BACHELOR THESIS

# Design of a dynamic diffuser for the aerodynamic improvement of a vehicle

*Author:*

Júlia Roca Ràfols

*Director:*

Jordi Vives Costa

UNIVERSITAT POLITÈCNICA DE CATALUNYA

Escola Politècnica Superior d'Enginyers de Manresa

Bachelor's Degree in Automotive Engineering

2021-2022

# Abstract

The impact on aerodynamic performance of incorporating plain floor and diffuser in a vehicle is well known. Normally diffusers are usually centered on the longitudinal symmetry axis of the vehicle. However, mass transfers in curved trajectories due to the effect of inertia, unbalance the normal forces on the inner and outer wheels. The objective of this project is the design of a dynamic diffuser, which is actuated by lateral displacement in response to side acceleration. This implementation would help to rebalance the normal forces between the inner and outer wheels of the rear axle.

# Resum

És ben conegut l'impacte que té sobre el rendiment aerodinàmic, el fet d'incorporar un sòl pla i un difusor a un vehicle. Normalment els difusors se solen centrar en l'eix de simetria longitudinal. No obstant, la transferència de massa en trajectòries corbes a causa de la inèrcia, desequilibren les forces normals a les rodes interiors i exteriors. L'objectiu d'aquest projecte és el disseny d'un difusor dinàmic, que actuï realitzant un desplaçament transversal com a resposta de l'acceleració lateral. Aquesta implementació ajudaria a reequilibrar les forces normals entre les rodes interiors i exteriors de l'eix posterior.

# Contents


**Abstract** **i**

**Resum** **ii**

**List of Figures** **v**

**Nomenclature** **vi**

**1. Introduction** **- 1 -**

*1.1. Background* *- 1 -*

*1.2. Motivation* *- 2 -*

*1.3. Objectives* *- 2 -*

**2. Aerodynamics background** **- 3 -**

*2.1. Flow conception* *- 3 -*

*2.2. Boundary Layer Theory* *- 4 -*
**2.2.1. Reynolds Number** - 4 -
**2.2.2. Shear Stress** - 5 -

*2.3. The Lift Force* *- 6 -*
**2.3.1. The Lift coefficient** - 7 -
**2.3.2. Aerodynamic load: Downforce** - 7 -

*2.4. The Drag Force* *- 7 -*
**2.4.1. Separation of the boundary layer** - 8 -
**2.4.2. Shape resistance and Drag Coefficient** - 8 -
**2.4.3. Coanda Effect** - 9 -
**2.4.4. Importance of Drag on cars** - 9 -

*2.5. The Venturi Effect* *- 12 -*

*2.6. Plain Floor and Diffuser application* *- 12 -*

**3. Kinematics and dynamics background** **- 14 -**

*3.1. Car configuration* *- 14 -*
**3.1.1. Axles distance** - 14 -
**3.1.2. Wheelbase** - 14 -
**3.1.3. Ride height** - 14 -
**3.1.4. Track Width** - 15 -
**3.1.5. Static weight distribution** - 15 -
**3.1.6. Center of gravity, CG** - 16 -
**3.1.7. Center of pressure, CoP** - 18 -

*3.2. Straight trajectories* *- 19 -*

*3.3. Curved trajectories* *- 19 -*
**3.3.1. Yaw and Roll motions** - 20 -

**4. Diffuser configuration** **- 22 -**

*4.1. Conventional diffuser* *- 22 -*

*4.2. Double diffuser* *- 23 -*

*4.3. Blown diffuser* *- 23 -*

**5. Design of a dynamic diffuser theoretical prototype** **- 24 -**

*5.1. Air flow path through the underbody* *- 24 -*

*5.2. Diffuser orientation* *- 26 -*

*5.3. Electro-hydraulic system* *- 26 -*

*5.4. Electronic position control* *- 27 -*

**5.4.1. Control systems** - 27 -

**5.4.2. Hydraulic servo-systems** - 28 -

**5.4.3. Position control systems** - 29 -

*5.5. Double-acting cylinder* *- 30 -*

*5.6. Modelling and simulation of the electro-hydraulic system* *- 31 -*

**5.6.1. PID Controller** - 32 -

**5.6.2. Power Unit** - 34 -

**5.6.3. Actuator** - 35 -

**5.6.4. System response** - 35 -

**6. CAD design of the diffuser prototype** **- 37 -**

*6.1. Diffuser design* *- 39 -*

*6.2. Cylinder* *- 40 -*

*6.3. Cylinder-diffuser assembly* *- 40 -*

**7. Conclusions** **- 43 -**

**Bibliography** **- 46 -**

# List of Figures


*Figure 1 – Boundary layer thickness [Sven De Schampheleire, 2017]* ................................... - 5 -
*Figure 2 - Velocity and Shear Stress acting in the Boundary Layer* [Kutateladze, 1990] ....... - 6 -
*Figure 3 - Airfoil profile and flow stream lines [Rex Lowe, 2020]* ......................................... - 6 -
*Figure 4 - Detachment of the Boundary Layer [Olivier Cleynen, 2015]* ................................. - 8 -
*Figure 5 - Fraction of aerodynamic drag D in total drag D+R [Fluid-dynamics notes, 2020]* - 10 -
*Figure 6 - Fuel consumption affected by drag, mass and power [Fluid-dynamics notes, 2020]* - 10 -
*Figure 7 - Powe-Speed curve [Fluid-dynamics notes, 2020]* ................................................ - 11 -
*Figure 8 - Speed-Time curve [Fluid-dynamics notes, 2020]* ................................................. - 11 -
*Figure 9 - Floor Effect on cars [Magenta Green, 2017]* ....................................................... - 13 -
*Figure 10 - Axles distance [Fluid-dynamics notes, 2020]* .................................................... - 14 -
*Figure 11 – Wheelbase [Fluid-dynamics notes, 2020]* ........................................................ - 14 -
*Figure 12 - Ride Height [Fluid-dynamics notes, 2020]* ........................................................ - 15 -
*Figure 13 - Track Width [Fluid-dynamics notes, 2020]* ...................................................... - 15 -
*Figure 14 - Static weight distribution [Fluid-dynamics notes, 2020]* ................................... - 15 -
*Figure 15 - Parameters for CG calculation [Fluid-dynamics notes, 2020]* ............................ - 16 -
*Figure 16 - Parameters for h2 calculation [Fluid-dynamics notes, 2020]* ............................. - 17 -
*Figure 17 - Important factors acting on a car regarding the CoP [own elaboration, 2021]* ..... - 18 -
*Figure 18 - ISO coordinate system [Mechanical Auxiliar Systems notes, 2021]* ................... - 20 -
*Figure 19 - Relating local and global systems with the angles [AutoMotorGarage, 2020]* ..... - 20 -
*Figure 20 - Basic configuration of car turning [M. Ouahi, J. Stéphant, D. Meizel, 2010]* ..... - 21 -
*Figure 21 - Basic configuration of car rolling [Hindawi, 2013]* ........................................... - 21 -
*Figure 22 - Double Diffuser and airflow path [WordPress, 2011]* ....................................... - 23 -
*Figure 23 - Curved-flow, side-slip and yaw-angle [Honda R&D Technical Review, 2009]* .... - 25 -
*Figure 24 – Diffuser conceptual representation [own elaboration, 2021]* ............................. - 26 -
*Figure 25 – Closed-loop control system blocks diagram [Dorf, 2011]* ................................... - 27 -
*Figure 26 - Directional valve and actuator [own elaboration, 2021]* ................................... - 29 -
*Figure 27 - Position control block diagram [own elaboration, 2021]* ................................... - 29 -
*Figure 28 - Single-rod double-acting cylinder [Festo, 2021]* ................................................ - 31 -
*Figure 29 – Simulink model of the electro-hydraulic system [own elaboration, 2021]* ........... - 32 -
*Figure 30 - Block Diagram of PID controller in Simulink [own elaboration, 2021]* ............. - 33 -
*Figure 31 - Simulink PID adjudged values [own elaboration, 2021]* .................................... - 34 -
*Figure 32 - Power Unit ensemble [own elaboration, 2021]* ................................................. - 35 -
*Figure 33 - Range of displacement (x) of the rod [own elaboration, 2021]* ........................... - 36 -
*Figure 34 - System response to an step input of 0.2 [own elaboration, 2021]* ....................... - 36 -
*Figure 35 – Front isometric view of the diffuser assembly prototype [own elaboration, 2021]* - 37 -
*Figure 36 – Rear isometric view of the diffuser assembly prototype [own elaboration, 2021]* . - 37 -
*Figure 37 - Exploded View [own elaboration, 2021]* .......................................................... - 38 -
*Figure 38 - Diffuser angle respect the x-axis [own elaboration, 2022]* ................................. - 39 -
*Figure 39 - Needle Bearing dimensions [TraceParts, 2019]* ................................................ - 39 -
*Figure 40 - Rod and Ledge displacement representations [own elaboration, 2021]* ................ - 41 -
*Figure 41 - Trigonometric variables [calculo.cc, 2021]* ....................................................... - 41 -
*Figure 42 – Ledge, bolt and bearing [own elaboration, 2021]* ............................................. - 42 -
*Figure 43 – Tenon [own elaboration, 2021]* ....................................................................... - 42 -
*Figure 44 – Exploded View of the cylinder and other components [own elaboration, 2021]* ... - 42 -

# Nomenclature

*BL = Boundary Layer*

*Re = Reynolds Number*

*x = horizontal position*

$x_{cr}$ *= critical x*

*y = vertical position*

*v = velocity*

$\tau$ = *Shear stress*

*P = pressure energy*

$\rho$ = *density*

*h = height*

*CG = Center of gravity*

*a = distance between CG and front axle*

*b = distance between CG and rear axle*

$L_{WB}$ = *Wheelbase*

$h_{RH}$ = *Ride height*

$t_F$ = *Front track width*

$t_R$ = *Rear track width*

*W = weight*

*CoP = Center of Pressure*

*D = Drag force*

*S = frontal area*

$C_D$ = *drag coefficient*

$\delta_f$ = *front wheel offset angle*

$\beta$ = *vehicle offset angle*

$F_{zo}$ = *external vertical reaction force*

$F_{zi}$ = *internal vertical reaction force*

# 1. Introduction

This project will focus in the study of aerodynamics, dynamics and kinematics applied in one of the most relevant components of competition vehicles: the diffuser. It will seek to address how this element works and what is its contribution to the vehicle performance.

Above all, this study will deal with the theoretical devising and developing of a new operating outcome for this element, seeking its best performance in vehicle turning situations.

At last, it will be intended to make a basic preliminary design of its structural configuration and its internal functioning, that allows us to obtain the desired result.

## 1.1. Background

This project unfolds into three main areas of knowledge when we talk about the automotive sector: aerodynamics, dynamics and kinematics. By knitting the three of them together, we have the required cognition to understand the functioning principle of the diffuser, and how to study its performance with the aim of improving it.

Focusing on vehicle aerodynamics, we harness it to explain the behaviour of a flow on the vehicle surface and, therefore throughout a diffuser. We draw on it to comprehend the dynamic and kinematic demeanour of the vehicle while its manoeuvrability. Therefore, we rely on the fact that the dynamical-kinematical output performance of a vehicle, is the response to the aerodynamical effects that act on it.

In addition, this project also relies on the implementation of a CAD software in order to develop a preliminary design of a dynamic diffuser. With this, we get to make use of the learned knowledge in the areas mentioned above, and apply them to develop a 3D virtual prototype.

## 1.2. Motivation

One of the main motivations on the realization of this project is to be able to understand more deeply the functioning of the diffusers, and to give them a new scope that has never been raised before. During my learning years in the automotive engineering, I have taken pleasure in learning fluid-dynamics and designing with CAD software, which are some of the subjects I have enjoyed the most. This has led me to feel truly interested in this specific project.

Moreover, I was enticed by the idea of looking beyond an already existing element, which has been used for many years now, and give it a new and more complex outlook. For me, it is a really good opportunity to enforce my knowledge in automotive notions and also let my creativity flow to contribute in new ideas.

## 1.3. Objectives

The main tasks to be carried out in this project are:

- ✓ Acknowledge in detail the dynamics, aerodynamics and kinematics on a car.
- ✓ Understand the role of a diffuser in a car.
- ✓ Get to know the different types of diffusers and its differences.
- ✓ Study and select a way of evolving the diffuser in order to get a lateral dynamic performance.
- ✓ Explain the obtained final result of the dynamic diffuser.
- ✓ Developing a simplified prototype in a CAD software.

# 2. Aerodynamics background

With the objective to dig into the functioning principle of a diffuser, it is highly important to understand the fundamentals of aerodynamics that take part in its performance.

## 2.1. Flow conception

We nominate flow as the technical concept of a moving fluid. There are many different categorizations to define a flow depending on its movement, composition, properties or behaviour.

- **Steady and Unsteady flow**: a flow might change its properties along time.
- **Inviscid and viscous flow**: in some fluid flows, viscosity can be neglected.
- **Compressible and incompressible flow**: depending on whether the fluid flow experiments a variation on density with time.
- **Uniform and non-uniform flow**: properties might be constant on all directions.
- **Laminar and turbulent flow**: Regarding the behaviour of the flow that overpasses an object, we can distinguish between laminar and turbulent flow. When the flow has a laminar predisposition, all particles waft on the same direction. Conversely, when the flow is turbulent the particles move freely, without a logic pattern.
- **Rotational and irrotational flow**: the flow boundary lines can move with different angles or rotate as a whole.

## 2.2. Boundary Layer Theory

In order to study the behaviour of a flow above a flat straight surface, the Boundary Layer Theory is used. We consider the generation of aerodynamic forces between the fluid and the body. The magnitude of the forces depends mainly on the shape of the body, and the viscosity and compressibility of the fluid.

When the flow of the fluid starts passing through the body, the particles that are closer to the surface stick on it. This fact originates a thin layer of flow close to the surface where the speed of the fluid oscillates from zero – at the surface – to the free stream value – away from the surface –. We define the free stream speed value $v$.

The change of velocity in the streamwise direction induces to the generation of the boundary layer. We define the thickness of the boundary layer as the distance until where this disturbance on the velocity takes place.

The aspect we consider in order to define a flow as laminar or turbulent, is the value of the Reynolds number.

### 2.2.1. Reynolds Number

The Reynolds number is the ratio of inertial forces to viscous forces. It is a dimensionless number used to categorize fluid systems in which the effect of viscosity is important in controlling the flow rates and patterns of a fluid.

For lower values of the Reynolds number, the fluid inside the boundary layer is laminar and the streamwise velocity changes uniformly between the surface of the body and the layer limit. This low values of the Re number appear near the first point of contact between the fluid and the body, where the fluid begins to stick to the surface. Referring to *Figure 1*, this laminar region is settled between $x = 0$ and $x = x_{cr}$.

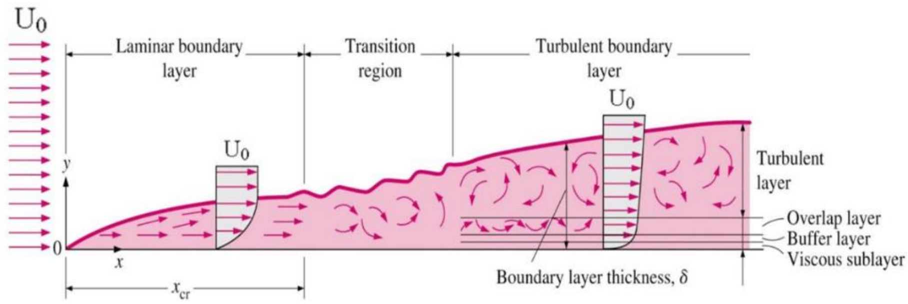


*Figure 1 – Boundary layer thickness* [Sven De Schampheleire, 2017]

As the value of $x$ increases, the Reynolds number does as well, and the flow becomes progressively more unstable. When the Reynolds number is on the order of $Re = 5 \cdot 10^5$, the critical point has been reached, therefore $x = x_{cr}$. Hereinafter, transition from laminar to turbulent occurs. This transition may take place earlier depending on the surface roughness and other boundary conditions.

### 2.2.2. Shear Stress

The shear stress generated between the fluid and the stationary surface when they come into contact, is the responsible of the velocity decrease down to zero at the surface, and the generation of the laminar layer at the beginning where the fluid increases its velocity uniformly.

At the outset of the boundary layer, the shear stress has its maximum value. In return, it leads to zero at the limit of the boundary layer. Regarding *Figure 2*, for a fixed value of $y$, the shear stress will decrease as $x$ increases, due to the growth of the boundary layer thickness.

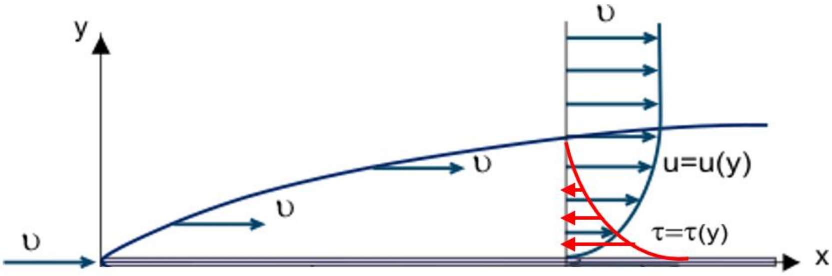


*Figure 2 - Velocity and Shear Stress acting in the Boundary Layer* [Kutateladze, 1990]

Getting to the point where $x = x_{cr}$, shear stress gets to its minimum value. It can't prevent the turbulence any longer, so the fluid loses its laminar structure and the particles start the transition to turbulent flow.

## 2.3. The Lift Force

We consider we have a cambered airfoil (non-symmetrical) and that it is upright (it has a positive camber on the upper part). Analysing the effect caused by the Bernoulli's Conservation of Energy Law on the airfoil, it can be noted the emergence of a force that pushes the body up. This is the result of creating a situation where the pressure above the airfoil is lower than the pressure below it.

When a flow is set in the direction of the airfoil profile, the flow will divide and part of it will pass along the upper surface and the other along the bottom. In the airfoil previously described, the fluid streams that pass above the airfoil will narrow due to the camber. This will generate a velocity increasing and, therefore, a pressure reduction. Meanwhile, at the lower part the flow will remain in its streamwise direction.

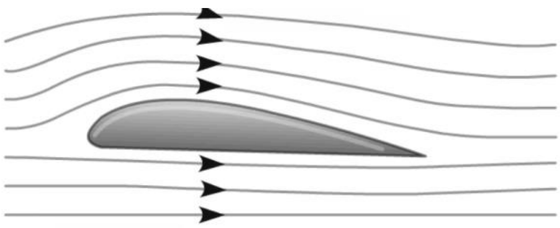

*Figure 3 - Airfoil profile and flow stream lines* [Rex Lowe, 2020]

This situation leads to a pressure difference between the upper and lower part, which generates a force that tries to re-balance this inequality. This force is called Lift.

### 2.3.1. The Lift coefficient

The coefficient of lift is a number aerodynamicists use to model all the complex dependencies of shape, inclination, and some flow conditions on lift. This coefficient depends on the lift force, the density, the velocity and the frontal area.

$$C_L = \frac{L}{\rho \cdot v^2 \cdot S}$$

### 2.3.2. Aerodynamic load: Downforce

Downforce is the name that designates the negative Lift. Considering the same airfoil on *Figure 3*, if we change its position upside down, the force generated for balancing the pressure difference would be in the opposite direction (downwards).

In the automotive sector, the downforce is considered an aerodynamic load. In order to compute the weight of a vehicle in different driving situations, it has to be added to the mass. It has been estimated that the effective weight of a racing car while cornering is 4 times its original weight, and while breaking 6 times it, approximately.

This makes the downforce a valuable and sought-after effect by car designers. It improves the stability and manoeuvrability without the need to increase the vehicle's mass.

## 2.4. The Drag Force

The Drag force can be described as the aerodynamic resistance a body provides. It appears when there is a detachment of the laminar layer with respect to the body surface. When this detachment occurs, the laminar structure is lost and uncontrolled big vortices are formed.

### 2.4.1. Separation of the boundary layer

The detachment of the boundary layer is a phenomenon that, in most technical applications, experts try to prevent. For example, if separation occurs prematurely in car surfaces or airplane wings, it will lead to a loss of lift (or downforce), an increased drag and less energetic efficiency.

The separation point (considering a 2D model) is defined as the point where the average shear stress tends to zero ($\bar{\tau} = 0$). This means the shear stress is partially positive and partially negative in the same proportion. As a consequence, part of the flow will be in the backward direction.

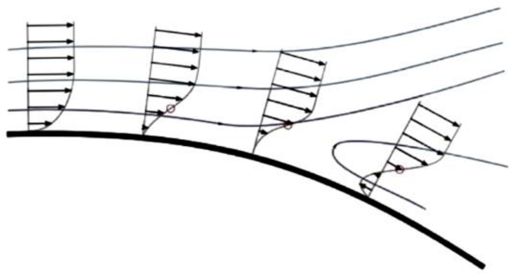

*Figure 4 - Detachment of the Boundary Layer* [Olivier Cleynen, 2015]

Hence, the detachment point is achieved when the 50% of the average flow is forwards and the 50% is backwards.

### 2.4.2. Shape resistance and Drag Coefficient

The shape of a body has a direct impact on the amount of drag. Aerodynamicists study the dependence of drag with the shape by comparing the values of drag coefficient for different objects.

The drag coefficient is a numerical value used to define the dependence of the drag with the shape, the density and the velocity of an object.

$$C_D = \frac{D}{\rho \cdot v^2 \cdot S/2}$$

In vehicles, the aerodynamic force and the required power are directly proportional to the frontal area.

$$F_T = C_D \cdot S \cdot \rho \cdot \frac{v^2}{2}$$

$$P_T = F_T \cdot v = C_D \cdot S \cdot \rho \cdot \frac{v^3}{2}$$

### 2.4.3. Coanda Effect

Coanda Effect defines the phenomenon where a moving fluid stream, in contact with a curved surface, will always tend to follow the curvature of that surface instead of continuing in a straight line.

Vehicles, especially race cars, adapt their shapes in order to take advantage of this property and thus reduce the drag.

### 2.4.4. Importance of Drag on cars

Is important to analyse the drag force effects on vehicles performance. Thereby, bellow we will see some graphs regarding the $C_D$ and its relation with fuel consumption and other remarkable parameters.

- **Fraction of aerodynamic drag in total drag**

The less aerodynamic the vehicle, the faster the $C_D$ increases as the speed increases. In the following graph we are comparing the percentage of drag a certain vehicle offers, in comparison with the instant velocity. The total value of the drag is defined as the drag force produced by the vehicle (D) plus the difference of force required to get to the total value (R).

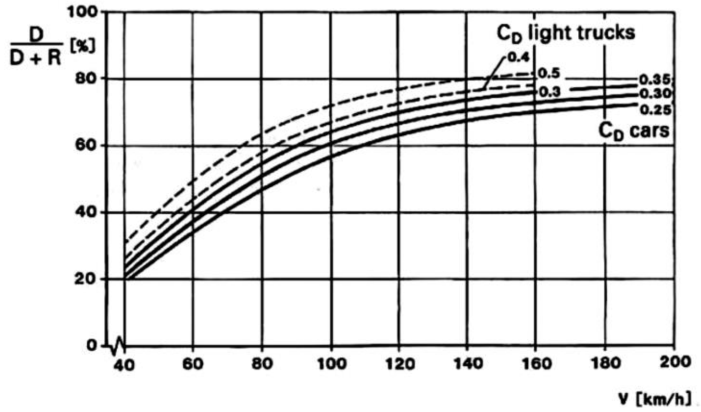


*Figure 5 - Fraction of aerodynamic drag D in total drag D+R* [Fluid-dynamics notes, 2020]

## • Impact on fuel consumption

In *Figure 6*, we can see how fuel consumption is affected by the drag, mass and supplied power. This study is carried out by the comparison of two different vehicle models.

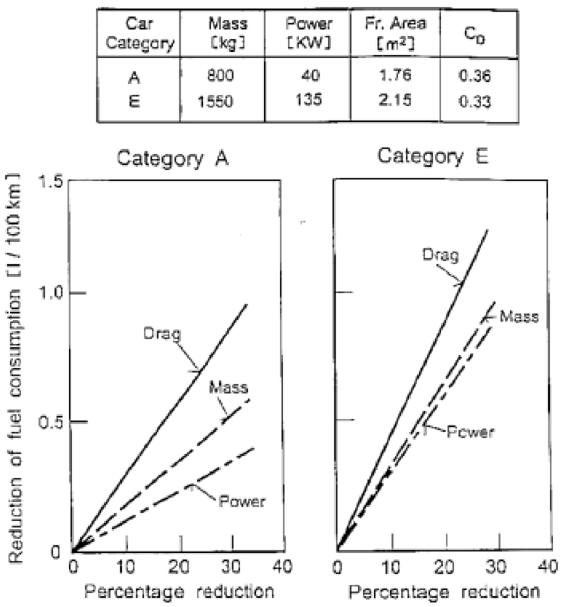


*Figure 6 - Fuel consumption affected by drag, mass and power* [Fluid-dynamics notes, 2020]

We can see in category E, even if we have a heavier vehicle with a larger front area, as the $C_D$ is lower the reduction of the fuel consumption is bigger.

- **Power vs. Speed**

At a common speed, the lower the $C_D$ the lower the power required. Therefore, there will be less fuel consumption.

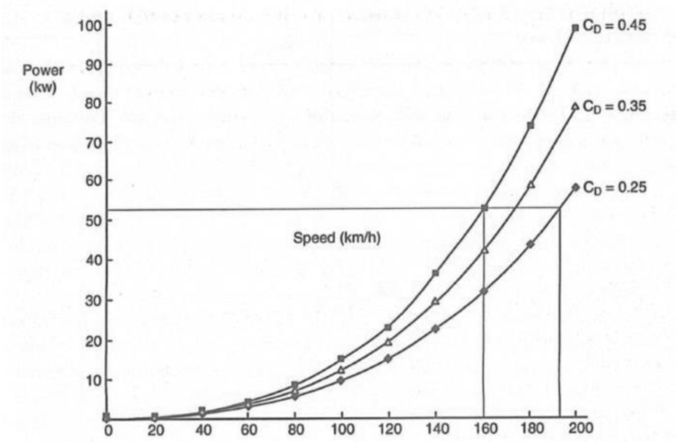


*Figure 7 - Powe-Speed curve* [Fluid-dynamics notes, 2020]

- **Speed vs. Time**

With a smaller $C_D$ it takes less time to reach a certain speed. This only happens when we talk about high speeds. At speeds lower than 100 km/h the difference is virtually negligible.

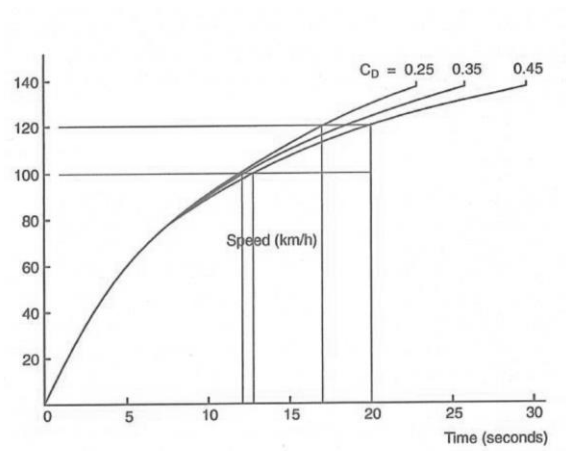


*Figure 8 - Speed-Time curve* [Fluid-dynamics notes, 2020]

## 2.5. The Venturi Effect

The Venturi Effect is the experimental outcome of reducing the section through which a fluid flow is carried out. When a fluid is outflowing across a constant section duct, and eventually this section decreases, the velocity will increase and the pressure will drop.

This phenomenon is theoretically transcribed by means of the Bernoulli's Equation. The equation uses the *Mechanical Energy Conservation Principle* to equalize the resultant energies generated before (1) and after (2) the section reduction.

$$P_1 + \frac{1}{2}pv_1{}^2 + pgh_1 = P_2 + \frac{1}{2}pv_2{}^2 + pgh_2$$

In the equation, the first factor (P) corresponds to the Pressure Energy, the second factor ( $\frac{1}{2}pv^2$) is the Kinetic Energy per unit volume, and the third factor ($pgh$) the Potential Energy per unit volume.

## 2.6. Plain Floor and Diffuser application

Car designers take advantage of the existence of a flat surface under the vehicle in order to generate downforce, due to the low air pressure. Applying Bernoulli, by reducing the lower section between the car floor and the ground, we have an increase in speed that leads to a reduction in pressure.

The car floor can be designed in such a way that, with a minimum resistance, we get this pressure reduction and therefore a downwards force generation, the negative lift or downforce. The goal is to find the best height between the ground (or track) and the vehicle, and that there is an adequate exit and entrance.

The ground effect, working together with the diffuser, can represent around 50% of the Downforce of a F1 car. Hence, this ground effect is essential because it is the one that generates most of the Downforce.

As always, there is a certain limit. There must be a gap through which the air passes, otherwise we would have a blockage. The air would not enter due to the boundary layer effect and would block the function of the ground effect and the diffuser.

To increase the floor effect, the diffuser is essential. However, if we increase the exit angle of the diffuser a lot, the detachment of the boundary layer may appear and this would break its operation. Therefore, the angles are limited. The objective is to give the diffuser the greatest possible exit angle, avoiding the detachment of the boundary layer and creating a more important ground and suction effect.

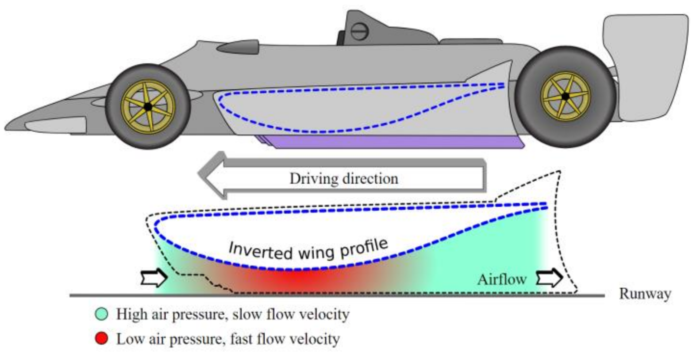


*Figure 9 - Floor Effect on cars* [Magenta Green, 2017]

# 3. Kinematics and dynamics background

## 3.1. Car configuration

### 3.1.1. Axles distance

The distance of the front and rear axle to the center of gravity is very important on a car. It defines how the loads will be distributed and it has a direct impact on the stability and the manoeuvrability.

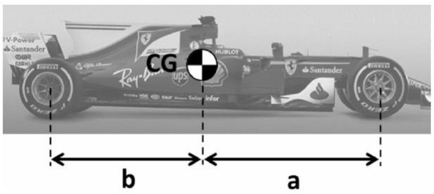


*Figure 10 - Axles distance* [Fluid-dynamics notes, 2020]

### 3.1.2. Wheelbase

The wheelbase is the perpendicular distance between the front and the rear axles. It gives an indication of the dynamic characteristics of the vehicle ride and handling. When a car has a larger wheelbase distance, it shows a better performance due to a reduced tendency to pitch and roll.

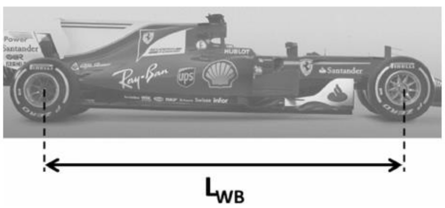


*Figure 11 – Wheelbase* [Fluid-dynamics notes, 2020]

### 3.1.3. Ride height

The right height, also known as ground clearance, corresponds to the distance between the lowest point of the vehicle and the ground. The longer a ride height is, the higher the position of the center of gravity, and consequently a worst handling performance.

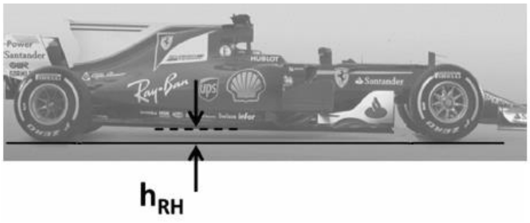


*Figure 12 - Ride Height* [Fluid-dynamics notes, 2020]

### 3.1.4. Track Width

Defines the distance between the centreline of the wheels located in the same axle. In a conventional vehicle we can distinguish between the *Front track width* ($t_F$) and the *Rear track width* ($t_R$).

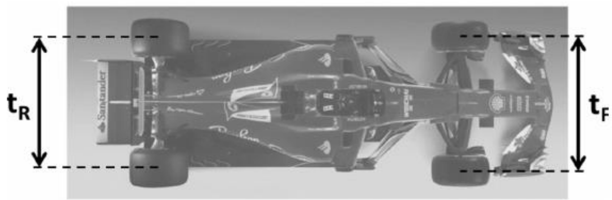


*Figure 13 - Track Width* [Fluid-dynamics notes, 2020]

### 3.1.5. Static weight distribution

The weight of the vehicle is distributed between the front and rear axles. Depending on the location of the center of gravity, the weight will tend to the front or the rear.

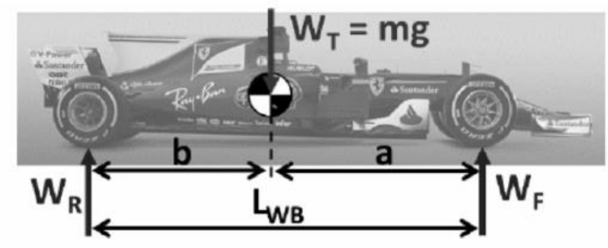


*Figure 14 - Static weight distribution* [Fluid-dynamics notes, 2020]

### 3.1.6. Center of gravity, CG

A center of gravity is a theoretical point where the resultant of all the forces that act on the body are applied. The torque generated by this resultant in any point of the body, is the same as the one produced by the weights of all the material masses that constitute said body.

The center of gravity depends on the shape of the body and how the mass is distributed. In a car, placing components up high, such as panoramic sunroof, will increase the CG height. In return, setting heavy subsystems at the bottoms of the vehicle, such as a battery pack, diminishes it.

To ensure good stability in vehicles, car designers will always seek for the lower CG possible, always regarding the shape and the requirements of it on its use.

In order to calculate the height of the CG, we define the variables displayed in *Figure 15*. To obtain these variables physically, the rear wheels of the car must be set onto platforms, so the front axle can be weight out.

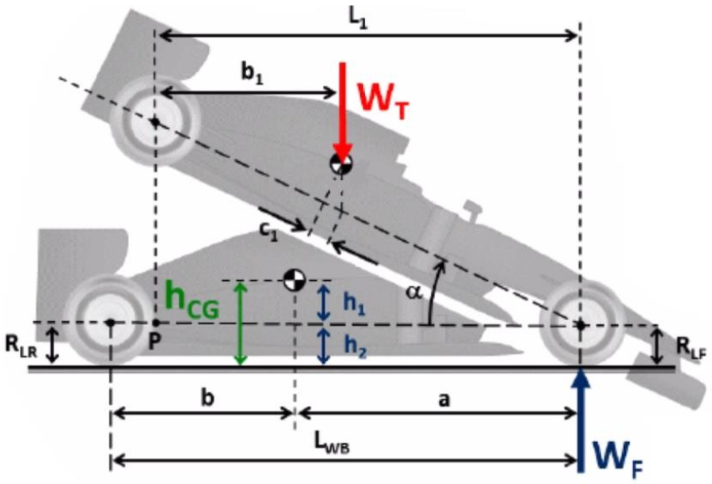


*Figure 15 - Parameters for CG calculation* [Fluid-dynamics notes, 2020]

As can be regarded, the height of the CG is discomposed in $h_1$ and $h_2$ in order to do the calculations.

Calculating $h_1$:

$$h_1 = \frac{c_1}{tan\alpha}$$

To calculate $c_1$ is used:

$$cos\alpha = \frac{b_1}{b + c_1}$$

For $b_1$ moments are taken around P:

$$W_F \cdot L_1 = W_r \cdot b_1$$

For $L_1$ is used:

$$L_1 = L_{WB} \cdot cos\alpha$$

$$h_1 = \frac{W_F \cdot L_{WB} - W_T \cdot b}{W_T \cdot tan\alpha}$$

Calculating $h_2$:

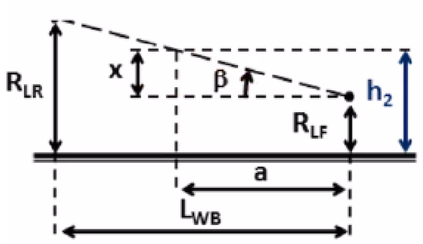


*Figure 16 - Parameters for h2 calculation* [Fluid-dynamics notes, 2020]

$$h_2 = R_{LF} + x$$

To calculate x is used:

$$tan\beta = \frac{x}{a} = \frac{R_{LR} - R_{LF}}{L_{WB}}$$

$R_{LF}$ = radius under front load
$R_{LR}$ = radius under rear load

$$h_2 = \frac{b \cdot R_{LF} + a \cdot R_{LR}}{L_{WB}}$$

### 3.1.7. Center of pressure, CoP

Generally, the center of pressure is the imaginary point where the total sum of aerodynamic forces applies. In a vehicle, it is the point where the resultant of the sum of the Drag and Downforce is located.

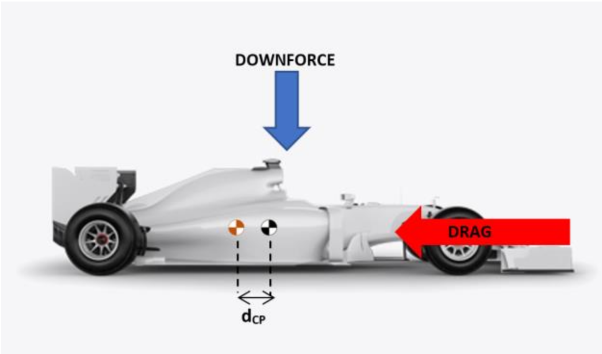


*Figure 17 - Important factors acting on a car regarding the CoP* [own elaboration, 2021]

In car aerodynamics analysis, we define a front-rear CoP. This way we can study the position of the CoP with regard to the CG and the axles. In the specific case where there is more front downforce, the CoP will be located between the front axle and the CG. In the other hand, if most of the downforce is applied to the rear, the CoP will be found between the CG and the rear axle.

This is important because, in the first case it will be more downforce on the front axle, in contrast with the second where the majority of the downforce will be applied on the rear axle.

In the first instance, there will be proportionately more aero grip on the front axle and we will have an oversteer vehicle. In the second instance, there is proportionately more aero grip on the rear axle and the vehicle is understeer. The aero grip defines the maximum lateral acceleration the axle can effectively take.

Furthermore, it is also important to consider the balance of the side forces. When the car is turning around, it experiences the action of lateral forces. These forces will act on the CoP. Owing to the distance between the CoP and the CG, a torque will appear.

Considering that the CoP is behind the CG, the car will always be stable in yaw. On the contrary, if the CoP is in front of the CG it will be unstable in yaw due to the generation of a torque that acts in the same direction as the car trajectory. For this reason, it is important that the CoP remains behind the CG.

## 3.2. Straight trajectories

When a vehicle is driving along a straight line, the forces that apply to it are the driving force from the engine in this same direction, and the air resistance and the braking force (sometimes) in the opposite direction.

Considering the crosswind forces negligible, in this situation there are no lateral forces acting on the vehicle. Accordingly, we have an ideal situation to obtain the maximum performance of the aerodynamic components on the car. Focusing on the lower part of the vehicle, the flow will get through the underbody of the car following its longitudinal axis along the flat floor and getting out through the diffuser.

Moreover, the lack of lateral actuators leads to a perfect compensation of the loads between the both sides of the vehicle, avoiding a transverse decompensation.

## 3.3. Curved trajectories

In contrast with the straight-line trajectory, in this situation there are lateral forces acting on the vehicle and affecting its manoeuvrability. As before, there will be the forces of the engine, the air resistance and the braking force (sometimes). In this case, they will act on the longitudinal axis on the rear wheels, but on the front ones they will take a certain degree offset orientation.

In order to study the behaviour of the car while cornering, it's important to define a vehicle coordinate system in which we specify the orientation of the x, y, z axis and the different angles assigned to them. These angles are the roll angle ($\varphi$) for the x-axis, the pitch angle ($\theta$) for the y-axis, and the yaw angle (ψ) for the z-axis.

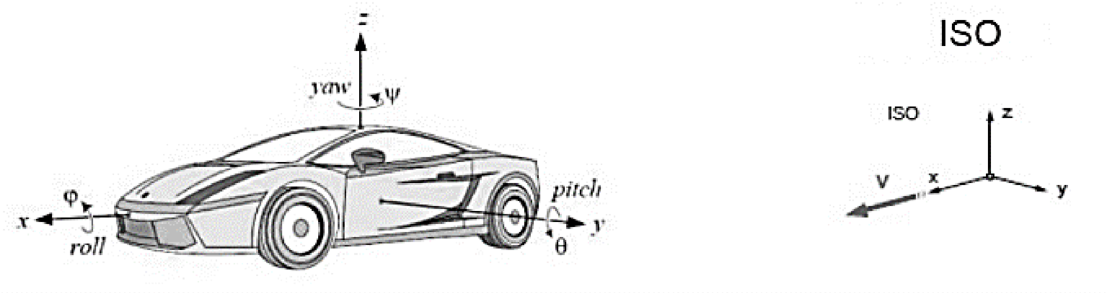


*Figure 18 - ISO coordinate system* [Mechanical Auxiliar Systems notes, 2021]

This is a local coordinate system that allows us to study the behaviour of the vehicle within the global coordinate system. The angles $\varphi$, $\theta$, ψ, are defined with respect to the direction shift between the local system and the global system.

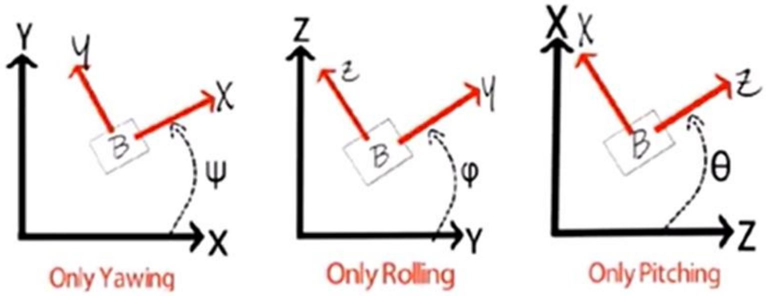


*Figure 19 - Relating local and global systems with the angles* [AutoMotorGarage, 2020]

### 3.3.1. Yaw and Roll motions

Yaw and Roll are two key lateral dynamic characteristics of the ground vehicles while cornering. The yaw angle describes the rotation of the car about the z-axis. As a consequence of the yaw motion, a side slip angle appears on the center of gravity ($\beta$) and on the front and rear wheels ($\beta_f, \beta_r$). This angle corresponds to the difference in orientation between the speed and the longitudinal axis of the car. We also define the angle between the axis just mentioned, and the wheel orientation ($\delta$).

When the car starts cornering, the yaw angle appears. Consecutively the velocity vector changes its orientation and creates the centripetal acceleration ($\ddot{y}$). This originates the centripetal force, which can be decomposed between the one that acts on the front wheels ($F_{yf}$) and the one that acts on the rear wheels ($F_{yr}$). For a car driving in a curve, the centripetal force corresponds to the frictional force between the tires and the road that pulls the vehicle towards the center of rotation.

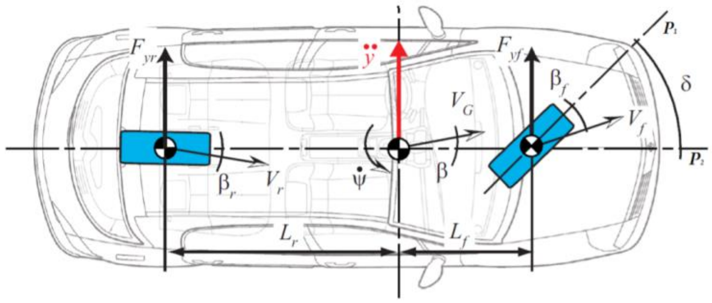

*Figure 20 - Basic configuration of car turning* [M. Ouahi, J. Stéphant, D. Meizel, 2010]

This system is subjected to an opposite reaction force which is the centrifugal force. This force acts on the vehicle pulling its mass away from the center of rotation. This leads to the emergence of a roll motion on the body and a displacement of the center of mass. Due to this phenomenon, there is an imbalance in the distribution of loads: an increase of the vertical force appears on the outer wheels, in comparison with the inner ones which have their vertical reactions reduced. In *Figure 21* the reaction force $F_{zo}$ will add up and $F_{zi}$ will diminish as the roll angle $\varphi$ increases.

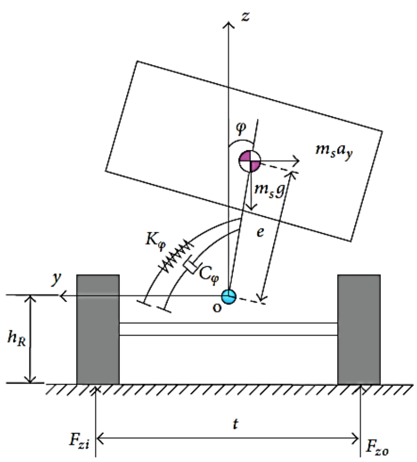

*Figure 21 - Basic configuration of car rolling* [Hindawi, 2013]

# 4. Diffuser configuration

As stated above, a diffuser main function is to lead the flow outward after being accelerated under the car by the flat bottom, creating an area of low pressure and, therefore, increasing the downforce and aerodynamic grip of the vehicle. Hence, it is important to see how a diffuser is shaped in order to obtain this outcome performance.

When the air flow collides with the front of the car, part of it heads towards the bottom and goes along between the flat floor and the ground. The flat bottom is vital in the generation of downforce as it is responsible to provide the so-called "floor effect". The diffuser is located at the end of the flat bottom, at the rear of the vehicle, and is the one in charge of controlling the air flow at its outlet and using the engine gases to generate pressure reductions.

There are different types of diffusers regarding its shape and the way they expand the flow from underneath the car to the rear. Nevertheless, all of them include a main element: the fences. All diffusers are equipped with vertical fences which can take different shapes. The main objective of the fences is to prevent the different types of flow that come from different areas of the car (the floor, the wheels, the exhaust system, etc.) from mixing.

Relying on the shape and the elements involved, we can tell apart between three different diffuser configurations: the conventional diffuser, the double diffuser and the blown diffuser.

## 4.1. Conventional diffuser

The conventional diffuser is the basic model of diffuser, which was first created and used until the appearance of the new improved models. This diffuser is modelled only with the fences, playing with its quantity, the distance between them and the geometric shape.

## 4.2. Double diffuser

With a double diffuser, the air can follow three different paths. In the first place, the one that circulates below the flat bottom can continue below the diffuser, passing along the fences, where it expands. Second, the air that circulates above the surface of the car, following its upper shape, ends up escaping from the car over the diffuser. Finally, thanks to the existence of some holes placed on the plain floor that exit above the fences, a certain volume of air seeps into the area delimited between the main diffuser and the elements of the rear impact structure. This generates extra downforce, not reachable by conventional diffusers.

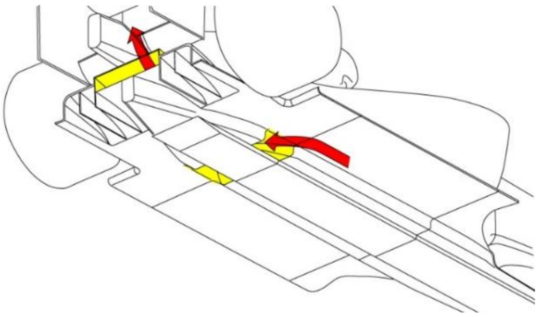

*Figure 22 - Double Diffuser and airflow path* [WordPress, 2011]

## 4.3. Blown diffuser

The blown diffuser operation principle is to reuse the gases that came out of the ducts exhaust by redirecting them towards the diffuser, and thus increasing the suction capacity of it. The system was much more complex than it seemed a priori since the gases blown depended on the intensity at which the motor was forced.

# 5. Design of a dynamic diffuser theoretical prototype

It is intended to design a theoretical prototype of a dynamic diffuser, which moves sideways throughout an established distance towards the left and the right of the longitudinal axis of the vehicle. We are going to use a conventional diffuser to carry out this study.

In order to accomplish this target, we need an additional functioning system which provides the diffuser with this transversal movement. Wherefore, after doing research on driving systems which provide linear motion displacements, an electro-hydraulic system has been adjudged to fulfil this purpose. This system incorporates a cylinder with a rod that will provide our diffuser with the linear motion we need. It will be attached to the car frame and also to the diffuser, so when the rod shifts, the diffuser will move with respect to the car body.

Moreover, in order to get a better performance of the dynamic diffuser, we don't want it to only move sideways in a perpendicular displacement with respect to the vehicle axis. We will get a sight better result if the linear motion provided by the rod of the cylinder is transferred to the diffuser as an angular movement. This will get a better orientation of the fences in the direction of the air flow.

## 5.1. Air flow path through the underbody

In a cornering condition, a car is subjected to a curved flow-field. If we switch to a reference frame in which the car is fixed but the air is moving, then the air has to follow a curved trajectory. When the car is in a mid-corner instant, the air at the front will be coming approximately from the direction of the inside front-wheel, while the air at the back will be going out through the inside rear-wheel.

In order to understand why it takes this behaviour, we are going to analyse the different components represented in *Figure 23*. In the picture on the left we can see:

- **White arrow**: instant velocity due to the side-slip
- **Curved blue arrow**: yaw moment (center of mass)
- **Green arrow**: additional velocity component induced by the yaw moment.
  - → The green arrows at the front of the car point in the opposite direction from the ones at the rear.
  - → The magnitude of the green arrows increases with distance from the center of mass.

The result, indicated by the red arrows, is a new component that has a different sign at the front and rear of the car, and its magnitude increases with distance from the centre of mass. Hence, the front of the car is rotating towards the inside of the corner, while the rear of the car is rotating away from it. The equivalent airflow direction (picture on the right) is obtained by reversing these arrows.

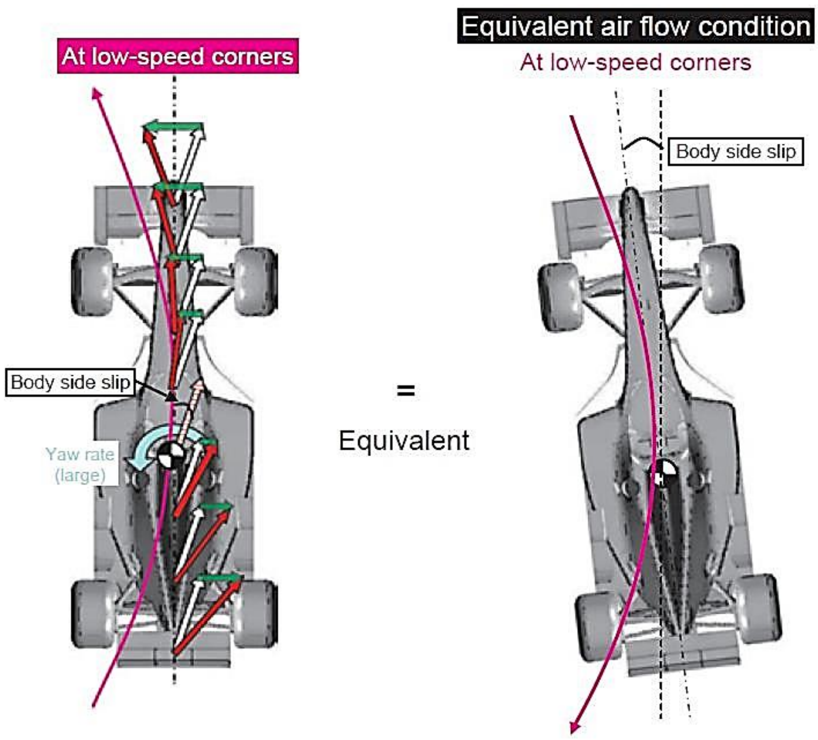


*Figure 23 - Curved-flow, side-slip and yaw-angle* [Honda R&D Technical Review, 2009]

## 5.2. Diffuser orientation

In order to adjust the performance of our diffuser to the curved airflow trajectory, in addition to the linear motion cylinder, we also need to incorporate a joint, which enables the diffuser to have a circular movement. By means of this incorporation, the fences of our diffuser will be oriented towards the outflow direction. In *Figure 24*, we can see a schematic representation of the performance of our dynamic diffuser. In the picture on the right, the cylinder is transferring a force to the left, this means the car is turning to the left as well.

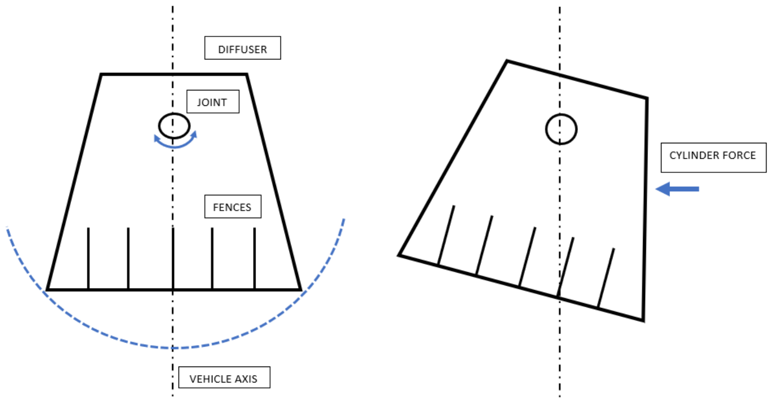


*Figure 24 – Diffuser conceptual representation* [own elaboration, 2021]

## 5.3. Electro-hydraulic system

Electro-hydraulic control systems are key components for industrial applications. They are used to replace the ordinary hydraulic systems, using actuators operated only by electrical power as an input. Therefore, this term covers all combinations of electronic signal processing with hydraulic drives.

With the development of automation, digital technology and communication technology, electro-hydraulic systems are becoming more digital, integrated and intelligent. We intend to make use of this modern developments to design an integrated electro-hydraulic control cylinder, which allows us to monitor the lateral movement of our diffuser.

In order to achieve this purpose, we draw upon the lateral acceleration sensor the car already possess, which measures the g-force from a turn and sends that information to the ECU. The ECU is in charge of translating this input data and sending it to the different actuators which act after it. In this case, we are interested in receiving this information at the electro-hydraulic control valve.

Owing to ECU sent data, our system will receive a certain electrical power input which will determine the magnitude of the cylinder displacement, and so, the diffuser's as well.

# 5.4. Electronic position control

## 5.4.1. Control systems

A control system consists of interconnected components, which allow us achieve a desired purpose. Control systems are based on the cause-effect relation, since they contain a process which, by introducing an input signal, responds with a specific output signal. Therefore, in a control system it is desired to find the input value that is going to provide the desired output.

There are two types of control systems: open-loop and close-loop. In open-loop systems there is no feedback of the output signal, for this reason we are going to use a close-loop control system.

In a closed-loop control system, a measurement of the actual process output is used to compare it with the reference input, determine the deviation and produce a control signal that reduces this deviation.

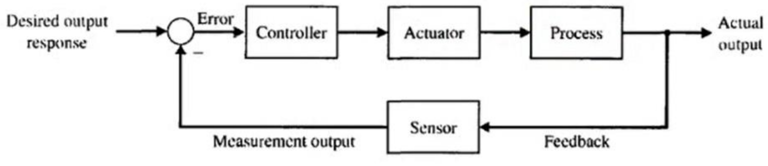


*Figure 25 – Closed-loop control system blocks diagram* [Dorf, 2011]

### 5.4.2. Hydraulic servo-systems

A servomechanism is defined in the Collins Dictionary as "an automatic control system in which the output is constantly or intermittently compared with the input through feedback so that the error or difference between the two quantities can be used to bring about the desired amount of control".

Hydraulic servo-systems are closed loop control systems involving electro-hydraulic elements (valves, actuators, etc.). They are associated with servo-hydraulics, which is the science that studies electrohydraulic regulation.

Hydraulic servo systems allow continuous monitoring and regulation of mechanical magnitudes, in our case the linear position of the cylinder.

They are made up of the following components:

- **Power unit**: supplies pressurized hydraulic fluid to the system. Usually includes a motor, a hydraulic bomb, filters and a heat exchanger.
- **Actuator**: the cylinder.
- **Control valve**: a continuous valve.

And also include these electronic elements:

- **Amplifier**: converts the control signal into a power signal, in order to control the continuous valve.
- **Control electronics**: contains the algorithm that allows to control the mechanical variable (position).
- **Sensor**: measures the variable to be controlled.

Hydraulic servo systems combine the advantages of hydraulics and control engineering to create the most powerful drive systems in modern industry.

The defining characteristics are:

✓ Excellent power-to-size ratio.

- ✓ Ability to apply forces indefinitely.
- ✓ High speed response.
- ✓ High tolerance to weathering due to the lubricating action of the hydraulic fluid.

## 5.4.3. Position control systems

The target of the position control is to move a load to a desired position in a precise way or, as in our situation, to move a load (the diffuser) along a desired linear trajectory of displacement. In order to get a linear position control, we use a hydraulic cylinder as an actuator. To get to control this actuator, a directional valve is required so we can inject the oil to make the cylinder go forward, backward and stop.

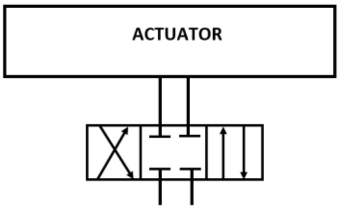


*Figure 26 - Directional valve and actuator* [own elaboration, 2021]

Next, we are going to build the block diagram for the position control (Fig. 27). To do this, we are going to represent each of the elements that compose it and interconnect them. These are the ones we have described above: power unit, actuator (cylinder), continuous valve, amplifier, sensor (attached to the cylinder rod), comparator and controller.

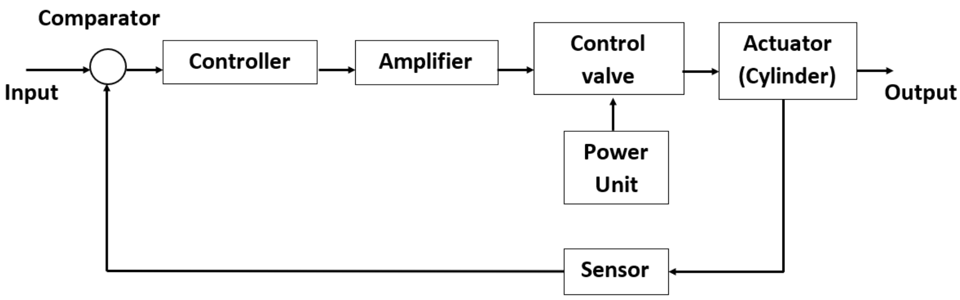


*Figure 27 - Position control block diagram* [own elaboration, 2021]

The power unit supplies the pressurized hydraulic fluid to the system, and it is the control valve that allows or not the crossing of this flow to the actuator. The input is responsible for providing the desired position value or values. Then we have the comparator, which is in charge of calculating the deviation between the input and the measured position (output). Then the controller or control algorithm is responsible for generating the control signal that is sent to the amplifier. This one takes the control signal and converts it into a power signal to control the continuous valve. The valve, depending on this signal, allows a specific amount of flow to pass to the actuator. The output variable is the linear position of the actuator rod, which will be measured by the sensor. Finally, the sensor sends that measured position signal to the comparator.

The initial electronic components (input, comparator, controller) are found implemented in the vehicle's control electronics.

# 5.5. Double-acting cylinder

The main characteristic of double-acting cylinders is they use hydraulic power (pumped oil) to extend and also to retract, in comparison with the single-acting cylinder which use the pumped pressure to extend, and then retracts by the weight of the load or by an inbuilt spring. A double acting cylinder alternates cycles of pressurized fluid to both sides of the piston and creates extend and retract forces to move the piston rod, permitting more control over the movement.

In order to get a uniform forward and backwards movement for our diffuser, which can be fully controlled by the amount of pumped oil inside the cylinder and without taking in consideration other variables (the mass of a load, the spring properties, etc.), the double-acting cylinder has come to be the best option for our application.

Moreover, a single-rod double-acting cylinder has been selected. Using a single-rod cylinder means both chambers don’t have the same free fluid storage space. For this reason, our electronic hydraulic system will take that in consideration in order to have the desired displacement in both directions.

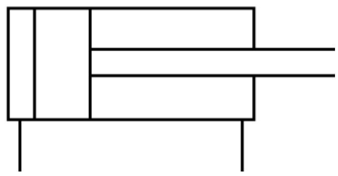

*Figure 28 - Single-rod double-acting cylinder* [Festo, 2021]

Using this kind of cylinder, we will have its body attached to the car frame, and the rod to the diffuser. The central position where the piston's head is exactly in the middle, would be the resting location when the car is not turning, so the diffuser is oriented towards the longitudinal axis. When the car starts to turn and the ECU sends an input voltage to our electro-hydraulic system, the oil will be pumped in one chamber or the other, depending if we want our diffuser to move to the left or the right, so the cylinder will extend or retract.

## 5.6. Modelling and simulation of the electro-hydraulic system

Once we have decided what kind of dynamic system will be used to make our diffuser move, and the different components of it, the next step is to study the behaviour of this complex system in a graphical programming environment.

This will be a primordial step in the design of our prototype, since it will allow us study the variables that define each component, and analyse the performance of the system depending on the different values we adjudge to these variables.

In order to develop this target, the add-on *Simscape Fluids* of MathWorks Simulink software has been used. Using this tool, we have been able to translate the previous block diagram represented in *Figure 27*, to the programming environment.

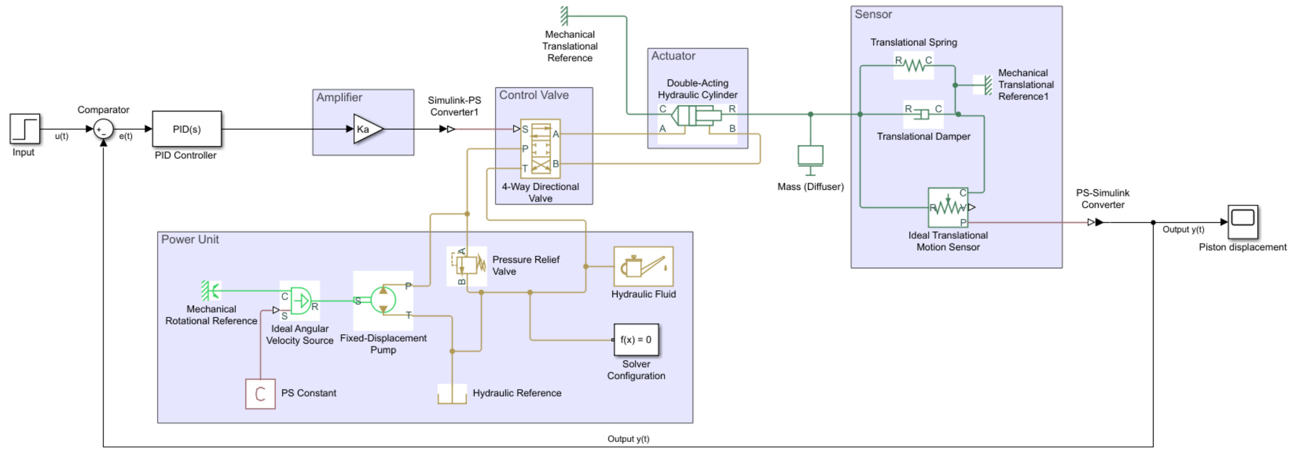


*Figure 29 – Simulink model of the electro-hydraulic system* [own elaboration, 2021]

As can be seen in *Figure 29*, the system consists on the same components previously mentioned and implemented in *Figure 27*. Next, we are going to analyse the most outstanding ensembles.

## 5.6.1. PID Controller

PID is an acronym that stands for Proportional, Integral, Derivative. It is used when is needed to keep a variable constant, or continuously control its variation. Essentially, it uses a control loop feedback to ensure the output wanted is what you will get. Simply, we put a setting in the controller and it will keep the output constant, based on feedback from some input, typically some kind of sensor.

A PID calculates the error by calculating the difference between the actual value and the desired value, and then sets the deciding parameters accordingly. This error is continuously being calculated until the process stops.

In order to analyse a little bit its inner working principle, we are going to analyse the block diagram developed in Simulink shown in *Figure 30*. On the PID Controller, the input (error) arrives to the P, I, D at the same time, and later their output is summed using an adder. This added output is applied to the system, in our case the Control Valve, after being processed through the amplifier.

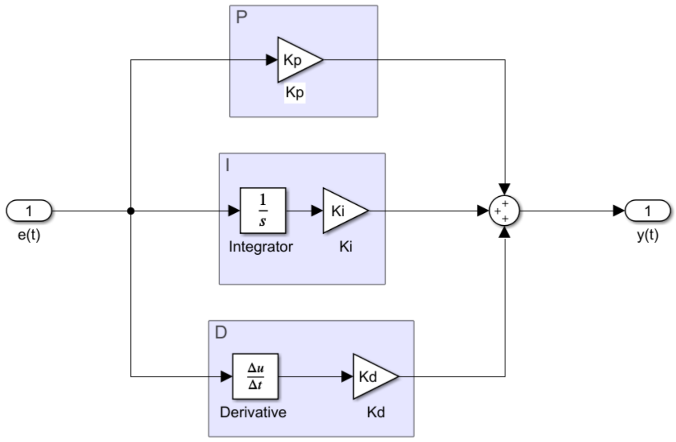


*Figure 30 - Block Diagram of PID controller in Simulink* [own elaboration, 2021]

First of all, the proportional component determines the ratio of output response according to the error signal (input). The output value produced by the proportional term is proportional to the existing error value. The error is multiplied by the coefficient "Kp" to tune the values. As the error decreases, the output we get from the proportional will decrease as well, until the error is zero so the system has the output desired.

The integral component sums the error term over time. The result is that even a small error term will cause the integral component to increase slowly. The integral response will continually increase over time unless the error is zero, so the effect is to drive the Steady-State error to zero and have a constant value of the integral response.

The derivative term causes the output to decrease if the process variable is increasing rapidly. It helps to improve the settling time of the system. The Derivative Response is highly sensitive to noise, which can make the system unstable, for this reason most practical control systems use very small derivative value or even zero (then we have a PI controller).

In our system, we have used the Simulink predefined block of a PID controller instead of using the ensemble on *Figure 30*. In *Figure 31* are shown the values we have assigned to the PID variables. As can be seen, the D is

0, so we technically are using a PI Controller. The filter is used to avoid high frequency gains.

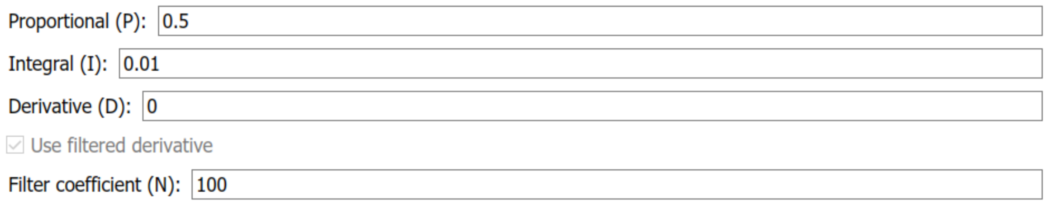


*Figure 31 - Simulink PID adjudged values* [own elaboration, 2021]

It is important to remark the fact that the values of the P and I are quite low. This is because we are working in meters and our cylinder will move between [0, 30] cm. So, we will define a maximum input of 0.3 to the initial step.

### 5.6.2. Power Unit

In our model of the hydraulic actuation system, the spool inside the valve controls the flow of pressure from a pump to either side of a hydraulic cylinder, which can extend and retract. A motor will drive the shaft of the pump, controlling the speed, and the control system (previously studied) will adjust the position of the valve.

The different components our power unit is equipped with are:

- **Pump**: we have selected a fixed displacement pump. This is because our system will work sporadically and it is really difficult it creates an over-pressured situation which provokes a use of the relieve valve and a loss of power for a long time.
- **Relieve Valve**: in the event that the system is over-pressurized, it is necessary to have this valve to divert fluid from the system.
- **Tank**: it represents an ideal hydraulic reference from where we obtain our fluid.
- **Ideal Angular Velocity Source**: it spins the pump at a fixed speed.
- **Hydraulic Fluid**: we specify the type of fluid.

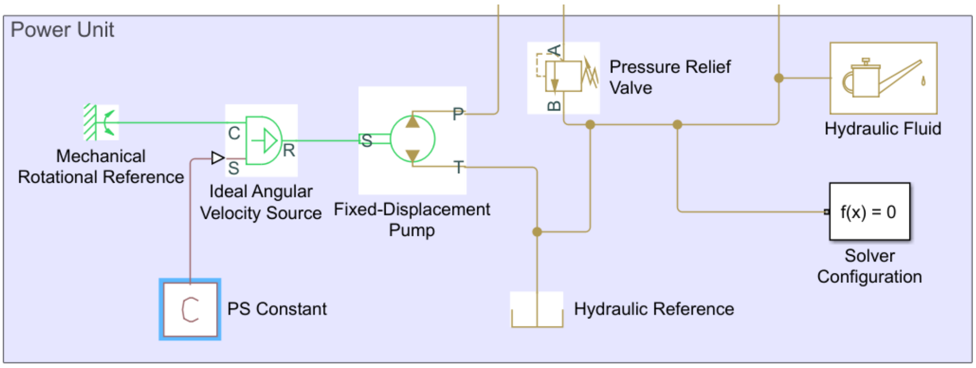


*Figure 32 - Power Unit ensemble* [own elaboration, 2021]

The other components of the ensemble which haven't been mentioned above, are structural non-relevant elements that are needed to make the system work.

## 5.6.3. Actuator

Our cylinder receives the fluid from the pump through the directional control valve (A and B). The cylinder has two connections (C and R) which represent the mechanical connections attached to it. The linkage C is connected to a point fixed in space, which corresponds to our car frame. The linkage R corresponds to the rod and it acts against a mechanical load, the diffuser (see *Figure 29*).

## 5.6.4. System response

Once we have modelled our system and we have studied its variables, we carry out a trail-error study until we get the response we deserve.

First of all, it is important to specify that our intention is to get a linear displacement of the rod in a range of [0, 30] cm. With this value, we can proportionate the diffuser with a linear displacement of [-15, 15] cm in the point of attachment with the rod. As we said before, our Simulink system is designed to work in meters, for this reason our input and output values will be inside the spectrum of [0, 0.3] m.

In *Figure 33* we can observe the range of displacement of the rod when we apply a sine-wave signal as an input.

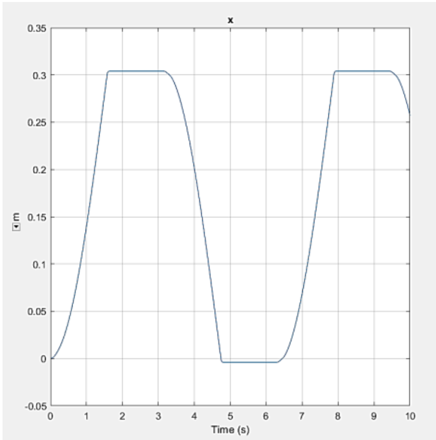


*Figure 33 - Range of displacement (x) of the rod* [own elaboration, 2021]

After doing this verification with the sine-wave input, we change it for a step. This step value would be the value of the position (x) of the rod our ECU has decided to send to the system after receiving a determined value of g-force form the sensor.

For example, if the ECU communicates a maximum displacement to the right of de diffuser, it will send an input of 0.3 m. In the other hand, if it wants our diffuser to be at its maximum position to the left, it will send an input of 0 m. If it wants it to remain in the resting position, the input would be 0.15 m.

In *Figure 34* we can see the input step signal (red) and the output (blue), and how this second one grows until matching its value to the first one. Once they have equalized, the signal is constantly being measured and corrected in order to remain (nearly) constant until a different input is received. In this example the input was 0.2.

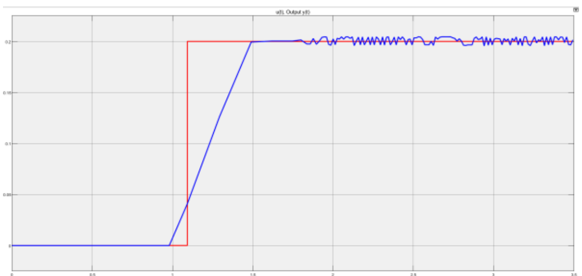

*Figure 34 - System response to an step input of 0.2* [own elaboration, 2021]

# 6. CAD design of the diffuser prototype

The final step that is needed to accomplish in this project, is the design of the theoretical prototype in a 3D CAD software. The goal of this task, is to have a visual reference of how the diffuser and the cylinder interact, and the different components that are needed to make this assembly work in a proper way.

The 3D model of the prototype has been developed in CATIA. CATIA is a computer-aided three-dimensional interactive application, developed by Dassault Systemes, a French software corporation.

In figures 35 and 36 the whole assembly is shown in two isometric views.

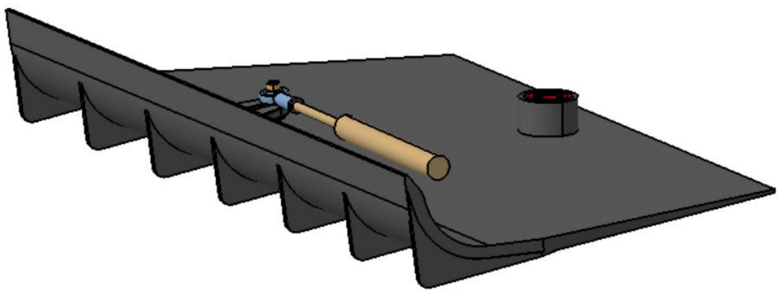

*Figure 35 – Front isometric view of the diffuser assembly prototype* [own elaboration, 2021]

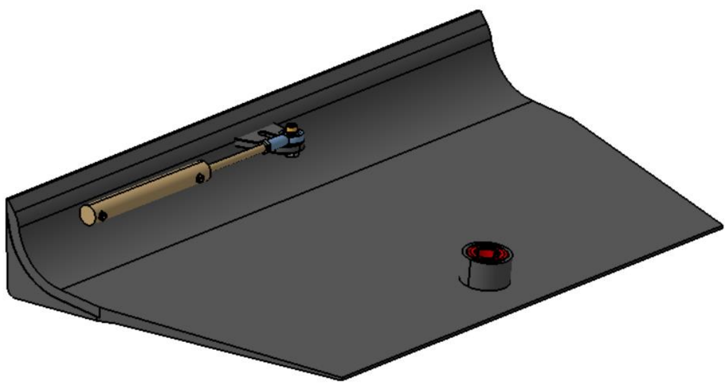

*Figure 36 – Rear isometric view of the diffuser assembly prototype* [own elaboration, 2021]

In the assembly we can see how the body of the diffuser is connected to the cylinder, and also where the join is set. In contemplation of having a better outlook of each one of the components that take part in this assembly, in *Figure 37* the exploded view is displayed. In this view, all the components are numbered, and then in Table 1 they are named and quantified.

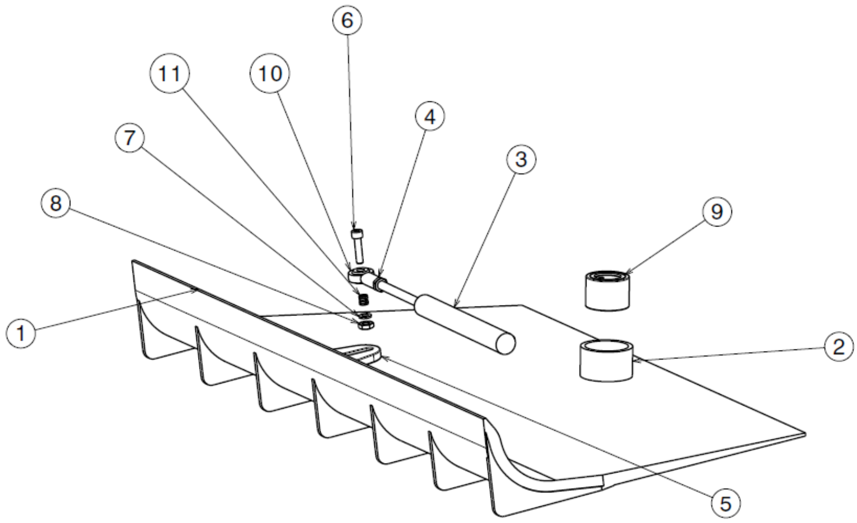


*Figure 37 - Exploded View* [own elaboration, 2021]

Table 1 – Assembly Component Descriptions

| Quantity | Part Number | Type | Number |
|---|---|---|---|
| 1 | Diffuser floor and fences | Part | 1 |
| 1 | Circular cavity | Part | 2 |
| 1 | Cylinder | Part | 3 |
| 1 | Tenon | Assembly | 10 |
| 1 | Locknut | Part | 4 |
| 1 | Ledge | Part | 5 |
| 1 | din_912-m6x1-25-10_9 | Part | 6 |
| 1 | Washer_iso_7089-6-a_200 | Part | 7 |
| 1 | Bolt | Part | 8 |
| 1 | Needle_bearing | Part | 9 |
| 1 | Needle Roller Bearing | Assembly | 11 |

## 6.1. Diffuser design

The diffuser body consists in three remarkable parts: the floor with the fences (1), the circular cavity (2) and the ledge (5). The floor has a certain angle of inclination, in order to use the Bernoulli Effect to create downforce and reduce the drag, as has been student in section 2.6.

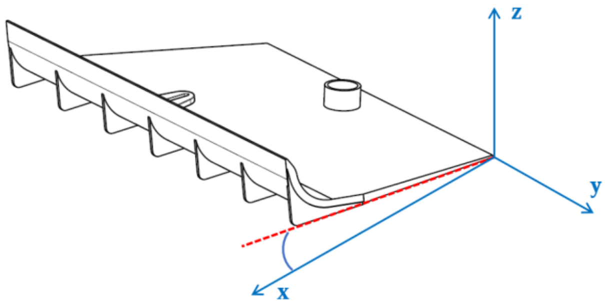


*Figure 38 - Diffuser angle respect the x-axis* [own elaboration, 2022]

The circular cavity is designed with the purpose of keeping in its interior a needle bearing (9). This bearing has a cylindrical hole in its central axis, which will embrace a cylindrical shaft from the car frame. A manufactured needle bearing has been chosen using the webpage TraceParts 3D Design Library, which includes standard supplier-certified 3D models. We have selected the bearing shown in *Figure 39* from the manufacturer Schaeffler Technologies AG & Co. KG. In Table 2 we can see the dimensions of the model used in the prototype. Using this bearing will enable an angular rotation of the diffuser around the car frame shaft. The shaft is static and the diffuser has a circular movement around it. The bearing prevents the friction between the two interacting surfaces.

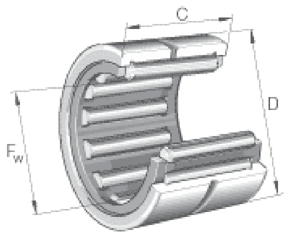


*Figure 39 - Needle Bearing dimensions* [TraceParts, 2019]

Table 2 – Needle Bearing Dimensions

| | |
|---|---|
| **C** | 72 mm |
| **D** | 100 mm |
| **$F_W$** | 60 mm |

The ledge is a prominence which stands out from the body of the diffuser. It is designed to be the point of interaction between the diffuser and the cylinder. It presents a grooved surface which enables the pass of a bolt throughout the opening. The bolt is the element in charge of transferring the liner movement of the cylinder to the diffuser. In section 6.3 the joining between these two elements is explained.

## 6.2. Cylinder

As the prototype designed is intended to be as realistic as possible, a double-acting cylinder (3) has been selected. The webpage TraceParts 3D Design Library has been used to choose our actuator. We have selected a double-acting cylinder from ANR Engineering. It has a stroke of 30 cm, in order to be in accordance with the control system we have previously designed in Simulink. The end of the rod is threaded, and we have used it to incorporate a tenon (10) with the magnitudes we need, in order to ensemble the actuator with the diffuser.

## 6.3. Cylinder-diffuser assembly

First of all, in contemplation of understanding the different components that take part in this joining, it is necessary to comprehend how the interaction of the cylinder and the diffuser works. We know the rod of the diffuser has a linear displacement. This displacement will be transmitted as an angular motion to the diffuser, due to the joint composed by the circular cavity (2) and the needle bearing (9). The result of this angular movement, is that the whole body of the diffuser moves along a determined angle instead of a linear distance, including the point where the cylinder and the diffuser connect.

In *Figure 40* we can see in red a representation of the range of action of cylinder rod. In green, the circular displacement of the upper part of the ledge gap. As can be observed, when the diffuser is in its central position, the rod of the cylinder would meet with the ledge at its lower part of the gap. Instead, when the rod is extended, it would be placed at the upper part of the gap. At this same position, the ledge would be inclined taking a diagonal orientation. This is the reason why an extended gap is needed instead of a simple hole.

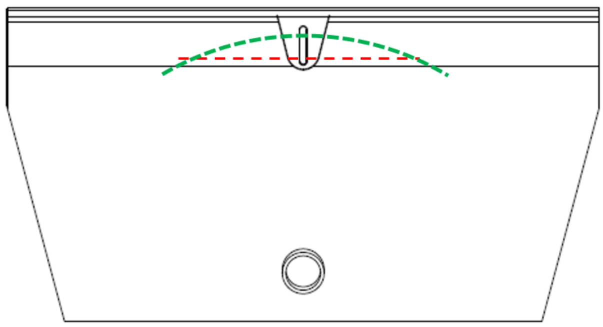

*Figure 40 - Rod and Ledge displacement representations [*own elaboration, 2021]

We use trigonometry to find the required length of the ledge gap. If we consider in *Figure 41* that **c** is half the stroke of the cylinder, and we have designed the prototype to have a distance from the ledge until the joint of 625.6 mm, we can calculate the values of **R** and **α**.

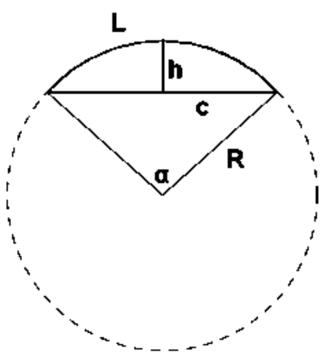


*Figure 41 - Trigonometric variables*
[calculo.cc, 2021]

Table 3 - Trigonometric Values

| | |
|---|---|
| **c** | 150 mm |
| **R** | 643.3 mm |
| **α** | 27º |
| **h** | ? |

In order to find **h**, which corresponds to the gap length, we use the following trigonometric relation:

$$h = R \cdot \left(1 - cos\frac{\alpha}{2}\right)$$

$$h = 17.8\,mm$$

Finally, to obtain a good performance in this operation, we use the following components:

- **Tenon**: it is threaded at one end to the cylinder rod, and at the other end it incorporates a hole with a bearing that allows rotational movements around the axis of the hole.
- **Bolt**: is the main component which connects the rod with the ledge. It goes through the tenon and the ledge gap, and then is fixed using a locknut.
- **Needle roller bearing**: it prevents the friction of the surfaces of the bolt and the ledge when the bolt moves along the gap (*Figure 43*).

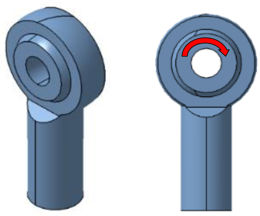

*Figure 43 – Tenon* [own elaboration, 2021]

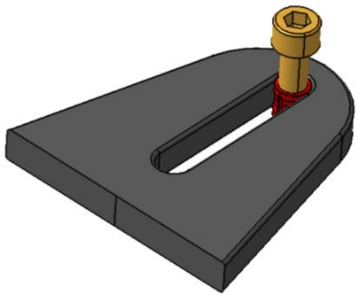

*Figure 42 – Ledge, bolt and bearing* [own elaboration, 2021]

In addition to this components, other elements are needed. In *Figure 44*, the cylinder along with all the components that take part in its joining to the ledge, are shown using an exploded view.

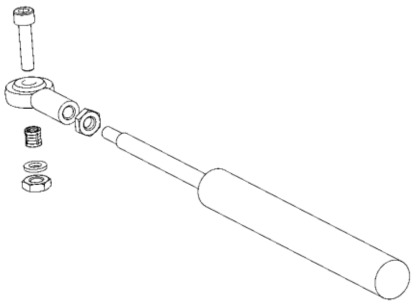

*Figure 44 – Exploded View of the cylinder and other components* [own elaboration, 2021]

# 7. Conclusions

This project has pondered the operation of car diffusers from the reason of its existence, until an improved still unfeatured version of it. Diffusers are nowadays one of the most relevant components of racing cars, thereupon they are constantly being modified and improved by aerodynamic engineers. At the starting point of this project, while doing some research on dynamic diffusers, I became aware of the fact that the main scope of this study had never been performed. Already existed investigations on developing dynamic diffusers that extended and retracted linearly towards the bottom of the car, but the aim of making it move transversely had never been propound.

The first objective was to understand how the airflow behaves when it gets through a car body, and more specifically along its underbody. It was essential to elaborate on the emergence of the downforce and drag forces, as they are the direct responsible of the diffuser implementation. Likewise, the understanding of the dynamical and kinematical behaviour of a vehicle and the resultant forces and motions when submitted in a cornering condition, where crucial to disclose why the dynamic diffuser was a reasonable new viewpoint for this existing static element.

Once the foundations were well settled, the first step was to contemplate which was the best course of action for our prototype, in order to get the best performance to obtain the higher downforce and the lower drag possible. Concerning this target, an analysis of the airflow trajectory along the underbody of the car while cornering was done, and the outcome was a curved flow field generated due to the centripetal force and the yaw motion. Using this knowledge, it was decided the fences of the prototype would front-face the outcoming airflow, owing an angular motion of the diffuser body. In order to do so, an approach was made proposing a hydraulic cylinder to provide linear motion, and a joint to convert it to an angular movement.

After adjudging the theoretical scheme of the prototype, the following step was to proffer a valid functioning system, which would be manufactural and functional. In order to accomplish this target, a servo-hydraulic system was developed. A study was made concerning the type of electronic system, hydraulic valve and actuator that would act on it. Next, this hydraulic system was developed using the interface *Simscape Fluids* from Simulink. By using this software, the final result was a functional hydraulic servo-system with a 30 cm stroke cylinder using a PI controller.

The ultimate step was to create a 3D CAD model which displayed the different elements needed to create the prototype. It was developed using CATIA from Dassault Systemes, and it was helpful to clarify how the assembly had to be performed in order to make the diffuser move in the desired way.

While performing the different sections of this project, I came across some complications. The first one was realising the diffuser needed to have an angular movement instead of a linear one, which was the initial idea. Next, it was difficult to choose which was the best cylinder, since in the first place I chose a double-rod double acting cylinder, and then I had to change it to a single-rode one, as the first one would make the care frame move. Later, while deciding the cylinder's dimensions, I designed it in Simulink to have a 50 cm stroke, but then in CATIA I realised it was way too long so I had to reduce it to 30 cm. The last complication I encountered was the ledge design in CATIA, because at first I designed it to have a cylindrical hole, but later I realized this would cause trouble due to the situation explained in *Figure 40*, so I redesigned it to have an extended gap.

Finally, it is important to catch sight on the different points developed in the project, regarding the initial objectives settled to be accomplished. These objectives were:

- ✓ Acknowledge in detail the dynamics, aerodynamics and kinematics on a car.

- ✓ Understand the role of a diffuser in a car.
- ✓ Get to know the different types of diffusers and its differences.
- ✓ Study and select a way of evolving the diffuser in order to getting a lateral dynamic performance.
- ✓ Explain the obtained final result of the dynamic diffuser.
- ✓ Developing a simplified prototype in a CAD software.

As can be seen, all of them have been accomplished throughout the development of this project. At this point, it remains open to be studied how this new designed prototype of a dynamic diffuser affects the vertical loads that appear in the rear axle of the car. The analysis of this aspect would be part of the aerodynamical analysis of the prototype, using a Computational Fluid Dynamics (CFD) software.

# Bibliography


Kang, Jun, S. O., Park, H. I., Song, K. S., Kee, J. D., Kim, K. H., & Lee, D. H. (2012). *Actively translating a rear diffuser device for the aerodynamic drag reduction of a passenger car.* International Journal of Automotive Technology

Fluids™, M. S. (2016, July 26). *MATLAB*. Retrieved fromhttps://www.youtube.com/watch?v=9LUuI9EIuAM

Hall, N. (2021, May 13). *NASA Official*. Retrieved from https://www.grc.nasa.gov/www/k-12/airplane/boundlay.html

He, D. T. (2017). *Aerodynamic analysis of an active rear split spoiler.* Oshawa, Ontario, L1H 7K4, Canada: University of Ontario Institute of Technology.

TecknoMechanics. (2020, March). Retrieved from https://www.youtube.com/watch?v=hmz1h5fk2bI

Hu, Zhang, R., Ye, J., Yan, X., & Zhao, Z. (2011*). Influence of Different Diffuser Angle on Sedan's Aerodynamic Characteristics*. Physics Procedia, 22, 239–245.

Porcar Galán. (2020). *Study of the functioning and importance of diffusers in Formula 1 cars*. Universitat Politècnica de Catalunya.

Hunsaker, Pope, O. D., Taylor, J. D., & Hodson, J. (2019). *Aerodynamic Centers of Arbitrary Airfoils Below Stall*. Journal of Aircraft, 56(6), 2158–2171.

Zhang, W. (2019). *Dynamics of Coupled Systems in High-Speed Railways: Theory and Practice*. Academic Press.

Schlichting (Deceased), & Gersten, K. (2017). *Boundary-Layer Theory by Hermann Schlichting (Deceased)*, Klaus Gersten. (9th ed. 2017.). Springer Berlin Heidelberg.

TecknoMechanics. (March de 2020). *Pneumatic Cylinder Working explained (Animation)*. https://www.youtube.com/watch?v=hmz1h5fk2bI

Uruba, V., & Knob, M. (2009). *Dynamics of a boundary layer separation*. *Engineering Mechanics*, *16*(1), 29-38.

Lögdberg, O. (2006). *Vortex generators and turbulent boundary layer separation control*.

*Zhifu, W., Yang, Z., Chaopeng, L., & Jun, F. (2016). Research on straight line stability control strategy of four wheel drive vehicle based on the sliding mode* *variable structure control and optimization algorithm*. Energy Procedia, *104*, 342-347.

Cao, J., Jing, L., Guo, K., & Yu, F. (2013*). Study on integrated control of vehicle yaw and rollover stability using nonlinear prediction model.* Mathematical Problems in Engineering, 2013.

Saber, F., Ouahi, M., & Saka, A. (2015, December). *Vehicle dynamics and steering angle estimation using a virtual senso*r. In *Xème Conférence Internationale: Conception et Production Intégrées*.

Rajamani, R., Piyabongkarn, D., Tsourapas, V., & Lew, J. Y. (2011). *Parameter and state estimation in vehicle roll dynamics.* IEEE Transactions on Intelligent Transportation Systems, *12*(4), 1558-1567.

López Vázquez, A. (2016). *Elements i dispositius aerodinàmics en la formula 1: EL DIFUSOR* (Bachelor's thesis, Universitat Politècnica de Catalunya).

Chiang, M. H. (2011). *The velocity control of an electro-hydraulic displacement-controlled system using adaptive fuzzy controller with self-tuning fuzzy sliding mode compensation.* Asian Journal of Control, 13(4), 492-504.

De Schampheleire, S. (2017). *Natural convection in classical heat sinks and open-cell metal foam* (Doctoral dissertation, Ghent University).

Xu, B., Shen, J., Liu, S., Su, Q., & Zhang, J. (2020). *Research and development of electro-hydraulic control valves oriented to industry* 4.0: a review. *Chinese Journal of Mechanical Engineering*, *33*(1), 1-20.

Xue, C., Zhu, H., & Yu, B. (2012). *Modeling and Simulation of Parameter Self-Tuning Fuzzy PID Controller for DC Motor Speed Control System*. In *Applied Mechanics and Materials* (Vol. 195, pp. 1003-1007). Trans Tech Publications Ltd.

Ogawa, A., Yano, S., Mashio, S., Takiguchi, T., Nakamura, S., & Shingai, M. (2009*). Development Methodologies for Formula One Aerodynamics*. Honda R&D Technical Review, F1.

Fluids™, M. S. (26 de July de 2016). *MATLAB*. Obtenido de https://www.youtube.com/watch?v=9LUuI9EIuAM