids. In this section, a description of them will be presented in order to understand the study that the computational fluid dynamics software uses in order to achieve a solution for the study case.

3.2.1 Mass Conservation

The conservation of mass, also known as continuity equation, is one of the fundamental equations governing fluid flow as it states that the total mass of fluid in a system remains constant as the fluid moves, so the quantity of air that is introduced into the system is the exact quantity of air that exits it. This is expressed by means of the following Equation 3.5:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \cdot v) = 0 \tag{3.5}$$

Here, the first term will represent the change of the fluid density along the time, while the second term represents the change in net mass flow through the domain.

However, when the flow is steady, the first term is zero, since density will not change with time. Then, when the flow is incompressible, meaning that density remains constant during flow, $\nabla(\rho \cdot v) = \rho \cdot \nabla(v) + v \cdot \nabla(\rho)$, with $\nabla(\rho) = 0$, giving as a result the following 3.6 equation:

$$\nabla \cdot v = 0 \tag{3.6}$$

3.2.2 Momentum Conservation

Momentum is the product of an object's mass and its velocity. The principle of momentum conservation asserts that, within a given problem domain, the total momentum remains constant. Momentum is not created or destroyed; rather, it can only be altered through the influence of external forces, in accordance with Newton's laws of motion. Then, conservation of momentum, mathematically described in equation 3.7, describes how momentum is transferred and conserved in a fluid flow.

$$\frac{\partial \cdot (\rho \cdot v)}{\partial t} + \nabla \cdot (\rho \cdot v \cdot v) = -\nabla \cdot p + \nabla \cdot \tau + \rho \cdot Fb \tag{3.7}$$

On the left hand of the equation, the first term is the rate of change of the fluid momentum as a function of time. The second term represents the variation of momentum due to convection.

Then, on the right hand of the equation, the first term represents the negative pressure gradient, representing the pressure forces acting on the fluid. The second term is the divergence of the stress tensor τ, representing the viscous effects in the fluid. Finally, the last term is the product of the fluid density and the gravitational acceleration, representing the influence of gravity on the motion of the fluid.

This equation states that the rate of change of the momentum of a fluid at a given point in space and time is related to the sum of the pressure forces, viscous effects and gravitational influence and it is fundamental, with other conservation equations, in order to understand and predict the behaviour of the flow in different situations and to analyse the interaction between the fluid and surrounding objects or surfeces.

3.2.3 Energy Conservation

The conservation of energy is based on the principle of the law of conservation of energy, also known as the first principle of thermodynamics. This principle states that the total energy of an isolated system remains constant over time, what implies that the total amount of energy remains constant, although it can change from one form to another, such as thermal energy, mechanical energy or chemical energy.

In a closed system where there is no exchange of energy with the environment, the total energy is conserved, meaning that the sum of the energy associated with motion (kinetic) and the energy associated with position (potential) remains constant.

The conservation of energy equation can be expressed as in the following equation 3.8, and states that the rate of change of total energy at a given point in space and time is related to the energy flow due to pressure forces, thermal conduction and the presence of heat sources or sinks.

$$\frac{\partial(\rho \cdot E)}{\partial t} + \nabla(\rho \cdot v \cdot E) = -\nabla(p \cdot v) + \nabla(k \cdot \nabla T) + q \tag{3.8}$$

For this equation, the first term of the left side is referred to the change of total energy per unit volume as a function of time, while the second term is the variation of the convected energy across the volume boundaries.

On the other side, for the right side of the equation, the first term represents the energy flow due to pressure forces. The second term is referred to the energy flux due to thermal conduction, being "k" the thermal conductivity and ∇T the temperature gradient and the third term represents the heat sources or skins in the system.

This fundamental principle provides an understanding of how energy is transformed and transferred in different processes and phenomena, and is fundamental to the study and development of sustainable and energy-efficient technologies.

3.2.3.1 Bernoulli's Principle and Venturi Effect

Bernoulli's principle can be deduced from the principle of conservation of energy. This implies that, in a steady flow, the sum total of all forms of energy present in a fluid along a streamline is constant at all points along the streamline. This means that the sum of kinetic energy, potential energy and internal energy remains constant.

In other words, Bernoulli's principle states that in a steady fluid flow, there is an inverse relationship between fluid velocity and pressure. When the velocity increases, the pressure decreases, and when the velocity decreases, the pressure increases.

Furthermore, the Bernoulli's equation 3.9 relates the pressure, velocity and head of a fluid in an incompressible, frictionless fluid flow and is fundamental in understanding the principles of aerodynamics and hydrodynamics.

$$\frac{1}{2} \cdot \rho \cdot u^2 + \rho \cdot g \cdot z + p = cte \tag{3.9}$$

Additionally, by combining the principle of conservation of mass, which states that the mass flow remains constant, with Bernoulli's principle, the phenomenon known as the Venturi effect can be explained. This phenomenon occurs when a fluid flows through a constricting duct. At the point of constriction, the fluid velocity increases and the pressure decreases. In contrast, in the larger cross-sectional areas of the duct, the fluid velocity decreases and the pressure increases. This effect is shown in Figure 3.3 and is of great importance for car aerodynamics because of its close similarity to that of a car.

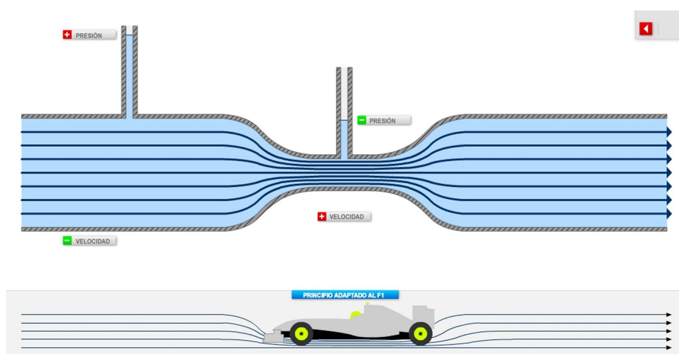


Figure 3.3: Venturi effect related to the aerodynaics of a car [20]

3.2.4 Navier-Stokes Equations

The Navier-Stokes equations (3.10, 3.11 y 3.12) are a set of partial differential equations used to describe the motion of viscous fluids, such as liquids and gases. These equations represent momentum balance and conservation of mass for Newtonian fluids, which are those in which the viscosity is proportional to the shear stress, and are a mixture of the equations above, expressed in three dimensions. These are of great importance in the field of fluid mechanics, as they provide a theoretical framework for studying and predicting the behaviour of fluids in a wide variety of situations.

However, due to their complexity, solving the Navier-Stokes equations analytically for general cases is a challenge, so numerical methods and simulation techniques are often used to obtain approximate solutions.

$$\rho \cdot (\frac{\partial u}{\partial t} + u \cdot \frac{\partial u}{\partial x} + v \cdot \frac{\partial u}{\partial y} + w \cdot \frac{\partial u}{\partial z}) = -\frac{\partial p}{\partial x} + \frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + \rho \cdot g_x \quad (3.10)$$

$$\rho \cdot (\frac{\partial v}{\partial t} + u \cdot \frac{\partial v}{\partial x} + v \cdot \frac{\partial v}{\partial y} + w \cdot \frac{\partial v}{\partial z}) = -\frac{\partial p}{\partial y} + \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + \rho \cdot g_y \quad (3.11)$$

$$\rho \cdot (\frac{\partial w}{\partial t} + u \cdot \frac{\partial w}{\partial x} + v \cdot \frac{\partial w}{\partial y} + w \cdot \frac{\partial w}{\partial z}) = -\frac{\partial p}{\partial z} + \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} + \rho \cdot g_z \quad (3.12)$$

In their most general form, the Navier-Stokes equations are a system of equations that take into account factors such as fluid compressibility, three-dimensionality of motion and time dependence. These equations can be supplemented by an equation of state relating the pressure, temperature and density of the fluid.

3.3 Aerodynamic Forces and Coefficients

When air flows around a body, it generates two main forces due to the distribution of pressure and shear stress on the surface of the body considered, Lift and Drag, which are perpendicular and parallel to the incidence of the air respectively. For racing cars, however, the importance, rather than lift, lies downforce. The resultant forces over an aerodynamic shape can be seen in Figure 3.4 below.

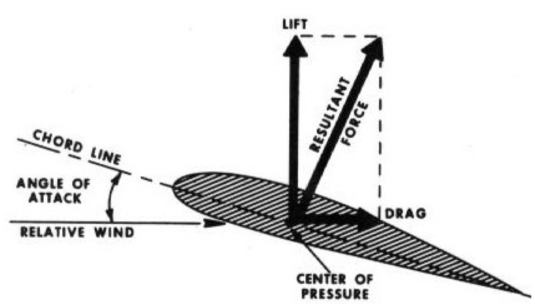


Figure 3.4: Force vectors resulting on an airfoil [21]

Lift can be defined as the pressure difference between the lower and upper surfaces of an airfoil (intrados and extrados, respectively), being its direction perpendicular to the direction of the incoming flow, and is described by the following equation 3.13.

$$L = \frac{1}{2} \cdot \rho \cdot S \cdot V^2 \cdot C_L \tag{3.13}$$

Analogously, the force opposed to the movement of a solid object though a fluid is the Drag, described by 3.14. There are different types of drag, that can be characterized in two main groups.

- Parasite drag: because of the air's inherent viscosity. It can not be eliminated or reduced.
  - Skin friction drag: as a result of the air molecules interacting with the surface of the solid object, leading to friction.
  - Pressure or form drag: produced by the wake of low pressure due to flow separation.
- Drag due to pressure difference.
  - Induced drag: due to the pressure disparity between intrados and extrados of the wing, what causes wingtip vorticities.
  - Wave drag: as a result of the entropy produced by shockwaves, leading to a reduction in total pressure.

$$D = \frac{1}{2} \cdot \rho \cdot S \cdot V^2 \cdot C_D \tag{3.14}$$

In these equations, C_L and C_D are the lift and drag coefficients, two dimensionless quantities used to characterize the aerodynamic forces acting on an object in a fluid flow, influenced by various factors such as the shape, size, and surface characteristics of the object, as well as the properties of the fluid, and are useful in evaluating the performance and efficiency and optimizing the aerodynamic characteristics of an object of study.

Chapter 4

Car Geometry

Since the aim of this study is to study the influence of winglets on the wheels of Formula 2 cars, this chapter will attempt to understand the design of the current F2 cars and explain the car design that will be used throughout this project given the simplifications that will have to be adopted in order to facilitate the calculation time to that of a personal computer.

4.1 Formula 2 Main Car

Formula 2 is a motorsport competition created in 2017, renaiming the previous GP2 Series, and it is considered a crucial stage in a driver's career, as it allows them to demonstrate their talent and skills in a competitive environment, and serve as a development platform for drivers aspiring to reach the top level of motorsport.

Cars used in this competitions are racing vehicles used in the second-tier single-seater category of motorsport promoted by the Fédération Internationale de l'Automobile (FIA). These cars are designed to offer high performance and exciting racing. F2 cars feature powerful engines, aerodynamic chassis and high-performance tyres. Their appearance and technical characteristics are similar to Formula 1 cars, although there are some differences in terms of power, technology and costs.

The single-seater used since the 2018 Formula 2 season is the 2018 Dallara F2, seen in Figure 4.1, which features an advanced aerodynamic design, with technical and performance characteristics suitable for this category.

Furthermore, Dallara is a renowned Italian company, specialising in the design and manufacture of racing cars, not only in F2 but also in other categories such as F3, 24 Hours of Le Mans among others.

Figure 4.1: Dallara F2 2018 prototype [22]

4.1.1 Technical Specifications

The Dallara F2 2018 is a vehicle that has been utilized by Formula 2 drivers to participate in races held at circuits worldwide, contributing to the training and honing of the drivers' skills.

Some of its main characteristics are the chassis, made of carbon fiber, providing an exceptional balance between stiffness and weight, and its turbocharged V6 engine, supplied by Mecachrome. The car boasts a displacement of 3.4 liters and an output of approximately 620 horsepower. The car is also equipped with high-performance Pirelli tires specifically designed for Formula 2 racing.

In terms of safety, the 2018 Dallara F2 incorporates various features, including the Halo system, a protective arch structure surrounding the driver's cockpit, meeting in that way the safety regulations set by the FIA.More detailed features about this single-seater will be visible in Table 4.1.

	Dallara F2 2018
Power	620 hp
Maximum speed	93.1 m/s
Weight (with pilot)	720 kg
Longitude	5.224 m
Width	1.900 m
Height	1.097 m

Table 4.1: Specifications of the 2018 Dallara F2 prototype [23]

4.1.2 Three Views of the F2 Prototype

The knowledge of the shape of the whole car, beyond its overall dimensions, is of vital importance to be able to design a CAD as similar as possible to the real car used in the F2 modality. To do this, the starting point is the real single seater three view drawings, seen in Figure 4.2, on which the design will be modelled to finally obtain the study model that will be shown in the following section 4.2.

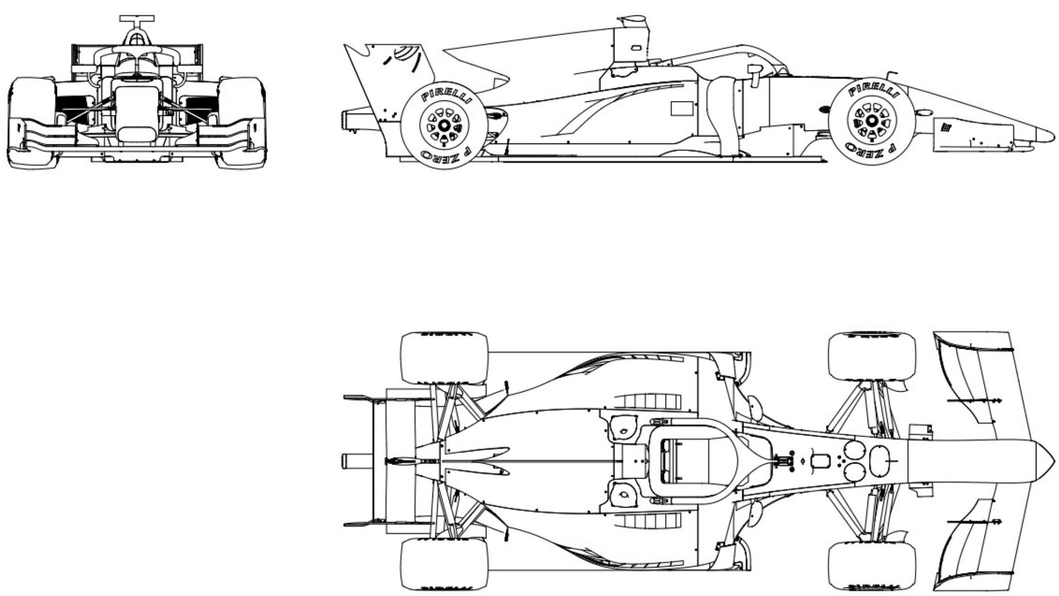


Figure 4.2: Three view drawing of the Dallara F2 2018 [Courtesy of Campos Racing]

It is important to emphasise that this design will not be exactly the one that will be studied, but that the designed vehicle will be as similar as possible to reality, always within the possibilities of the study.

4.2 Formula Car for Study

Once the current design of the F2 prototypes is known, the aim is to design a car as similar as possible within the foreseen constraints, in order to facilitate the calculation time and mesh for the CFD simulation that will be observed in later chapters.

In this design, the most general measurements will be accomplished, such as the height, length and width of the car, as well as the measurements of the wheels and surfaces, but other elements will be eliminated to facilitate the design.

The elements that will be eliminated are the ones that are not a mainly important part in the aerodynamic study over the over-wheel winglets. In Figure 4.3, the designed car is observed, where the main differences can be seen.

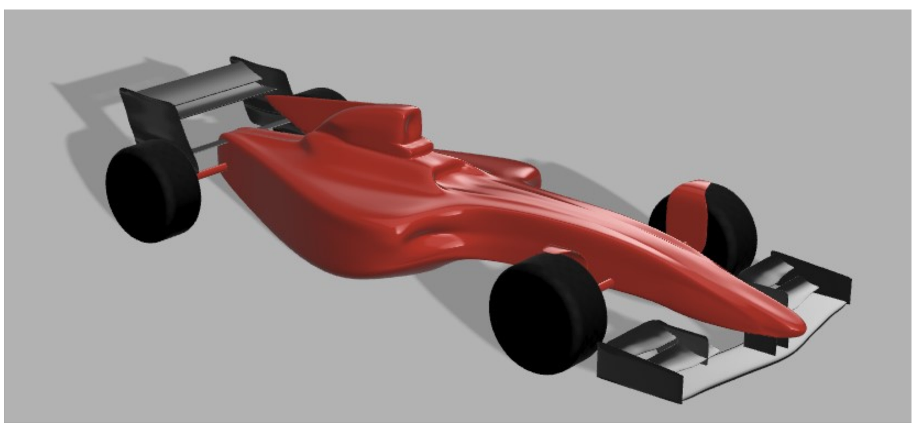


Figure 4.3: Car designed for aerodynamic study

This image clearly show that the car is very simplified with respect to the one seen in Figure 4.1. These differences are such as the following:

- The flat bottom will not be designed as in the real Dallara F2 2018 as the real design is not available in any reliable source of information, so an approximated design will be taken.
- Air ducts are also elements that are not taken into account in the aerodynamic study the car as a whole is going to be studied. Meaning, it is preferable to design a recess in the fairing of the car rather than a duct which in this case, does not lead to engines or cooling benefits.

- Similarly, gills and the pilot's seat will not be considered as well as Halo security system because its main objective is not aerodynamic but about security.
- The last of the non-considered elements are the systems that allow wheel direction, so a fixed tube will be considered to support the wheels.

As the main aerodynamic element of study in this project is the over-wheel winglets of the cars, a comparison between aerodynamic values of cars with and without them will be studied, what will lead to an understanding of the real effects this element has on the car, which clearly depend on the design of both, the car and over-wheel winglets.

Both CADs need to be considered because as these simplifications will be done, aerodynamic data will change with respect to the real F2 prototypes data. This explains the usage of non-over-wheel winglets car geometry, since it is the one that will be used to later compare the differences that these elements generate over the car. In the following sections, the design with no over-wheel winglet will be seen in 4.2.1, while the design of the winglet will be described in section 4.2.2, giving a understanding of the cases that will be simulated via CFD.

4.2.1 Design with no Over-Wheel Winglet

Regarding the car, the design of the joint between the car and wheel is key in order to adjust the setup of the simulation to later allow the wheel to move in a simple manner, so, in order to allow a movement without interfering in the design, a space as small as possible, as seen in the Figure 4.4, is left in between.

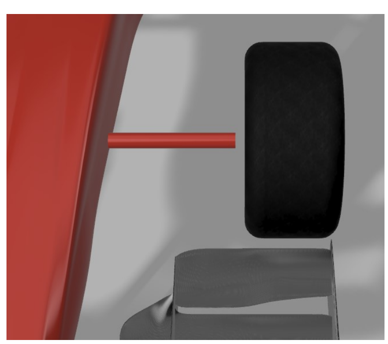


Figure 4.4: Space left between the wheels and the car

This design is the one intended to resemble the case currently found in this motorsport category, where in a closer way, it would look like in Figure 4.5. The results obtained though the use of this design will be taken as ”real data” despite all the simplifications made.

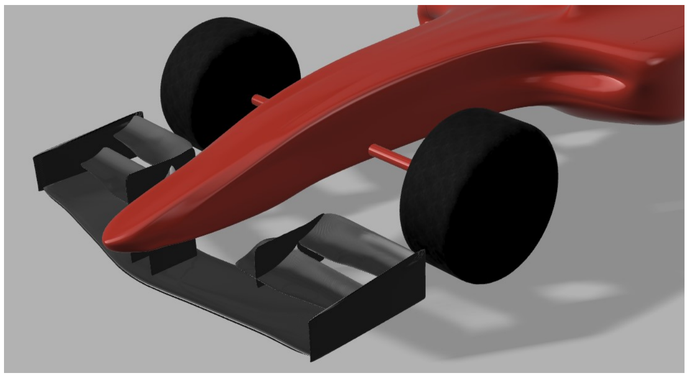


Figure 4.5: Car designed for aerodynamic study

4.2.2 Design with Over-Wheel Winglet

For the case where over-wheel winglet will be considered into study, the design and dimensions of this are fundamental for the aerodynamic results that these produce over the car, as the contribution could vary in an opposite manner with different designs. The main dimensions for the winglet are the shown in Table 4.2, and this will take on a trapezoid-like shape, even though this shape is not a completely symmetric and which does not have straight sides.

In order to understand the following table, it is important to know that the ”chord” is the length between the given Leading edge (most initial part of the shape) and Trailing edge (most rear edge of the shape).

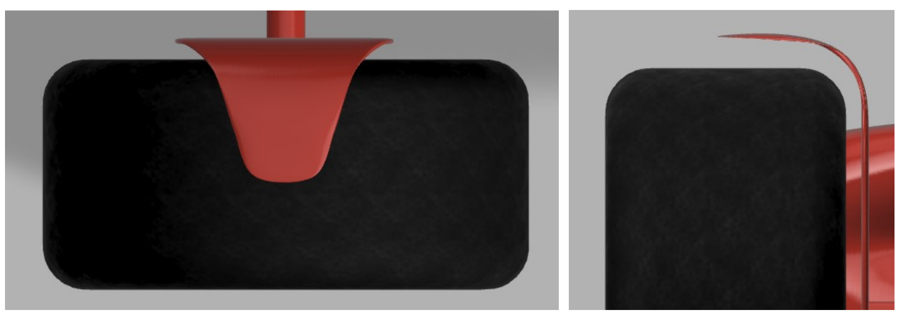
	Measurements
Distance from the wheel to the winglet	0.04082 m
Tip chord	0.10000 m
Root chord	0.31000 m

Table 4.2: Important measures of the designed over-wheel winglet

These distances are seen in the Figure 4.6, where both, a top and front view are provided in order to understand in a better way the shape considered for this project. Here, the trapezoid-like shape is easily seen in Figure 4.6(a).

The vertical distance between the wheel and the winglet is a very important factor to reaping the benefits that this aerodynamic element can offer. As seen in 4.6(b), a distance between these of 40.82 mm is left in order to allow a comfortable movement of the wheel as well as to consider a slight distance for the moment of changing wheels as pit-stop.

This distance is the one that has been considered far enough from the wheel to allow the computational mesh to not generate any problems between the two, but also to make its effects as beneficial as possible. Likewise, the distance should also be small enough to be affected by the air coming from the wheels.



((a)) Top view ((b)) Front view

Figure 4.6: Different views of the designed winglet with respect to the wheel

It may be noted that the distance considered is completely designed by the user, taking as an example various images of F1 teams to finally choose the design that apparently would be the most beneficial, given that no real measurements are known on the cars, and even less so in F2 as this aerodynamic element does not exist.

Chapter 5

Computational Fluid Dynamics (CFD) Simulation Characteristics

This chapter will study how the CFD program has been set up and will explain the models and fluid properties that have been used to obtain a result that is as accurate as possible. Every simulation has been run in STAR-CCM+, which is a simulation software provided by Siemens.

5.1 Fluid Properties and Modelling

To start the configuration of the simulation, one must first know the properties of the physical medium under study and, in turn, the properties of the fluid. For this, the characterization of fluids explained in Section 3.1 will be taken into account.

First, it is important to consider the altitude of the circuit where the car is eracing, which in this case will be the new Las Vegas circuit 5.1, placed at 610 m [24]. Las Vegas circuit gives the atmospheric conditions shown in the following Table 5.1.

	Atmospheric conditions
Pressure	94204.4 Pa
Density	1.15481 kg/m^3
Temperature	284.193 K
Speed of sound	338.003 m/s
Dynamic viscosity	$1.04282 \cdot 10^{-6}$ $kg/(m \cdot s)$
Reynolds	$4.628 \cdot 10^{8}$

Table 5.1: Atmospheric conditions at Las Vegas circuit altitude

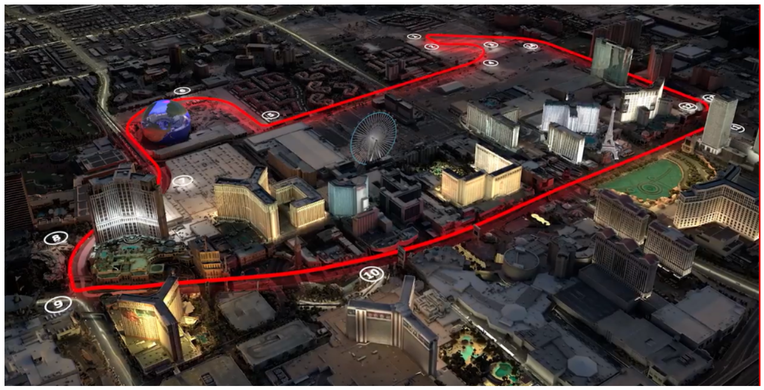


Figure 5.1: Las Vegas circuit [25]

Regarding the working fluid, air is selected, with both constant density and viscosity, resulting in a steady fluid. Since average speeds in F2 cars is around 288 km/h (80 m/s), using the Mach equation 3.3, Mach number for F2 cars is around 0.23 for the racing conditions, which gives subsonic and non compressible regimes.

Taking the racing velocity, the longitude of the car and the given atmospheric conditions in the racing circuit, Reynolds number needs to be calculated by means of equation 3.4 in order to indicate the flow regime, and as seen in the previous Table, the value is high enough to consider turbulent regime.

This turbulence needs to be reflected in STAR-CCM+ by means of using turbulence models. There are several models, but K-Omega SST is taken, which belongs to the Reynolds-averaged Navier-Stokes (RANS) turbulence models.

K-Omega is a two-equation model used to calculate in the inner part of wall regions and boundary layers due to its automatic wall treatment, in which transport equations are solved for both, the kinetic energy (k) and ω, the specific dissipation rate. This model has the disadvantage of not being precise for free-stream turbulence. Then, shear stress transport (SST) considers k-ω but also considers k-ϵ, which works precisely in outer regions. To this, a compressibility correction which enables the effects of dilatation dissipation will be allowed.

Using the described turbulence model will be of particular interest to observe turbulence and vortexes in the air, as well as the wake generated behind the car, which will give the famous "slipstream" effect. Then, k-ω SST gives an improved performance for both, near and far from the boundaries.

For the numerical aspects, second order convection segregated flow is considered, allowing secondary gradients. This model is preferable for non-viscous and low Mach numbers, which is the case of study.

5.2 Meshing Strategy

Once the continua has been set, the mesh need to be designed in order to have cells in which to calculate. The first step involved in this procedure is the selection of the domain and its dimensions, seen in Figure 5.2, where a distance of three cars is left between the starting wall of the domain and the car, and distance of 12 cars is left downstream. In addition, a distance of 18 m around the car is left.

The distances left between the object of study and the rest of the domain is very important in order not to have any interferences and reflections between the subsonic regimes and these faces. In addition, the edge effects that can be generated by the walls of the domain over the fluid are also reduced, thus allowing the flow to develop and stabilise by a sufficiently large distance.

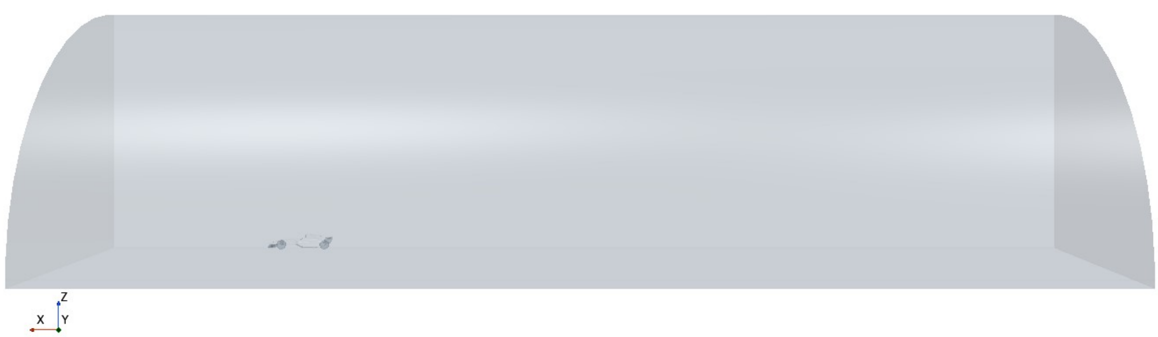


Figure 5.2: Domain considered for study

Regarding the design of the mesh, the type of cells used for this study will be Polyhedral, since for such big domains where a high level of detail is needed near the object of study, the less quantity of cells with the best resolution as possible is key in order to achieve the less calculation time as possible, and Polyhedral cells presents the best balance between these [26]. This type of cells also have more interactions with its neighbours, making better approximations of gradients but having the disadvantage of arriving to a small fluctuation in calculation values.

Other type of cells such as Tetrahedral give a better convergence in both higher calculation time, iterations and cells number, so, if exact convergence is to be obtained regardless of all other factors, this type of cell should be chosen. However, taking into account the complications of the study and the use of an own computer, the complete balance must be considered, and given that with Polyhedral, a very accurate convergence is reached (despite slight fluctuations in the values that can be assumed), this balance will take precedence.

A base size of 1 m and Target Surface Size of 35 (0.35 m) is set in order to achieve an accurate resolution, with a Minimum Surface Size of 1 (0.01 m) in order to get a very high level of detail. It is also very important to set the growth rate of the cells, as these parameters will allow the correct transfer of information between cells. Then, Surface Growth Rate and Volume Growth Rate are set to 1.3 and 1.1 respectively. To all these, an automatic surface repair and surface remeshers are enabled so the mesh is as smooth as possible, and then, make it easier to transfer information between cells.

5.2.1 Additional Custom Controls

To this mesh, some additional controls have been applied to several areas of special interest where the mesh is needed to be more accurate because of the importance of the given aerodynamic object of study. This technique is done in order to refine the most important areas, normally near the object of study, in a different way than the rest of the domain in order to achieve the wanted accuracy but with less number of cells (since refinement can be applied in the wanted areas). These areas are the following:

The first and biggest control is the Car Block created around the geometry of the car as seen in Figure 5.3, where a remeshing control is applied with a custom size of 9 relative to base size. This is, that in this area, the mesh will be refined over the rest of the domain.

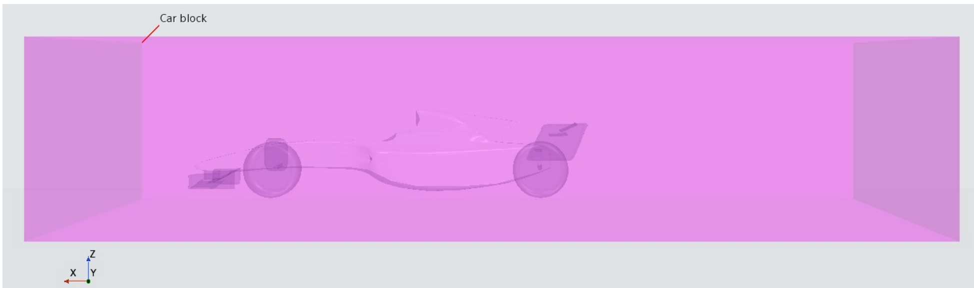


Figure 5.3: Car block refinement area

Within the car block, two further areas of special aerodynamic interest will also be refined, since as explained earlier in the section 2.2, spoilers are some of the main aerodynamic elements of a formula car. It is for this reason that a remeshing control with a custom size of 7 relative to base size is applied. These blocks surrounding the ailerons are as shown in the Figure 5.4.

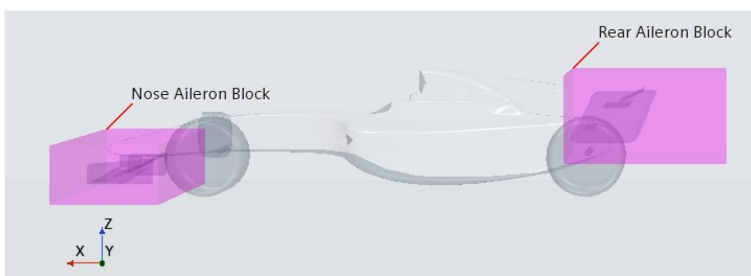


Figure 5.4: Ailerons block refinement area

The last additional block where the mesh is refined will be the one that encompasses the over-wheel winglet which is the main object to be studied in this project, given that it is an area of special interference and is wanted to calculate as accurately as possible over this area. The block is seen in Figure 5.5 below, where a refinement custom size of 0.35 relative to base size will be applied.

Figure 5.5: Over-wheel winglet block refinement area

It may be noted that in this Figure, the car with over-wheel winglet is considered, as this will be the case where the study area will be most affected and therefore where the area to be refined should be measured. However, for the case where the car is studied without this aerodynamic appendage, this volume will still be taken into account for two reasons.

The first reason will be to have the same mesh quality in both cases in order to obtain equally accurate values to study the case. The second reason is that this area remains of interest as it is where the air incident on the wheel meets the top of the wheel, moving in an opposite direction.

In addition to these created areas, there is one more factor that is very important to consider in studies where the wake left by the main object is of interest for the scope of study. In formula racing, in addition to the aerodynamics that occur on the car itself, the wake left by the car is also of great importance as it generates differences in both pressure and speed that affect the car and in the case of another car behind, it would also affect that car (the so-called slipstream). As these detachments are highly significant, the mesh to which the wake is expected to be directed shall also be refined.

This area will be done by means of a Surface control with wake refinement in the direction in which the airstream moves (this is the following vector (-1,0,0) as the direction of the airstream is opposite to the x axis). The wake refinement will have a distance of 15 m and a spread angle of 20º in order to cover all the area around the car that could be affected by the wake left by the car. However, an isotropic size with a percentage of base of 20 is set with a growth rate of 1.12.

With all these conditions explained, the complete mesh will look like this as shown in Figure 5.6, where the refinement boxes and the wake can be easily observed and differentiated from the rest.

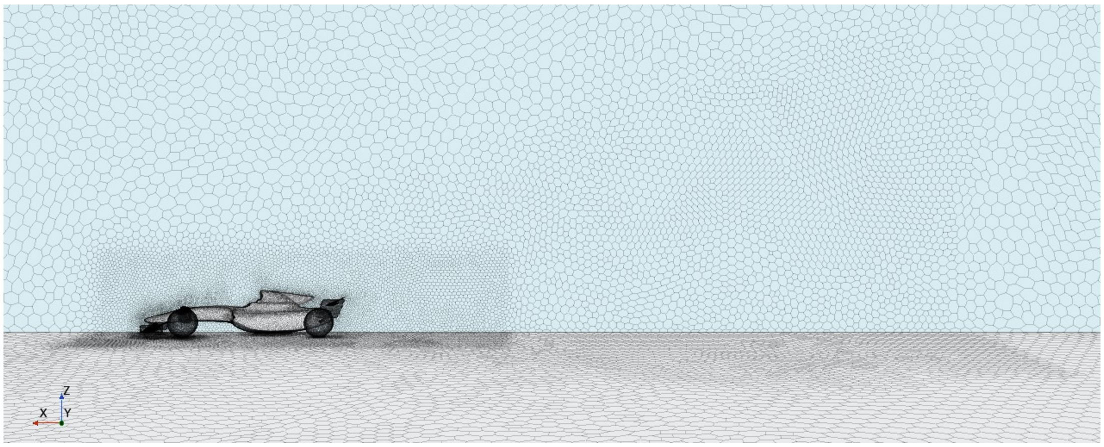


Figure 5.6: Image of the mesh around the car

With all these conditions, as expressed above, what is achieved is a greater calculation capacity in the areas of special interest, without the need to add cells to the whole domain, reaching a good level of accuracy and acceptable calculation times for this type of project.

5.3 Boundary Conditions

Boundary conditions are fundamental in computational fluid dynamics simulations for several reasons;

These are mainly the conditions that determine the problem that is going to be calculated, including inlet and outlet conditions among others, but they are also key to study the flux surrounding the objects of study, which influence velocity, temperature and pressure gradients in the regions close to the body.

In addition, they also have a great impact on the stability and convergence of the simulations at the time of calculation, and can affect the results as a whole, influencing the near-surface environment and the far flow, at the boundaries of the domain.

In STR-CCM+, the boundary conditions are selected via the "Regions" section, where the following boundary conditions shall be set:

The condition applied at the entry point is **Velocity Inlet**. This condition is applied when the velocity of the fluid at that point is known and it is desired to simulate its propagation and behaviour in the domain.

With the use of this boundary condition, the fluid velocity at the entry surface is specified, which in this case will be constant, and the direction that the velocity will take, which in the case of the study will be considered perpendicular to the entry surface on which the condition is applied. For this study case, a constant velocity magnitude of 80 m/s is applied.

Then, the condition used to specify the fluid pressure at an outlet sorface of the domain is **Pressure Outlet**. In this, the pressure of a fluid at the outlet point must be specified, which at a point where the flow is already fully developed (unaffected by reflections or interactions), will be the atmospheric pressure at the study conditions.

Since the racing circuit is Las Vegas, the constant atmospheric pressure shall be set to -7120.55 Pa with respect to the reference atmospheric pressure of 101325 Pa.

For the **"Farfield"** area surrounding the car, Velocity Inlet is also selected, but this time, several differences are taken. Since the air flowing in a parallel direction to the car needs to be simulated in this boundary, a flow direction by components is considered. The velocity will have a constant magnitude of 80 m/s but now in the direction of (-1,0,0) according to the axis used.

As the case study is a car with symmetric geometry along a longitudinal plane, the **Symmetry plane** boundary condition can be utilised, only having to simulate half of the car, reducing the size of the simulation domain.

By using this condition, it is established that the properties and behaviour of the air are symmetric on both sides of the plane, considering the velocity in a direction parallel to the plane (as it does not allow flow penetration through this condition).

The car, floor and wheels, will use a condition of **Wall**, but since the car is running at 80 m/s, both the floor and wheels will be also moving, being the car itself the only surface of these that will not move.

In order to simulate the movement of the floor, its boundary condition can be specified with a constant velocity in the form of a vector, simulating then that the wall surface is moving with (-80,0,0) m/s.

For both, the rear and front wheels is more complex because they are surfaces that rotate rather than move along an axis.

To simulate the rotation of the wheels, a reference frame will have to be created for each of the wheels, as the motion of the wheels will vary the behaviour of the simulation. Therefore, the reference frames will be located on the axis of each of the wheels, in the positive direction of the y-axis.

The rotation rate will be calculated by means of the following equation 5.1, where ω is the angular speed, V is the linear speed, and R is the radius of the wheels.

$$\omega = \frac{V}{R} = 233.3307 rad/s \tag{5.1}$$

Once all these boundaries conditions are set, one can proceed to simulate the case in the CFD program, as both the mesh and the conditions are already entered.

5.4 Mesh Independency

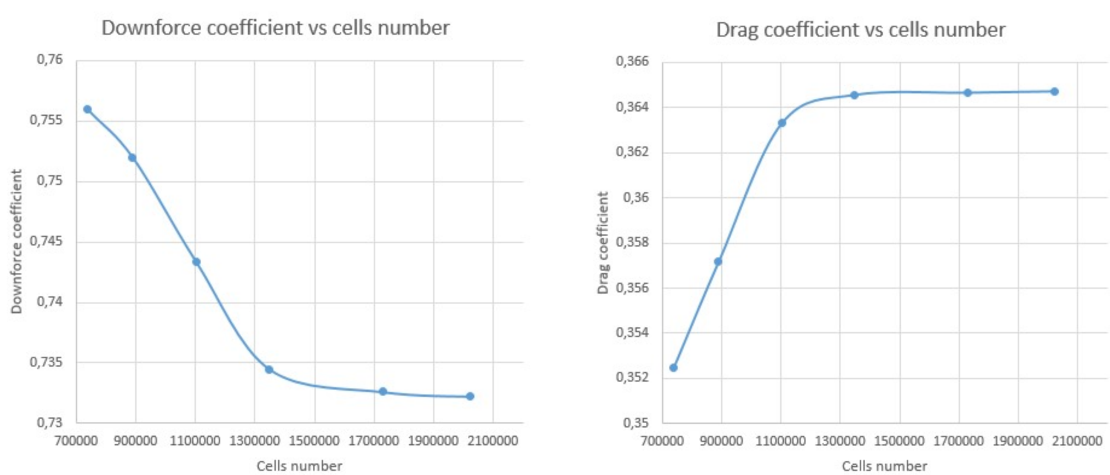
Mesh independence in computational fluid dynamics refers to the state where the results of a simulation remain unchanged with respect to changes in the mesh resolution. To determine mesh independence, a convergence study is typically performed by running the simulation with different grid resolutions, achieving a case which is not significantly affected by the choice of grid size.

Having a mesh result independent from the grid size taken provides confidence in the accuracy and reliability of the results. It ensures that the solution is not excessively influenced by the grid and that further refinement of the mesh will not significantly alter the outcome. This also allows for more efficient simulations by enabling the use of coarser grids without sacrificing accuracy.

Before proceeding to the study of the results obtained with the simulations, it is very important to know that the study grid is sufficiently accurate to be considered independent. That is, even if the number of cells is increased by 50 %, the values of the main parameters do not change substantially.

It may be noted that for mesh independence study, in order to have a lower computational effort than with the previous conditions explained, the movements are not considered, so none of the values seen in this section will be the final ones used for the study.

To ensure mesh independence, several calculations of both, drag and downforce coefficients will be done with different meshes, changing the values for the target surface size (in the three refinement blocks and the overall domain). The variations of the coefficients are shown in Figures 5.7(a) and 5.7(b).



((a)) Downforce coefficient

((b)) Drag coefficient

Figure 5.7: Forces coefficients acting on the car for different meshes

Even though the values calculated in these six different meshes seen in Figure 5.7 do not differ much from one another, a stabilising trend is observed for both coefficients for cells number over 1300000.

Then, in Table 5.2 the values of the most accurate mesh and the fourth mesh seen in the figure are stated in order to establish how cost-effective each one is.

	Cells number	**Downforce coefficient**	**Drag coefficient**
Fourth mesh	1348686	0.73445	0.36457
Refined mesh	2023087	0.73222	0.36482
Difference	40 %	0.3045 %	0.042 %

Table 5.2: Mesh results for the two main meshes considered

Once the variations in the results are seen, being small enough to consider both of them accurate, and considering that increasing significantly the number of cells is not time-effective, increasing the computational effort and time of simulation, the fourth mesh seen in the figures is taken, with a number of cells of 1348686. The mesh characteristics explained in previous sections refer to the chosen mesh.

Chapter 6

Sensitivity Study

Once an independent mesh is generated and verified to converge, the results obtained from the STAR-CCM+ simulations can be presented and analysed. In this section, the main focus will be on the velocity and pressure fields.

To ensure the reliability of the simulations, the convergence criteria considered for the cases will be explained in more detail in Appendix A below. Literature will serve as a slight comparison, but no precision will be achieved as many simplifications over the geometry have been done.

As the study is performed in 3D, vortices will be an important factor to take into account in this simulation, so new views and planes will have to be generated to observe more clearly the behaviour of the air flow around the F2 prototype. Furthermore, since the car is in subsonic regimes, no shock waves or transonic effects will be seen in this study.

In this chapter, two simulations with two different geometries will be studied, for the same atmospheric conditions and simulating the same car speed, leading to the only difference between the two simulations being the absence or presence of the object of study, the over-wheel winglet, leading intuitively to a small difference in the number of cells due to the absence of refinement in the vicinity of this surface for the case without winglet.

The comparison between the two geometries will be the subject of the next Chapter 7, where results will be obtained for the car as a whole.

6.1 Analysis of Aerodynamic Performance with no Over-Wheel Winglets

First, the analysis of the case study of the car without over-wheel winglet will be carried out separately from the case with over-wheel winglet, that will be studied below.

Before starting the study only on the area of special interest, the whole car will be considered in order to understand in a simpler way both velocity and pressure fields, which will be observed in Figures 6.1 and 6.2.

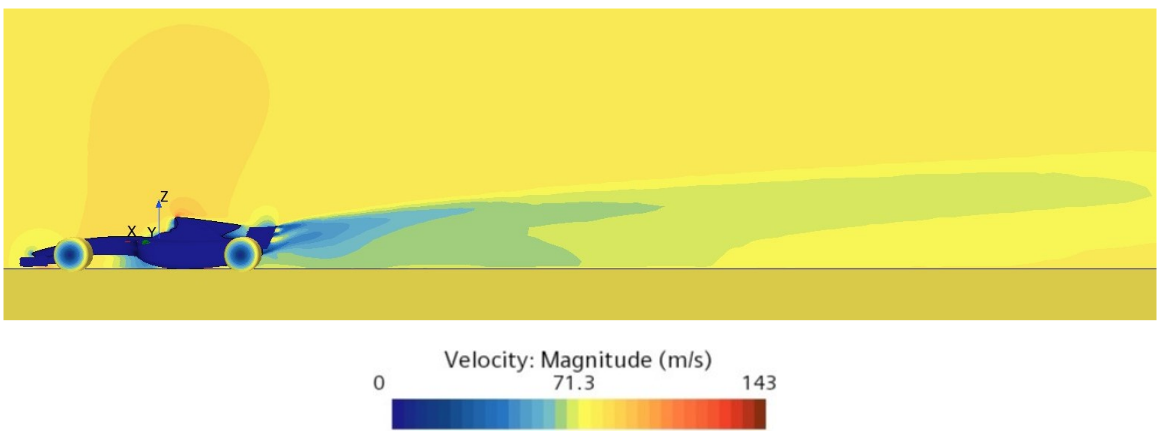


Figure 6.1: Velocity field of the F2 prototype with no over-wheel winglet

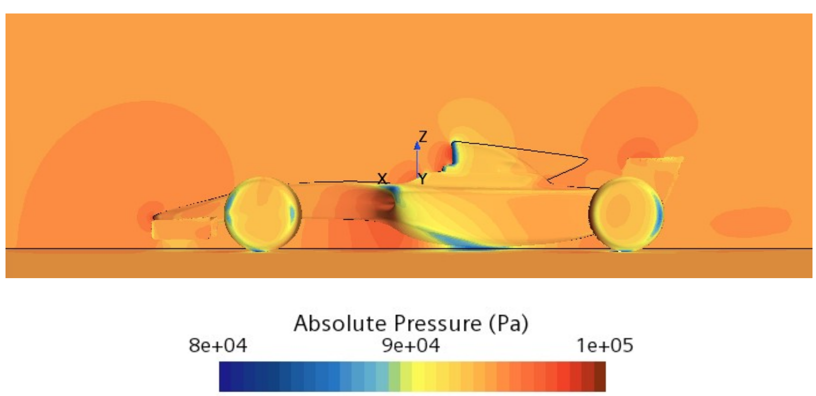


Figure 6.2: Pressure field of the F2 prototype with no over-wheel winglet

As expected, the magnitude that changes the most is the velocity, so the project will focus more in detail on this for the studies, because after all, the aim is to study the streamlines, which in some way are represented by the behaviour of the air, seen in the velocity field. Nevertheless, the car will also be studied more closely to prove the main aerodynamic elements of the car.

The main purpose of Figure 6.1 above is to observe the effect of the wake left by the car moving at 80 m/s, which can be said to measure a total of three and a half cars. That is to say, apart from generating this flow separation at the rear of the car, which will generate a pressure drag that will slow it down, opposing the movement, it will also affect the cars that are placed at this distance behind the main car.

This wake left, affects the car placed behind because the car behind recives the dirty air from the car at the front, so that pressure difference between front and rear of the second car is reduced, and therefore, its total pressure resistance, leading to an increase of increasing the top speed of this one.

It may be noted that both, the floor and wheels are moving in the given direction, which in the case of the floor, this will be the same as for the airstream, whereas for the wheels, it is important to make sure that they are moving in the direction desired. In Figure 6.3, the direction of rotation is proved to be as anti-clock wise.

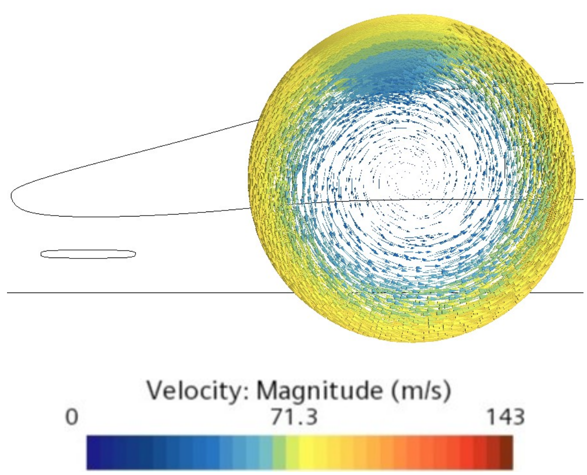


Figure 6.3: Velocity vectors over the wheels

Looking closer at the car, as shown in the following Figure 6.4, some important studies can be carried out on the theory explained above.

The first area to focus on would be the first incident point on the front wing, where a stagnation point is seen. This is, a point where the fluid comes to a complete stop and its velocity is reduced to zero due to the presence of an obstacle or a sudden change in the flow direction. This is what happens in the incident point of the wing (and other surfaces, also seen in the rear wing and intake of the car).

There is another area of interest at the top of the car's intake, where high speeds are reached. This is the same as in the airfoils because of the curvature of this surface and the difference in pressure compared to the lower area. An increase in curvature, generally increases the lift generated by an airfoil at a given speed.

It is very important to consider also the lower velocity areas, where there is either a stagnation zone as explained above, or a detachment zone. This second possibility is the given in several main areas. One of the areas would be in the front part of the cockpit, where the helmet of the pilot would be placed, where a stagnation area is produced by the presence of the solid itself. Then, another area is placed behind the so-called "shark fin", affected by the wake left by the cockpit.

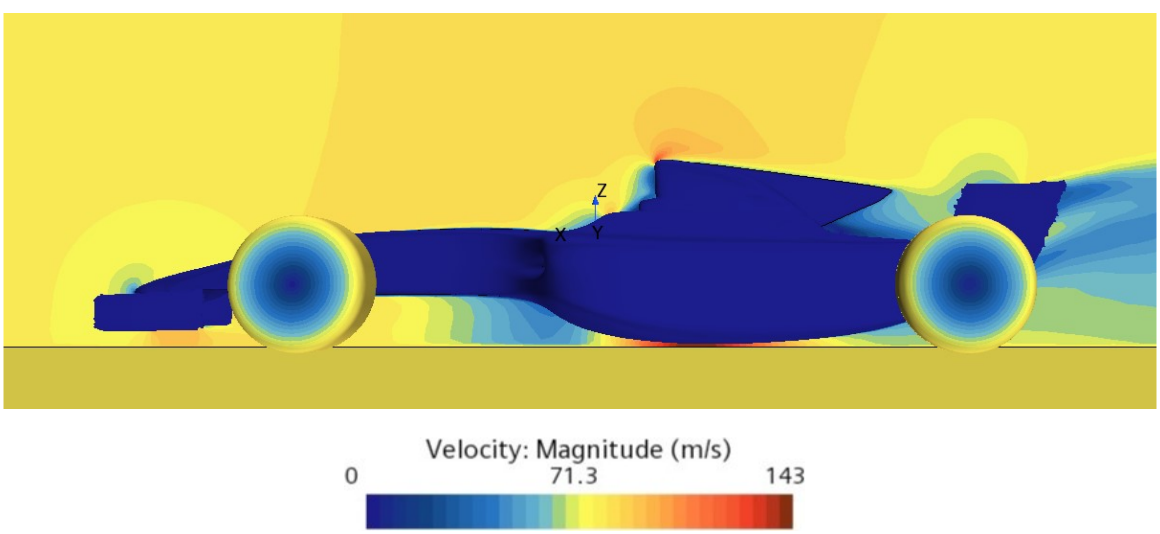
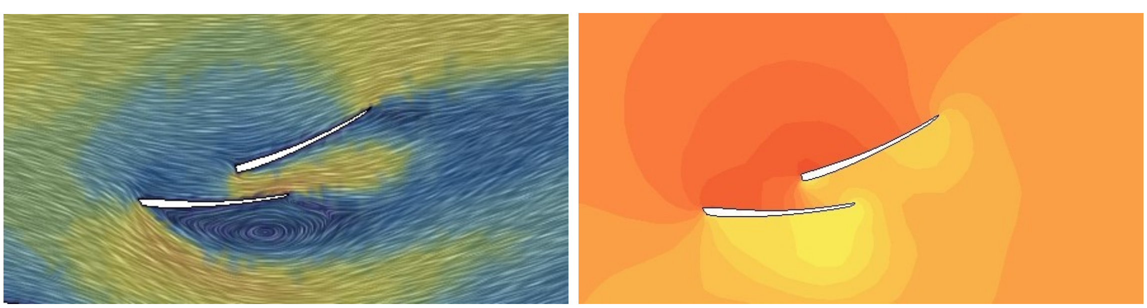


Figure 6.4: Closer velocity field of F2 prototype into study

On the other hand, one of the most important aerodynamic zones is the lower area of the car, where the Venturi effect generated in this area has already been explained in 3.2.3.1. The image shows this increase in speed that is generated in the lower area of the car, reaching a speed close to double the speed of the air incident on the car. This increase in speed generates a lower pressure in this area of the car (as can be seen in Figure 6.2), thus generating a "suction" in this area, generating a downward force that will keep the car stuck to the ground.

The other main aerodynamic area is seen behind the rear wing, which generates a wake that clearly influences the performance of the car. In order to see in a more understandable way the detachment of the air over the rear wing of the prototype car, Figure 6.5 will provide the streamlines and the pressure field on the symmetry plane. In this area, a zone of higher velocity is observed between the two airfoils, being influenced by the channel formed between the two, as well as a large recirculation occurring in the area below the first aileron, where the air is stripped from the surface, generating vortices that lead to a suction area in the lower side of the second aileron, generating also downforce in the rear of the car (it may be noted that this same effect occurs on the front wing, proving the theory explained). Finally, the airflow is displaced slightly upwards at the rear, this being the wake left by the car.



((a)) Streamlines and velocity overlapped ((b)) Pressure field

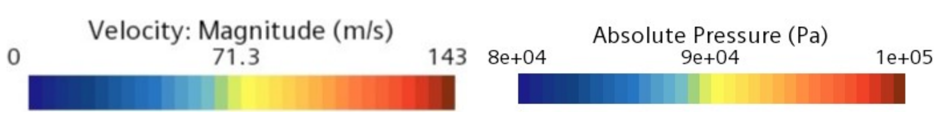


Figure 6.5: Velocity and pressure fields for the car with no over-wheel winglet in the area close to the rear wing

6.1.1 Main Area of Study

Once the general aerodynamic effects on the car with no-winglet are seen, a more focused study over the area of interest for this project will be done.

In order to study better the influence of the wheel over the air incident on it, an auxiliary plane parallel to the plane of symmetry passing through the wheel shall be created. In this way, the behaviour of the air in this area will be known, and later it will be possible to compare it with the case with over-wheel winglet that will be studied in the following section, observing the differences that this element produces over the airflow surrounding it.

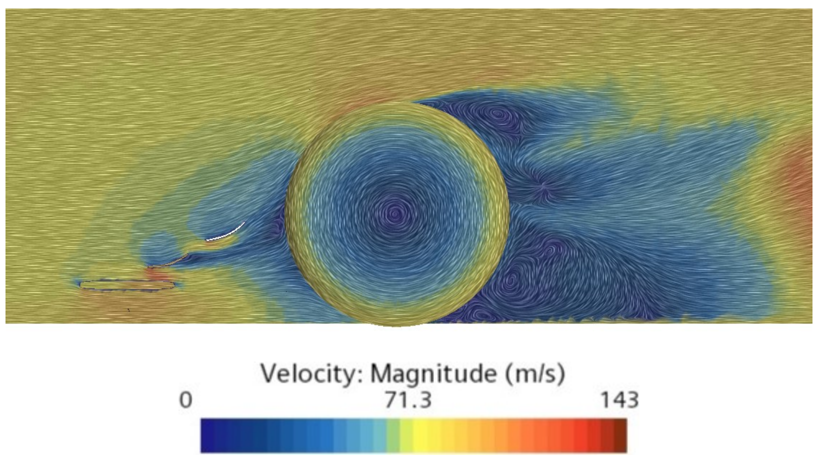


Figure 6.6: Streamlines and velocity overlapped on the front wing of the prototype

By means of this auxiliary plane created, it is observed that the air passing through the top of the wheel is displaced slightly upwards in the area of interest. In addition, there is a large stall area where very large swirls are formed, generating high aerodynamic drag in the rear of the wheels.

A similar behaviour to this is created in the rear wheels of the car, but these are not highly relevant in the overall aerodynamics of the car since the air deflected here does not affect other areas (while the air deflected by the front wheels and other aerodynamic elements such as the front wing, do affect the rest of the car).

Now, to better understand the behaviour, the following Figures 6.8 and 6.7 with the 3D streamlines on the front wheels will be observed, in order to see not only how the flow behaves on the same plane, but also along the wheel and the movement of this flow towards the car.

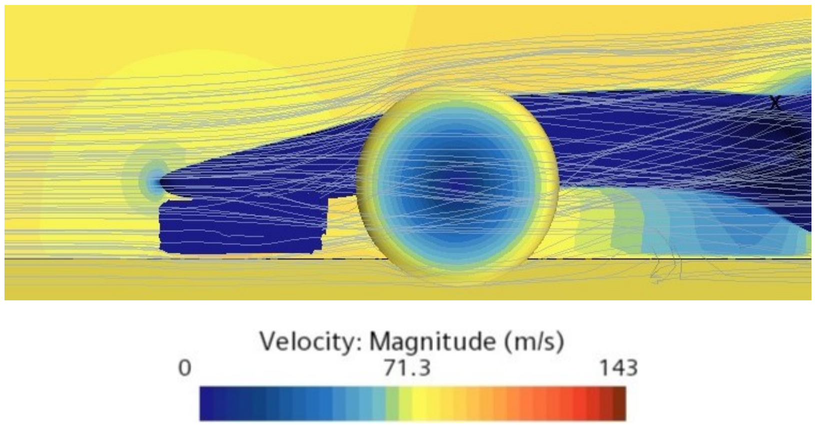


Figure 6.7: 3D streamlines over velocity field view from the side of the car

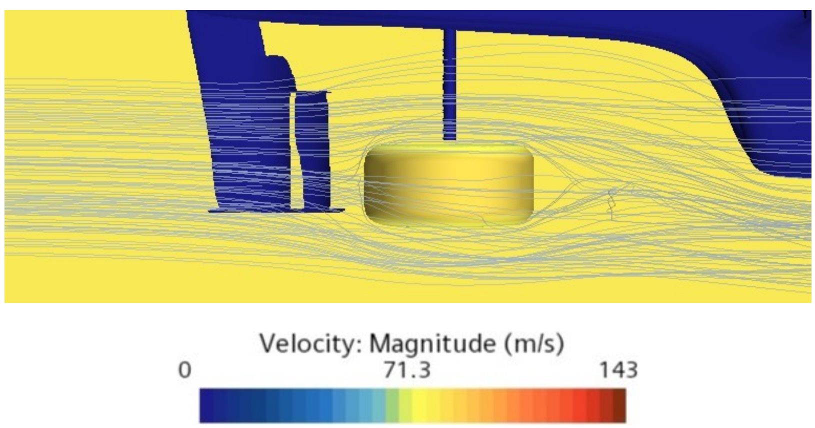


Figure 6.8: 3D streamlines over velocity field view from the top of the car

These images show that the airflow incident on the wheel, as expected and as seen in igure 6.7, is displaced upwards, thus no longer impinging on the car and cannot be redirected to other aerodynamic surfaces of interest. In other words, the flow that is displaced upwards by the effect of the wheels is not reattached to the car, but leaves the area of interest for the car, but without generating high turbulence.

On the other hand, in the top view, it can be seen how, due to the influence of the surfaces and the bodies, the incident flow is displaced towards the outside of the car, and can be affected again by the pontoons of the car and thus take advantage of the air to generate the desired forces on the car.

It is important to know that the air that is released from the car cannot be recovered and therefore used by the car. This air is the one shown in Figure 6.8, which does not follow the shape of the car but follows a straight direction. This is generated due to the difference in directions between the incident air velocity and the effect of the wheel, whose velocity at this point is opposite, thus generating a vertical displacement.

Finally, once it has been possible to observe and study the behaviour of the flow in these subsonic conditions, one can proceed to obtain the values of the coefficients of interest of the prototype studied for its configuration without over-wheel winglets, where the reference area shall be the frontal area of the car. These coefficients obtained by CFD simulation can be seen in Table 6.1. It should be emphasised that when talking about downforce, the sign has been inverted (z axis was positive upwards) to make the concept more understandable.

	Values obtained
Downforce coefficient	1.113196
Drag coefficient	0.338652

Table 6.1: Values obtained by CFD for the case with no over-wheel winglets

It should be noted that, among the coefficients calculated by means of the STAR-CCM+ software, with the considerations explained in previous sections, and as expected, the downforce coefficient (the force opposite to lift, which is the generally studied in aerodynamics) is higher than the drag coefficient.

A big difference between the both of them is seen, being Downforce coefficient (positive downwards) one order of magnitude higher than Drag coefficient. It is of great importance the fact that the Downforce coefficient is higher than 1, that when applying the forces expression seen in Equation 3.13, an increase in this force will be seen. Oppositely, the Drag coefficient will decrease the rest of the term seen in Equation 3.14, giving as a result a lower force.

6.2 Analysis of Aerodynamic Performance with Over-Wheel Winglet

Once the case without an over-wheel winglet has been studied in depth. That is, what would be the case of the current F2 prototype that has been in use since 2018, a study of the main case of study of this project (with the over-wheel winglet designed), will be done.

First, a general study of the complete car will be done, which will not differ much from the seen in Section 6.1. This is, the difference will not be noticeable among the far areas, but in the closer zones to the main area of study, generating slight differences in the air flowing around.

Then, in Figure 6.9, the whole velocity over the car with this main aerodynamic appendix element will be seen in order to understand the behaviour of the velocity around the car. A very similar velocity field to the previous case, seen in Figure 6.1 is seen, knowing then that this over-wheel winglet does not change the aerodynamics of the car drastically.

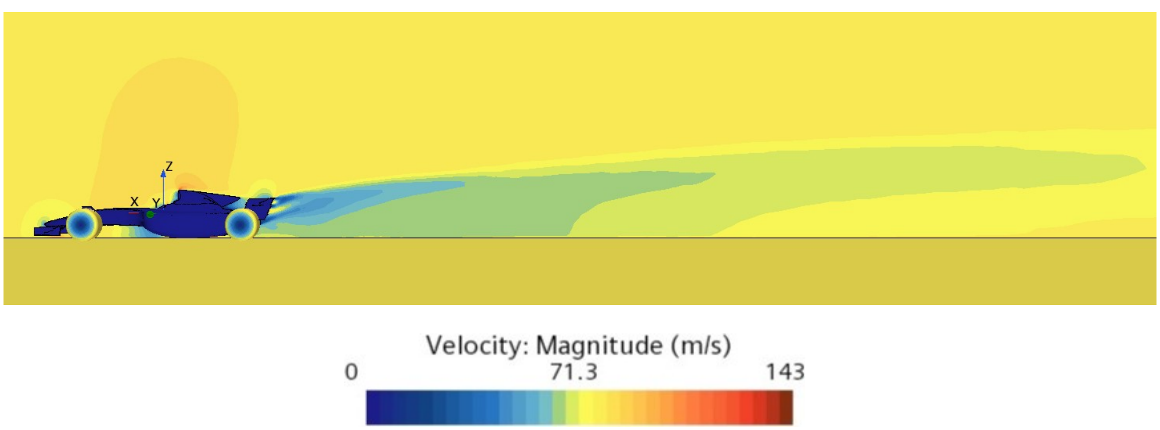


Figure 6.9: Velocity field over the F2 prototype with over-wheel winglet

The main purpose of the previous figure is to observe the wake left by the car moving at a velocity of 80 m/s, which length is of about four car distances behind.

Apart from the length of the wake, it is also important to observe the velocity gradients on it, that appear to be smooth, therefore, the F2 prototype with over-wheel winglets generates a wake of great distance but that affects smoothly. This is, that wake, as explained before, affects also the car that can be behind the main study car since, it affects the objects placed at a distance similar to the one of this lower velocity area.

For a car, it is very important to know the air flowing through it, as the direction or development of the air will play a key role in the behaviour and forces of the car itself, and hence its performance. So, if one car encounters the wake left by another, the more laminar it is, the more it will benefit the second car, leaving more possibilities for overtaking, as less dirty air will be encountered.

It can therefore be said that the wake left by a car is highly dependent on the geometry of the car and its ability to carry the air towards the areas of interest, both for the car itself (due to the Drag force the car will generate from both the length of the wake and the turbulence on it) and for possible cars behind it, which can benefit them more or less, and therefore reach different top speeds that can be used to overtake rivals.

In addition to studying the velocity, although the project will not focus on the study of pressure, the pressure field will be observed in the following Figure 6.10.

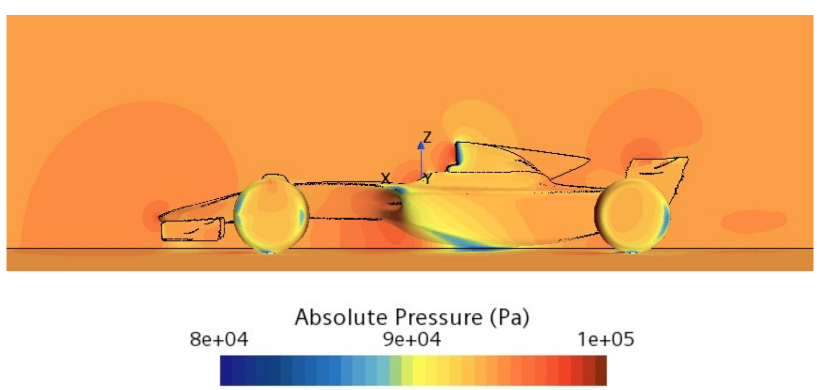


Figure 6.10: Pressure field over the F2 prototype with over-wheel winglet

Bernoulli principle can be corroborated in the vicinity of the car, as it can be seen that in areas of higher velocities, lower pressure will be found. However, in the zone affected by the wake, something different occurs, as this zone is influenced by the flow detachments on the different surfaces instead of being affected by the velocities of the flow itself, not leading to the given relationship (the higher the velocity, the lower the pressure).

Since this study has been carried out under the same conditions as the case observed in the previous section, it can be assumed that the wheel rotation is correct for a forward movement of the car, as it occurs in the same direction as before, as explained in 5.3, meaning a positive direction to y-axis. As well as the wheels movement, the floor will also follow the direction of the airstream to simulate that the object moving is the car, following a direction opposite to the x axis direction and a magnitude of 80 m/s.

Now, in Figure 6.11, a closer view over the car can be taken, where as in the previous section, important aerodynamic concepts can be studied. The overall behaviour of the car is very similar to what has been previously studied, leading to small differences due to the fact that the air affected by the newly introduced aerodynamic element can be redirected towards surfaces of interest.

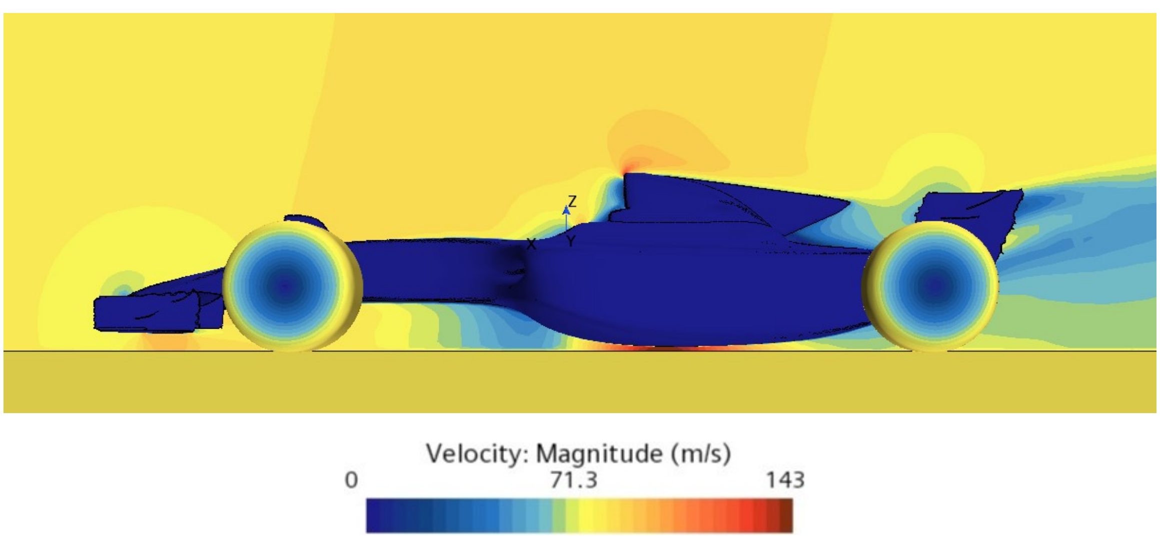
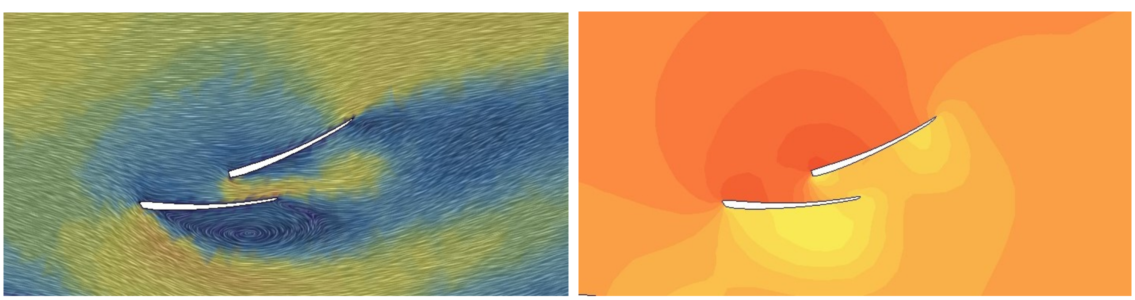


Figure 6.11: Closer velocity field of F2 prototype with over-wheel winglet

In this image, as in the previous case study, the areas of interest are the same, with special relevance being given to the stagnation areas where the air speed is reduced to zero because it meets a body at that point. in addition, the behaviour of the air in the upper part of the cockpit is the same as before, increasing the speed in the initial part of the "shark-fin".

But, the two main areas are again the flat bottom, creating a Venturi effect and therefore generating a remarkable force towards the ground as already explained, and the wake left at the rear of the car, where it displaces the air upwards due to the diffuser and the expansion of the airflow that comes compressed from the flat bottom, in addition to the air that passes through the rear wing. Both airflows come together to create a single wake.

Due to the effect of the new aerodynamic appendage, it is worthwhile to look again at the streamlines on the rear wing, as the air may have been influenced by the geometry of the car and ended up in this area, which was the main theoretical objective. In the Figure 6.12 these streamlines and the pressure field are observed, showing a large acceleration in the lower parts of the airfoils.



((a)) Streamlines and velocity overlapped ((b)) Pressure field

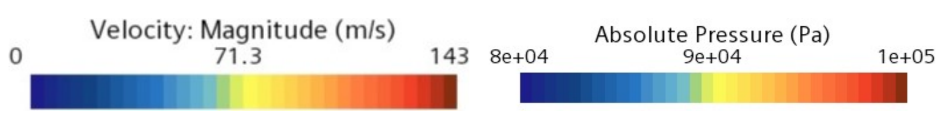


Figure 6.12: Velocity and pressure fields for the car with over-wheel winglet in the area close to the rear wing

6.2.1 Main Area of Study

Once the most relevant aspects of the aerodynamics of the F2 prototype designed with the over-wheel winglets have been studied, the area around these new aerodynamic elements can be studied to better understand the effect that this generates on the air in the vicinity of the winglets.

To enter into the study of this area of interest, we will follow a script quite similar to the one used in the previous section, where the first thing to do is to project the streamlines together with the velocity on an auxiliary plane located in the middle of the wheel, as shown in the following Figure 6.13. Although the winglet above the wheel is not well observed, if you look closely, one can see its location as well as its laminating effect over the air flow.

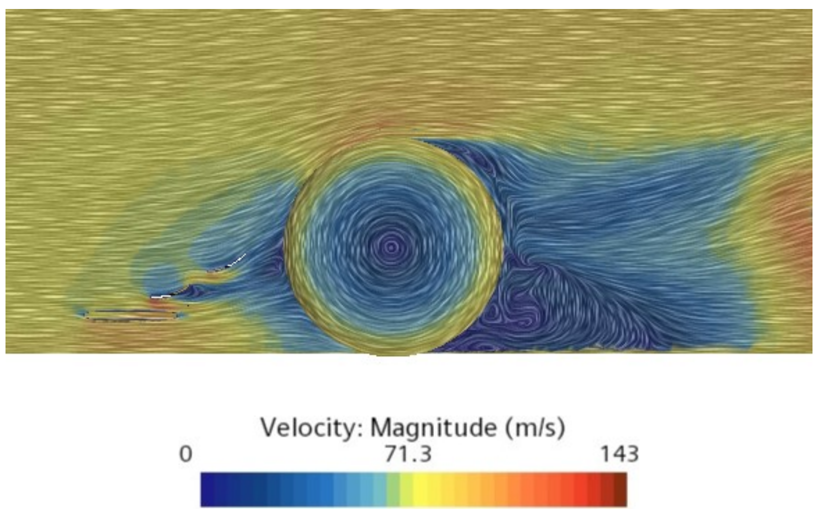


Figure 6.13: Streamlines and velocity overlapped on the front wheel of the prototype with over-wheel winglet

In addition, it is observed that due to the effect of the aerodynamic surfaces of the front wing, the air passing through the front wing moves towards the top of the car, generating a downward force (as is being sought with this design). Once it passes this surface, the air meets the wheel, which rotates in the opposite direction, generating stagnation points in the areas of the wheel where this air ”collides”.

The air that moves towards the upper part of the wheel is affected by the designed over-wheel winglet, diverting this air towards a new direction of displacement practically horizontal, observing its laminating effect which, as will be seen in the next images, will be of great interest, being able to be used by the rest of the car to redirect it as desired.

In order to better observe this effect not only on the created auxiliary plane but also in three dimensions, the following Figures 6.14 and 6.15 are obtained.

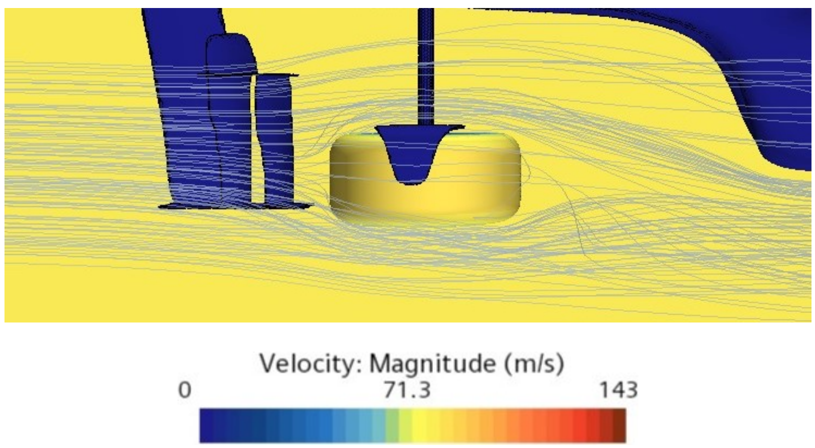


Figure 6.14: 3D streamlines over velocity field view from the top of the car with over-wheel winglet

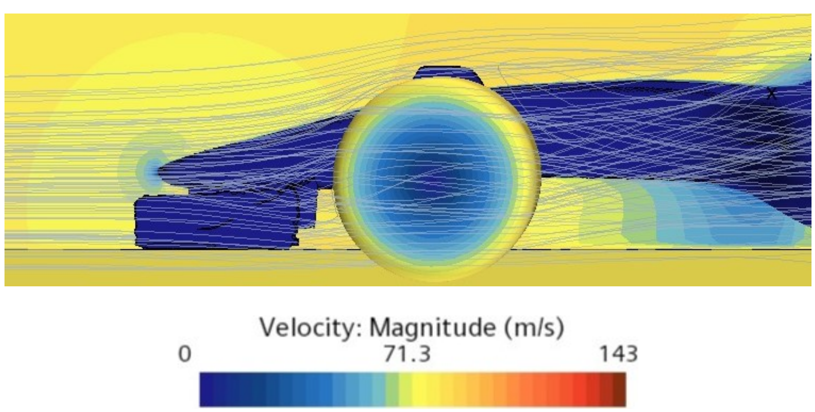


Figure 6.15: 3D streamlines over velocity field view from the side of the car with over-wheel winglet

Along the area encompassing the wheel, it is observed that this air, as seen above projected on the plane, tends to move upwards until it is impacted by the presence of a smooth element in a given position, so that by means of the boundary layer, the flow will stick to this horizontal surface and it can be said that it re-laminates the flow coming in another direction.

Once all these studies have been carried out and the effect of this surface is understood, where what was theoretically intuited has been fulfilled, the coefficients of interest can be calculated for the case of the prototype created with the over-wheel winglet. These coefficients calculated by means of Computational Fluid Dynamics STAR-CCM+ software will be seen in the Table 6.2 below.

	Values obtained
Downforce coefficient	1.126040
Drag coefficient	0.351189

Table 6.2: Values obtained by CFD for the case with over-wheel winglets

As seen, again, differences are observed between the orders of magnitude of the coefficients of interest, with Downforce (which has been defined positive downwards since this is what is intended with these racing cars) being greater than the drag coefficient. These coefficients will be used in Equations 3.13 and 3.14 in order to obtain a dimensional value for the forces that the single-seater suffers at the given study velocity.

Chapter 7

Results and Discussion

In the previous chapters, the main concepts of aerodynamics have been explained in order to understand the case study as well as to define how to introduce the conditions in the Computational Fluid Mechanics simulation program, STAR CCM+. Thus, with the values and simulations observed in the previous chapter, it is possible to obtain several results and discussions on the subject.

In this chapter, both results obtained in the previous sections 6.1 and 6.2 will be compared in order to observe the differences between the two cases and finally to make decisions on what is better for the case study and for the F2 prototypes taking into account the advantages and disadvantages of each of the case studies, and being aware that the study geometry is not exactly the same as the geometry of the current F2 as well as the design of the over-wheel winglet, which is a design that could be optimised.

The comparison between the cases studied will be done by means of both, values calculated for the coefficients and images in order to understand the differences of the values and the way the airflow affects each of them. To understand the values given by each case study, it is very important to know how the new surface created on the wheel affects it, since its objective is to laminate and redirect the car towards areas of greater aerodynamic influence in order to obtain various benefits, so as to better understand how these values have changed and not to make decisions only by observing the values.

7.1 Comparison over the General Areas of the Car

In order to make the comparison between the two case studies, first, aerodynamic differences will be observed on the car in general, without focusing on just one specific area.

For this study, it is reasonable to start by comparing the total velocity field, looking at images large enough to focus the study on the behaviour of the wake left by the geometry of the car, which despite having been seen previously, in this section we will look at both cases together and over an area where the differences are sufficiently notable to draw conclusions. The Figures to focus on are 7.1 and 7.2, where velocity fields over both figures are seen.

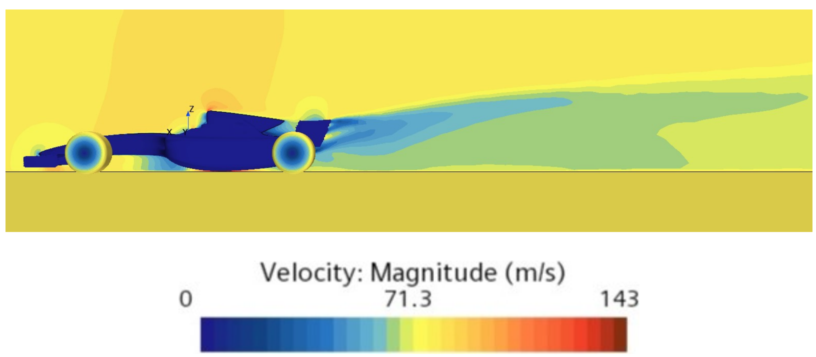


Figure 7.1: Velocity field over the car with no over-wheel winglet

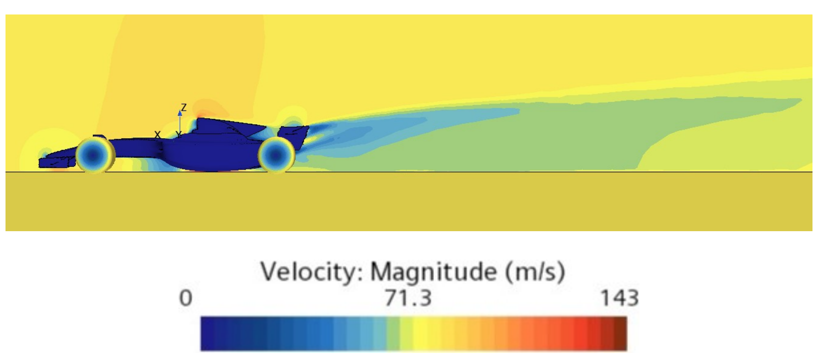


Figure 7.2: Velocity field over the car with over-wheel winglet

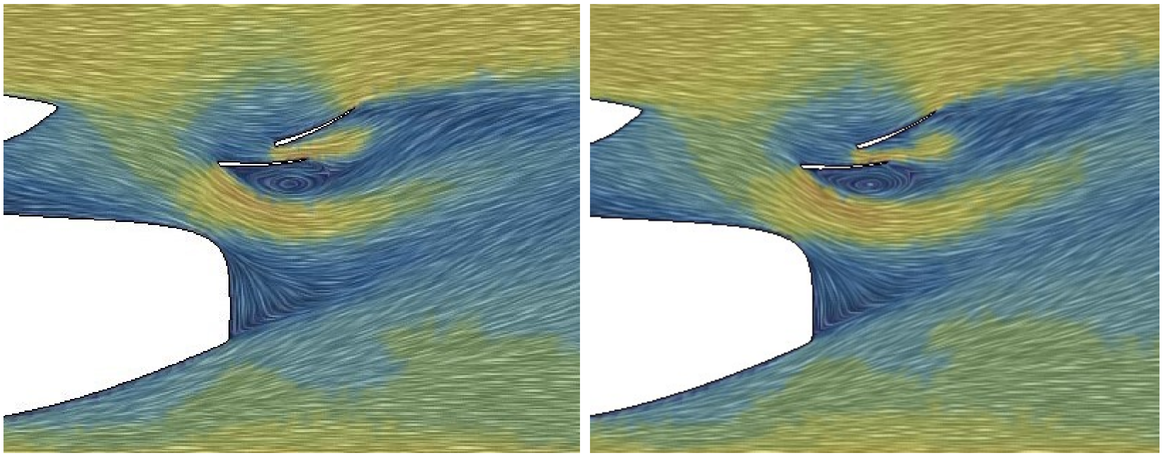
Key behaviours over the wake can be observed in these figures, including the following:

- The wake left by the car with no over-wheel winglet is shorter than the wake left by the car with this aerodynamic element. Measuring the area at approximately 70 m/s (the darkest green seen in the images) for a length of two and a half cars and three cars, respectively.
- In addition, the velocity field is more irregular in Figure 7.1 as can be seen in the images, since it varies less linearly, leading to a more gradual and therefore, a more linear velocity field in Figure 7.2 over the wake.

It is important to emphasize that a larger wake left by the car means a higher drag of the car since in this case, the car leaves a higher turbulence. In addition, in the car that produces a larger wake, it will give rise to greater slipstream effects since it will generate the same suction effect but at a higher distance. This means that a car that is behind at a greater distance than in the first case (without over-wheel winglet), will be able to obtain the benefits that the slipstream gives.

In both geometries, the airflow that forms the wake comes from two specific zones, mixing the air coming from the top of the car, passing through the rear wing, and the airflow coming from the flat bottom and diffuser, which has a lower velocity (due to the expansion function of the diffuser). These two airflows can be said to come together forming the wake in its entirety. So, the differences observed by the previous images are clearly influenced by the air coming from the car, which can be influenced by the existence of the over-wheel winglet.

To better understand the way in which the air flow reaches these areas, the following Figures 7.3(a) and 7.3(b) will be studied, where the streamlines overlapped with the velocity will be presented to observe the differences whose origin will be verified in the following Section 7.2. These images show not only the streamlines on the rear wing, but also on the air coming out from the diffuser and the behavior that the air follow as well as their direction, arriving to differences in velocity and detachment areas, as well as recirculating and vortices, affecting to both, drag and downforce.



((a)) For the car without over-wheel winglet ((b)) For the car with over-wheel winglet

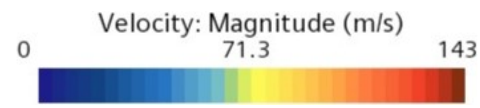


Figure 7.3: Streamlines overlapped with velocity in the rear part of the cars

Two small differences can be seen in these images. These are, on the one hand, the increase of velocity in the lower part of the rear wing for the case seen in 7.3(b), and so, the smaller recirculation area on the extrados of this airfoil, as well as the upward displacement of the air passing through the wing.

On the other hand, a greater lamination is also observed on the lower part (where the air coming from the flat bottom and the diffuser affects the most) in the second image, showing more gradual changes of direction and giving rise to a more gradual wake as observed in the first image.

With this, it can be intuited that these differences are generated by the effect of the designed over-wheel winglet , but for this it will be better to study it in the next section, where comparisons on the main area of interest between both geometries will be carried out.

7.2 Comparison over the Main Area of Study

Once the main comparison of the total effects on the car has been made, it will be of vital importance to study the areas close to the zone of study (upper area of the front wheels of the prototype) to determine the reasons for the effects previously described.

For that purpose, the comparison between the streamlines (overlapped with the speed) will be studied first on the additional plane of study created in the middle of the wheel, an area where there is still an over-wheel winglet in order to observe the effect of this element. These images are seen in the following Figures 7.4(a) and 7.4(b).

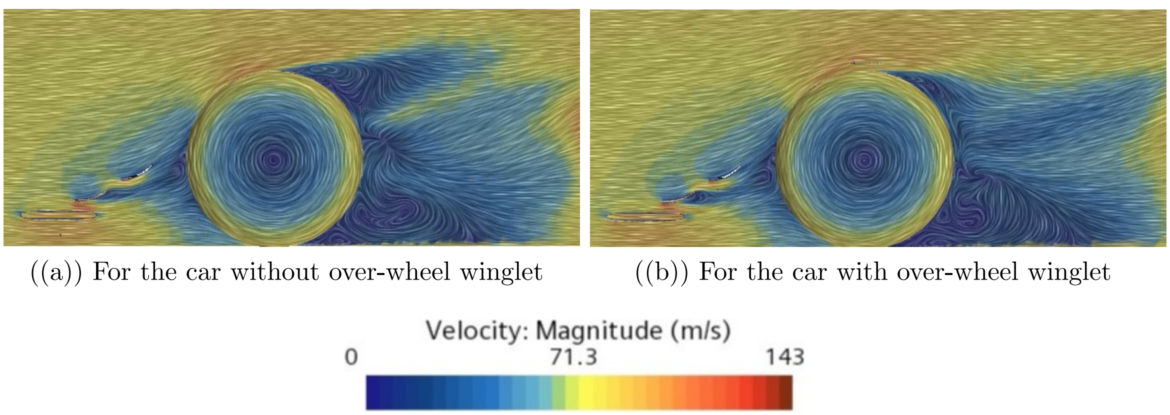


Figure 7.4: Streamlines overlapped with velocity in the front part of the cars

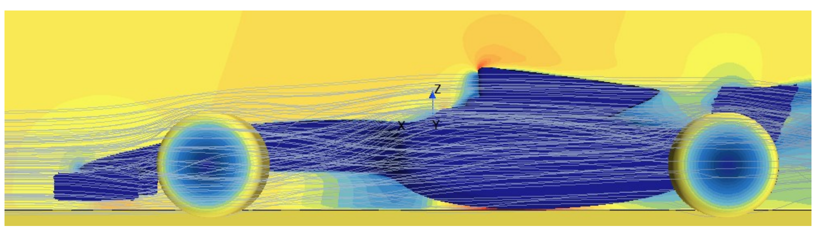
In these images, the same behavior is observed in the incident part on the front wing and wheel. As expected, it is from the over-wheel winglet that the distribution of the streamlines changes (and with them the velocity).

The winglet is observed to create a laminating effect on the airflow incident on the upper part of the wheel, being that what in case 7.4(a), the crossing between free flow and the effect of the wheel, which moves in the opposite direction on this point, creates a vertical velocity component, in case 7.4(b) no longer, because the new aerodynamic element is in charge of eliminating it.

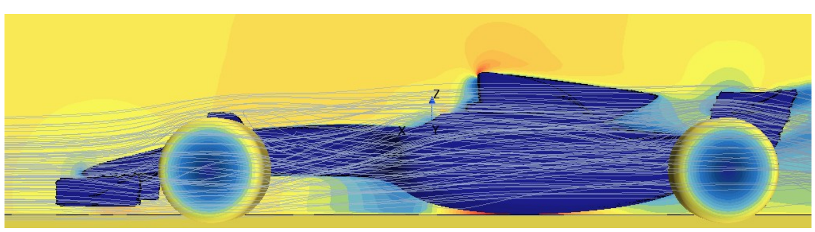
For the first case, however, the fact that there is a vertical component on the incident air, at the top of the wheel, causes the air flow to detach from the boundary layer of the wheel following this direction. As a consequence, vorticity and stall zone are generated in this area.

On the contrary, for the second case, since there is a fixed surface on the wheel that forces the air passing between these two to laminate and leave this area in a horizontal direction, it generates a much lower vorticity than in the first example. In addition, over the top of this winglet, the air attaches to the surface through the boundary layer, thus causing a completely horizontal flow over the top of the wheel that can be redirected and used in later parts of the car.

To finish observing the full effect of having or not having this surface on the wheel, it will be interesting to observe the three-dimensional streamlines not only on the top of the car but also on the rest of the car. Therefore, the comparison over the streamlines passing through exact areas for both geometries seen from a lateral view will be first observed in 7.5.



((a)) For the car without over-wheel winglet



((b)) For the car with over-wheel winglet

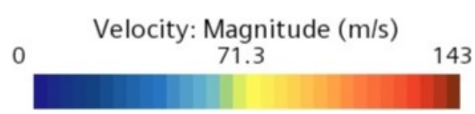
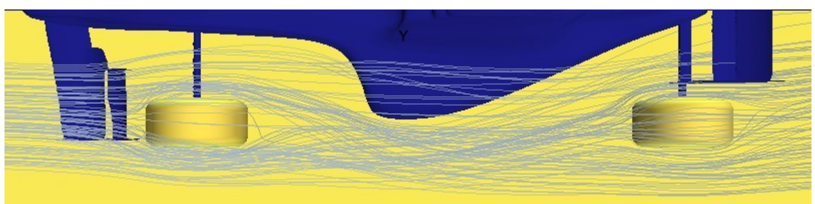


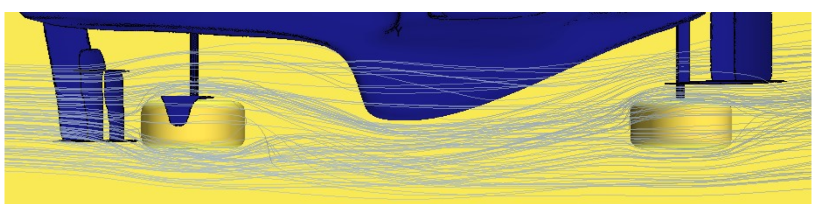
Figure 7.5: 3D Streamlines over the cars seen from a lateral view

In these, you can see the streamlines running through the entire car from a side view. As the main conclusion in this view, it is observed that first, the streamlines ascend (influenced by the front wing) until later, they are affected by the wheel. It is from this moment that the air direction changes between both geometries, following in 7.5(a) a trajectory with a vertical component, while in 7.5(b), the flow remains mainly horizontal until it is affected by the presence of another geometry such as the cockpit.

The designed over-wheel winglet will not only affect the flow in its vertical component but also horizontally as both depend on each other. Thus the effect of the winglet will also affect the way the air sticks to the rest of the car when it crosses it longitudinally, thus affecting the effect of the rear wing as well. This is best seen in the following figure 7.6, where a comparison of both geometries seen from above can be studied, and conclusions can then be drawn from both views.



((a)) For the car without over-wheel winglet



((b)) For the car with over-wheel winglet

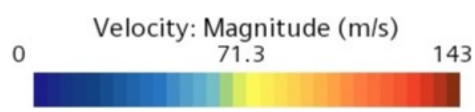


Figure 7.6: 3D Streamlines over the cars seen from a top view

These images show that the over-wheel winglet case causes the air, since it does not move towards the parts furthest from the ground, to be affected by the sidepods of the single-seater, following the geometry of these surfaces, which conduct the air towards the rear wing area. This means that car 7.6(a), causes the air to move further away from the area close to the car, while 7.6(b) causes this airflow to remain attached to the car surface and therefore end up in the rear wing, which will lead to higher downforce and drag coefficients as a result.

7.3 Differences in Values and Explanation of these

Once all the relevant comparisons taken into study have been carried out, the values of the aerodynamic coefficients of both geometries will be taken again in order to draw conclusions from both and finish understanding the case studies of the design prototypes.

To proceed, the values of the aerodynamic coefficients previously presented in other sections will be reconsidered in order to draw conclusions. These will be observed in the following Table 7.1.

Car with no over-wheel winglets		**Car with over-wheel winglets**	
	Values		**Values**
Downforce coefficient	1.113196	Downforce coefficient	1.126040
Drag coefficient	0.338652	Drag coefficient	0.351189

Table 7.1: Values obtained by CFD for the main aerodynamic coefficients on both geometries

From these data, it can be concluded that in the case where the over-wheel winglet is added, both downforce and drag coefficients are larger. This is because although it is positive for the car to have more downforce, this coefficient is also related to aerodynamic drag, causing the airflow through the surfaces that help the car to stick closer to the ground to also generate more aerodynamic drag as expected, as more air passes through these surfaces and also, the air is affected by a new element itself.

Furthermore, it is very important to recognize that the car will not only run straight, because on the circuits (and on the new Las Vegas circuit under consideration), the cars encounter curves, so it is also important to observe the way the streamlines are distributed in Figure 7.6, since in 7.6(b), the fact that the air attaches more to the surface will also cause less lateral displacements on the car.

Not only the coefficients must be considered, but also the differences of each one of them between both cars, which will be observed in the following Table 7.2.

	Measurement difference	**Percentage**
Downforce coefficient	0.012844	1.147 %
Drag coefficient	0.012537	3.635 %

Table 7.2: Difference between the coefficients of both car geometries

In this table it can be seen that although it seems negative that the drag coefficient has increased, the downforce has also increased, which is positive for motorsport, since the car is safer both in straights and in curves (where this fact allows to go a few tenths faster), also allowing to reduce the weight of the car to achieve a given downforce and thus have an engine that will push more (since less weight could be taken).

The drag coefficient, it is important to note that it will not always be the same, since when the DRS is activated, the coefficient will be reduced as this is an area where the air affected by the new element of study flows after going through the car, giving an extra thrust to the cars in the car-car fight.

Thus, it could be concluded that the introduction of the over-wheel winglet, despite involving minuscule differences in the overall aerodynamics of the car, could mean considerable differences between cars fighting for a place, and therefore between teams, since a few tenths in the times of racing cars can mean many places in the standings.

It should be noted that since the geometry used for the study is very simplified, its effect may not be fully appreciated in this project, and that this effect may be greater in reality. Furthermore, the designed over-wheel winglet is not optimised, which could be a factor why the difference in downforce is not so high.

Chapter 8

Practical Implications and Future Directions

Throughout the study done in this project, it has been possible to ”introduce” the concept of the over-wheel winglet already used in Formula 1, on Formula 2 cars (or other categories which have not introduced it yet) that have been designed taking into account the difficulties.

This part has real drag and downforce implications as already noted in previous chapters given the influence it has on the air around the top of the front wheels, but it is also important to consider the influence of this surface on weight and safety for the car.

In terms of weight, this element would be manufactured (and indeed is currently manufactured in F1) in carbon fiber, which would have a very low weight that could be considered negligible. While in terms of safety, it is also important to consider the wheel change that the teams' mechanics encounter, since having such a thin surface in an area so close to the wheel can pose dangers to the tire, which can be punctured if the change is not precise, and to the mechanics and racers in the event of hitting it, although with a good design at a sufficiently safe distance, these disadvantages would be completely eliminated.

In addition, if left as an element of free study for each Formula 2 team, since all cars are the same with small differences in non-aerodynamic elements such as brakes and suspensions between them, it could lead to a greater aerodynamic study within each team and could result in large differences between cars, and so, in the championships, leading to more aerodynamicists in the teams, where many studies of this element could be done.

8.1 Potential for Future Aerodynamic Innovations

It is important to note that the study conducted, in addition to all the constraints listed at the beginning of the project that led to simplifications in the design of a Formula 2 prototype, also suggested a geometry for the over-wheel winglet that apparently lead to be good, as the surface was placed at the distance considered optimal for the greatest benefits.

Many different designs of this surface could be made, both in terms of geometry itself and distance from the wheel, from which deliberations could be made to choose the one that would give the most benefits to the car. In addition, the length of the element across the width of the wheel would also be important as this affects the amount of air that is laminated as it crosses the study area.

In the last year of the F1 category, small grooves have also been observed on this surface, as seen in Figure 8.1, which would be interesting to study as it could be used as an additional wing to further optimize the aerodynamics of the car.

Figure 8.1: Slot left in the over-wheel winglet of the Mclaren 2022 car [27]

In addition to contributing to all this, new geometries could also be considered, not only to laminate the flow but also with the main objective of redirecting the air to the areas that each team sees as optimal and thus having more aerodynamic forces on certain areas of interest.

All this could mean a big change in the way of seeing the Formula 2 format, where currently there are no aerodynamic differences between cars, leading to a greater spectacle not only for fans of the category but also for the drivers and teams themselves as they would be able to contribute in a small way to the performance of their single-seaters.

8.2 Proposal for Future Research Directions

In addition to contributing to future aerodynamic research and studies in various motorsport categories on aerodynamic elements that help to laminate the air passing through the wheels and redirect it to the areas of interest, other elements could also be added, in addition to even reconsider this geometry to even increase the benefits that the over-wheel winglet produces. Also, as a big recirculation area is seen in Figure 7.4, a rear tip shaped surface could even be placed to separate the laminar flow from the turbulent flow coming from the wheels to better contribute to the car's performance, as long as it is not detrimental to the pit-stops.

For the case of the study prototype, the geometry of the single-seater itself could also be redesigned in the part where the engine is positioned, behind the driver, which could mean that the air, if considered beneficial, would stick to the surface again and could reach the ailerons or other aerodynamic elements. In addition, the winglet under study could also undergo geometric optimizations.

In addition to the aerodynamics of the main area of study, thermal effects on vehicles are also very important to consider since the safety of the pilots depends on all the possible factors affecting the care. Consequently, the design of the connection to this element is located in an area very close to the brakes, which could be redesigned to cool the high temperatures that these parts reach through the air in the car.

Therefore, this study could be left open for a possible next master's thesis or other studies where the author would focus on the aerodynamic optimization of the geometry of interest, on both, the over-wheel winglet design and the geometry of the cockpit itself, where elements to optimize have been observed. These studies could lead to optimal cases in which it could be said that the perfect car has been found for a team during the season.

Chapter 9

Relationship to the Sustainable Development Goals of the 2030 Agenda

It is very important, in addition to studying the project itself, to relate it to the sustainable development goals of the 2030 agenda in order to understand not only the implications that the case study may have on racing car competitions, but also on the environment. This is what will be studied in this chapter.

The sustainable development goals of the 2030 agenda are seen in Figure 9.1, and the ones that could be related to the case of study will be stated next.

Figure 9.1: Sustainable Development Goals of the 2030 agenda

Although it is a project that does not focus on any of the development objectives in particular, to some extent, this study affects some of them to a certain extent. Therefore, all the objectives will be described below and all of them will be described in order to better understand which of them can be affected to a greater or lesser extent.

1. End of poverty: the relationship to this objective is not applicable.

2. Zero hunger: the relationship to this objective is not applicable.

3. Health and well-being: the relationship to this objective is not applicable.

4. Quality education: the relationship to this objective is not applicable.

5. Gender equality: in the automotive world, since previous decades, most of the representation has been male, both in the paddock and in the studios run by the teams. By being a woman doing this project I believe it can give other women hope to enter this world and thus, companies can see the equal potential that both genders can offer to this field of study.

6. Clean water and sanitation: the relationship to this objective is not applicable.

7. Affordable and clean energy: the relationship to this objective is not applicable.

8. Decent work and economic growth: the world of racing is very important for economic growth, as teams rely on sponsors to help them make possible the research and improvements carried out each season. In addition, as with many large-scale events, the countries that host grand prix events have very large profits, which leads to the whole country benefiting in terms of improving the lives of their citizens.

9. Industry, innovation and infrastructure: as might be expected, the study directly affects innovation, in this case in aerodynamics, which can be extrapolated to many other aspects that could be affected by knowledge of air behaviour, such as different means of transport.

10. Reducing inequalities: the relationship to this objective is not applicable.

11. Sustainable cities and consumption: the relationship to this objective is not applicable.

12. Responsible production and consumption: the relationship to this objective is not applicable.

13. Climate action: Given that car racing is not likely to come to an end any time soon, more aerodynamic cars may result in lower fuel consumption, which will lead to less pollution. In addition, this factor has led to the consideration of new engines such as the hybrids currently in use. From the point of view of the project carried out, the vast majority of the project has been carried out with the use of electricity, as the simulations must be done on computers connected to the electricity supply for the hours that the calculation lasts. Therefore, if a greater commitment to the environment is sought, simulation plants with solar energy as a source of electricity can be set up in the racing teams.

14. Underwater action: the relationship to this objective is not applicable.

15. Life of terrestrial ecosystems: low relation to this objective, although the project indirectly affects it through the previous goal 13.

16. Peace, justice and strong institutions: the relationship to this objective is not applicable.

17. Partnerships to achieve goals: given that studies of great importance in the engineering sectors are not carried out by a single person, but by a group of people or even by different companies seeking the same objective, it helps to ally with others for a common goal.

Chapter 10

Project Specifications and Cost

This chapter is intended to describe the conditions under which the project was carried out and the cost of its implementation. The project budget will take into account human, software and equipment costs, while the conditions will determine the manner of realization and the conditions of the space used for it.

10.1 Specifications

The conditions under which a project must be carried out must also be taken into account, always in accordance with Royal Decree 486/1997 and 488/1997 in order to ensure the safety of the personnel carrying out the work.

Royal Decree 486/1997 is the one establishing the minimum health and safety provisions for workplaces. In this, aspects such as structural safety are published, including the safety of the electrical installation and a safe and comfortable working environment for the workers of a company, where environmental conditions that do not pose a risk to the health of workers must be provided, where both temperature and humidity play a fundamental role. In addition, the workplaces must also be sufficiently clean and tidy, with adequate lighting for the characteristics of the activity carried out, avoiding reflections and glare that other sources may cause.

On the other hand, Royal Decree 488/1997 establishes the minimum health and safety provisions for the use of display screen equipment by workers. In this, it states that the display screen must be adjustable and tiltable for the user's comfort, as well as being luminously adjustable. Likewise, the keyboard must be tiltable and independent of the screen and its keys must be easily readable. The desk provided should be large enough to allow for a comfortable position and the user should be provided with a height and backrest adjustable working seat.

10.1.1 Implementation Conditions

Once the specifications set by the Spanish government for a company have been defined, the implementation conditions used to carry out the project will be taken into account. Among these, the following stand out:

10.1.1.1 Technical Conditions

The hardware used, as set out in the budget, is as follows:

- **Brand:** Lenovo
- **Model:** Ideapad 330-15ikb core i7
- **Processor:** Intel Core i7-8550U
- **Processor speed:** 1.80 GHz
- **Operating system:** Windows 11
- **Graphics card:** AMD Radeon 530
- **RAM:** DDR4 8 GB
- **RAM speed:** 2133 MHz
- **Screen size:** 15.6 ”
- **Resolution:** 1366 x 768 pixels

The computer programs used are those described above, including STAR CCM+, a Siemens program used for computational fluid dynamics, in addition to the use of Microsoft 365 equipment for teams, a program that allows connecting and chatting with other users, as well as making calls and sharing files. Excel has also been used for data processing. Finally, the memory has been made in latex, a text composer oriented to the creation of high quality written documents.

10.1.1.2 Working Conditions

In addition to the technical conditions, the conditions of the workplace in which the study has been carried out must also be considered.

On this point, the user must consider the previously mentioned royal decrees since the author is exposed to various risks when carrying out the project. Among these risks are ergonomics, visual fatigue and working posture, as well as electrical contacts, lighting and noise.

Therefore, in spite of being an own project and therefore, having been carried out in conditions not as safe as in a professional workplace, it has been provided with a work table, seat, work environment and adequate lighting. Although it is true that the laptop screen can be considered somewhat small for a project that involves so many hours, that is why an additional screen has been used for comfort.

10.2 Project Cost

Finally, the cost of the study will be presented, which is organized into different types of costs, such as labor and equipment used or necessary for the realization of the case, as well as the software to be used.

10.2.1 Personnel Costs

In this section, the cost of the people who have contributed to the development of the study should be considered, where in this case it will be the author of the project and her tutor. It should be noted that the personnel cost it is not only what the two people will be paid, but also what the company that hires them will be paying for these people. These expenses include social security and salary, among others.

In the case of the author, the time invested in the project, and therefore to be taken into account, is as follows:

- **Learning and research:** 30 h
- **Preprocessing:** 73 h

- **Analysis:** 72 h
- **Postprocessing:** 60 h
- **Writing the report:** 132 h

It may be noted that the sum of time spent by the author is approximately the expected amount of time of an average Final Degree Project, which is supposed to take about 300 h (12 credits · 25 h/credit). The total personnel costs will therefore be shown in the following Table 10.1.

	Time [h]	**Cost [€/ h]**	**Value [€]**
Author of the project	367	33	12111
Tutor of the project	25	50	1250
		TOTAL	**13361**

Table 10.1: Personnel costs

Thus, the cost of personnel for the project carried out would be **THIRTEEN THOUSAND THREE HUNDRED AND SIXTY ONE EUROS (13361 €)**.

10.2.2 Cost of Computer Equipment and Software

The cost of the computer equipment must take into account the computer under consideration, which in this case is a LENOVO ideapad 330-15ikb core i7, whose value is 590 €.

For the softwares to be used, different ones will be considered. For STAR CCM+ (SIEMENS), it is known that the annual licence costs 180000 € per year, of which the proportional share of usage will be considered. For the use of Fusion 360 (AUTODESK), a monthly licence can be contracted which costs 73 €/month and which is enough to realise the 3D designs used for the project.

In addition, for the online meetings between the author and the tutor, Microsoft Teams was used, which is part of the Microsoft 365 package and costs 64.03 € per year. Within this package other programs have also been used. However, the report was written in LaTeX, which is a free license program.

The computational costs as a whole can therefore be seen in the following Table 10.3.

	Time [h]	Cost [€/h]	Value [€]
Computer	-	-	590
Star CCM+	205	20.54	4210.7
Fusion 360	-	-	73
Microsoft 365	-	-	64.03
LaTeX	-	-	-
		TOTAL	**4937.73**

Table 10.2: Computational costs

Thus, the cost of the computer equipment and software used for the project carried out would be **FOUR THOUSAND NINE HUNDRED AND THIRTY-SEVEN EUROS WITH SEVENTY THREE CENTS (4937.73 €)**.

10.2.3 Electrical Consumption Cost

For the cost of the electricity used for the project, the current rates and the complete number of hours that the author has used for the study will be taken into account. the cost can be seen in the following Table 10.3.

	Consumption [kW h]	Cost [€/kW h]	Time [h]	Value [€]
Laptop	0.360	0.1101	367	14.15
			TOTAL	**14.55**

Table 10.3: Electrical consumption cost

Thus, the cost of the electricity consumption would be **FOURTEEN EUROS AND FIFTY FIVE CENTS (14.55 €)**.

10.2.4 Total Cost

Therefore, the sum of the different costs of the project, will be the cost shown in the following Table 10.4. To the total cost of the project, the Value Added Tax must be added.

	Cost [€]
Personnel	13361
Computer equipment	590
Software	4347.73
Electrical consumption	14.55
Subtotal	18313.28
VAT (21 %)	3845.79
TOTAL	**22159.07**

Table 10.4: Total cost of the project

Therefore, in total, the project has a total value of **TWENTY-TWO THOUSAND ONE HUNDRED AND FIFTY NINE EUROS WITH SEVEN CENTS (22159.07 €).**

Chapter 11

Conclusions

To finish with the project, some final conclusions can be drawn on the total impacts observed on the aerodynamic study of the effect of the over-wheel winglets on Formula 2 cars.

Among the aspects to be considered, and as mentioned above, it is of vital importance to observe the results of the aerodynamic analysis, where the case with the aerodynamic element of study added presents higher values of drag coefficient and downforce. This result is something very considerable, as it may be a factor to be considered for the FIA's decision to include or not these aerodynamic appendages in more racing car modalities.

Some additional areas of improvement for possible new projects that could be carried out, such as aerodynamic optimization of the car design and possible slots for the study winglet, have been observed, leaving an open door towards a possible future master's thesis or further studies.

It is very important also to consider that if a more aerodynamic car or with more downforce is made, the weight of the car and the consumption may vary, which in the case of a more aerodynamic car, leads to a decrease in fuel consumption, causing great benefits on the environment, besides being able to cause greater similarity between cars and more entertainment in the races, helping to create a greater following of competitions, which leads to economic growth of the countries.

Regarding the cost of the project, it should be noted that like most engineering projects, it involves a prior investment in equipment and personnel, which can be very different depending on the aspects considered, so it will be important to consider the most realistic situation possible to be able to have the right perspective.

That is why this project gives a real vision of the repercussion of aerodynamics in F2 cars and helps to understand the importance of the knowledge of air behavior among all the aspects that encompass it, and the importance that this investment has had over the years.

References

[1] D. Durruty, "McLaren presentó el MCL60, todo un tributo al red bull de 2022", https://automundo.com.ar/formula-1-mclaren-mcl60-2023/, Feb.2023.

[2] E. Garayar, "Alfa romeo 158 alfetta", https://vidaalvolante.wordpress.com/tag/alfa-romeo-158-alfetta/.

[3] R. Blancafort, "1968: El año en el que nace la publicidad", https://soymotor.com/blogs/rblancafort/1968-ano-nace-publicidad.

[4] J. Rubio, "La historia del lotus 79, 'padre' de los próximos fórmula 1, y por qué serán tan fascinantes", https://www.elconfidencial.com/deportes/formula-1/2022-01-22/lotus79-monoplazas-adrian-newey-efecto-suelo_3362486/, Jan. 2022.

[5] M. F. Osma, "Historia de la aerodin´amica en la F1", https://www.aerodinamicaf1.com/2019/09/historia-de-la-aerodinamica-en-la-f1/, Sep. 2019.

[6] G. Series, "GP2 Series renamed FIA Formula 2 Championship", https://web.archive.org/web/20170314171211/http://www.gp2series.com/News-Room/News/2017/03_March/GP2-Series-renamed-FIA-Formula-2-Championship-/

[7] D. Plaza, "Así ha cambiado el pilotaje con los nuevos Fórmula 1 de 2022", https://www.motor.es/formula-1/cambios-pilotaje-formula1-2022-202285249.html, Feb. 2022.

[8] J. Krewson, "Fast, present, future: 1967 lotus 49 vs. 2013 corvette ZR1", https://www.roadandtrack.com/car-culture/a3982/fast-present-future-64-8-roa0513/, Apr. 2013.

[9] "Lotus 79", https://iracing.fandom.com/wiki/Lotus_79, Jul. 2010.

[10] "McLaren MP4-8 - F1technical.Net", https://www.f1technical.net/f1db/cars/740/mclaren-mp4-8

[11] S. Fans, "Onboard: Carlos Sainz reaches record top speed of 343km/h at Paul Ricard track in France", https://rb.gy/8uauw, Aug. 2022.

[12] F. Nugnes, "Técnica f1: Ferrari cambiaraáde color para reducir el peso en españa", https://es.motorsport.com/f1/news/analisis-tecnico-ferrari-pintura-color-diferente-bajar-peso/10303511/, May. 2022.

[13] M. F. Osma, "El alerón delantero en la Fórmula 1 (I): Partes. *Aerodinámica F1*" https://www.aerodinamicaf1.com/2019/11/el-aleron-delantero-en-la-formula-1-partes/, 2022.

[14] M. White, "Bahrain F2: Pourchaire gana a Lawson en una carrera caótica", https://es.motorsport.com/fia-f2/news/cronica-resultados-carrera-formula2-bahrein-2022/9163406/, Mar. 2022.

[15] M. F. Osma, "El alerón trasero en la Fórmula 1 (I): Partes. *Aerodinámica F1*", https://www.aerodinamicaf1.com/2019/10/el-aleron-trasero-en-la-formula-1-i-partes/, 2022.

[16] J. L. Ruiz, " Los secretos de Mercedes, al descubierto", https://twitter.com/MercedesFRteam/status/1662427288695078914/photo/1, May. 2023.

[17] M. A. Herguedas, "Esperanzador debut de Fernando Alonso con Aston Martin, a 29 milésimas de Verstappen", https://www.elmundo.es/deportes/formula-1/2023/02/23/63f7946ae4d4d8150c8b45a5.html, Feb. 2023.

[18] V. A. P. B. J. Vagabond, "*Turbulent Flow and Viscosity.* Physics and Chemistry for IG and A level", https://esfsciencenew.wordpress.com/2012/10/26/turbulent-flow-and-viscosity/, Oct. 2012.

[19] K. Boldt, "Aerodynamic measurements: Boundary layer components", https://www.grasacoustics.com/blog/aerodynamic-measurements-components-of-the-boundary-layer, Mar, 2022.

[20] C. S. Martínez, "Técnica F1: ¿Qué es y cómo funciona el efecto suelo?. *Planeta del Motor*." http://planetadelmotor.com/efecto-suelo-f1-que-es/, 2016.

[21] "Forces on an Airfoil", http://avstop.com/ac/flighttrainghandbook/forcesonanairfoil.html.

[22] Tok. Wiki, "Campeonato de Fórmula 2 2020 EntradasyCalendario", https://www.actualidadmotor.com/ver-f2-gratis/.

[23] "F2 2018 unveiled in Monza - Formula 2", https://web.archive.org/web/20171105173813/http://www.fiaformula2.com/News-Room/News/2017/08_August/F2-2018-unveiled-in-Monza/, 2018.

[24] Zaca, "4 Tips To Avoid Las Vegas Altitude Sickness", https://zacalife.com/blogs/blog/las-vegas-altitude-sickness.

[25] "Formula 1 announces tickets to start at $ 500 for Las Vegas Grand Prix", https://www.fox5vegas.com/2022/10/26/formula-1-announces-tickets-start-500-las-vegas-grand-prix/, Oct. 2022.

[26] Simscape, "Polyhedral, Tetrahedral, and Hexahedral Mesh Comparison", https://www.symscape.com/polyhedral-tetrahedral-hexahedral-mesh-comparison.

[27] S. Giorgi, "F1 McLaren, che passo indietro! *Formula Uno Analisi Tecnica*", https://www.funoanalisitecnica.com/2022/08/f1-mclaren-passo-indietro.html, Aug. 2022.

Appendix A

Convergence of Cases

Analyzing the convergence of the solution is a crucial step in validating the simulation and progressing to post-processing. To ensure accuracy, several key convergence criteria have been established:

- Residuals: lower than 10-3 for continuity and lower than 10-6 for the rest.
- Rate of variation of key parameters: low enough.
- Satisfaction of the conservation of mass equation: low enough.
- Satisfaction of the conservation of momentum equation: low enough.
- The flow does not have any discrepancy.

The pressure and velocity profiles obtained in chapter 6 from the solution, were thoroughly examined to eliminate the possibility of any unusual or extraneous phenomena. Streamline analysis was also employed to detect such occurrences. When applied to this specific case, no undesired or anomalous events were observed, suggesting that the solution may have reached convergence. However, it is important to verify convergence using additional criteria.

It is also important to assess the rate at which essential parameters, such as mass flow, force, inlet pressure, and outlet velocity, change. This assessment is done by comparing the data obtained from different iterations.

Additionally, the examination of the parameter of interest, which in this study refers to the aerodynamic coefficients, is crucial. It is necessary to observe a consistently stable trend across the iterations in order to determine that the case study has converged and reached its final value. With each iteration, the values are recalculated until the solution aligns with the actual value, indicating the convergence state.