UNIVERSITAT POLITÈCNICA DE CATALUNYA
BARCELONATECH
Escola Superior d'Enginyeries Industrial,
Aeroespacial i Audiovisual de Terrassa

PERFORMANCE AND IMPROVEMENTS OF A FORMULA ONE CAR

Document:
Memory

Author:
Alex Bauza Frostad

Director:
Roberto Castilla Lopez

Degree:
Industrial Technology Engineering

Announcement:
Autumn 2021-2022

FINAL DEGREE PROJECT

RESUMEN

El siguiente estudio hace referencia al análisis del funcionamiento y mejoras de la competición de automovilismo más famosa del mundo, conocida como Formula 1. Dentro de este deporte, se engloban numerosas secciones y especialidades como la mecánica, chasis o motores, pero en este caso, nos introduciremos en la investigación del alerón delantero, que corresponde a uno de los elementos más importantes de la aerodinámica.

Como todos los deportes, la Fórmula 1 también ha evolucionado, y en este caso centraremos nuestro proyecto en la creación de un nuevo diseño de alerón delantero adaptado a las nuevas normativas impuestas por la FIA en 2022. Por ello, nuestros principales objetivos son crear este nuevo Modelo CAD 3D en SolidWorks, gracias a otros prototipos e imágenes obtenidas, y también iniciando un proceso de debate donde se comparan las mejoras que aporta este nuevo diseño con respecto a años anteriores.

La metodología comienza con la creación del prototipo 2022 y reproducirlo en un software de simulación CFD estudiando el comportamiento dinámico del aire al igual que hace un túnel de viento. El proyecto explica en detalle el funcionamiento de estas simulaciones, y la metodología utilizada en su creación, a partir de imágenes, tablas y conclusiones obtenidas en el transcurso del trabajo. Una vez vistos los aspectos aerodinámicos más importantes, situados en la parte inicial de la memoria, y la evolución de la aerodinámica en el deporte a lo largo de la historia, se realizan algunas comparativas con un modelo 3D obtenido de 2018 (antiguo reglamento) gracias a las simulaciones ya comentadas.

Las simulaciones generan numerosos datos y resultados que deben ser considerados y comentados para encontrar las diferencias entre estos dos alerones, donde un valor simbólico es la eficiencia aerodinámica numérica, que incluye información sobre los coeficientes de sustentación y arrastre, que pueden llegar a aumentar considerablemente con el nuevo modelo. Además, con los resultados llegamos a la conclusión de que la configuración de un coche es crucial para sacarle el mejor rendimiento, y pequeños cambios, como la altura o la geometría del alerón delantero, pueden ser decisivos.

En resumen, nuestras conclusiones finales generalizarán y compararán cuál es la mejor configuración de alerón delantero, y si se puede lograr el objetivo de la FIA de reducir la cantidad de aire sucio con el nuevo reglamento. Además, se comentan algunas otras ideas futuras para mejorar nuestro estudio, como diferentes efectos cambiantes del rendimiento o consideraciones avanzadas.

ABSTRACT

The following study refers to the analysis of the performance and improvements of the most famous motorsport competition in the world, known as Formula One. Within this sport, numerous sections such as mechanics, chassis or engines are included, but in this case, we will introduce ourselves in the investigation of the front wing, one of the most important elements of aerodynamics.

Like all sports do, Formula One has also evolved, and in this case, we will focus our project on the creation of a new front wing design adapted to new regulations imposed by the FIA in 2022. So, our main objectives are creating this new 3D CAD model in SolidWorks, thanks to other prototypes and images, and also starting a discussion process where the improvements provided by this new design are compared to previous years.

The methodology starts with the creation of the 2022 prototype and reproduce it in a CFD simulation software studying the dynamic behaviour of the air like a virtual wind tunnel. The project explains in detail the operation of these simulations, and the methodology used in their creation, based on images, tables and conclusions obtained in the course of the work. Once seen the most important aerodynamic aspects, located in the initial part of the memory, and the evolution of aerodynamics in the sport throughout history, some comparisons are made with a 3D model obtained from a 2018 front wing (old regulation) thanks to the CFD simulations already commented.

Simulations generate numerous data and results that must be considered and commented to find the differences between these two ailerons, where a symbolic value is the numeric aerodynamic efficiency that includes information about lift and drag coefficients, which can arrive to increase considerably with the new model. Also, with the results, we arrive at the conclusion that a car’s configuration is crucial to find the best performance, and small changes, like the height or geometry of the front wing, can be decisive.

To sum up, our final conclusions will generalize and compare which is the best front wing configuration and if the FIA’s objective to reduce the amount of dirty air with a new regulation can be accomplished. Besides, some other future ideas to enhance our study, like different changing performance effects or advanced considerations, are remarked.

INDEX

TABLE INDEX

Title and number of all the tables in order of appearance in the text.

FIGURE INDEX

Title and number of all graphics in order of appearance in the text.

GLOSSARY

Aerodynamics: fragment of the fluid mechanics studying and analysing the movement and the combination of a solid piece with the surrounding interacting fluids [1].

AIACR: known as Association Internationale des Automobile Clubs Reconnus, was an organisation created in Paris during 1904 and the objective was to regulate the most important automobile competitions in the world.

Airflow: defines the air movement between two different points as a result of a pressure differential with the flow direction pointing from the highest to the smaller pressure point.

Brake caliper: is the set of parts executing the final mechanism of a car's braking system, the one that pushes the brake pads against the disc to slow down the velocity or even stop the vehicle completely.

Cockpit: one of the Formula One parts, is a compartment for the pilot to sit and drive comfortably inside the car [1].

Dirty air: we are talking about dirty air when a racing car follows another from behind and the wake airflow coming from the one in front affects the performance with a turbulent and chaotic air pattern.

Drag: aerodynamic force or resistance generated in the horizontal axis in the opposite direction to the car movement that appears when the vehicle moves through the air [1].

DRS: acronym for Drag Reduction System, is a mechanism using a moving flap on the rear wing of a racing car. The functionality is to change its inclination with an electric sensor to reduce the drag resistance during the movement and gain extra speed in the straights, where the car needs less downforce [1] [18].

FIA: is the evolution of the AIACR, and has the acronym of Fédération International de l'Automobile. As the name says, is the highest international influence in the automotive sector, governing the corresponding teams and associations in more than 90 countries, and in charge of the rules established in every regulation change during the years.

Ground effect: is a characteristic from the car's aerodynamics and appears when the body has a differential of pressure between the top and bottom zone, and is running near the ground, this aerodynamic phenomenon causes alterations in the airflow that are normally used to improve the performance.

Halo: introduced in 2018, is a car part located on the top of the driver's head, and has the objective to protect the cockpit from outside hits or crashes. Consists in three flexible and resistant titanium bars that weigh approximately 10 kg and can resist huge impacts during a race.

Hybrid era: considered one of the biggest changes in the regulation of the sport, in 2014 the cars were changed from a 2.4-liter V8 with only combustion power, to a different power unit with the addition of an energy recovery system and a 1.6-liter turbocharged V6 [11].

Pontoons: an aerodynamic element located on the vehicle's sides, with the mission of changing the direction of the airflow to the back of the car and stabilising the turbulent airflow to help the performance of the other aerodynamic devices.

Set-up: is a very common word used in Formula One, and refers to the way the car has been changed and designed to improve the performance of a specified condition. Normally, the teams adapt their car to different circumstances like the weather or a track to improve efficiency and final execution.

Side skirts: used in different periods during the sport evolution, are aerodynamics shape designs that reduce the pressure from the borders of the car and moves to the under part of the vehicle to enhance the ground effect efficiency.

Spoiler: this concept changes depending on where it is used, but talking about racing cars and Formula One, a spoiler is operated to change how the airflow is affecting, with the increasing speed, to the final performance of the vehicle with the acceleration of the aerodynamic flow.

Turbulence: it's a disorderly movement of the different fluids where their molecules are describing chaotic and irregular tendencies instead of following parallel and steady trajectories. This way of behaving is called laminar movement, and as commented, has no disorder between the layers [1].

Vortex: is used in flow dynamics referring to a turbulent flux that has a spiral rotation movement and closed current paths. This concept creates the idea of vorticity, which is a physical magnitude to quantify how much rotation a fluid has, and is established and calculated as the circulation per unit area that has the flow in one determined position.

Wake: trace that a moving body leaves on his back in different fluids like air or water, in racing most commonly used to refer to how the airflow behaves and changes after a vehicle that passes through.

1 INTRODUCTION

1.1 OBJECTIVE

Create a modern and experimental front wing for a Formula One racing car in 2022 and analyse its performance using simulations with a CFD software. Actually, in this sport, aerodynamics is a crucial factor that helps the teams to create a competitive car, so with qualified simulations the main objective is to sketch a new front wing prototype according to new technical regulations.

1.2 SCOPE

To get to the proposed objective, the working process will be divided into several procedures of different areas:

- Introduction of a Formula One racing car to understand how some variables can affect the velocity and car's development during a race.
- Some aerodynamics explanations related to the knowledge collected in fluid mechanics to see the physical and mechanical use of a front wing.
- The study of the different front wings during history. In this case I will be analysing the state of the art and the improvements that can help me with the creation of the new design.
- A sketch creation with a computer-aided designer (CATIA, SolidWorks…). Based on the study made before, create a new front wing, according to the recent specifications to improve the aerodynamics.
- Simulate the object with computational fluid dynamics (CFD) to see how the piece works in different conditions in airflow.

Finally, working with front wing prototypes, there can emerge other sort of analysis, like an experimental results validation in wind tunnels, or compare and work with different fluids and not only with air (water in rainy conditions). Also, consider some wind lateral consequences in the performance, and air compressibility repercussions can modify the dynamic of the studies. With extra time and information, this could be interesting information that can be added at project's end, but in my case, are just commented.

1.3 REQUIREMENTS

There are some specifications in the working process that this project needs to guarantee, and here are going to be resumed.

First of all, the simulation will be done with CFD SimScale, an online free software, so it is important to have some notions about the program to ensure precision in the simulations and the analysis of the results.

Then, one of the restrictions we need to consider is to know and respect the front wing design regulations that Formula One has for 2022, to make sure that our model can be established in a team the next year.

Finally, some of the ideas studied during the degree in subjects as fluid mechanics or fluid technology to understand the aerodynamics from the car.

1.4 JUSTIFICATION

Formula One is one of the most technical sports in the world, and a lot of teams invest big quantities of money to create a competitive car to reach their objectives, so the design and the study of one of the most important parts, like the front wing, can be very helpful.

Also, the front wing is the first part in contact with the air, so an optimal design can help to have a great balance and generate the downforce required from the car. The simulations are the best option to avoid spending money and resources in a wind tunnel, this aspect is pleasant for a F1 team, and help them save money for other improvements in the car.

To conclude, an optimal air flow circulation in the car helps to have higher speed in the straights and also being faster in the curves with extra grip on the track, so the influence is enormous on how the car is set-up.

2 FORMULA ONE CAR PERFORMANCE

Designing a Formula One racing car is the most difficult part of this worldwide sport, and there is a lot of study and dedication behind each final piece of the car. It's a combination of engineering and complex mathematics to simulate the performance in different areas; like the chassis, the aerodynamics, tire degradation or set-up design.

In this case, the study will be focused on the aerodynamic section, which is the one in charge to define the airflow direction when a car is racing in the track. A racing car has infinite sectors dedicated to the improvement of the aerodynamics, but it's important to know the basics and see how they work in terms of the final performance of the car.

2.1 FRONT WING

This is one of the most important parts of a racing car, because it is the first section of the car in contact with outside's airflow. The creation of the front wing was to define the direction of the airflow generated in high speed velocities, but actually, this part of the car has two main objectives. Those two main objectives are to avoid the contact between the airflow and the front wheels, which reduces considerably the progression of the car, and also generate the optimal downforce to improve the subjection of the car on the track. In this project, will be the complement studied and analysed with a new prototype.

Figure 2. Ferrari SF-21 front wing 2021. (Source: [3])

Figure 1. Mercedes AMG front wing 2021. (Source: [3]).

2.2 REAR WING

This is one of the parts that receives the most significant changes during the years, because it has to adapt to the regulation. Formula One regulation modifies yearly, and the rear wing can completely modify the performance of a racing car. This wing generates a great part of the car's downforce, and has a drop-down wing which can be opened in the straights to give a higher top speed to the cars avoiding resistance generated from the airflow. [2]

The drop-down wing is called DRS, which means “Drag reduction system” and as the name says, works reducing the drag generated at that high speed. [2] [18]

Figure 3. Ferrari SF-21 rear wing 2021. (Source: [2])

2.3 DIFFUSER

A diffuser is located on the car’s rear floor, and works with all the accelerated airflow arriving from the car’s front area. This accelerated flow creates a lower pressure area down the car and increases the downforce produced by the car compared with the pressure there is on the top of the car. In this case, the diffuser has the function of guiding this accelerated airflow through the outside of the car with the highest acceleration possible to increase the downforce [3].

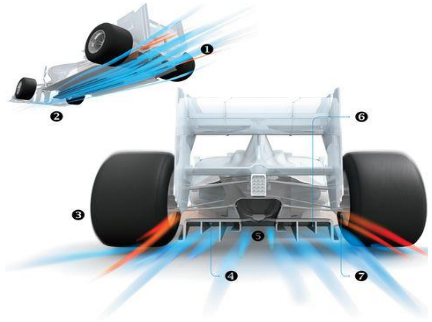


Figure 4. Airflow accelerated in a diffuser. (Source: [3])

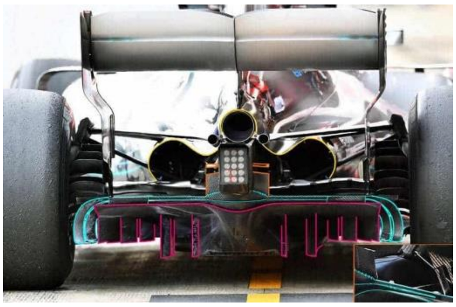


Figure 5. Mercedes AMG F1 diffuser. (Source: [3])

2.4 BARGEBOARDS

The bargeboards are located in the body of the car, behind the front wheels, and they work to improve the aerodynamics of the car helping to control the turbulence generated by the front tires [4]. With this part of the car, we can redirect the aerodynamic flow around the car, and not only the one coming from the tires. Related to the front wing, the bargeboards are also created to redirect the vortex created in the front wing of the car as we can see in the picture. In our case, this element is not going to be studied, but can improve also our front wing effect on the car.

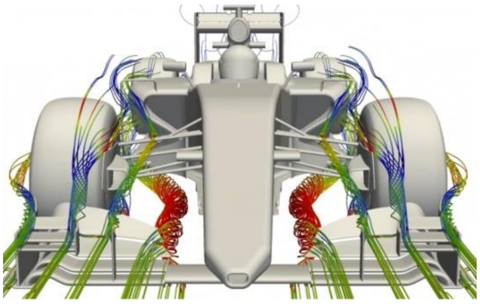


Figure 6. Aerodynamics function of the bargeboards. (Source: [6])

The red airflow shown in the picture is the vortex generated from the end side of the wing, and is called Y-250 because the distance between the vortex and car's centre is 250 mm. At the end of the project, this action is considered in our prototype.

2.5 ENDPLATES

These are vertical pieces located on both sides of the wings to improve the performance reducing the vortex creation. This phenomenon appears when the high-pressure air that we have on the top of the wings wants to connect with the low pressure located on the bottom [5]. The connection between those two different pressure areas at the end of the wing creates vortices, which generates turbulence reducing the capability of the car. With these endplates, we can't reduce a complete vortices formation, but we can mitigate the aerodynamic effect to the car behind reducing turbulences, which is one of the objectives from the new prototype.

Figure 7. Vortices creation at the end of the endplates. (Source: [4])

2.6 FLOOR

The floor in a Formula One car has a high relevance in the aerodynamics, because it is the nearest part of the car to the track. To improve the downforce of the car, it's important to have the minimum low pressure on the bottom of the car, and that's why the shape of the floor has to be designed and studied, to also reduce the drag and improve this downforce.

Normally the floor is designed with a front inclination to reduce the entrance of air, and also use the vortices created in the front wing to expel the air from the lower section of the car to the outside. A wood panel is included in the inclined part to avoid the direct contact between the carbon fiber and the track and reduce the wear.

Figure 8. Mercedes AMG floor's car. (Source: [3])

2.7 SHARK FIN

The regulation prohibited it in 2018, and was a vertical plate located in the symmetry plane of the car covering the motor, behind the cockpit and extended to the rear wing. A shark fin located at the end of the car helped to stand against the centrifugal force generated by the car in fast curves. Some of the Formula One teams saw that they could reduce the weight that the tires had to support in a fast corner, but also the main inconvenience was the drag generated in lateral airflows, where the shark fin worked as a parachute and reduced the high speed in the straights [6].

Figure 9. Williams F1 team's shark fin. (Source: [6])

These are generally aerodynamic crucial components of a Formula One car, and each has a different function depending on the car’s needs. Also, the regulation changes every year, so the teams need to adapt their car to the new rules imposed by the FIA, which is the governing body for world motor sport since 1904. In fact, an introduction with the most important aerodynamics compounds is key to the work’s development and to start the analysis from our new aileron.

3 BACKGROUND AND REVIEW OF THE STATE OF ART

3.1 HISTORICAL IMPROVEMENTS

Formula One is the most popular and prestigious motorsport competition in the world, and is known all over the world as the highest speed cars racing in all the continents. This sport started the competition in 1950, but the antecedents came from 1894, when in France began being popular to organize races between villages in the connecting roads. These types of races were terribly unsafe with frequent accidents for the drivers and also with the spectators in the surroundings.

The first Grand Prix started in 1906 and was organized by “Automobile Club de France” (CAF) and the duration of the race was two days, where the drivers had to race six laps in Le Mans, with a 105 kilometres distance per lap. The winner between 32 contenders was a Renault car driven by a Hungarian pilot called Ferenc Szisz, showed in figure 10, who finally died in 1944.

Succeeding France, other European countries started organizing races with the name Grand Prix, and each territory with their personal rules and differences in the cars. Italy, in the location of Monza, where there's still a circuit, in 1922, or Spain in 1924, are some of the examples of races before the modern era of the Formula One arrived [7].

Figure 10. Ferenc Szisz, first Grand Prix winner in 1906. (Source: [7])

The Second World War (1939-1945) was an inflection point to create the world drivers' championship, in 1950. The rules of this world championship title were defined before the war, but it was during 1947 where the old AIACR changed the name for FIA and implemented the regulation.

Silverstone, in the United Kingdom, was the initial race of the first Formula One World Drivers Championship, and the winner was the Scuderia Alfa Romeo, with the Italian Giuseppe Farina as the driver. At that time, a racing competition began and has persisted over time to the present with a far-reaching evolution in the cars.

3.2 RACING CAR EVOLUTION

From 1950 to the present, the automobile has changed drastically with improvements in all the different sectors. Every year, Formula One teams create several strategies to upgrade a car and compete for the world championship, and that has created a technical evolution shown in the difference between the first racing car and the actual ones.

Figure 11. 1950s Formula One car. (Source: [9])

In 1950, as we can see in figure 11, the cars were quite different from the cars we know nowadays. Motor was located in front of the driver, and didn't have any aerodynamic system as wings or flaps, so the model was very simple and created an orderly wake behind the car that was only affected by the pilot at the rear part of the car [8].

The first decade is not remembered for the aerodynamic improvements, some rules were changed and improvements in the engines started in 1954, when the Mercedes team ended the successful streak of Ferrari during two years. The cars started to change and one significant alternative was to install the engine behind the driver and in 1958, Stirling Moss won the first Formula One race with a rear engine car [9].

Figure 12. First race win with a rear engine car in 1958. (Source: [9]).

In the 60s, the front of the car started to be more compact and smaller due to the new engine location. Teams established a thinner front part, but without adding any aerodynamic element that could help in the performance [10]. With the new design, the car was aerodynamically stronger, but it wasn't until 1968 when one of the biggest revolutions in a Formula One car appeared, and totally changed the working process in a car design. That event was the addition of small wings on the front of the Lotus 49B car, impulsed by Colin Chapman.

Figure 13. Lotus 49B car in 1968. (Source: [9])

The Monaco Grand Prix in 1968 was the place where Colin Chapman started the race on the aerodynamic design in Formula One, helping Graham Hill win the race with the Lotus 49B. With that innovative design, all the teams started to set up new front wings in different positions and with several sizes to create the pressure differential needed of Bernoulli's principle to increase the speed.

As all the teams started to copy their strategy, the next year (1969), Lotus created a new aerodynamic system based on a double wing to innovate and stand out from their competitors. In this case, there was a front wing and a rear wing, but in a higher position to try to avoid the dirty air coming from the other cars competing.

Figure 14. Double wing Lotus car in 1969. (Source: [11])

This aerodynamic improvement helped Lotus to create a faster car, but the problem was the vortex appearance due the height of the wings, which created a turbulent wake behind making the way to drive harder for the other cars on the track.

After some races, the rear wings were prohibited from the FIA because of the turbulent wake produced which caused some accidents in different races. The teams had to change it for something not too dangerous and with less generation of turbulent airflow to the cars behind, so the solution was a design of new spoilers that could add an aerodynamic improvement without being a danger for the competition. These spoilers were similar to a rear wing, but with a smaller height and close to the surface of the car.

Figure 15. New spoilers in the F1 car. (Source: [11])

As in all the historical moments in Formula One, the aerodynamics started to change due to the regulations. All the teams had to check if their pieces were designed according to a pattern, and that happened during the 70s. Some teams tried to introduce new methods, but there wasn't any huge innovation system because of all the dimension stipulations.

The biggest improvements in this decade came from the British Scuderia Lotus, one of the pioneers of the aerodynamic changes in the past, with the introduction of a pointed nose car and side pontoons in 1972 and the introduction of the ground effect in the 1978s vehicle.

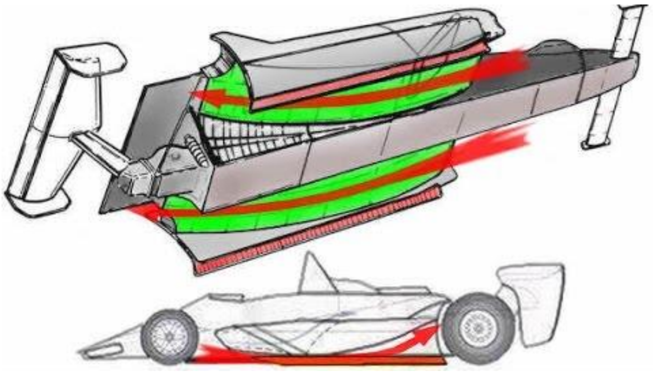


Figure 16. Ground effect picture. (Source: [3])

This ground effect made a suction consequence on the car's floor, and increased the downforce to improve the performance in the curves. Lotus had changed not only the aerodynamics of the front and rear of the car, they also took advantage of the floor to make their car quicker. The design of the floor to improve the car's downforce is still a strategy nowadays in the new hybrid era of Formula One, so that was also considered a great progress in the aerodynamics.

Figure 18. Lotus 72 design. (Source: [10])

Figure 17. Lotus 78 racing car. (Source: [10])

Once finished in the 70s, the next decade of the 80s was related because cars also received a progression in the aerodynamic system due a new regulation. The ground effect gave the teams a new way to create a faster car in the curves, and that increased the chance of accidents in the fast corners during the next year. The safety of the drivers was more important than the speed for the FIA, that's why they prohibited the use of side skirts in 1981, and the flat bottom obligation arrived in 1983.

Also, in 1981, McLaren implanted the first racing car with a monocoque chassis made of carbon fiber, the same material as the actuality. This material has a considerable strength, that's why they could build a smaller robust chassis with almost any shape possible without affecting the aerodynamic design of the car [12]. In the Annex document, there is a brief introduction about our theorical material and properties of our new front wing creation.

Figure 19. McLaren with a carbon fiber chassis 1981. (Source: [11])

To increase the downforce and improve fast corner's speed, the teams had to introduce more air at high speed below the car, that's why some groups experimented removing the front wing or simply having a dimension reduction in the nose of the car. This evaluation didn't last long and during the continuous seasons, the car design was recreated with a smaller front and rear wing more refined.

Formula One was increasing their world effect in the 90s, and the cars started to be safer, with the FIA adding safety elements on the track to reduce the damage of an accident. With these changes, the front part of the car also started the evolution of a new design using a higher nose and a greater distance from the asphalt. That change helped to introduce more fast airflow on the bottom of the car without the necessity of removing the front wing. As we can see, the front part has made an evolution during the last decades, but substantial technical changes are going to appear during the hybrid era (2014-2020) [11] [13].

Figure 20. Tyrrell 019 with a higher front wing in 1990. (Source: [13])

The 2000 arrived and the cars started to be more laborious with difficult designs and some additional fins on the sides. Teams wanted to take advantage of every part and centimetre of the car to create a competitive car to fight against their challengers. One of the most relevant improvements arrived in 2005, when the French team Renault created a system called Mass Damper, which consisted in the addition of a 10 kg mass between the two hooks working as a shock absorber [14].

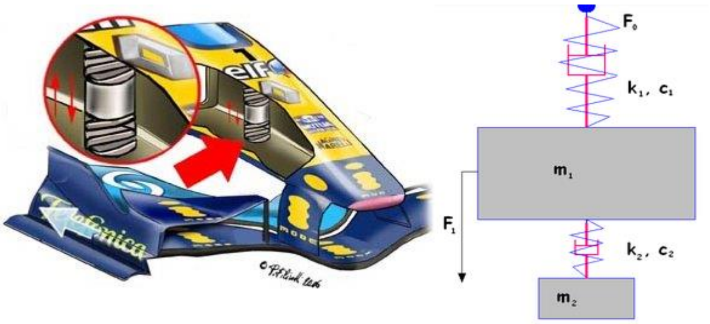


Figure 21. Mass Damper system in the Renault car. (Source: [14])

Owing to this new change, the system was capable of absorbing the vibrations caused by the irregularities in the asphalt with the high speed of the cars making it easier to stay stable in the front wing part and increased the aerodynamic efficiency. That helped Renault to win the world championship in 2005 and 2006 with the Spanish driver Fernando Alonso [15].

Figure 22. Renault R25 car in 2005. (Source: [15])

Next years were defined as the baroque aerodynamics, because all the engineers started to implant new aerodynamic appendages in every part of the car. The teams didn't worry about the aspect of the car, they only wanted to take advantage of every section, adding a new type of flap or wing to increase the downforce and avoiding the dirty air from the car in front. Some teams changed their front wing and fins, creating new versions which were more resistant and some of them had new elements added, such as horns, which created new vortices affecting the car behind.

Despite the new creative ideas, those aerodynamic changes didn't last much time, because in 2009, the FIA changed the regulation prohibiting the possibility to add aerodynamic elements in the body of the car. Teams had to adapt to the new rules, and that's what Brawn GP did, creating the new double diffuser, helping Jenson Button to win the world champion title the first year (2009) [16].

Figure 23. Double diffuser Brawn GP. (Source: [16])

The regulations allowed bodywork elements to be placed in the area close to the rear impact structure, and some teams took advantage of this using the structure as a new diffuser channel. The new design had two separate channels and the key to its operation were the holes made at the junction between the flat bottom and the diffuser.

One of the most visual changes during the decade was the shark fin, an idea created by Milton Keynes. This new aerodynamic element was located in the engine hood area and arrived with the utilization of the F-Duct, a system monitored by the driver which reduced the drag effect of the rear wing in some cases. The drivers could open the F-Duct manually to unblock an extra air jet to change the laminar flow of the rear wing, which produces the necessary load in the curves, for a turbulent regime to reduce the drag [17].

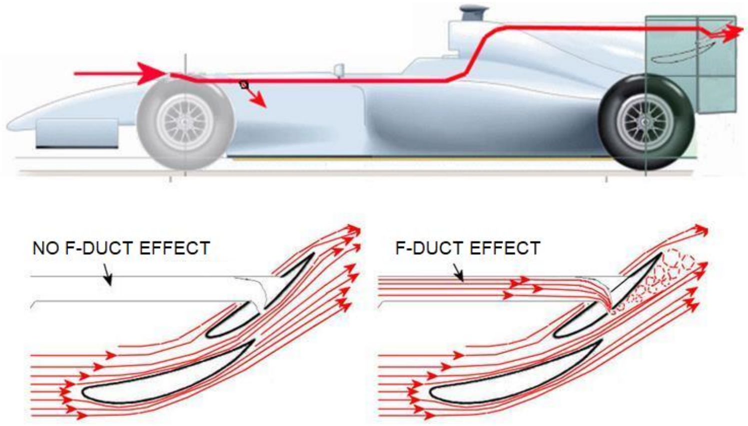


Figure 24. Aerodynamic effect caused by the F-Duct. (Source: [17])

Next year, in 2011, the FIA changed the regulation and all the aerodynamics of the car couldn't be monitored manually by the driver during the race, so this modification ended with the F-Duct idea created by McLaren the year before. Nonetheless, this year arrived to F1 one of the most important changes in the history, to promote racing and make it easier for the drivers to follow the car in front helping the overtaking, this element was called DRS and placed on the rear wing.

The Drag Reduction System, allows the upper flap of the rear wing to pivot on its trailing edge, reaching a nearly horizontal position. That creates an air entrance in the rear wing on the straights that reduces the drag of the car and helps arriving to a higher top speed in the linear sectors of the track.

This element was created by the FIA, so it has some stipulations to be used in a race. First of all, the DRS wants to help the overtaking and racing, that's why it can only be activated when the car behind is less than a second from the car in front, and only in the sectors specified in each track. Normally the DRS zone is located on the straights of the track, and there are between two or three different places in the same lap.

Figure 25. Mercedes AMG DRS opened against McLaren with DRS closed. (Source: [2])

Activating the DRS is done by the driver at the steering wheel with a personal button, and in a race can only be activated before the third lap and only one second behind a car. Also, the weather conditions affect the use of the system, because the FIA has the possibility to disable the DRS during a race if it's dangerous for the drivers, as it can be a wet track. Some studies say that you can obtain an estimated bonus of 10-15 Km/h in the straight compared with the other cars [18].

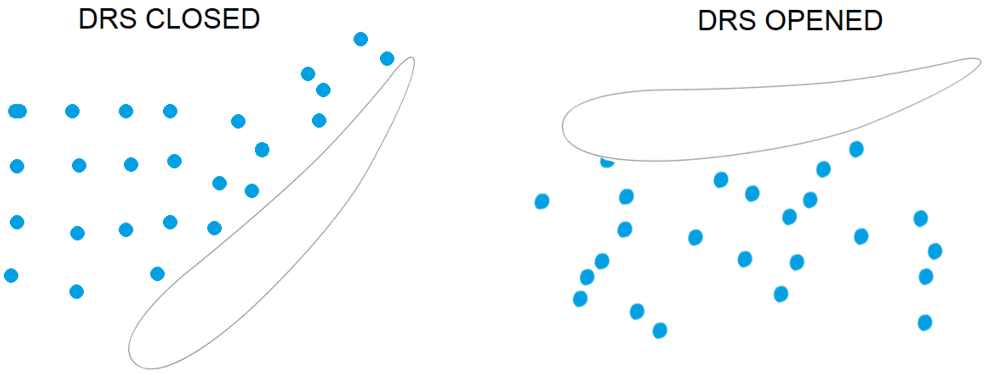


Figure 26. Comparison of the air molecules with DRS opened and closed.

Figure 26 shows us the difference of performance between an opened and closed flap in a straight line. In the first case, the flap is closed and the airflow can't pass through the wing generating drag and downforce to the car, which is important to have in the curves and not in the straights. Then, when the vehicle opens the flap in a straight (DRS zone), it reduces considerably the drag and downforce because the airflow reduces the contact with the surface of the rear wing, so the car has less resistance and can arrive at a higher top speed.

3.3 TURBO HYBRID ERA

Actually, Formula One has arrived to the recognized V6 turbo hybrid era (2014-2021), where the cars have been improving the potential with a clear team dominance, Mercedes AMG Petronas. The sport wanted to create a new image reducing the environmental impact and creating an electrical power unit called ERS, which gives more power to the car when the battery is charged [11].

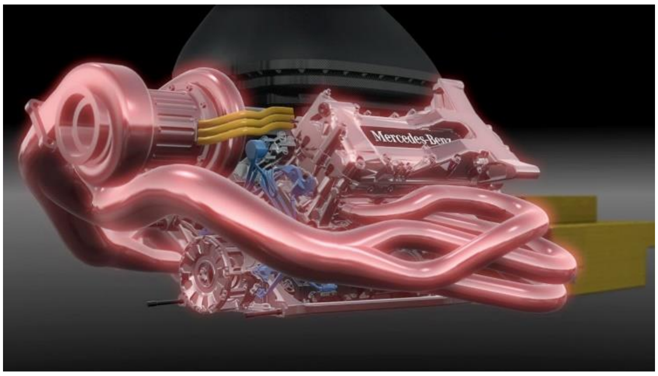


Figure 27. F1 Mercedes Hybrid turbo V6 power unit. (Source: [19])

These new types of motors allow the teams to reduce the fuel used in a race car, and that decreases the weight of the cars, creating an upgrade speed. The combustion is much more efficient than the old V8 motors, and the new rules helped Mercedes to win from 2014 to 2020 every single world championship title [19].

One of the last improvements of the car arrived in 2018, when from the hand of the FIA, with the help of Mercedes and other teams, decided to add a safety system used in motorsport cars that protects the cockpit from impacts from external objects or collisions. This is a titanium element anchored in the front part of the cockpit of the single-seaters, dividing the vision of the pilots in two to protect the heads of the drivers.

The Halo protecting system passed several simulations to ensure the safety of the drivers, and the final piece was made of titanium because it's an ultra-resistant material with a weight of around 10 kilograms. From here, each side had to adapt their aerodynamics to the new shape of the cockpit, and FIA decided to allow the addition of small wings on the top of the halo with some specifications [20].

Figure 28. Red Bull's halo in the racing car. (Source: [20])

3.4 NEW 2022 REGULATION

Summarising, we can see that all the elements have variate and the cars that were designed in the beginning of the sport are not the same as the ones we have in the actuality. In 2022, the rules will make an enormous change on the cars to increase the fights between the cars and see more overtakes on track. When the turbo hybrid era arrived, overtaking was one of the most difficult things caused from dirty air and turbulence generated from the cars in front [21]. Current F1 cars, when followed by another, generate turbulence that can cause losses of up to 50% of the downforce in the car that is trying to overtake. In this way, overtaking often does not take place, or the drivers are forced to lift their foot to not affect their tyres. The new cars in 2022 are trying to reduce the turbulence with a new design studied in wind tunnel and computational fluid dynamics (CFD), as we are going to do with the front wing, which will be designed from some pictures posted in the official F1 webpage as in figure 29.

Figure 29. New design for 2022 regulation. (Source: [36])

4 FORMULA ONE AERODYNAMICS

During a competition, there are a lot of loads created by the motion of air, affecting the performance of the automobiles drastically. The best way to study is with a cross section of a wing, and see how moving the car with high speed can generate different alternatives.

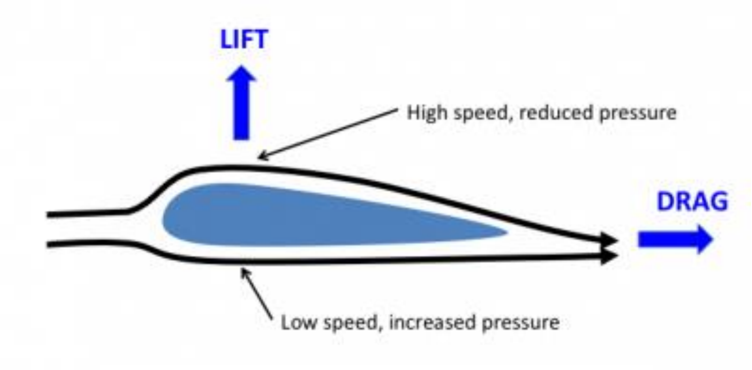


Figure 30. Forces generated in the F1 wing. (Source: [47])

Dimensions and sections have a crucial effect in this case, because the angle or shape can affect the area in the upper and lower surface and change the physical properties. As we can see in figure 30, the difference of speed that arrives to the faces of the wing creates a higher pressure on the lower surface and a lower pressure on the top of the cross section. This pressure differential comes with a force that lifts the vehicle, but also with a new force called drag, which is usually smaller, but affects negatively to the performance. A force in the opposite direction of the movement must be mitigated to go faster, and that's what the teams do with the aerodynamic improvements.

Talking about the forces generated in a racing car, there is also a side force also generated moving the car at high speeds, but this force affects less to the performance. So, at this point, the three forces affecting a car are the drag, the lift and the side force, as we can see in figure 31 [47].

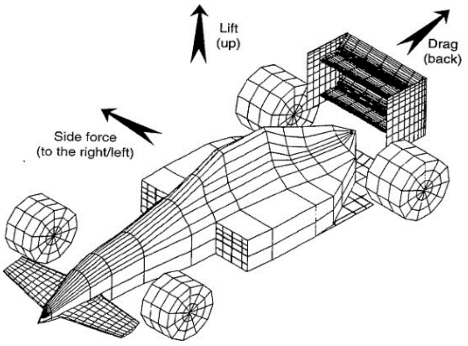


Figure 31. Directions to identify the aerodynamic forces. (Source: [47])

Side force doesn't have a critical impact on the performance, and in the case of Formula One, the most important point is to generate an opposite force to the lift load, which is naturally known as downforce. The creation of downforce is extremely important and leads to major improvements in a race, that's why the cars have created inverted wings (like the front or rear wings) to face the lifting impact of high-speed air motion. The aerodynamic downforce helps to increase the performance of the tyres in a fast corner and in high-speed tracks reducing considerably the time per lap.

4.1 LAMINAR AND TURBULENT FLOW

Once seen the forces affecting, there is a substantial difference depending on which air flow arrives to the car and changes increasing the speed on the track. A Formula One car has a lot of aerodynamic components that are in contact with the exterior, creating different streamlines shapes. Those are easy to study in a wind tunnel with the addition of smoke, because it shows the traces of the airflow near the cars that are being studied, but also can be analysed with CFD simulations, like it could be in the real wind tunnel.

These curves describing the pictorial movement of a fluid can appear in two different ways, considering one of them much more aerodynamically efficient. The difference appears in the way these streamlines adapt to the car, because in the first case, the lines follow exactly the shape of the body, so the steady-state flow is attached to the surface. Otherwise, the flow lines can be separated to the surface or vehicle's body, causing a disorder in the air passed through the car. These two ways of finding a streamline separates the airflow in two possibilities, laminar or turbulent flow, as we can see in the figure 32.

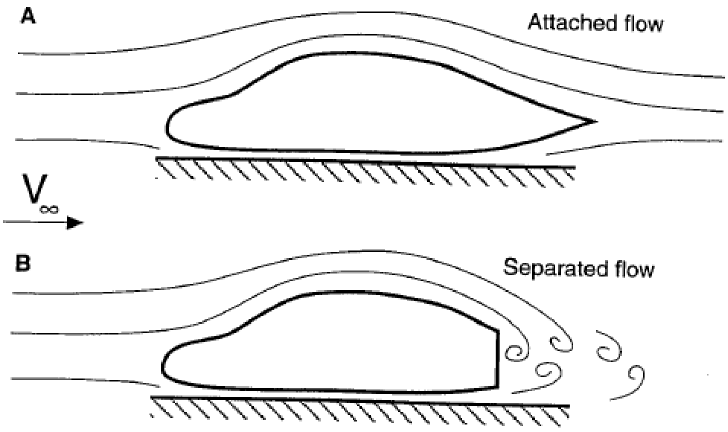


Figure 32. Two types of streamlines. (Source: [35])

In the first case, the car has a beneficial figure because the airflow continues attached to the surface and generates a laminar flow at the end, making it easier to follow for other drivers. Compared to the first one, in the second picture, the streamlines start to separate from the car and create a turbulent area at the back of the car that generates instability and vibrations to the other cars.

Laminar stream or laminar flow refers to all the ordered fluids moving in parallel streamlines following the direction of the motion and velocity of the fluid, and the turbulent flow is when the fluid particles are moving in different directions generating turbulence and unsteady movements that can be unpredictable. In figure 33 it's clearly shown the entrance of laminar flow (probably because there is no obstacle in front) and the turbulent wake generated from different parts of the car like the tyres or the front wing [22].

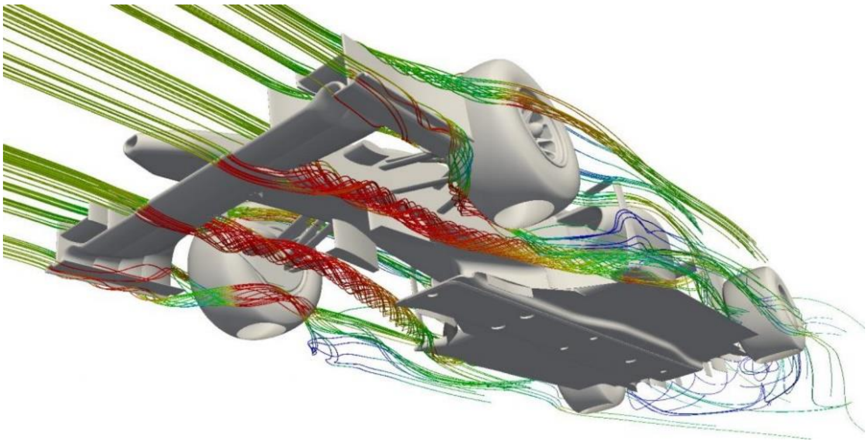


Figure 33. Laminar and turbulent flow over a Formula One car. (Source: [22])

4.2 REYNOLDS NUMBER

These types of flows are defined due the Reynolds number, which is a dimensionless number created by Osborne Reynolds, and is one of the most commonly used to talk about Formula One aerodynamics. Knowing this value helps to predict the type of flow patterns affecting the car and can be useful to compare the results from unlike models and speed in a testing session. When the value is over 10^5, the flow is considered turbulent and can negatively affect the wings and planes of the car with vibrations, and if it's less than 10^5, the drag and lift forces can be considerably different [23] [47].

$$\mathrm{Re}_D = \frac{\rho V D}{\mu} = \frac{V D}{\nu}$$

Later in the CFD simulations, we are going to consider a same type of flow because it doesn't change so considerably. In table 1 there is a brief description for both possibilities:

Table 1. Different Reynolds value effects.

Re VALUE	***PREDOMINANT***	***DESCRIPTION***
Low	Viscous force	The fluid viscosity creates an ordered particle movement separated in layers and generating a laminar flow.
High	Inertial force	Opposite to the previous, inertial forces predominate and we have a high velocity fluid that even the viscosity cannot avoid the chaotic movement of the particles.

4.3 DIMENSIONLESS NUMBERS

The Reynolds number is not the only value used in the world of racing cars, there are also different alternatives to define fluid properties or behaviours, and one these are the dimensionless numbers. A dimensionless number is a value without physical units that is created from other variables, and has a physical meaning for the system studied. In this case, as we are studying the aerodynamics of a Formula One car, they will have a relation to fluid mechanics and the airflow affecting the car [24] [47].

Skin-friction coefficient (C_f) is a non-dimensional number that is also used in analysing the vehicles because it gives information about the level of friction between the surface and the outside fluid, in this case air. This value doesn't have units and depends on the friction resistance and the dynamic pressure:

$$C_f = \frac{\tau}{\frac{1}{2}\rho V_\infty^2}$$

In the numerator we have the surface shear force per unit surface (N/m2) and in the denominator is the value of the dynamic pressure depending on the density of the fluid (Table 2). This value will be interesting to study the drag conditions and it's easier to use than international units to compare different situations.

Table 2. Density and viscosity at 20ºC and 1 atm.

FLUID	***ρ [kg/m³]***	***μ [Pa · s]***
Air	1,22	$1{,}8 \cdot 10^{-5}$
Water	1000	$1{,}0 \cdot 10^{-5}$
SAE 30 Motor Oil	919	$4{,}0 \cdot 10^{-5}$

Once we have seen the aerodynamic forces, shown in figure 31, engineers also use different non-dimensional numbers referring to the resistances on the vehicle's surface. To avoid depending on the speed of the car, these values are divided by the square of the velocity making it depend only on the shape. There are three of them, one for each force, but normally the one related to the side is not important compared with the other two [24].

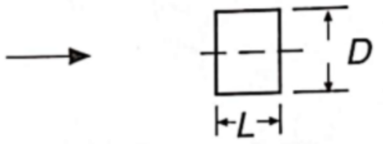
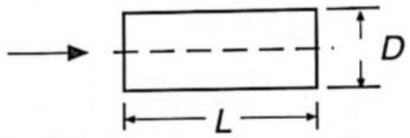
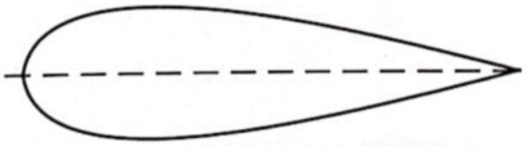
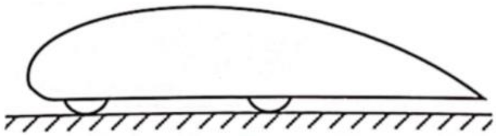
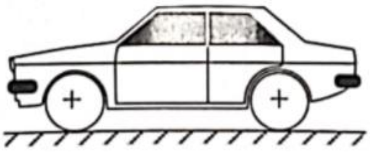
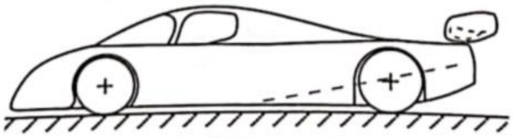
$$C_d = \frac{D}{\frac{1}{2}\rho V_\infty^2 A} \; ; \qquad C_l = \frac{L}{\frac{1}{2}\rho V_\infty^2 A} \; ; \qquad C_y = \frac{Y}{\frac{1}{2}\rho V_\infty^2 A}$$

Each formula refers to one different type of force, C_d is the drag coefficient, C_l defines the lift coefficient, and the last one is the side-force coefficient C_y. The numerator is the value in newtons of the force affected by the air, and each one has an impact on the final vehicle's performance. Value mentioned A, located in the denominator, is a reference area (in m^2), and for a vehicle will be the projected frontal area if we consider that the air comes directly from the front part. Tendency is to reduce the coefficients to the minimum expression, but it's important to have a balance between them to arrive at the optimal value in each case, because not all the tracks and conditions are the same. In table 3 we have some configurations with the corresponding coefficient that show us the shape's dependence.

The relation between lift and drag coefficient is called aerodynamic efficiency, and it's a very important value for the teams because it shows them how their car will be affected. The objective is to have a balance between both values and maximize the value with less drag and more negative lift units [25]. The formula can be used for the force values and with the coefficients, but the final value has to be also non-dimensional or in percentage.

$$Aerodynamic\ efficiency = \frac{L}{D} = \frac{C_L}{C_D}$$

Table 3. Drag and lift coefficients in different situations.

FIGURE	***CONFIGURATION***	***C_D***	***C_L***
Circular plate		1,17	0,00
Circular cylinder with L/D < 1		1,15	0,00
Circular cylinder with L/D > 2		0,82	0,00
Low drag body		0,04	0,00
Low drag vehicle		0,15	0,18
Conventional car		0,43	0,32
Racing car		0,75	-3,00

Once seen the dimensionless numbers associated with the forces, there is also a physical aspect that can affect the performance of a high-speed vehicle. In order to find in each case, the aerodynamic load it's important to know the pressure distribution on the outside of the car to see how this factor can affect the final result.

$$C_p = \frac{p - p_\infty}{\frac{1}{2}\rho V_\infty^2}$$

C_P is the pressure coefficient that gives information about the pressure distribution in the car, and as the others, it's independent of the speed. Actually, this coefficient is directly related to the pressure *p*, because the other values maintain constant, also the density, which is considered incompressible (ρ = constant) for velocities less than the supersonic.

These dimensionless numbers are useful to study the aerodynamic performance of the car in several conditions, and are constantly compared to find the best possible scenario for their vehicle. Finally, as we have commented recently, there are some situations where the density of the air can't be considered constant, and to find the information, the Mach number is very useful. This value makes a relation between the velocity (U) and the sound velocity in a fluid (U_s), and shows if the working fluid can be considered incompressible or not, and therefore if the density is constant or not [27].

$$M = \frac{U}{U_s}$$

Table 4 represents the type of air regime depending on the Mach value, and normally, even in Formula One, the value is smaller than 0,3 and can be considered incompressible air. This will make the calculations and the analysis easier, and in our case, will simplify the CFD simulations, but in some cases, can appear in the exhaust pipes of a racing car [26].

Table 4. Mach values and their regime in air.

REGIME	**SPEED VALUES**		
	MACH NUMBER	***KM/h***	***MPH***
Subsonic	< 0,8	< 980	< 609
Transonic	0,8 - 1,2	980 - 1.470	609 - 914
Supersonic	1,2 - 5,0	1.470 - 6.126	915 - 3.806
Hypersonic	5,0 - 10,0	6.126 - 12.251	3.806 - 7.680
High-hypersonic	10,0 - 25,0	12.251 - 30.626	7.680 - 19.031
Re-entry speeds	> 25,0	> 30.626	> 19.031

A lot of non-dimensional numbers are used in this technical sport, but these are the most common ones, and also the ones we need to understand the study of the computational fluid dynamics (CFD) of the front wing design.

4.4 BOUNDARY LAYER

Talking about aerodynamics, it's crucial to understand the concept of boundary layer and the importance related to the two types of flow we have explained (laminar and turbulent). First of all, in the boundary layer, the effects of viscosity are significant and the velocity on the surface of the stationary plate becomes zero. As mentioned, there are two types of boundary layer, the laminar and the turbulent, and in each case the flow will be as explained in the point 4.1.

Airflow particles moving through this layer start first in a laminar flow, where they are parallel to each other, and later, on its trajectory the deposition starts to be more chaotic with a direct increase of thickness. Figure 34 shows how the boundary layer looks on a car's surface, and the different shape possibilities in a flat plate, with a transition zone between the laminar and turbulent flow.

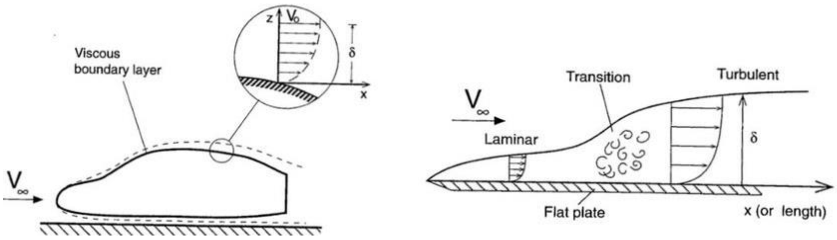


Figure 34. Boundary layer on a vehicle's surface. (Source: Researchgate Boundary layer)

In relation to a Formula One car, this boundary layer, and especially its thickness (δ), can modify the performance, because it's proportional to the viscous friction drag that the car generates. So, when an engineer has to create new aerodynamic models for their car, they have to check how the boundary layer state will be, to take advantage of each regime in different situations.

The increase of thickness can generate friction drag, and a too-steep layer can create a flow separation and an addition of drag with a loss in the wing's downforce, so it's important to arrive at a laminar flow to avoid losing time during a race [28]. To conclude, more aerodynamic information is located in the annexes and can be useful to our final simulations.

5 2022 NEW REGULATION CHANGES

Next year the sport will have one of the biggest set-up rules changes in history, and that's going to cause a huge difference in the way the teams design and change some aspects in their cars. There have been innumerable modifications and adaptations in the regulation since this sport was created in August of 1950, and these changes are all created to have a closer, fairer and more exciting racing between the cars on the track.

2022 Formula One cars will have a radical new design philosophy compared to the actual season (2021), with a difference in the looking aspect and aerodynamic changes that want to achieve more competitive racing and a reduced gap between the cars, to improve the racing and spectacle for the fans. Actually, the new design is fully focused on promoting better racing, and that will be possible with a stunning revolution in the aerodynamics, starting from the front wing.

This new design has been studied with the ultimate engineering to make more efficient the way a car in front can affect the one behind during the race. In actuality, the cars can currently lose up to half of the downforce that gives them grip around the corners when they are following behind another car, and that appears because of the dirty air from a car in front making them unstable to drive. On the 2022 car, the nose concept and front wing has been completely rethought to help avoid these problems, and also the rear wing will try to push the airflow higher, to avoid them affecting the performance of the car behind [30] [31].

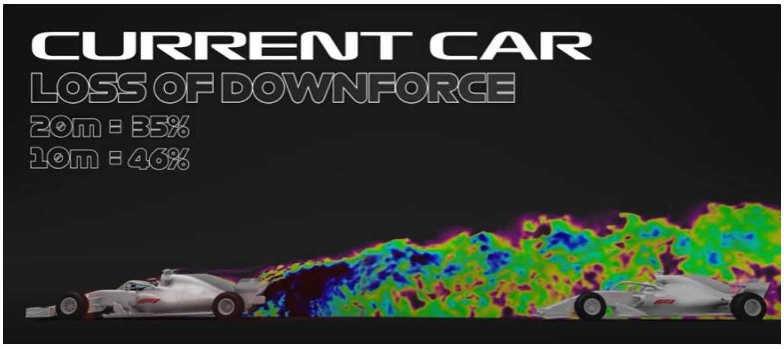


Figure 35. Loss of downforce in 2021 cars. (Source: [31])

Figure 35 gives us evidence on how much loss of downforce the car that is racing behind receives, and this percentage of value is higher as closer the vehicles are on track.

Actually, an estimation done with all the data collected during the year, shows that a car that is 20 meters ahead creates the sufficient dirty air to make the follower lose 35% of the downforce, and 46% if the car is only 10 meters of distance.

This is the effect that wants to be changed in 2022, to reduce the dirty air and the turbulent flow created from a racing car in front, and to arrive at this objective, a lot of aerodynamic changes must be done in relation to the new regulation.

Figure 36. Hypothetical loss of downforce in 2022 cars. (Source: [31])

Obviously, the new designs have not been done yet, and all the information is not real and comes from different simulations, but the objective is to reduce the values to 4% of downforce lost with a 20-meter distance, and 18% with a 10-meter distance, which is a clear difference compared to the actual results [30] [31].

To reduce these values, the car has to be changed completely, starting from the length and weight of the cars, but also the shape and figure will be totally new with other elements never seen before, and with a simplification of all the aerodynamic elements. Increasing the simplicity of the aerodynamics, all the parts from the car that used to create turbulence to the car behind will be changed or removed.

5.1 2021 FRONT WING

This last year of the regulation (before the change in 2022), the front wings have been one of the most technical and complicated elements in terms of aerodynamics, every single part of the nose was designed for a specified reason and with a final objective.

Figure 37. Red Bull 2021 front wing. (Source: [2])

It’s true that, during the start of the new turbo hybrid era, the noses were different and more complicated than the ones used today, but these 2021 front wings are even less simple that the ones we’ll have the next year in 2022.

5.1.1 FLAPS

Figure 37 shows the Red Bull F1 team’s front wing during this season, and as we can see, there are different parts to study and see how they work. First part to analyse are the flaps, which are the biggest part and where the maximum quantity of airflow is hitting during a race. These are aerodynamic wings that can be modified its position during a race depending on the conditions, if the car needs more speed on straights, they will be more bent (low downforce), and if the priority is to increase curves performance, the flaps will have more inclination (high downforce).

Normally, the low downforce is needed on tracks where the major parts are straights or fast corners, like Monza (Italy) or Silverstone (United Kingdom); meanwhile on tracks with a lot of corners and curves, such as Monaco, will be interesting to have a high downforce set-up with the wings with a higher inclination.

The most important function of the flaps is to avoid the direct contact, from the airflow coming, with the front tyres because that could generate turbulence and vibrations on the car’s movement and make it really difficult to drive. Even though this is the most important purpose, it is also important to generate pressure in opposition to the lifting force (explained in point 4) and create downforce to press the car to the track.

After the changed regulation in 2019 (which is still used in 2021), it was only allowed to use four different flaps in every section, adding the main plane.

5.1.2 MAIN PLANE

Is the front wing's lowest part and where all the other components are fastened, so is considered the nose's main and largest element. This section has the wing inverted in comparison with the flaps, and is the most resistant part because it is the first contact from the car in a frontal crash or collision with other cars. In terms of regulation, the central zone is a common design for all the teams that cannot change and with stipulated dimensions (figures 38 and 39 show us the explanations in the rulebook 2021), and the rest of the main plane can be a new design, but according to some dimension criteria.

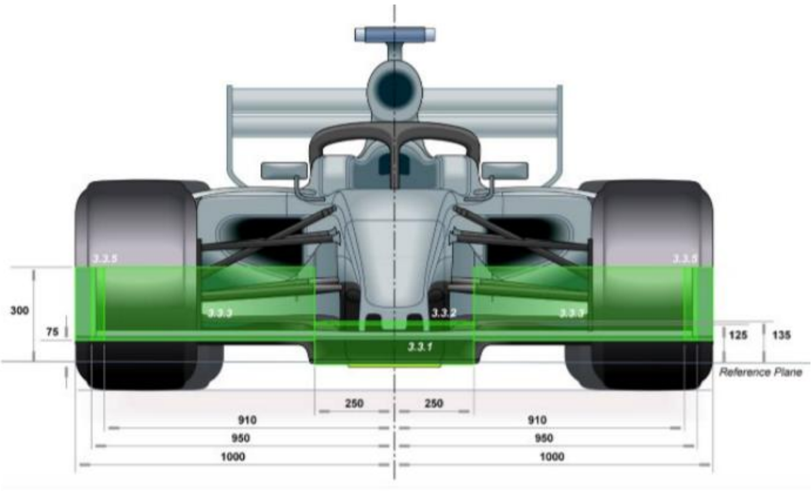


Figure 38. Rulebook page 17. (Source: 2021 FORMULA 1 TECHNICAL REGULATION).

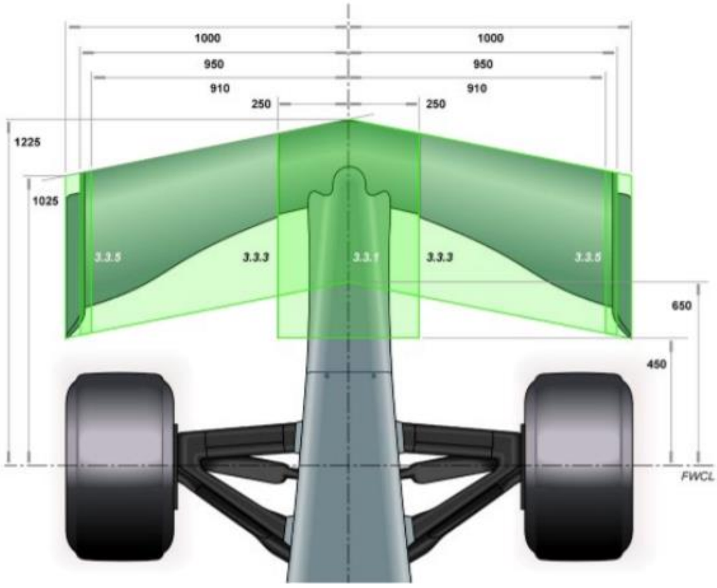


Figure 39. Rulebook page 17. (Source: 2021 FORMULA 1 TECHNICAL REGULATION).

As we are going to see during the evolution of the front wings in 2022, there will be one of the biggest regulations changes in this sector, also replacing how the car will change the airflow direction.

5.1.3 ENDPLATE

This was one of the last improvements of the front wing, with the addition of these vertical profiles at the end to reduce the vortex creation. With the old designs, as the air with higher pressure tends to move towards areas with lower pressure values, without this profile, the combination of air would be done on the edges, creating vortices and turbulent flow that is not ideal for the performance of the car. This is why the experts consider that the modelling of the endplates made cars much more efficient.

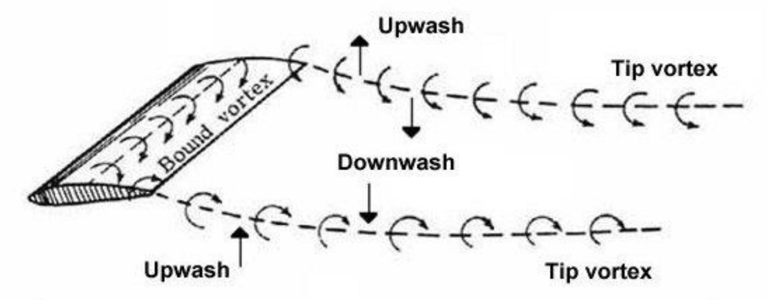


Figure 40. Vortex creation in a wing without endplates. (Source: [35])

Without this vertical plane located on each side of the wing, the vortex creation could be critical for the aerodynamic efficiency, because it decreases the downforce that the car can generate. Furthermore, the engineers also use these endplates to avoid hitting the tyres with the direct airflow, including an inclination to the outside to expulse the air of the wheel's trajectory, as is shown in figure 41. In this case, it's an old front wing version that is not used in the 2021 season, but easily shows how this vertical piece also has a slope to the side.

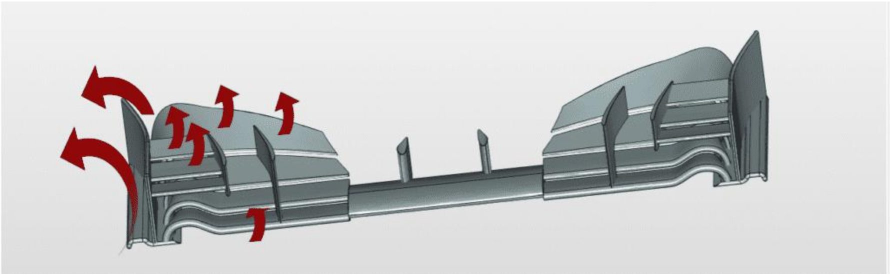


Figure 41. Airflow direction with the addition of the endplates. (Source: SolidWorks)

As with all the nose components, the endplates will also change with the new regulation and we are going to check it in the next section, with the differences between one and the other.

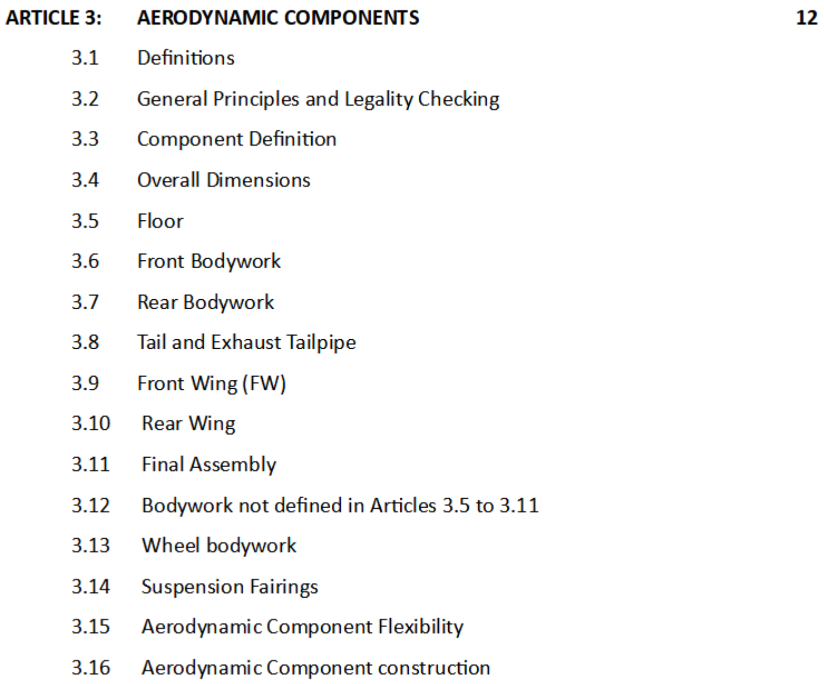
5.1.4 FOOTPLATE

This is the last recognized element of a Formula One front wing during the 2021 season, and has the least implication in the performance of the car, although it remains important. In this case, it's located at the end of the endplate, along the horizontal axis, and is used to create a new vortex on the side of the car that affects positively to the tyres. Each team had different strategies during 2021, but the most frequent shape was a curved semicircle carbon fiber piece, to bring the vortex to the basal part of the front wheel.

Figure 42. RedBull F1 team's footplate. (Source: [8])

5.2 2022 FRONT WING

With the start of the new season, all the information collected and the data saved during the past years won't be needed. Every team must create a completely new and contrasting model according to the articles that the FIA has stipulated in the rulebook (figure 43). This new regulation, a version of the 19th February of 2021, has a complete article talking about the conditions the aerodynamic components must achieve (article 3), and will be used to take some ideas to design the sketch of the new prototype of the 2022 front wing [31] [32].



ARTICLE 3: AERODYNAMIC COMPONENTS **12**
3.1 Definitions
3.2 General Principles and Legality Checking
3.3 Component Definition
3.4 Overall Dimensions
3.5 Floor
3.6 Front Bodywork
3.7 Rear Bodywork
3.8 Tail and Exhaust Tailpipe
3.9 Front Wing (FW)
3.10 Rear Wing
3.11 Final Assembly
3.12 Bodywork not defined in Articles 3.5 to 3.11
3.13 Wheel bodywork
3.14 Suspension Fairings
3.15 Aerodynamic Component Flexibility
3.16 Aerodynamic Component construction

Figure 43. Copy of the Article 3 of the 2022 F1 Technical Regulations.

All the information needed is collected in the report, but the front wing specifications are explained in the articles: 3.6, talking about the car's front bodywork; 3.9 where the different parts of the front wing commented before are explained; and 3.11, considering how the final assembly and manufacture has to be done [32].

This is a very complex and time-consuming task where teams work for years with groups of professionals, new designing technologies and spend large quantities of money, so the final front wing design of 2022 will be quite different from the model used to do the simulations. Four principal sections seen from 2021's noses must be created together to design the best possible way to satisfy the two principal ambitions for 2022 front wings:

- Generate the needed downforce to increase the speed on the curves to reduce the tyre degradation with the prevention of car sliding. It's considered that the front wing, which is the first part of the car in contact with the outside, can generate between 25-30% of the total downforce. (Table 5)

‣ Redirect the airflow passing through the wing to avoid affecting negatively other car aerodynamic branches or components. In this case, with the new simplified designs, will be arduous to complete because teams are not allowed to use cascade winglets or other type of winglets like they could before the 2019 changes.

Table 5. Approximations of downforce generation changes.

CAR COMPONENT	***DOWNFORCE 2021 CARS***	***DOWNFORCE ESTIMATED 2022 CARS***
Front wing	25%	15%
Bodywork	5%	5%
Floor and diffuser	45%	65%
Rear wing	25%	15%

As expected, the simplifications in 2022 will reduce the impact of the downforce generated on the car from the front wing, and will give more significance to the new floor and diffuser component, which will experience an extensive improvement in the ground effect.

5.2.1 FLAPS FROM 2022

The first element that has changed are the flaps, with a simpler design and less technical sections defined. Objective is to reduce the turbulence during racing, so there won't be any small winglets or extra aerodynamic elements between the four flaps that make up the new nose. Obviously, the most important change that arrives in 2022 is that the flaps will be directly connected to the front part of the car, and the initial reason is to avoid the creation of vortices that can create chaotic wake flow [32].

Because of the lack of information, we can't know how the teams will use this opportunity to take an advantage and create their designs, but in the CAD model used to do the simulations, the wings are all connected to the front part directly, as the figure 44 shows.

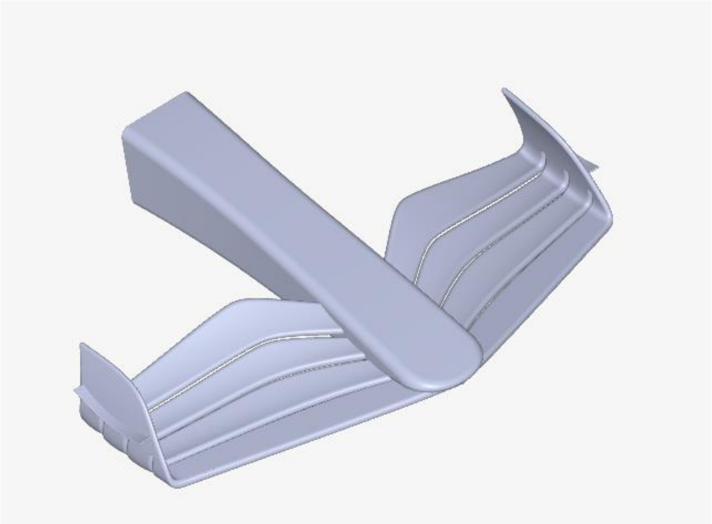


Figure 44. Flaps connected directly to the car in 2022.

5.2.2 MAIN PLANE FROM 2022

In 2022, any car will have a main plane that helps to connect to the chassis of the car, so the idea of this initial plane is removed, in this case, because the wings will be directly connected to the car. After all these years, where the main plane was the first constituent looking at the front plane and had the initial contact with outside's conditions, the new cars will be designed to have a wing on the starting line of the nose.

Avoiding a horizontal plane at the front of the vehicles can also help to reduce the irregular and the unbalanced flow that cars leave during a race. This will also be shown in the regulation of 2022, where they have deleted the section where the required dimensions were explained the past year in 2021 [32].

5.2.3 ENDPLATE FROM 2022

Endplates will also change with the new designs; the old vertical and curved pattern will be removed for a new curved and pointed end sketch where some of the rules have been changed. New regulation's article 3.9.2 shows the compulsory dimensions of the virtual surface that must be defined to create the endplates. Also, our design used in CAD wants to be the most realistic possible compared with pictures (figure 45) that the Formula One has shown to the public, but there is no sufficient information yet about the designs and dimensions.

Figure 45. New car drawing sketch. (Source: [36])

5.2.4 FOOTPLATE FROM 2022

Finally, even if its function is maintained, the new footplates will also change the shape and figure compared with the actual ones. In this case, the design has been simplified as a basic extrusion piece on the endplate's side with a curvature defined by recreating one wing to create a similar aerodynamic effect on the car. As we can see in figure 45 is a picture of the design provided by Formula One showing the initial sketch, and compared, in figure 46 we can find the CAD model created out of the pictures.

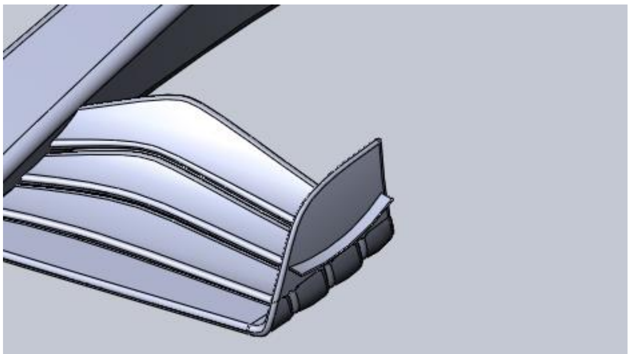


Figure 46. CAD model simulated.

6 CFD STUDY AND SIMULATION

Once we have introduced the fundamental ideas of the project, the aerodynamic influence in the sport and a new CAD model design of the 2022 front wing, we are able to start a performance analysis that will give us some information necessary to finally understand the behaviour in conditions similar to the reals. Formerly, to obtain the necessary results and find the final and optimized equipment, the selected choice was testing in a wind tunnel because it was useful to understand the aerodynamic problems and receive incremental data that helped to achieve improvements with the minimum resources. Nowadays, this procedure has changed due to the rapid evolution of a software capable of simulating the conditions of the flow movement called CFD (Computational Fluid Dynamics).

6.1 CFD INTRODUCTION

Computational fluid dynamics, also known as CFD, is a computer-based tool from the fluid mechanics branch that is used for the study and simulation of a fluid conduct with numerical calculations and mathematical algorithms. This software can be used for a lot of different studies with the possibility of having its results over and over to solve the problems that can appear during the different simulations.

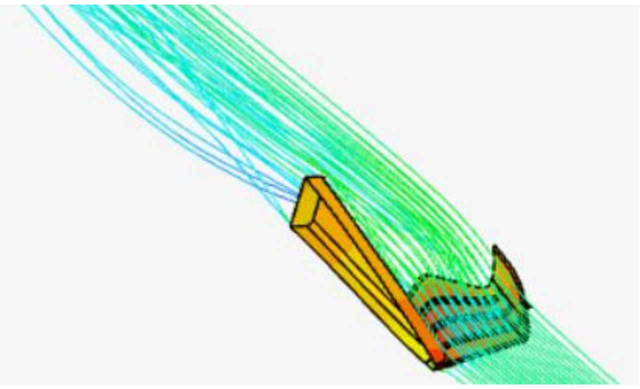


Figure 47. Simulation run in the 2022 prototype.

6.2 SIMSCALE SOFTWARE

Founded in Munich (Germany) in 2012, is the first cloud-based simulation platform used to perform simulations in the field of computational fluid dynamics, but also there is the possibility to use solid mechanisms or thermodynamics simulations.

Also, SimScale is used to avoid difficult and expensive software, because it's a free platform where the only tool needed is a computer with access to the internet.

At this point, there are different CFD capabilities to create the simulation, starting from internal or external study, laminar or turbulent flow, steady-state or transient, and other options like use rotary machinery or compressible and incompressible fluid simulation. In our case, we are going to see that the simulation will be studied as the external flow in contact with the front wing, with steady-state to reduce the time of simulation and demanding calculations, and an incompressible fluid with constant density because in Formula One the Mach number is smaller than 0,3 comparing the car speed with the sound speed.

Figure 48. SimScale logo. (Source: [43])

6.3 CFD PROCEDURE IN FORMULA ONE

Actually, in the sport, CFD methodology is a tool used in different areas to simulate the behaviour of the fluids related to the car, starting from the gas coming out of the exhaust pipe to the oils that lubricate the motor [26]. In this case, it is used to understand the airflow in contact with the car and see how it performs along the different areas of the front wing. Obviously, the simulations done from the working engineers are more complicated and closer to reality than our brief investigation about a possible design of the front wing, but they still use the same steps that we have done from the beginning of the examination.

Starting from the CAD designing model, the next step is to set the important points to simulate creating the mesh of the structure. Normally, there needs to be a balance between the number of points studied and the time the software needs to simulate the meshing process, because a too specific mesh structure can be impossible to run.

Next parallel step is to have a value configuration of the fluid parameters that can be even very extended and precise, or simplified with generic considerations. Once these three points are made, the next move is to run the simulation and wait for the program to define all the simulation steps and study the fluid conduct. When the simulation has finished, the results need to be analysed and understand the components like velocity and pressure or the fluid behaviour. The next point defines how this process has been made to find the final results of the simulations and see the performance of this new front wing available in 2022.

6.4 PREVIOUS CALCULATIONS

6.4.1 VELOCITY CALCULATIONS

Each circuit has a different value of high and low speed, and this number depends on infinite factors like the distance, height, shape of the circuit, and obviously the cars that are racing. In this simulation, we are going to see three different cases and compare them to find the variations between high speed traps, that are obtained in the straights, with medium or low speed sectors, which are usually the corners.

Table 6. Speed trap values in Monza 2021 (Italy).

MONZA 2021 SPEED TRAP		
Driver	***Team***	***Speed (Km/h)***
N. Latifi	Williams F1 Team	344,6
A. Giovinazzi	Alfa Romeo Racing	342,3
G. Russell	Williams F1 Team	341,9
L. Norris	McLaren Racing	341,4

To simulate a real high-speed value, we are going to take the 344,6 km/h of the Williams car in one of the fastest races of the year, the circuit of Monza. Table 5 selects the higher speed traps of the race in Italy, and this value will be introduced as a velocity inlet in the simulation with the approximation of 96 m/s [34] [35].

Then, an estimated low speed corner value can be 120 km/h, where the downforce of the car is very important to improve the aerodynamic of the car without affecting the tyres. In the International System of Units, we are talking about approximately 33 m/s.

Finally, a study arrived at the mean speed value in Formula One which is around 210 km/h or 58 m/s, so this value will be also simulated to see the differences with the others commented. Velocities that we are going to study in a simulation are resumed in table 7[35]:

Table 7. Simulated speed values.

Case	***Type of speed***	***Km/h***	***m/s***	***Approx. (m/s)***
1	Low speed corners	120	33,33	33
2	Average speed	210	58,33	58
3	High speed trap (Monza)	344,6	95,72	96

6.4.2 REYNOLDS NUMBER CALCULATION

Before creating the simulations, some parameters and values need to be found to introduce them as specific conditions and guide the reproduction of the real car on track. To arrive to the result, we need first to know the air physical properties in environmental conditions of pressure and temperature (T = 20ºC; P = 1 atm) [35]:

Table 8. Air properties at T=20ºC and P=1 atm.

Physical property	***Value***	***Units***
Pressure	101.325	Pa
Temperature	293	K
Density	1,196	kg/m^3
Kinematic viscosity	1,5295E-5	m^2/s
Dynamic viscosity	1,82E-5	Pa·s
Speed of sound	343,15	m/s

Now we can calculate the estimated Reynolds number for the maximum and minimum speed, and in the case of the average speed commented also in table 7. To select the value of length L, we are going to use the maximum value permitted in the 3.9.1 rulebook article, which is 300mm.

$$Re_{L\,1} = \frac{\rho VL}{\mu} = \frac{1{,}196\,\frac{kg}{m^3}\cdot 33{,}33\,\frac{m}{s}\cdot 0{,}3\,m}{1{,}82\cdot 10^{-5}\,Pa\cdot s} = 657.077{,}14 \approx 6{,}57\cdot 10^{5}$$

$$Re_{L\,2} = \frac{\rho VL}{\mu} = \frac{1{,}196\,\frac{kg}{m^3}\cdot 58{,}33\,\frac{m}{s}\cdot 0{,}3\,m}{1{,}82\cdot 10^{-5}\,Pa\cdot s} = 1.149.934{,}30 \approx 1{,}15\cdot 10^{6}$$

$$Re_{L\,3} = \frac{\rho VL}{\mu} = \frac{1{,}196\,\frac{kg}{m^3}\cdot 95{,}72\,\frac{m}{s}\cdot 0{,}3\,m}{1{,}82\cdot 10^{-5}\,Pa\cdot s} = 1.887.051{,}43 \approx 1{,}9\cdot 10^{6}$$

6.4.3 MACH VALUES

This paragraph has the function to demonstrate that Formula One cars are racing below the limit value of 0,3 commented in the point 4.3 to be able to simulate with incompressible airflow.

$$Ma_1 = \frac{U}{U_S} = \frac{33{,}33\,\frac{m}{s}}{343{,}15\,\frac{m}{s}} = 0{,}09713$$

$$Ma_2 = \frac{U}{U_S} = \frac{58{,}33\,\frac{m}{s}}{343{,}15\,\frac{m}{s}} = 0{,}16998$$

$$Ma_3 = \frac{U}{U_S} = \frac{95{,}72\,\frac{m}{s}}{343{,}15\,\frac{m}{s}} = 0{,}27895$$

As we can see, the numbers represent the case defined in table 7, and all the Mach numbers are below the reference number 0,3 ($Ma < 0{,}3$), which means that the air can be considered incompressible in every case [27].

6.5 SIMULATION RUN

6.5.1 INTRODUCTION TO THE SIMULATION

Once finished with the initial calculations, the next step is to start the simulation introducing the parameters checked in the pre-simulation process. Considering that SimScale works as a CFD software, the first step is to find the CAD model created and introduce it. In my case, to import the design to SimScale, I have changed the (.sldprt) file format to the (.STEP). The main reason for this change is because the files with this extension, also known as Standard for Product Data Interchange, contain 3D CAD object data saved in text format that comply with ISO 10303-21.

When the CAD model is prepared to start the study, it's important to know the basic fundamentals of the Navier-Stokes. These are crucial because they define any single-phase fluid behaviour (liquid or gas) and have the possibility of being simplified depending on the computational complexity levels wanted. Normally, these simplifications appear with the suppression of compressibility, vorticity and viscous effects from our fluid.

And then, when all the information is collected, it starts the practical analysis, which is divided into four different parts. First of all, the pre-processed creation goes ahead with the solid geometry design, where the fluid and the simulation domain selected defines the space where we are going to simulate. It's important to create the external flow volume for our front wing and to maximize the similarities with a possible wind tunnel study.

When the first part is finished, SimScale has the option of introducing the material necessary to start simulating, which is the second step. In this case, as we are working as a wind tunnel does, the fluid in contact with the car is going to be the air, with its common values commented in 6.4.2. The third step is to create the boundary conditions to assign to every geometry section some attributes to generate the most realistic simulation and the closest to reality.

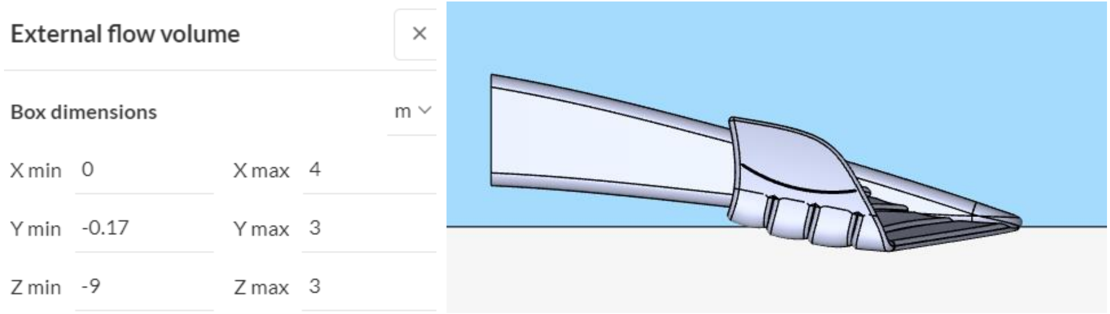
Finally, when all these steps are completed, we arrive at the most crucial point, where the mesh needs to be defined. The geometry is divided into small elements connected by nodes, and depending on how this mesh is created, the results will be more or less precise. So as is commented in the introduction, with a more precise result, the time needed to run the simulation can be too extensive, so it's important to find a balance and compare the results with different meshes. Once finished, the post-processing starts and the solutions will give us the information needed to find our conclusions.

6.5.2 GEOMETRY

The first step to define the geometry is to create a domain that is representing the wind tunnel section, and as our front wing is symmetric, the calculations will be divided in two creating a symmetry plane where the domain section is crossing the nose.

With SimScale, the steps are: creating the external flow volume and deleting the remaining bodies to not affect the simulation time and final results.

The best option to create the domain box dimensions is to use the new minimum value of distance between the ground and the edge of the front wing that is stipulated with the entrance of the new regulation, which is 100 mm [36]. Considering that this edge is 170 mm below the centre of mass from our CAD design, as is shown in figure 49, the minimum value of the domain has to be 270 mm negative to the Y axis.

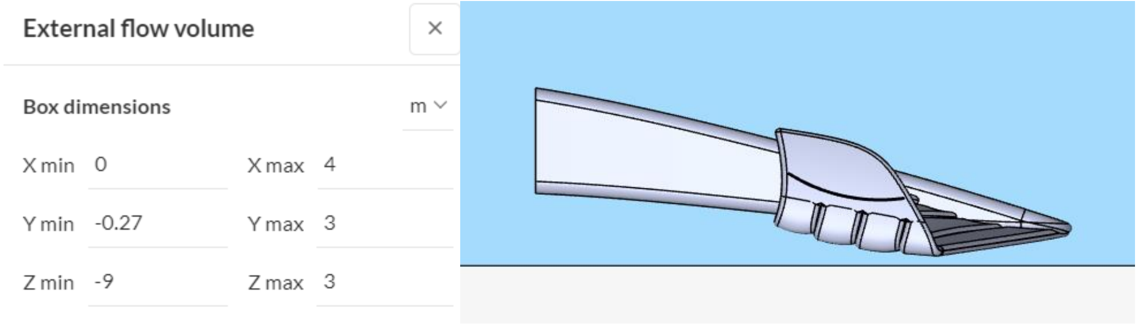

External flow volume

Box dimensions m

X min 0
X max 4
Y min -0.17
Y max 3
Z min -9
Z max 3

Figure 49. Plane in contact with the front wing edge in CAD design.

Figure 49 is just representing where the external flow volume had to be if we wanted to simulate only the nose part of the wing, but in this case, we also want to see the ground effect with the distance between the track and the front wing starting point, so the dimensions defined in figure 50 are the most representative.


External flow volume

Box dimensions m

X min 0
X max 4
Y min -0.27
Y max 3
Z min -9
Z max 3

Figure 50. Domain dimensions used in the simulation.

Also, when the domain is created, it’s important to leave a distance to the object on the opposite side of the symmetry plane and on the top because these are areas that are not affected by any restriction, and must simulate an open space, like it would be on the circuit.

Considering that the length of the CAD model is L = 1.5 m, our domain has a relation with this value (figure 51):

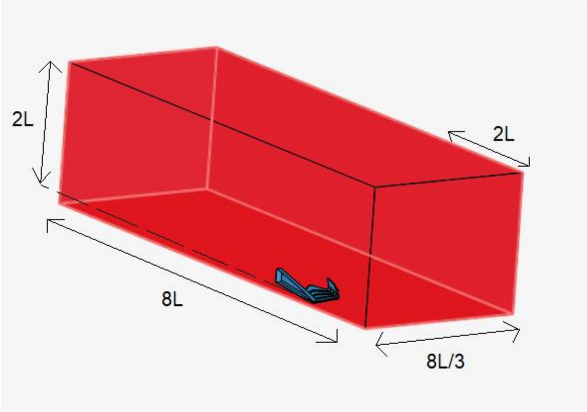
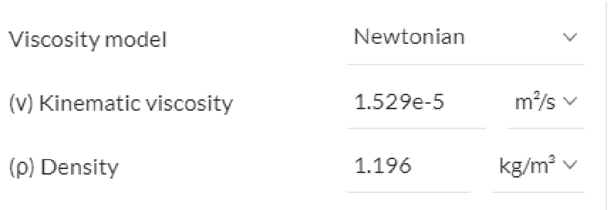


Figure 51. Box dimensions from our domain.

6.5.3 MATERIALS

A CFD is a program that comes from the fluid mechanism brand, so one of the predominant pieces of information is to find the fluid that is going to be transported inside the domain. Normally, to see the streamlines passing through the Formula One car's shape, wind tunnels add smoke to the practice day and that lets the engineers and designers make it easier to extract rewarding information. As the wind tunnel tests are very expensive and unreachable in this educational investigation, the material used in SimScale is the air, with its properties defined in figure 51, and corresponding with table 8 [35]:



Viscosity model	Newtonian	
(ν) Kinematic viscosity	1.529e-5	m^2/s
(ρ) Density	1.196	kg/m^3

Figure 52. Air properties in SimScale.

6.5.4 BOUNDARY CONDITIONS

In this part of the process, every geometry position has to be defined to allow SimScale to create a simulation with accurate details and reliable data. The reason for not having any moving body inside our virtual wind tunnel, makes it easier to define the variables and recreate the front wing analysis situation. Normally, when the teams have to work with the aerodynamics, they also need to include an angular speed created from the wheels, which have a drastic influence on the final performance. To define our boundary conditions, SimScale gives different types of different analysis depending on the project we are working on, and in this case, they are:

6.5.4.1 INLET

First condition that creates an airflow entrance on the front wall with a stipulated velocity (velocity inlet). To create different scenarios, there are going to be contrasting speed values in the negative Z axis direction, according to the coordinate system generated.

6.5.4.2 OUTLET

When the entrance conditions are defined, the outlet can help us to see how the flow is leaving, considering a pressure outlet with a fixed value of 0 Pa. This contemplates a relative pressure of zero, and simulates the real atmospheric pressure at the back.

6.5.4.3 SYMMETRY

This tool can be used in several ways, but in this case, we use this condition as a symmetry plane to create an imaginary infinite object in the eyes of the program in the X direction to avoid affecting in the final analysis with SimScale.

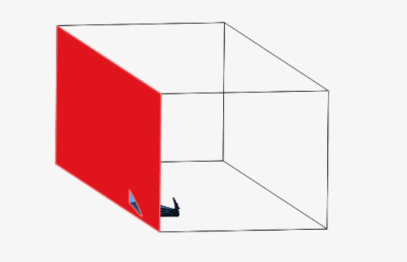


Figure 53. Symmetry wall condition.

6.5.4.4 FRONT WING AND BOTTOM WALLS

Wall condition to the entire front wing surface without velocity, and applying a no-slip setting to avoid any effect on the final results. Then, the bottom plane also has a wall condition, but in this case as a moving wall, with the same velocity value as the inlet velocity to have the same environment of the wind tunnel.

6.5.4.5 LATERAL WALLS

Finally, the lateral walls are the same as an open area where the airflow can move without being affected by any part of the solid.

In this case, the flow can go in and out and to create this situation in SimScale the consideration applied is a wall boundary condition with slip velocity. These lateral walls are located on the top and opposite side of the symmetry wall.

6.5.5 MESH CREATION

The last step, before checking the results, is to create the mesh and make a discretization in the domain defined in 6.5.2. In order to compare different results and see how the mesh influences on the final solution, there are three different simulations with a changing fineness, starting from a coarse mesh, then a moderate mesh, and finally a fine mesh. These values affect the number of cells and will have an impact on the results accuracy [43], but also have been changed from the initial study because the results were confusing. After a first try with lower cell values, table 9 describes the final mesh characteristics, which were finer than the starting ones, but also with a three differentiation:

Table 9. Meshing differences in the 2022 prototype.

MESH	***CELLS***	***NODES***
Coarse	3,9M	1,9M
Moderate	4,2M	2,0M
Fine	4,7M	2,1M

There can be more specific studies with extra advanced meshes, but the lack of information and time makes it impossible, nonetheless, are enough to see the performance of the 2022 front wing.

6.5.6 RESULTS VALIDATION

6.5.6.1 GRID CONVERGENCE ANALYSIS 2022 PROTOTYPE

There are different ways to see if the results are correct, or if there is a dependence on the size and geometry of the mesh. In this case, one of the most important is the grid convergence analysis [44], where we find if the aerodynamic coefficients are dependent or independent from the disparate mesh studies.

Table 10. Lift and drag coefficient values at 58 m/s for the 2022 prototype.

Type of grid	***Cd***	***Cl***
Extra fine (8)	-0,16787	-0,62632
Extra moderate (7,5)	-0,16945	-0,60699
Extra coarse (7)	-0,16831	-0,60535

Calculations are made in the Annex IV document, and the results will be resumed in the following table with the consideration of a non-dependent mesh values, where there exists a convergence.

Table 11. Grid converge results for the 2022 prototype.

Coefficient	***p***	$f_{h=0}$	$GCI_{2,3}$	$GCI_{1,2}$	**Range**
C_d	0,825	-0,16377	2,18%	3,05%	1,0096
C_l	6,240	-0,62811	0,031%	0,357%	0,9766

The same results have been made for the old front wing design to compare the values and find some conclusions. In this case, the results are also explained in the Annex IV and we can consider a grid convergence and discard a dependence on the mesh.

6.5.6.2 RESIDUALS

Once the first validation is clear, there is another way to see if the results are representative or not, and that's using the residual values coming from the simulation. That means checking how the results perform with the number of iterations and if there exists a considerable change during the simulation. Not only the residuals, also the convergence plots for the boundary conditions will give us information at the end of the iterations (Annex V), which in this case are a maximum of 1000 repetitions to smooth the number of simulations created.

$$N^{\underline{o}}iterations = \frac{End\ time[s]}{Delta\ t\ [s]} = \frac{1000[s]}{1[s]} = 1000$$

All the residual graphs give us information, but in this case, figure 54 and figure 55 represent the results of a 58 m/s car speed (considered the mean value) simulation for the new 2022 prototype and the 2018 front wing, respectively. Both of them, with the finest mesh established, which give us the most accurate result.

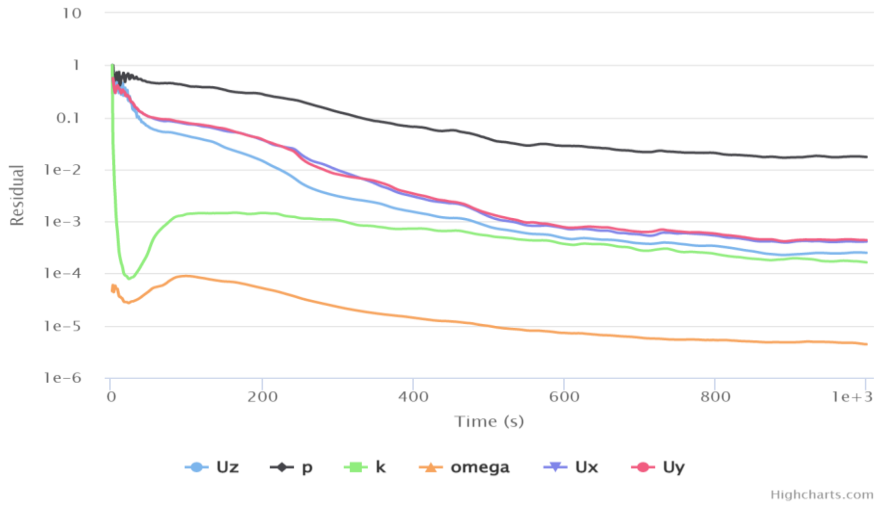


Figure 54. 2022 prototype residuals.

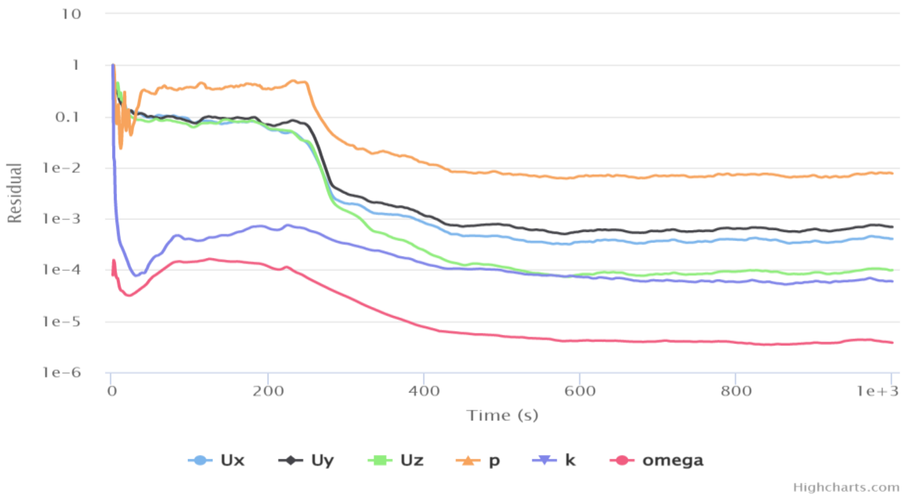


Figure 55. Old front wing residuals.

When the values are passing the iteration number 400, the residuals, which are the solution imbalances, start to have a constant value smaller than 10^{-2}, and for the velocity and turbulence parameters less than 10^{-3}. Normally, it's considered a good starting point with similar values, so we can accept our simulation results, because as smaller the residuals are, more realistic the solution can be contemplated.

7 RESULTS SUMMARY

Arriving at this point, and with all the information already schematised, it’s important to see how the newly created design works on a SimScale simulated track, and if the regulation changes have affected or not to the front wing. And as commented before, this comparison will be between our 2022 prototype created in CAD and a 2018 front wing from a web page in the following references [45].

7.1 PRESSURE DISTRIBUTION

First important result is to see the pressure values that are affecting the front wing surface, because the main objective is to create downforce with a higher pressure on the top of the body and lower on the bottom to counter the lift force explained in point 4. To check how this performs, figure 56 provides the difference between the two faces with a 58 m/s speed:

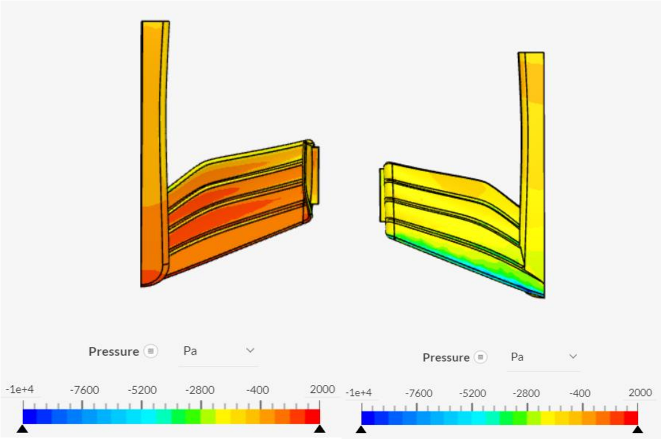


Figure 56. Pressure distribution of the 2022 front wing.

With this new spoiler system, we arrive at a higher-pressure value on the top of the nose that helps increase the stability and the downforce of the car. The values are estimated from -10.000 Pa to 2.000 Pa, and the location that has a lower pressure is the front side of the first flap, with a blue and green colour combination. The main reason is because the distance is lower to the track, and the ground effect is enhanced creating a lower pressure on the bottom because the speed is higher.

Compared with figure 57, where it shows the old design with extra spoilers and small wings, the pressure values defined with the colours can have a close first impression, but there are important differences to comment on.

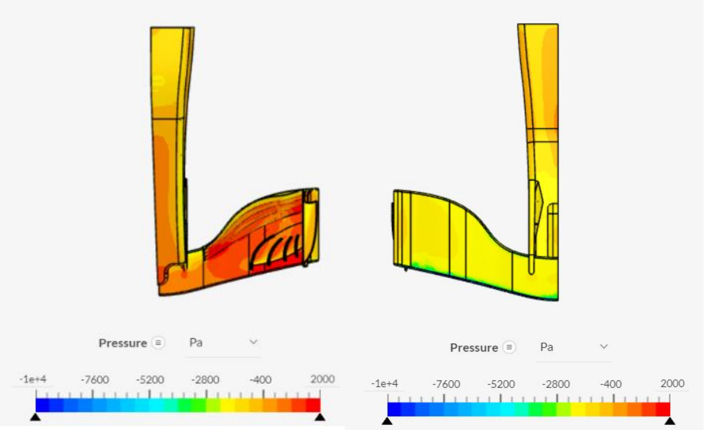


Figure 57. Pressure distribution of the 2018 front wing.

First of all, talking about the new design, the simulation shows a higher surface affected for the low-pressure distribution, which will help the ground effect and start a downforce generation. Also, the main characteristic of the new regulation is to increase the simplicity of the model, and with this new elementary CAD it provides a much more homogeneous pressure distribution, where there are no excessive differentials of pressure. In the old structure, as is shown in the picture 57, there are different strategic points where the value is higher, and that can help the apparition of vortex and affect to car racing.

To study the wake effect and see the airflow characteristics, it's necessary to create a situation with the completed structure of the racing car, including tires and body to simulate the real effect on the drag forces. Such s complex study needs extra time and deeper information, but our simulations can give us some crucial information to compare the performance between the two regulations.

7.2 VORTEX CREATION

Before the new regulation changes, laborious and exigent front wing designs created vortices with shapes that as it passed through the air, it caused a differential of pressure making a circular path of the airflow. When the laminar flow hits the front wing, it changes to a more chaotic and disorderly movement that negatively affects to a car standing behind, and the most drastic example is the Y-250 vortex. As commented before, this effect appears in the four flaps union above the main plane (figure 6), which is characterized as the old design. Figure 58 introduces how the particle traces create a circular movement when it has been in contact with the strategic location at 250 mm from the symmetry axis.

Figure 58. Particle traces creating vortices.

In comparison to this case, the idea that the FIA wants to introduce is to expel the air to a higher level and remove it from the way and wake of the cars. A simple design is the main reason, because all the small edges and winglets will be eliminated and the car will redirect the airflow to the top of the car reducing the amount of turbulent air affecting the back. This 2022 prototype is just a testing model and there will be a lot of differences with the final result, but simulations indicate in figure 59 that the vortex creation is considerably narrow.

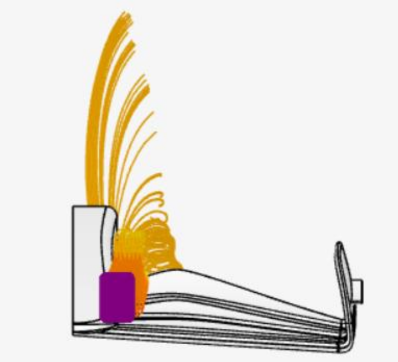


Figure 59. Redirected airflow with new design improvements.

7.3 LIFT AND DRAG COEFFICIENT VALUES

The set-up is also crucial in terms of improving the performance of the car, and most of the time it's the track that imposes the final configuration. Formula One has the resources and the possibility to adapt every vehicle part to the most efficient conditions to reduce the gap with competitors, and one of these possibilities is to change the flaps inclination to change the lift and drag coefficients. Normally, these are simulations that teams have really easy access to, working with different profiles like NACA or NLF combining the angle inclination and changing airflow effect. [46]

In this case we are not studying the different profile's lift and drag, just the influence of the complete front wing, but figure 60 can show how these coefficients can oscillate according to the angle that the incident wind makes with respect chord of the profile, and when both values are defined, aerodynamic efficiency can be calculated.

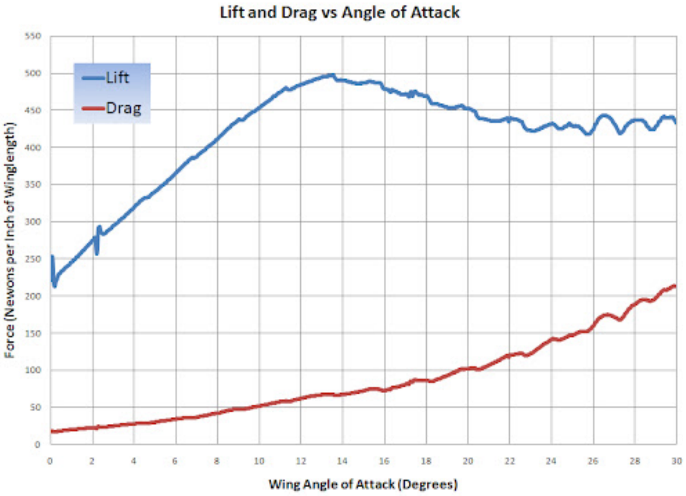


Figure 60. Lift and drag force values changing the angle of attack. (Source: [46])

7.3.1 VELOCITY INFLUENCE FOR THE 2022 PROTOTYPE

For the 2022 CAD model created, the angle of attack has a constant value, because are finished designs, but it's also important to see if the velocity changes the lift and drag repercussions. To sort it out, the three speed cases will be simulated in SimScale.

Table 12. Lift and drag coefficients at different speeds 2022 front wing.

Case	***Type***	***Track speed***	**C_L**	**C_D**	**C_L / C_D**
1	Low corners	33 m/s	-0,6185	-0,1689	3,6619
2	Average speed	58 m/s	-0,6263	-0,1679	3,7302
3	Straight	96 m/s	-0,6140	-0,1659	3,7010
Difference between case 1 and 3 (%)			0,73%	1,81%	1,06%

Considering the same mesh, these three values are maintained relatively constant, and show us that the vehicle's car doesn't affect to the coefficient. The error between the values is too small to consider that affects to the performance, and now, these values will be compared with the 2018, which also have no significant difference with speed changes.

7.3.2 VELOCITY INFLUENCE FOR THE 2018 PROTOTYPE

Once found a 2018 front wing model that has a convergence behaviour, the simulations will have the same purpose, consider different values to see if the coefficients change, and then compare the new with the old design. That will help us to understand how the new regulation is going to change the way how the car progress on track.

Table 13. Lift and drag coefficients at different speeds 2018 front wing.

Case	**Type**	**Track speed**	C_L	C_D	C_L/ C_D
1	Low corners	33 m/s	-0,3940	-0,1763	2,2348
2	Average speed	58 m/s	-0,4007	-0,1782	2,2490
3	Straight	96 m/s	-0,4085	-0,1792	2,2798
Difference between case 1 and 3 (%)			3,55%	1,62%	1,97%

As in table 14, values are negative because the strength works in the opposite coordinates' axes direction of the movement, that means the lift force works in the negative Y direction and the drag in the negative Z direction, so the values can be considered as absolute value. For these two models created, we can see that the new 2022 design has a higher lift coefficient and a lower drag effect, but the lift coefficient can change considerably depending on the wind angle attack, so the objective of the teams is to find the best angle that fits the race conditions. In terms of the drag coefficient, we can see that there exists a fairly large difference that doesn't create a big impact on the efficiency value.

7.3.3 COMPARATIVE DIAGRAMS

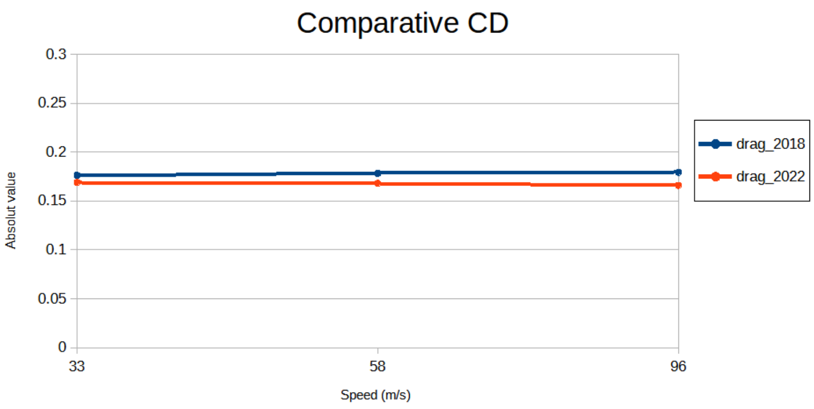


Figure 61. Diagram with drag coefficients against the speed.

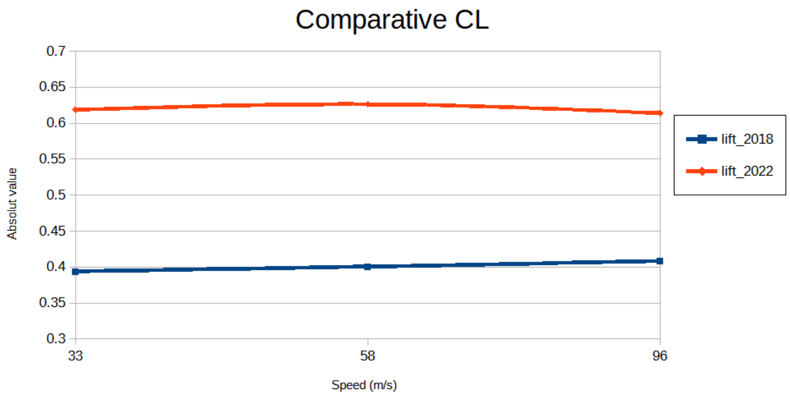


Figure 62. Diagram with lift coefficients against the speed.

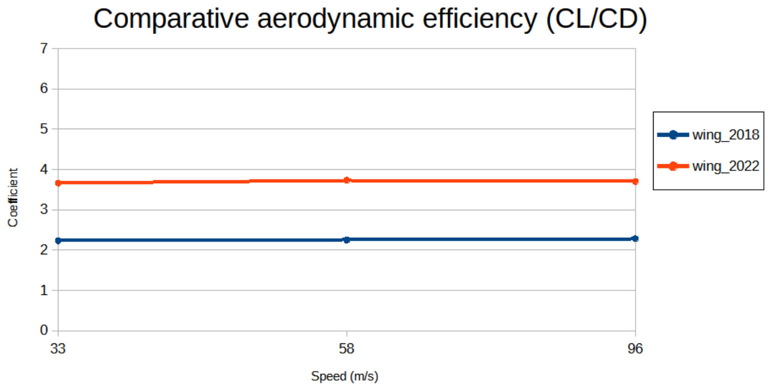


Figure 63. Diagram with aerodynamic efficiency against the speed.

Diagrams represent how the coefficients are changing depending on the speed, which as we can see is negligible. Lift and drag coefficients only depend on the geometry and the attack angle, so for different speeds the value maintains constant. In this case they are not exactly the same but the error is so small that can be assumed exact values. Also, the diagrams are created to compare both models, with an orange line to refer to the new front wing created for 2022, and a blue line for the old design. As we have commented, the drag coefficient maintains relatively stable during both simulations, but is the lift coefficient who receives a supplementary conversion increasing two tenths. That generates a differentiation on the aerodynamic efficiency, which is higher for the new regulation, but can change with the possibility of adjusting the wing's angle of attack.

7.4 GROUND EFFECT IN 2022 PROTOTYPE

Considered one of the most important aerodynamic consequences, we need to guarantee that the new design generates the correct velocity and pressure distribution to start the ground effect that is maximized with other body components.

As is the first part in contact with the exterior when the movement has started, the theoretical and ideal situation is to have a higher velocity value underneath, with a lower pressure that creates a pressure differential above and increases the downforce.

Figure 64 demonstrates that also the new design is capable of generating the effect necessary to start the most effective distribution, which is the ground effect. The simulation has been generated with the finest mesh created and with a velocity inlet of 58 m/s, and the speed range goes from 0 to 100 m/s.

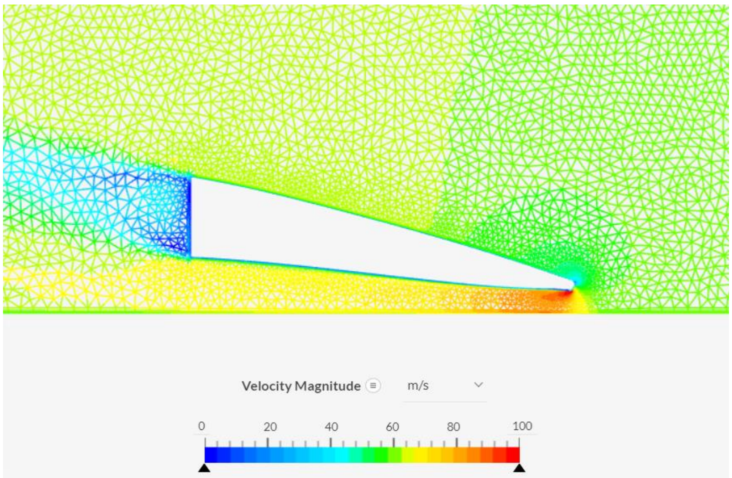


Figure 64. Symmetry velocity vision from ground effect generation.

In the picture we can also find the boundary layers generated in the surface and how the intense colours like red and orange are located under the front wing. That creates a lower pressure area (figure 65) where the car will start to gain a suction strength that will be increased along the body shape.

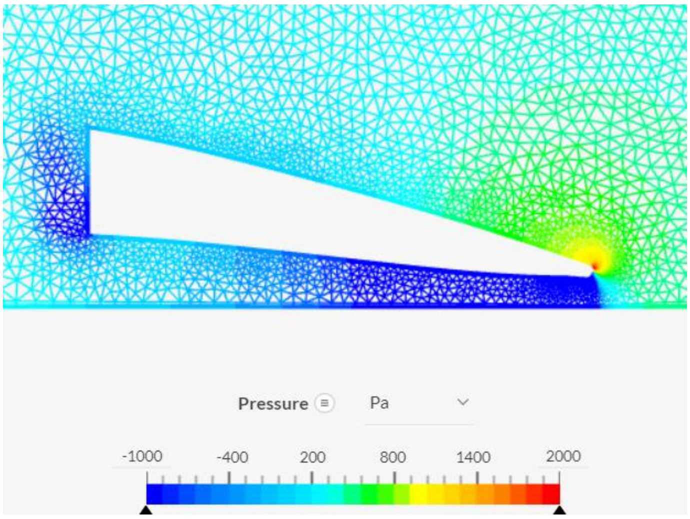


Figure 65. Symmetry pressure vision from ground effect generation.

At this point, a factor that has to be taken into consideration is the distance that exists between the car and the asphalt, which is also a value that can be modified depending on track factors, the vehicle's benefits and assembly components. With the new regulation, this distance from the Z_0 has changed, and now the cars can't be so close to the ground, with a minimum value of 100 mm, which is 25 mm higher than the older regulation [36]. This factor evidences that the suction generated will be lower in 2022, but also the influence of the wake from other cars on track, which was one of the main objectives to accomplish to increase the racing spectacle.

Finally, before starting to compare how the coefficients change with the distance with Z_0, we don't have to neglect from figures 64 and 65 that there is a very important factor implicit in all the simulations that is wrong. By simulating a separated aileron and not the whole vehicle, lift and drag are disturbed by the rear from the nose. The turbulence indicated in figure 66 is caused by the rough geometry of the nose, that in reality would not exist. For this reason, the support data and above all resistance are not exact. However, they do help us to compare since the effect affects both ailerons.

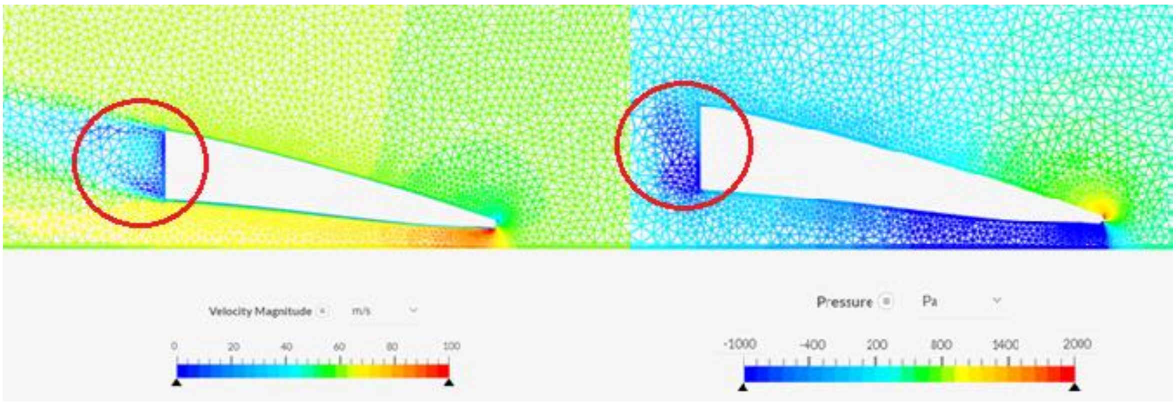


Figure 66. Turbulence caused due separated front wing simulation.

7.5 COEFFICIENT EVOLUTION ACCORDING TO GROUND DISTANCE

Not always the best option is to have it positioned at the lowest distance allowed from the regulations, because some circuits have aggressive curves, pianos or bananas front wings can hit and crash these components and turn on into cracked sections. A broken front wing can generate high turbulence and vibrations that reduce considerably the efficiency, that's why in some cases the height from the horizontal line is better to have it higher than 100 mm, even if it reduces the ground effect generation [36].

Now, we are going to simulate different scenarios where the domain has been modified and see how this ground effect mitigates with the distance between the track and the car. If they reproduce the theoretical situation, both coefficients should be bigger in the nearest part to the asphalt, and reduce its absolute value with the increment of height. To simulate this effect, it's important to create domains at different Y values, starting from the minimum legal distance (100 mm) and adding distance progressively.

Table 14. Lift and drag coefficients at different ground distance.

Case	***Distance (mm)***	***Y (m)***	***C_L***	***C_D***	***C_L/ C_D***
1	100	-0,270	-0,6185	-0,1689	3,6619
2	150	-0,320	-0,6157	-0,1686	3,6518
3	200	-0,370	-0,6009	-0,1652	3,6374
4	500	-0,670	-0,5446	-0,1613	3,3763
5	2.830	-3,000	-0,4706	-0,1542	3,0512
Difference between case 1 and 5 (%)			23,91 %	8,70%	16,68%

Simulated conditions are the same in every case, with a speed constant of 58 m/s, but in the case 5 from table 14, it has used the finest mesh possible without exceeding its maximum time of simulation. Also, in all the cases, the residuals have been examined to see if there was a convergence conduct. The conclusions we can extract from table 14 is that the distance can affect the performance, but with this model created as a prototype for 2022, significant variations appear when the distance exceeds the 200 mm.

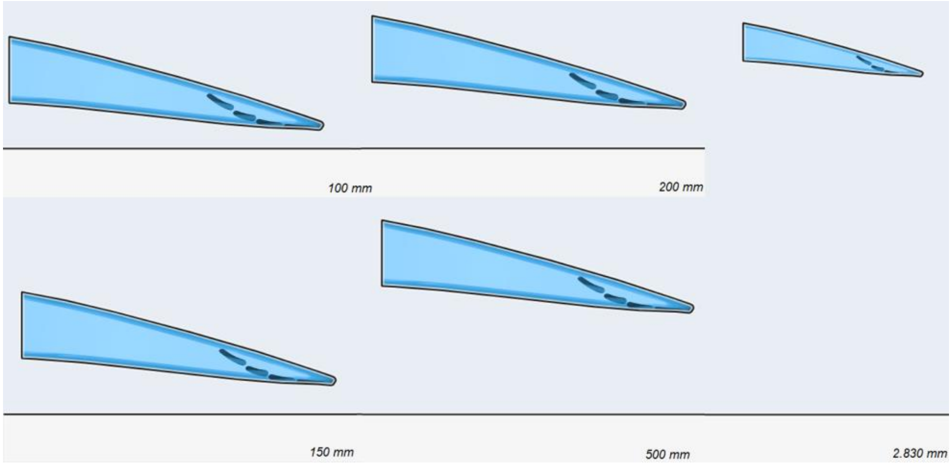


Figure 67. Distance from the ground differentiation.

Higher values than 200 mm are not realistic because they require a car extremely elevated (figure 67), but are used to compare and create the following diagrams.

7.5.1 HOW C_L AND C_D CHANGE

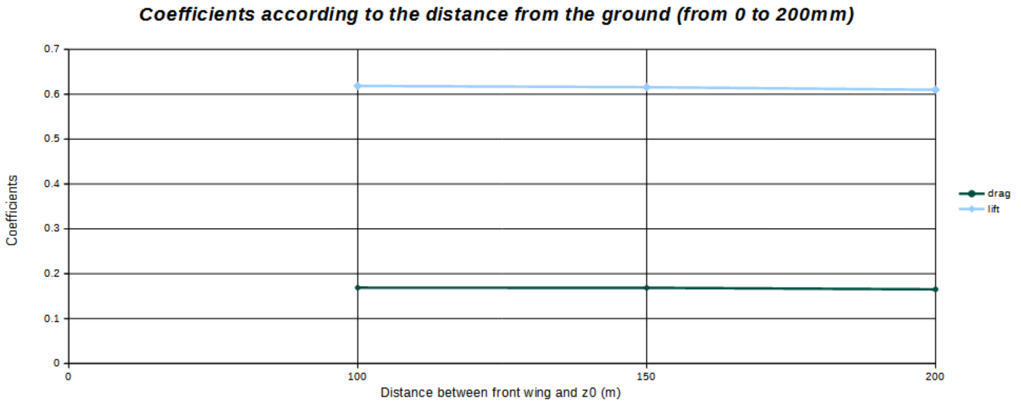


Figure 68a. Coefficients according to the distance from the ground (from 0 to 200 mm).

A representation of table 14 values demonstrates that lift and drag changes if we increase the distance from the horizontal plane of the ground to the front wing's edge. This difference grows working with long distances, and at 2.830 mm, where we are talking about fictitious numbers because can't be applied to the Formula One vehicle, the slope gets an extensive inclination. To represent it, there is a resumed diagram between 0 and 200 mm that represents possible values, and a larger diagram to show how this dependence changes increasing the distance. Figure 68a reveals that lift has a higher dependence, but in both cases, the variation is not as high compared to bigger distances. From 100 mm to 200 mm, lift reduces only a 2,85% the value, and drag a 2,19%.

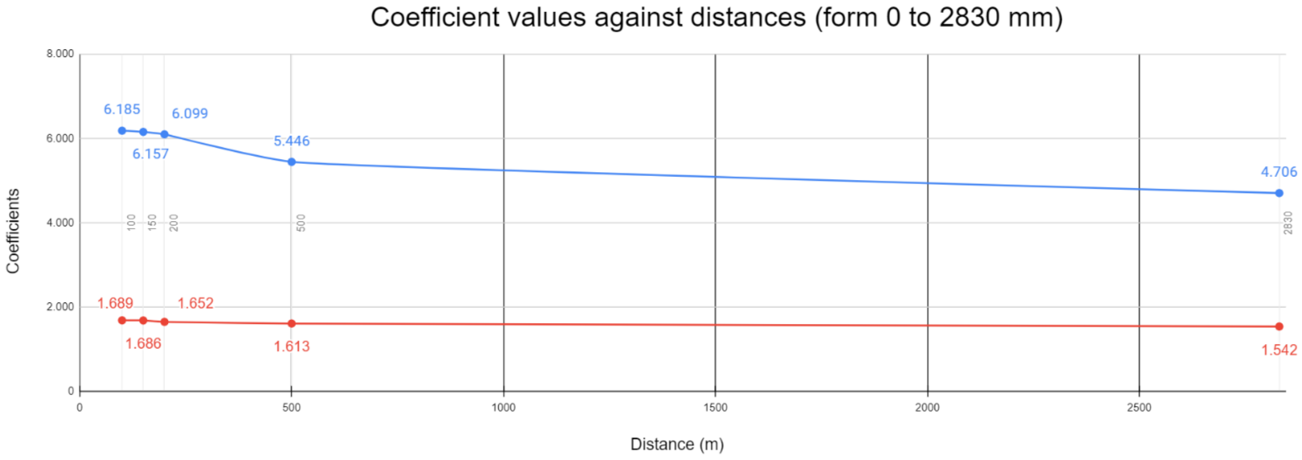


Figure 68b. Coefficients according to the distance from the ground.

Finally, with figure 68b, the results commented before are demonstrated, and as higher is the distance, lower are the aerodynamic coefficients. Now, the procedure will be done for aerodynamic efficiency quotients, but obviously behave equal to the graphics indicated. (Blue line: C_L) (Red line: C_D)

7.5.2 C_L/C_D EVOLUTION

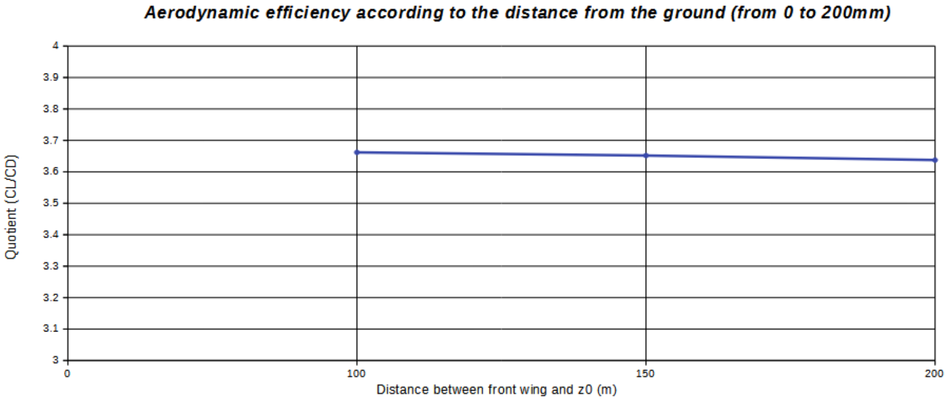


Figure 69a. Aerodynamic efficiency according to the distance from the ground (from 0 to 200 mm).

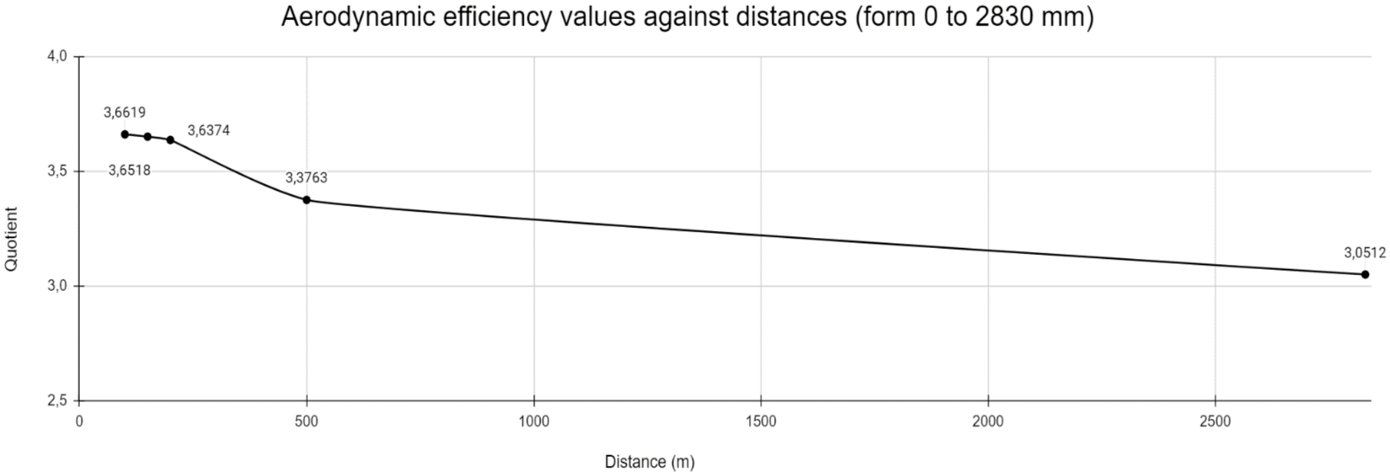


Figure 69b. Aerodynamic efficiency according to the distance from the ground.

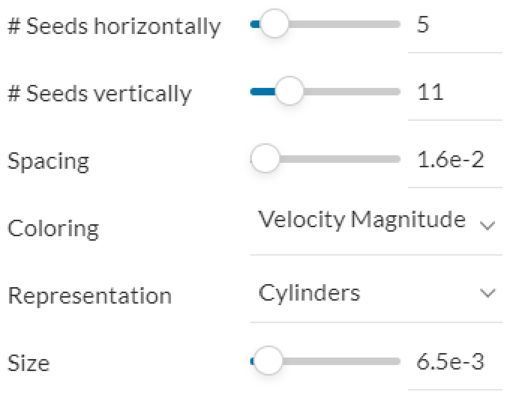
Arriving to the investigation’s end, they have shown us that the speed is not a differentiating factor that can change aerodynamic effects, and the drag and lift coefficients generated with the same car geometry or set-up will be the same for low corners, average speed, and long straights, because the error is less than a 5%.

Similarly, this point 7.5 compares the effects with different domains to simulate the possibility of adjusting the height from the ground, and as the results from SimScale demonstrate, it's a factor that has to be taken into consideration.

Figure 69a and 69b indicates that aerodynamic efficiency also depends on this factor, having a same conduct as lift and drag coefficient because it's the quotient. In this case, from the lowest position possible (case 1) to the most extreme simulation (case 5), showed in figure 69b, there is a difference of 16,68%. Everything evidences that the distance has to be studied for every independent circuit, to find a balance between necessary ground effect generation for lift and drag effects.

7.6 NEW ENDPLATES CONFIGURATION

Solution Fields is a forceful branch that can be used in SimScale to extract crucial information to compare different models, and as it has been done in the vortex creation, the particle trace illustrates airflow's shape and direction. From this point, the idea is to originate an oriented airflow coming from the velocity inlet face pointing directly to the endplate and footplate and see how this new design performs. Comment that the particle trace conditions are the same for both CAD designs to make the simulation more realistic and powerful.



# Seeds horizontally	5
# Seeds vertically	11
Spacing	1.6e-2
Coloring	Velocity Magnitude
Representation	Cylinders
Size	6.5e-3

Figure 70. Particle trace conditions.

Small-scale size and particle spacing is used (figure 70) to guarantee that our fluid only changes the direction and is being influenced by the endplate and footplate.

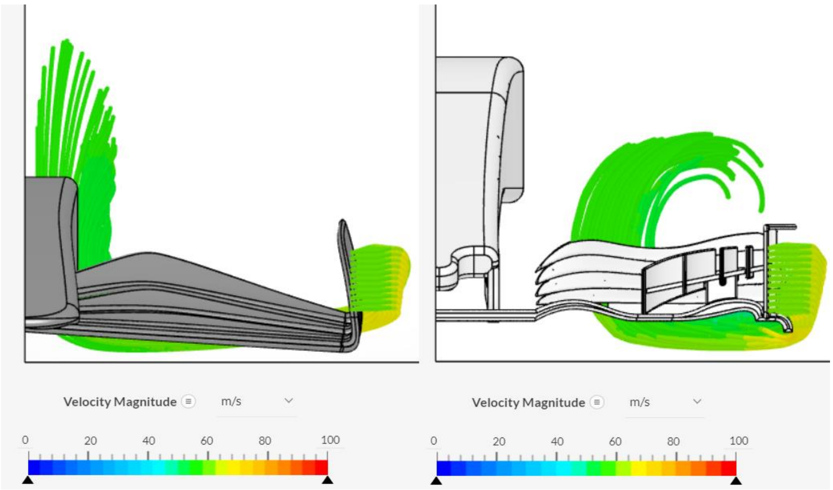


Figure 71. Front view from the comparing endplates effect.

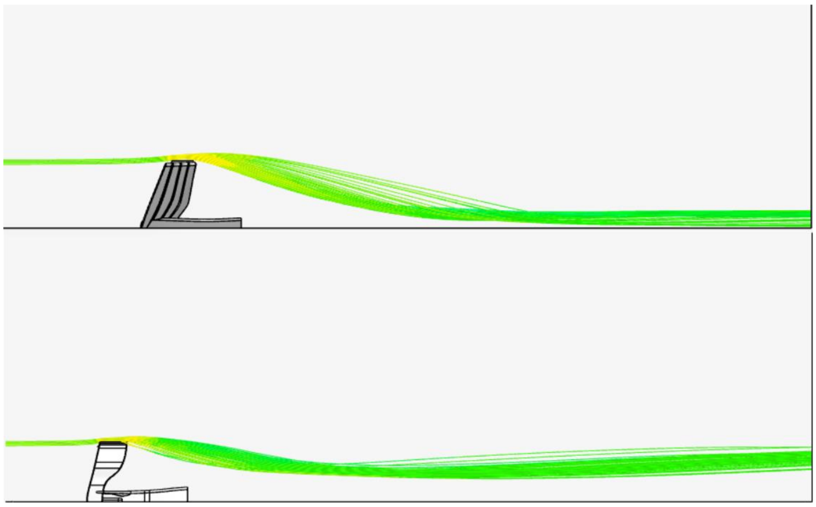


Figure 72. Top view from the comparing endplates effect.

Before the introduction of endplates, the front wing created vortices by the fluid's contact on its surface because of a pressure differential between upper and lower position, and with the creation of these vortices, the performance worsens with a negative result in the downforce generation and an increasing induced drag. Then, after the introduction of this aerodynamic element and contrasting regulations, arrives a new changing methodology. With the short information there exists, we can find a clear differentiation in the images.

In the analysis, figure 71 gives us a configuration and figure reference, and figure 72 shows the airflow direction influenced by the front wing. Starting in order, we can compare the wake from both models and see that 2018's aileron generates a higher chaotic and circular airflow than 2022. The new regulation tries to avoid the creation of disordered influence to the car behind, and that's started from the front wing.

Curved and with a simple configuration is the reason why this wake is not so negative and, with other aerodynamic elements located along the vehicle, the efficiency will be improved in order to reduce the dirty air.

Finally, airflow's direction is also very significant, meanwhile the new created nose conducts the air to the rear toward the symmetry axis, 2018's changes minimally the path of motion without arriving at the domain's symmetry wall. This can be a problem because moving it's wakes directly to the front wheels can encourage an increase in dirty air affecting cars following (figure 73).

Figure 73. Dirty air affecting a 2022 prototype. (Source: [36])

8 CONCLUSIONS

Going back to the project's beginning, the major objective was trying to explain and analyse the performance of the new front wing model that arrives with the new FIA regulation. The idea was to create a CAD prototype with the brief and summarized information available due the team's confidential information, and try to compare, with CFD simulations, how this new mock-up behaved in real race track conditions. At the end, this purpose has been completed with a broken-down explanation of all the necessary steps to generate the simulations.

From here, and putting all the factors and studies together, there are different conclusions obtained from this research. First of all, Formula One is a high-tech and engineering sport where a lot of resources need to be available to find an efficient solution that leads you to success. There are a lot of ideas and factors to consider during a designing process, that's why it can be considered very important to work as a team.

Secondly, talking about Formula One and after studying its performance, front wings are one of the most important aerodynamic elements from a racing car because they generate the necessary downforce and distribute the airflow through the rest of the body, so a correct and efficient modulation can be crucial for a team. Then, analysing our results can help us to understand that there's a deal of influences that can affect the final drag and lift generation, and contribute directly to the car performance. Even arrived to the interpretation that the speed doesn't affect to the coefficients (C_D and C_l), others, like the attacking angle of a moving flap, dimensions (longer nose), geometry (simplified respect other regulations) and distance from the ground, make the correct set-up a combination of infinite factors.

Talking about the coefficients, and once seen which changing effects affect the performance, we have also found numerical solutions interpreted. First of all, a changing velocity, from 33 m/s to 96 m/s (approx. 225 km/h) doesn't affect the lift and drag formation, because the analysis shows that a 65,6% modification of the car's speed generates a 0,73% maximum lift error and 1,81% maximum drag error. In contradiction, an increase of height of only 100 mm between the nose and ground, can affect losing more than 5%, and in extreme cases, a difference of approximately 3 meters, can reduce up to 15% the aerodynamic efficiency in 2022's new model.

Moreover, CFD simulations have been a great discovery working with aerodynamics in Formula One because there exists the possibility of simulating and comparing various aerodynamic designs, in a relatively short period of time, without spending resources and in a much efficient way. Nowadays, this methodology can be considered indispensable in the car design, to start later with the wind tunnel testing that will give the teams a reliable analysis solution.

In addition, even a time dedicated to check if simulations converged to find if they were secure, there is information missing from theoretical values to be compared with the ones obtained from simulations. Normally, when a value is obtained after an individual calculation, it's important to find this equivalence that gives you information if the results can be accepted or not. There are many ways to create and design a front wing, but all the F1 teams avoid showing their results to the users and specially to an opposite Scuderia, so that's why it's arduous to find abstract solutions.

Comparing the two models, a difference of approximately -6% in the drag and +36% of the lift, creates an approximate +40% of increased value for 2022's aerodynamic efficiency against the 2018 design. Our simulation also gives us some ideas about how the wake will behave in new cars, where the dirty air is one of the objectives to reduce significantly.

Finally, it's important to consider that this project could be extended because there are different points that are commented on but could be validated and increased with extra information and resources. These points include a study and simulation of different aerodynamic profiles used in this area like NACA airfoils and adapt them to the prototype, compare with older front wing designs, or find moving wing's influence according to the angle of attack. Also, it could be interesting to see the effect of the velocity inlet not just coming from a straight line, i.e. include an incidence angle between fluid and car, or work also with possible body's vibrations

9 BUDGET SUMMARY

In order to finance a project, there are a good deal of aspects that must be taken into consideration to finally include, in the total amount of price, every individual peculiarity where it's important to introduce economical resources. Considering this project, which is a study combination between information research and designing processes, the best way to create a budget summary is starting with the professional fees and start annexing all the extra costs that can appear during the development.

All this information is collected in the budget deliverable, where each of the costs is broken down with its corresponding justification and demonstration and finishes with table 15, where each amount is collected with the corresponding taxes and amortizations. The final cost estimated is 15.000€, but these could change due to numerous factors.

Table 15. Final project costs with taxes and amortizations.

<table>
<tr><th colspan="6"><i>BUDGET SUMMARY</i></th></tr>
<tr><td><i>Type</i></td><td colspan="2"><i>Time</i></td><td colspan="2"><i>Price per hour</i></td><td><i>Total price</i></td></tr>
<tr><td>Research</td><td colspan="2">70 h</td><td colspan="2">25 €/h</td><td>1.750 €</td></tr>
<tr><td>CAD work</td><td colspan="2">85 h</td><td colspan="2">40 €/h</td><td>3.400 €</td></tr>
<tr><td>CFD simulations</td><td colspan="2">90 h</td><td colspan="2">50 €/h</td><td>4.500 €</td></tr>
<tr><td>Development</td><td colspan="2">105 h</td><td colspan="2">25 €/h</td><td>2.625 €</td></tr>
<tr><td>TOTAL</td><td colspan="2">350 h</td><td colspan="2"></td><td>12.275 €</td></tr>
<tr><td><i>Tariff</i></td><td colspan="2"><i>Price</i></td><td><i>Possible use</i></td><td><i>Project use</i></td><td><i>Total price</i></td></tr>
<tr><td>Standard</td><td colspan="2">6.600 €/year</td><td>8.760 h</td><td>85 h</td><td>64,04 €</td></tr>
<tr><td><i>Power</i></td><td><i>Time used</i></td><td><i>Energy losses</i></td><td><i>Consump-tion</i></td><td><i>Price</i></td><td><i>Total price</i></td></tr>
<tr><td>120 W</td><td>350 h</td><td>3,54%</td><td>43,5 kWh</td><td>0,29452 €/kWh</td><td>12,81 €</td></tr>
<tr><td colspan="3">PRICE WITHOUT TAXES</td><td colspan="3">12.351,85 €</td></tr>
<tr><td colspan="3">21% I.V.A</td><td colspan="3">2.593,89 €</td></tr>
<tr><td colspan="3">PRICE WITH TAXES</td><td colspan="3">14.945,74 €</td></tr>
<tr><td><i>Amortization product</i></td><td><i>Purchase price</i></td><td><i>Project use</i></td><td><i>Useful life</i></td><td><i>%</i></td><td><i>Total price</i></td></tr>
<tr><td>HP Pavilion 15-bc400ns</td><td>800 €</td><td>350 h</td><td>13.000 h</td><td>2,69 %</td><td>21,54 €</td></tr>
<tr><td colspan="3"><i>FINAL PROJECT PRICE</i></td><td colspan="3"><i>14.967,28 €</i></td></tr>
</table>

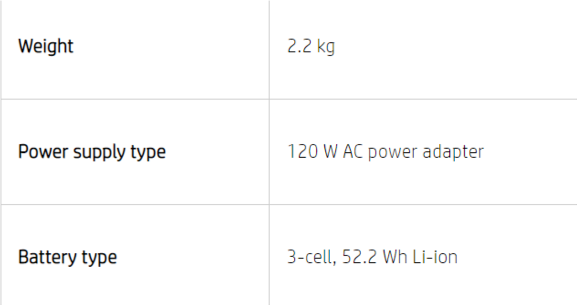
10 ANALYSIS AND ASSESSMENT OF ENVIRONMENTAL IMPACT

10.1 ELECTRICITY CONSUPTION

Formula One is considered one of the most polluting sports in the world, and a study made in 2019, considered that approximately 256.000 tons of CO_2 were emitted during the season [37]. Despite this information, teams, engineers and the FIA are working every year to change the situation and convert the motorsport to a more sustainable spectacle.

In this case, a project created for the design and computational simulation of a racing car piece can be considered as a low environmental impact work category, because all the processes are generated from the investigation and development with computers. Actually, one of the purposes of these previous investigations in CFD is to avoid spending resources that can make the teams lose money and create an impact on the environment.

Even with the high impact reduction, avoiding cycles and simulations in a real wind tunnel, it's important to create a brief calculation with the consumption of the computer and its influence on the environment. This study has been made with the computer HP Pavilion 15-bc400ns, that has a power adapter of 120W [38], so we can estimate the consumption and find the impact of the estimation.



Weight	2.2 kg
Power supply type	120 W AC power adapter
Battery type	3-cell, 52.2 Wh Li-ion

Figure 74. HP computer battery specifications. (Source: [38])

This final project is worth 12 ECTS, which is considered an approximate value of 300 hours of individual work. To make a more realistic calculation, I'm going to consider a 350 hours' time period because the time used with the simulations and with the CAD design were longer than I expected.

$$Consumption = 120\,W \cdot 350\,h = 42.000Wh = 42\,kWh$$

At this point, it will be interesting to find the electricity emission factor in Spain, where this project has been created. Normally, this value changes, so it's important to find the most updated. Information extracted from the institutional webpage of Cataluña [39] show that the actual emission factor is:

Table 16. Electricity emission factor 2021.

Emission factor (EF)	***Units***	***Updated date***
0,25	kg CO_2/kWh	16/04/21

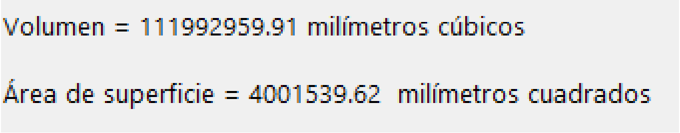
To estimate the energy dropped, in the official Spanish power grid web page [40], there are different excel files with the energy losses from every day at different hours. Making the mean value of one December symbolic day, the coefficient obtained is -0,0354, which is a 3,54% of the total.

$$Carbon\,footprint \;=\; (Cons \cdot EF) \cdot (1 + \;|losses|) \;=\; (42\,kWh \cdot 0,25 \frac{kgCO_2}{kWh}) \cdot (1 \;+\; 0,0354) \;=\; 10,87\;kgCO_2$$

The total amount of CO_2 emitted to the environment is 10,87 kg which is a small-scale value compared to the possible wind tunnel simulation and the prototype construction. The fact that the CAD model is created with SolidWorks lets us find the total amount of material used, and estimate the real impact of the manufacturing and designing of the carbon fiber nose.

10.2 CAD CREATION IMPACT

Directly, the online creation of a CAD model doesn't have any impact apart from the already calculated before, but the estimation of the real impact from the impression of the front wing can affect the environment considerably. In this case, the front wing has been designed with the final dimensions, but to simulate it in the wind tunnel it can also be a prototype.



Volumen = 111992959.91 milímetros cúbicos

Área de superficie = 4001539.62 milímetros cuadrados

Figure 75. Properties from CAD's 2022 front wing.

Figure 75 represents the volume and area of the design, and the volume value is 0,112 m^3, that can be passed to the total amount of mass with the density of the carbon fiber specification.

$$Mass\,(carbon\,fiber) \;=\; Density \cdot Volume \;=\; 1.780\,\frac{kg}{m^3} \cdot 0,112\,m^3 \;=\; 19,936\,kg\;carbon\;fiber$$

When the total mass is defined, the last step is to find how much CO2 comes from the fabrication of this material, which is going to be considered the basic carbon fiber in the market. As there is not a significant difference between all the types of fibers, the conversion factor used is [42]:

$$Conversion\,factor\ (CF) \ = \ 12,5\ kg\ CO_2 eq\ /\ kg\ carbon\ fiber$$

$$Carbon\,footprint \ = \ CF \cdot Mass\ (carbon\,fiber) \ = \ 12,5\ kg\ CO_2 eq\ /\ kg\ \cdot 19,936\ kg \ = \ 249,2\ kg\ CO_2$$

Finally, as the results show, the creation of the front wing to be used in wind tunnel simulations generates approximately 250 kg CO_2 to the planet, which is a substantial value compared with the amount polluted during the development of the entire project investigation. The exact percentage can be found with the following methodology:

$$Percentage = \frac{10{,}87\ kg\ CO_2}{249{,}2\ kg\ CO_2} \cdot 100\ \% \ = \ 4,36\ \%$$

Considering the results, an approximate 350-hour investigation is less than a 5% of the front wing fabrication, so that shows the importance of the previous computer study and CFD simulations, not only for the impact on the environment, but also because of the amount of economic resources that can be saved.

11 REFERENCES

11.1 ONLINE REFERENCES

[1] Formula 1. Glossary [online]. Formula 1. Formula One World Championship Limited, 2003 [Consultation 27th October 2021]. Available in:
<< https://www.formula1.com/en/championship/inside-f1/glossary.html >>

[2] Aerodinámica F1 [online]. Aerodinámica F1: alerón trasero by Mario Fernández, 2019 [Consultation: 26th October 2021]. Available in:
<< https://www.aerodinamicaf1.com/2019/10/el-aleron-trasero-en-la-formula-1-i-partes/ >>

[3] Autosport. Motorsport Network [online]. UK: Advertise with Autosport, 2022 [Consultation: 27th October 2021]. Available in:
<<https://www.autosport.com/f1/news/how-does-a-formula-1-car-work-wings-diffusers-and-more-explained-4982275/4982275/>>

[4] Youtube: Racing Atmosphere [online]. Spain: Formula 1 atmosphere, 2018 [Consultation: 27th October 2021]. Available in:
<< https://www.youtube.com/watch?v=jYoopIzhxFA&ab_channel=RacingAtmosphere >>

[5] Motor.es [online]. Motor.es: Formula 1 by José D. Pascual [Consultation: 27th October 2021]. Available in:
<< https://www.motor.es/formula-1/que-es-endplate-f1-202065766.html >>

[6] Aerodinámica F1 [online]. Aerodinámica F1: aleta de tiburón by Mario Fernández, 2020 [Consultation: 28th October 2021]. Available in:
<< https://www.aerodinamicaf1.com/2020/05/la-aleta-de-tiburon-en-la-formula-1/ >>

[7] RedBull [online]. Wikimedia Commons, 2015 [Consultation: 29th October 2021]. Available in:
<< https://www.redbull.com/es-es/f1-2015-historia-szisz-mansell-frentzen-clark-herbert-tambay >>

[8] Motorsport. Motorsport Network [online]. Madrid: Motorsort.com by Stefan Ehlen, 2021 [Consultation: 29th October 2021]. Available in:
<<https://es.motorsport.com/f1/news/galeria-evolucion-historica-coches-formula-1-866338/866338/#gal-866338-m0-giuseppe-farina-alfa-romeo-158-11334442>>

[9] Diariomotor.com [online]. Diariomotor, competición by Alex Garcia, 2017 [Consultation: 29th October 2021]. Available in:
<< https://www.diariomotor.com/competicion/breve/curiosidades-f1-el-cesped-de-la-victoria/ >>

[10] Aerodinámica F1 [online]. Aerodinámica F1: historia de la aerodinámica en la F1 by Mario Fernández, 2019 [Consultation: 29th October 2021]. Available in:
<< https://www.aerodinamicaf1.com/2019/09/historia-de-la-aerodinamica-en-la-f1/ >>

[11] Motorsport. Motorsport Network [online]. Madrid: Motorsort.com by Franco Nugnes, 2021 [Consultation: 2nd November 2021]. Available in:
<< https://es.motorsport.com/f1/news/carga-aerodinamica-formula1-record-comparativa/5171817/ >>

[12] Diariomotor.com [online]. Diariomotor, competición by Sergio Álvarez, 2017 [Consultation: 2nd November 2021]. Available in:
<< https://www.diariomotor.com/2016/05/03/monocasco-fibra-carbono-historia-mclaren />>

[13] Wikipedia. Wikimedia Foundation [online]. UK, Wikipedia, 2021 [Consultation: 3th November 2021]. Available in: << https://en.wikipedia.org/wiki/Tyrrell_019 >>

[14] Aerodinámica F1 [online]. Aerodinámica F1: Mass Damper by Mario Fernández, 2021 [Consultation: 4th November 2021]. Available in: << https://www.aerodinamicaf1.com/2021/01/el-mass-damper-de-renault-en-la-formula-1/ >>

[15] Wikipedia. Wikimedia Foundation [online]. UK, Wikipedia, 2019 [Consultation: 4th November 2021]. Available in: << https://es.wikipedia.org/wiki/Renault_R25 >>

[16] Aerodinámica F1 [online]. Aerodinámica F1: Doble difusor by Mario Fernández, 2020 [Consultation: 6th November 2021]. Available in:
<< https://www.aerodinamicaf1.com/2020/05/el-doble-difusor-en-la-formula-1/ >>

[17] Motorpasión [online]. España, WeblogsSL by Igor Zamorano, 2010 [Consultation: 9th November 2021]. Available in: << https://www.motorpasion.com/formula1/como-funciona-el-f-duct >>

[18] Ledalap Academy [online]. Badajoz, Ledalap Academy by Abel Caro, 2022 [Consultation: 10th November 2021]. Available in: << https://lebalap.academy/f1/aerodinamica/drs/ >>

[19] Youtube: Fullboost [online]. Australia, fullboost, 2009 [Consultation: 10th November 2021]. Available in: << https://www.youtube.com/watch?v=I7MOlYhLZDY >>

[20] Motorsport. Motorsport Network [online]. Madrid, Motorsort.com by Sergio Lillo, 2021 [Consultation: 10th November 2021]. Available in:
<< https://es.motorsport.com/f1/news/halo-formula1-que-es-seguridad/4918513/?nrt=54 >>

[21] Autobild.es [online]. España, Autobild: Aaxel springer by Adrián Mancebo, 2021 [Consultation: 10th November 2021]. Available in:
<< https://www.autobild.es/noticias/coches-f1-2022-llega-nuevo-reglamento-899503 >>

[22] Technical [online]. Zottegem, Technical F1 by Vanja Hasanovic, 2018 [Consultation: 26th November 2021]. Available in: << https://www.f1technical.net/features/21778 >>

[23] Aerodinámica F1 [online]. Aerodinámica F1: Numero de Reynolds by Mario Fernández, 2020 [Consultation: 26th November 2021]. Available in:
<< https://www.aerodinamicaf1.com/2020/10/que-es-el-numero-de-reynolds/ >>

[24] The Constructor, building ideas [online]. The constructor by Gopal Mishra, 2009 [Consultation: 29th November 2021]. Available in: << https://theconstructor.org/fluid-mechanics/dimensionless-numbers-importance-fluid-mechanics/34265/ >>

[25] Sciencedirect [online]. ScienceDirect, Elsevier B.V. by T.Tanuma, 2017 [Consultation: 29th November 2021]. Available in:
<< https://www.sciencedirect.com/topics/engineering/aerodynamic-efficiency >>

[26] Aerodinámica F1 [online]. Aerodinámica F1: Numero de Mach by Mario Fernández, 2021 [Consultation: 1st December 2021]. Available in:
<< https://www.aerodinamicaf1.com/2021/06/que-es-el-numero-de-mach/ >>

[27] Wikipedia. Wikimedia Foundation [online]. UK, Wikipedia, 2021 [Consultation: 1st December 2021]. Available in: << https://en.wikipedia.org/wiki/Mach_number >>

[28] Aerodinámica F1 [online]. Aerodinámica F1: Capa límite by Mario Fernández, 2019 [Consultation: 1st December 2021]. Available in: << https://www.aerodinamicaf1.com/2019/09/la-capa-limite/ >>

[29] Autobild.es [online]. España, Autobild: Aaxel springer by Adrián Mancebo, 2021 [Consultation: 8th December 2021]. Available in:
<< https://www.autobild.es/noticias/coches-f1-2022-llega-nuevo-reglamento-899503 >>

[30] The race [online]. UK, the race by Edd Straw, 2021 [Consultation: 9th December 2021]. Available in: << https://the-race.com/formula-1/newey-f1-2022-rule-changes-the-biggest-in-four-decades/ >>

[31] Youtube: Formula 1 [online]. Official F1 Channel, 2005 [Consultation: 9th December 2021]. Available in: << https://www.youtube.com/watch?v=hBmWJOy9vT4&ab_channel=FORMULA1 >>

[32] FIA [online]. FIA, 2022 F1 technical regulations, 2021 [Consultation: 13th December 2021]. Available in: <<https://www.fia.com/sites/default/files/2022_formula_1_technical_regulations_-_iss_3_-_2021-02-19.pdf >>

[33] Fi.com [online]. Fi, digital motorsport media by Michael Delaney, 2021 [Consultation: 19th December 2021]. Available in:
<< https://f1i.com/news/418207-monza-speed-trap-who-is-the-fastest-of-them-all-3.html >>

[34] Motormario [online]. Italy, motormario.com by Ignacio Bettosini 2021 [Consultation: 19th December 2021]. Available in: <<https://motormario.com/analisis-los-circuitos-velocidades-y-promedios#:~:text=La%20mayor%20parte%20de%20los,la%20FIA%20ha%20considerado%20ideal>>

[35] Wikipedia. Wikimedia Foundation [online]. UK, Wikipedia, 2022 [Consultation: 19th December 2021]. Available in: << https://es.wikipedia.org/wiki/Aire >>

[36] Technical [online]. Zottegem, Technical F1 by Joshua Newbon, 2021 [Consultation: 21th December 2021]. Available in: << https://www.f1technical.net/news/23344 >>

[37] La Verdad [online]. Cancún, La Verdad GELV Comercializadora by Marina Rosales, 2020 [Consultation: 23th December 2021]. Available in: << https://laverdadnoticias.com/ecologia/Impacto-ambiental-en-la-Formula-1-Como-y-cuanto-contamina-20201210-0117.html >>

[38] HP, HPsupport [online]. HP Development Company, 2015 [Consultation: 23th December 2021]. Available in: << https://support.hp.com/es-es/document/c06040688 >>

[39] Gencat [online]. Cataluña, Gencat Canvi Climatic, 2021 [Consultation: 24th December 2021]. Available in:
<<https://canviclimatic.gencat.cat/es/actua/factors_demissio_associats_a_lenergia/#:~:text=El%20mix%20de%20la%20red%20el%C3%A9ctrica%20espa%C3%B1ola%20publicado%20por%20la,25%20kg%20CO2%2FkWh>>

[40] Red eléctrica de España [online]. España, Grupo Red Eléctrica, 2022 [Consultation: 24th December 2021]. Available in:
<< https://www.ree.es/es/actividades/operacion-del-sistema-electrico/coeficientes-de-perdidas >>

[41] UCO, huella de carbono [online]. Córdoba, La huella de Carbono de la UCO by Ana de Toro 2013 [Consultation: 24th December 2021]. Available in:
<< https://www.uco.es/servicios/sepa/images/documentos/descargas/huellaC2013.pd f>>

[42] AIMPLAS [online]. San Sebastián, AIMPLAS by Eva Verdejo, 2016 [Consultation: 24th December 2021]. Available in: << https://life-future-project.eu/descarga_web.php?id_pub=146 >>

[43] SimScale GmbH [online]. Munich, SimScale member of NAFEMS, 2021 [Consultation: 24th December 2021]. Available in: << https://www.simscale.com/docs/simulation-setup/meshing/ >>

[44] Curiosity Fluids [online]. Curiosity Fluids by STF solutions, 2016 [Consultation: 26th December 2021]. Available in: << https://curiosityfluids.com/2016/09/09/establishing-grid-convergence/ >>

[45] GrabCad Community [online]. Brazil, Ferrari SF71 by Rafael Rodrigues, 2018 [Consultation: 28th December 2021]. Available in: << https://grabcad.com/library/ferrari-sf71h-1 >>

[46] Wordpress [online]. España, Wordpress perfil alar, 2014 [Consultation: 28th December 2021]. Available in: << https://jasf1961.wordpress.com/tag/perfil-alar/ >>

11.2 OTHER REFERENCES

[47] Race Car Aerodynamics: Designing for Speed, by Joseph Katz, 1996. ISBN 9780837601427

Also, some references that must be taken into consideration are the ones that gave me the base knowledge about aerodynamics, fluid mechanics or fluid technology. Although the information has not been directly extracted from its content, the following books are cited:

- Red Bull Racing F1 Car 2010-2014 by Owners' Workshop Manual. ISBN 9780857338013
- Computational Methods for Fluid Dynamics by Joel H. Ferziger. ISBN 9783540420743
- Computational Fluid Dynamics by John D.Anderson, JR. ISBN 9780070016859
- Mecánica de Fluidos by Irving H.Shames. ISBN 958600246
- Mecánica de fluidos. Problemas y soluciones by Julio Hernández. ISBN 9788436270037
- Mecánica de Fluidos by Frank M.White. ISBN 9788448166038
- Mecánica de Fluidos. Fundamentos y aplicaciones by Cengel Yunus. ISBN 9789701056127