

Review

The diffuser of a racing car is responsible for collecting all the air flowing under the car and drive it to the outside through the rear part of it. It is a characteristic element that, despite it is true that at first glance it maintains certain similarities in all automobile modalities, it becomes a very particular piece according to the nature of each competition. Any slight modification or variation of the design parameters can make what was considered a simple car a race winner and it is for this reason that this report lays out a very instructional strategy that an aerodynamics engineer could perfectly follow when trying to accomplish the goal of ensuring that the air stays stuck to the bottom of the car and never detaches, which is one of the key components to build a successful diffuser.

We are used to focus on which are the problems and what the final solutions to this problems are, but it will be worth it to see and navigate through all the steps that we have to take to go over a path that might seem simple at first. The report that follows is based on one hand on theoretical aerodynamic definitions, concepts and principles to introduce the reader in the field and help him get a full understanding of the work. On the other hand it advances to the detail of the iterative methodology and computer programming used to study the aerodynamic behavior of the diffuser. To contextualize, Formula 3 competition has been chosen. There are a lot of racing car competitions where a diffuser is used but F3 will allow us to see the more important characteristics of this part of a vehicle and at the same time will make the study feasible as its design won't be as complicated as the one we would require for an F1 car for example letting us focus on the basic but most important settings of a diffuser.

Summary

REVIEW	1
SUMMARY	2
1. GLOSSARY	5
2. PREFACE	7
2.1. Motivation.....	7
3. INTRODUCTION	9
3.1. Goals of the project.....	9
3.2. Scope of the project.....	10
4. AERODYNAMICS	11
4.1. History of aerodynamics	11
4.2. Introduction to aerodynamics	14
4.3. Aerodynamics most important principles and equations.....	15
4.3.1. Continuity equation	16
4.3.2. Bernoulli's equation	16
4.3.3. Venturi effect	18
4.3.4. Coanda Effect.....	19
4.4. Boundary layer.....	20
4.4.1. Boundary layer separation	23
4.5. Vortex.....	25
4.6. Aerodynamic forces	26
4.6.1. Downforce.....	27
4.6.2. Drag force.....	29
5. FORMULA 3 INTRODUCTION	32
6. DIFFUSER	33
6.1. Performance parameters of a diffuser:.....	35
6.2. Downforce mechanism on a diffuser	35
7. FLOW SIMULATION PROGRAM	38
7.1. ANSYS – Computational Fluid Dynamics	38
7.2. CFX – Software	39
7.3. CFX process	41
7.3.1. Creating the Geometry/Mesh.....	41

7.3.2.	Defining the Physics of the Model.....	43
7.3.3.	Solving the CFD Problem	44
7.3.4.	Visualizing the Results in the Post-processor	45
8.	CFX SIMULATION – FLOW ANALYSIS OF THE DIFFUSER	46
8.1.	First Analysis	46
8.1.1.	Geometry	46
8.1.2.	CFX Analysis	47
8.1.2.1.	Geometry	47
8.1.2.2.	Mesh	47
8.1.2.3.	Boundary conditions.....	48
8.1.2.4.	Run	49
8.1.2.5.	Results	50
8.2.	Second Analysis.....	52
8.2.1.	Geometry	52
8.2.2.	Mesh, boundary conditions and run.....	53
8.2.3.	Results.....	53
8.3.	Third Analysis.....	55
8.3.1.	Geometry	55
8.3.2.	Results.....	55
8.4.	Fourth Analysis.....	58
8.4.1.	Geometry	58
8.4.2.	Results.....	58
8.5.	Fifth Analysis	61
8.5.1.	Geometry	61
8.5.2.	Results.....	61
8.6.	Possible ways to further improve on the design of this diffuser.....	62
8.7.	The importance of the mesh and Turbulence Model.....	63
8.7.1.	Mesh importance	63
8.7.2.	Turbulence models	64
8.8.	Comparative analysis and numerical conclusions	67
9.	ENVIRONMENTAL IMPACT	69
10.	BUDGET OF THE PROJECT	71
11.	PROJECT PLANNING	74
	CONCLUSIONS	75

ACKNOWLEDGEMENTS	77
-------------------------	-----------

BIBLIOGRAPHY	78
---------------------	-----------

Bibliographic References	78
--------------------------------	----

Complementary bibliography	78
----------------------------------	----

1. Glossary

Viscosity: Viscosity is a measure of a fluid's resistance to flow. It describes the internal friction of a moving fluid. A fluid with large viscosity resists motion because its molecular makeup gives it a lot of internal friction. A fluid with low viscosity flows easily because its molecular makeup results in very little friction when it is in motion.

Compressibility: A measure of how easily a gas can be forced into a smaller volume.

Turbulent flow: type of fluid flow in which the fluid undergoes irregular fluctuations, or mixing. In this type of flow the speed of the fluid at a point is continuously undergoing changes in both magnitude and direction.

Laminar flow: the flow of a viscous fluid in which particles of the fluid move in parallel layers, each of which has a constant velocity but is in motion relative to its neighboring layers

Streamline: is a line that is tangential to the instantaneous velocity direction (velocity is a vector, and it has a magnitude and a direction)

Streakline: A streakline is the line traced out by all the particles that passed through a particular point at some earlier time

Reynolds: The Reynolds number is an experimental number used in fluid flow to predict the flow velocity at which turbulence will occur. It is described as the ratio of inertial forces to viscous forces and helps us determine if a flow is laminar or turbulent.

Sidepods: Aerodynamic device to improve airflow between front and rear wheels on open wheel racing car which also covers ancillary equipment within car, most often water radiators which are air cooled by ram scoops at the open front of the sidepods.

Bargeboards: They are curved vertical planes situated longitudinally, between the front wheels and the sidepods, held away from the chassis at the front on struts or other connectors, and connecting to the sidepods or extensions of the floor at the rear.

Compressible flow: flow having significant changes in fluid density.

Incompressible flow: flow in which the fluid density is constant.

Subsonic flow: directed motion of a fluid medium in which the velocity is less than that of sound in the medium throughout the region under consideration.

Supersonic flow: directed motion of a fluid medium in which the velocity is greater than that

of sound in the medium throughout the region under consideration.

Ride Height: is the height of clearance the car has between the bottom of the car and the road. The ride height has an impact on the car's center of gravity, and thus on its behavior when cornering or braking.

Mach number: is a dimensionless quantity representing the ratio of flow velocity past a boundary to the local speed of sound. It helps us determine if a flow is compressible or incompressible, subsonic or supersonic.

Oversteer: handling of an automotive vehicle that causes turns that are sharper than the driver intends because the rear wheels slide to the outside of the turn before the front wheels lose traction.

Understeer: a handling characteristic of an automotive vehicle that causes it to turn less sharply than the driver intends because the front wheels slide to the outside of the turn before the rear wheels lose traction.

2. Preface

2.1. Motivation

Often students choose a certain kind of project because they think it will not take much to carry it out, others do it as an obligation and just because they need it to finally get a degree while others choose a topic because they already know a lot about it and want to turn that knowledge into a great project. These are none of the reasons that got me to do this project.

Since I was a young kid I have sat every Sunday with my brother and father to watch Formula 1 races on television. I have always been really into car competitions but I used to enjoy them from a spectator standpoint, just because all those cars competing against each other was thrilling and exciting for me. But the time came I started to know more about all the sciences behind those cars, clearly it was not just a powerful engine that allows racing cars take turns at such high velocities.

I believe I can say that curiosity and intrigue and the enthusiasm to know more were my main reasons to start this project, it was not enough to know that a F1 car could all it does because of the aerodynamic components on it, I wanted to know how it works. So this project is the result of a future engineer that decided to stop wondering about all he did not know about racing cars and planned to give answers to all the questions.

I do not intend to work on this field in a few years but I will not say it can never happen, I just want to highlight the fact that I chose this topic because I saw a great opportunity to learn about something I like and at the same time recognize some credits for it.

3. Introduction

The racing cars competition has always been in the forefront of technological developments and, aside of always being one step ahead in terms of applications on vehicles, has always relied on the best techniques to get good results as fast as possible and before the opponent does. The process of obtaining a good result before the rivals is synonymous of time, and if we talk about time and aerodynamics is necessary to mention the wind tunnel and its importance in knowing if the results obtained will be certainly satisfactory. CFD, or computational fluid dynamics, is the tool that has also allowed us to reach this point, causing large structures of engineering experts to be built in this specific area and who are capable of getting very accurate results in a cheaper way.

The methodology followed by CFD studies is very clear but even within the world of engineering is unknown to many people. It's worth getting a touch of the typical problems that arise in these analyzes in order to value the work that lies behind all the aerodynamic components of a racing car that we normally see on television on Sunday's midday.

This project presents the aerodynamic diffuser as a key element on the behavior of a F3. An element that often goes unnoticed simply because it is actually difficult to realize it's there. The fact that it is hidden under the car does not help to make it visible as a front or rear wing is although in recent years there have been Formula 1 teams that have placed the focus of all the criticism on the diffuser due to the advantages it gave when a double diffuser was used (using the air circulating along the sides of the car to create an even bigger pressure gradient).

Formula 1 is the automobile discipline that has popularized the concept of aerodynamics, therefore we must thank this sport because it has played a major role in making this matter well known. In return we must say that it is not the only modality that uses air to make the car faster on a circuit, and it is also not true that only diffusers are mounted on these cars as many people might think. There are many competitions like Formula 3, Formula E or Nascar that use the very same techniques and concepts to make their cars faster and faster but show different designs due to the fact that every competition has its different rules and car specifications.

3.1. Goals of the project

The main objective of this study is for the author to be able to capture and express everything learnt through extended research and for the reader to be able to absorb all that information presented with ease, which means the author has done a good job on

elaborating with clarity an accuracy.

Some other goals to be achieved are related with the computational analysis and simulation. The author has to be successful on finding the appropriated mesh for the geometry and the final result of the different parameters studied to converge and have minimal error. It is also needed to distinguish which of the models that the simulation program offers have to be selected in order to accomplish a good simulation of the real problem, and las but not least to be able to explain and understand the result obtained in accordance with what has been said or stablished previously in the theory sections.

And finally we can say that the ultimate goal is for the author to gain experience carrying out projects, getting to improve a third language such as English and also be ready to edit bigger and more extensive projects thanks to the knowledge provided by the realization of this one.

3.2. Scope of the project

As it has already been mentioned before, this project will focus on analyzing through a computational fluid dynamics software the behavior of one aerodynamic component of the many that exist on a Formula 3 racing car: the diffuser.

The simulation of this component will be carried out in 3 dimensions with the CFX software of ANSYS. Although a 2 dimensions analysis would have been enough to understand the basic mechanism of a diffuser, the 3D analysis will let us complete a more detailed report and study of how a diffuser works in a real car. This part is the most complicated part of the project and the one we will have to put more effort in order to obtain a successful result due to the fact that during the four grades of university there has not been a whole course dedicated to CFD. On the contrary, most of the theory required to understand the function of a diffuser and how it works has been covered on the course Fluid mechanics, and therefore, will represent a not so difficult part of the project.

4. Aerodynamics

Aerodynamics, literally “air in motion,” is the branch of the larger field of fluid dynamics that deals with the motion of air and other gaseous fluids. It concerns the forces that these gaseous fluids, and particularly air, exert on bodies moving through it. Without the science of aerodynamics, modern flight would be impossible.

During this part of the project we will take a quick look to the history of aerodynamics, review some of the basic knowledge of aerodynamics that early physicians and mathematicians used to achieve important breakthroughs in this field, deeply explain the most important principles and equations and define those clue concepts that will take parts in the understanding of the results obtained in the simulation.

4.1. History of aerodynamics

The word “aerodynamics” itself was not officially documented until 1837. However, the observation of fluids and their effect on objects can be traced back to the Greek philosopher Aristotle in 350 B.C. Aristotle conceived the notion that air has weight and observed that a body moving through a fluid encounters resistance.

Archimedes, another Greek philosopher, also has a place in the history of aerodynamics. A hundred years later, in 250 B.C., he presented his law of floating bodies that formed a basic principle of lighter-than-air vehicles. He stated that a fluid—either in a liquid or a gaseous form—is continuous, basically restating Aristotle's theory of a hundred years earlier. He comprehended that every point on the surface of a body immersed in a fluid was subject to some force due to the fluid. He stated that, in a fluid, “each part is always pressed by the whole weight of the column perpendicularly above it.” He observed that the pressure exerted on an object immersed in a fluid is directly proportional to its depth in the fluid. In other words, the deeper the object is in the fluid, the greater the pressure on it. Deep-sea divers, who have to accustom themselves to changes in pressure both on the way down into the sea and again on the way up to the surface, directly experience this phenomenon.

A direct proportional relationship means that if one part increases, the other will increase by the same factor. Physicists and mathematicians use the Greek letter alpha (α) to denote such a relationship. Applied to pressure and depth, if the depth of an object is doubled, the pressure exerted on the object would double as well. The opposite would also be true. As altitude increases (negative depth), pressure decreases. Archimedes also demonstrated that, in order to set a stagnant fluid in motion, the pressure on the fluid must be increased or decreased. The resultant movement will take place in the direction of the decreasing

pressure.

The next contribution to aerodynamics did not occur until the end of the 1400s. In 1490, the Italian painter, sculptor, and thinker Leonardo da Vinci began documenting his aerodynamic theories and ideas for flying machines in personal notebooks. An avid observer of birds and nature, he first believed that birds fly by flapping their wings, and thought that this motion would have to occur for manmade aircraft to rise. He later correctly concluded that the flapping of the wings created forward motion, and this forward motion allowed air to pass across the bird's wings to create lift. It was the movement of the wing relative to the air and the resulting reaction that produced the lift necessary to fly. As a result of his studies, he designed several ornithopters—machines that were intended to copy the action of a bird's wing with the muscle power being supplied by man. But these designs did not leave the drawing board. His other designs included those for the first helicopter and a parachute.

Leonardo noticed another phenomenon that would prove useful in the study of aerodynamics. He noticed that water in a river moved faster—at a greater velocity—where the river narrowed. In numerical terms, the area of a cross-section of a river multiplied by the velocity of the water flowing through that section equals the same number at any point in the river. This is known as the law of continuity ($\text{Area} \times \text{Velocity} = \text{constant}$ or $AV = \text{constant}$). The law of continuity demonstrates the conservation of mass, which is a fundamental principal in modern aerodynamics and will be explained further on in this project. He also observed the different ways in which a fluid flowed around an object—called a flow field.

Leonardo also stated that the aerodynamic results are the same if an object moves through the fluid at a given velocity or if the fluid flows past the object at rest at the same velocity. This became known as the “wind tunnel principal.” For example, the results are the same aerodynamically whether a runner moves at 10 miles per hour in calm air and if the wind is blowing at 10 miles per hour past a stationary person. He also determined that drag on an object is directly proportional to the area of the object. The greater the area of an object, the greater the drag. Further, Leonardo pointed out the benefits of streamlining as a way to reduce an object's drag.

However, Leonardo's notebooks were not discovered until centuries later, and his ideas remained unknown until the 19th century.

Scientists working in the 17th century contributed several theories relating to drag. The Italian mathematician and inventor Galileo Galilei built on Archimedes' work and discovered that the drag exerted on a body from a moving fluid is directly proportional to the density of the fluid. Density describes the mass of an object per unit volume. A very dense fluid produces more drag on objects passing through it than a less dense fluid. The density of air

(a fluid) changes with its distance from the Earth's surface, becoming less dense the farther it is above the Earth's surface and, as such, exerting less pressure. Thus, an object passing through air high above the Earth's surface will encounter less drag than the same object passing through air close to the Earth's surface.

In 1673, the French scientist Edme Mariotte demonstrated that drag is proportional to the square of the velocity of an object. Dutch mathematician Christiaan Huygens had been testing this theory since 1669 and published his results with the same conclusion in 1690. The English scientist and mathematician Sir Isaac Newton presented a derivation of the drag equation of a body in 1687.

In 1738, the Dutch scientist Daniel Bernoulli published his findings on the relationship between pressure and velocity in flowing fluids. Other scientists used his research as a foundation for further research. The French scientist Jean le Rond d'Alembert, an associate of Bernoulli's, introduced a model for fluid flows and an equation for the principle of the conservation of mass. He further presented the idea that velocity and acceleration can vary between different points in fluid flow.

Swiss mathematician Leonhard Euler, also an associate of Bernoulli, derived equations from Bernoulli's and d'Alembert's principles. The most famous of these became known as "Bernoulli's Principle." It states that, in a flowing fluid, as velocity increases, pressure decreases. This became a key concept for understanding how lift is created and will be deeply commented on the next chapter of this project. Euler also introduced equations for fluid flow, though at the time they could not be solved and applied.

Italian mathematician Joseph Lagrange and French mathematician Pierre-Simon Laplace studied Euler's findings and tried to solve his equations. In 1788, Lagrange introduced a new model for fluid flow as well as new equations for calculating velocity and pressure. In 1789, Laplace developed an equation that would help solve Euler's equations. It is still used in modern aerodynamics and physics. Laplace also successfully calculated the speed of sound.

In addition to these theoretical advancements, experiments in aerodynamics were also producing more practical results. In 1732, the French chemist Henri Pitot invented the Pitot tube, a device that enables the calculation of velocity at a point in a flowing fluid. This would help explain the behavior of fluid flow. The English engineer Benjamin Robins performed experiments in 1746 using a whirling arm device and a pendulum to measure drag at low and high speeds.

In 1759, the English engineer John Smeaton also used a whirling arm device to measure the drag exerted on a surface by moving air. He proposed the equation $D = k \cdot S \cdot u^2$, where D

is the drag, S is the surface area, u is the air velocity, and k is a constant, which Smeaton claimed was necessary in the equation. This constant became known as Smeaton's coefficient, and the value of this constant was debated for years. Those making the first attempts at flight, including the Wright brothers, used this coefficient. The French scientist Jean-Charles Borda published the results of his own whirling arm experiments in 1763. Borda verified and proposed modifications to current aerodynamic theories and was able to show the effect that the movement of one object had on another nearby object.

Sir George Cayley of England is generally recognized as the father of modern aerodynamics. He understood the basic forces acting on a wing and built a glider with a wing and a tail unit that flew successfully. He realized the importance of the wing angle of attack and that curved surfaces (camber) would produce more lift than flat ones. Stability in his designs came with the use of dihedral—an important concept still used today. He first made public the notion that a fixed-wing aircraft was possible in 1804 in his major publication, "On Aerial Navigation," which described the theoretical problems of flight.

The contributions of all of these thinkers, mathematicians, and scientists are part of the foundation of the science of aerodynamics. They paved the way for the aerodynamic developments that would occur during the nineteenth century, as well as for those who would eventually achieve heavier than air flight.

4.2. Introduction to aerodynamics

All physical objects on Earth are subject to gravity, but gravity is not the only force that tends to keep them pressed to the ground. The air itself, though it is invisible, operates in such a way as to prevent lift, much as a stone dropped into the water will eventually fall to the bottom. In fact, air behaves much like water, though the downward force is not as great due to the fact that air's pressure is much less than that of water. Yet both are media through which bodies travel, and air and water have much more in common with one another than either does with a vacuum.

Liquids such as water and gasses such as air are both subject to the principles of fluid dynamics, a set of laws that govern the motion of liquids and vapors when they come in contact with solid surfaces. In fact, there are few significant differences—for the purposes of the present discussion—between water and air with regard to their behavior in contact with solid surfaces.

When a person gets into a bathtub, the water level rises uniformly in response to the fact

that a solid object is taking up space. Similarly, air currents blow over the wings of a flying aircraft in such a way that they meet again more or less simultaneously at the trailing edge of the wing. In both cases, the medium adjusts for the intrusion of a solid object. Hence within the parameters of fluid dynamics, scientists typically use the term "fluid" uniformly, even when describing the movement of air.

The study of fluid dynamics in general, and of air flow in particular, brings with it an entire vocabulary. One of the first concepts of importance is viscosity, the internal friction in a fluid that makes it resistant to flow and resistant to objects flowing through it. As one might suspect, viscosity is a far greater factor with water than with air, the viscosity of which is less than two percent that of water. Nonetheless, near a solid surface—for example, the wing of an airplane—viscosity becomes a factor because air tends to stick to that surface.

Also significant are the related aspects of density and compressibility. At speeds below 220 MPH (354 km/h), the compressibility of air is not a significant factor in aerodynamic design. However, as air flow approaches the speed of sound—660 MPH (1,622 km/h)—compressibility becomes a significant factor. Likewise temperature increases greatly when airflow is supersonic, or faster than the speed of sound.

All objects in the air are subject to two types of airflow, laminar and turbulent. Laminar flow is smooth and regular, always moving at the same speed and in the same direction. This type of airflow is also known as streamlined flow, and under these conditions every particle of fluid that passes a particular point follows a path identical to all particles that passed that point earlier. This may be illustrated by imagining a stream flowing around a twig.

By contrast, in turbulent flow the air is subject to continual changes in speed and direction—as for instance when a stream flows over shoals of rocks. Whereas the mathematical model of laminar airflow is rather straightforward, conditions are much more complex in turbulent flow, which typically occurs in the presence either of obstacles or of high speeds.

4.3. Aerodynamics most important principles and equations

As explained in section XXX, during the last centuries, various contributions to flow modelling were achieved. The biggest and more important contributions of those are now explained in this section. This equations and principles allow us nowadays to study all kinds of fluid flows and build the equations to simulate and analyze them with computer programming.

4.3.1. Continuity equation

One of the fundamental principles used in the analysis of uniform flow is known as the Continuity of Flow. This principle is derived from the fact that mass is always conserved in fluid systems regardless of the pipeline complexity or direction of flow.

If steady flow exists in a channel and the principle of conservation of mass is applied to the system, there exists a continuity of flow, defined as: "The mean velocities at all cross sections having equal areas are then equal, and if the areas are not equal, the velocities are inversely proportional to the areas of the respective cross sections." Thus if the flow is constant in a reach of channel the product of the area and velocity will be the same for any two cross sections within that reach. Looking at the units of the product of area and velocity leads to the definition of flow rate. This is expressed in the Continuity Equation:

$$m = \rho_{i1} v_{i1} A_{i1} + \rho_{i2} v_{i2} A_{i2} + \dots + \rho_{in} v_{in} A_{in} \\ = \rho_{o1} v_{o1} A_{o1} + \rho_{o2} v_{o2} A_{o2} + \dots + \rho_{om} v_{om} A_{om} \quad \text{Eq. (1)}$$

Where:

m = mass flow rate (kg/s)

ρ = density (kg/m³)

v = speed (m/s)

A = area (m²)

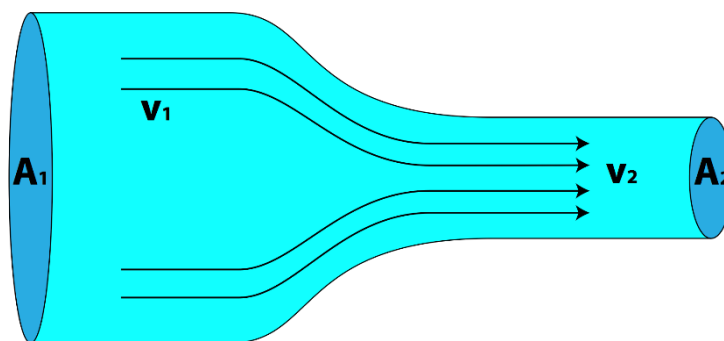


Figure1: Pipe of two different areas.

4.3.2. Bernoulli's equation

The simplest and most popular explanations of aerodynamic lift invoke the Bernoulli principle, which, in turn, is derived from Bernoulli's theorem. Investigated in the early 1700s by Daniel Bernoulli, his equation defines the physical laws upon which most aerodynamic rules exist. This now famous equation is absolutely fundamental to the study of airflows. Every attempt to improve the way a racing car pushes its way through molecules of air is governed by this natural relationship between fluid (gas or liquid) speed and pressure. There are several forms of Bernoulli's equation, three of which are discussed, in the succeeding paragraphs: flow along a single streamline, flow along many streamlines, and flow along an airfoil.

All three equations were derived using several assumptions, perhaps the most significant being that air density does not change with pressure (i.e. air remains incompressible). Therefore they can only be applied to subsonic situations.

Being that F3 cars travel much slower than Mach 1, these equations can be used to give very accurate results.

Low speed fluid flow along single or multiple streamlines is interpreted in Figure 1. The presumptions regarding the application of Bernoulli's equation to this scenario are listed in the figure. In this situation, there exists a relationship between velocity, density and pressure. As a single streamline of fluid flows through a tube with changing cross-sectional area, (for example an F3 air inlet), its velocity decreases from station one to two and its total pressure equals a constant. With multiple streamlines, the total pressure equals the same constant along each streamline. However, this is only the case if height differences between the streamlines are negligible. Otherwise, each streamline has a unique total pressure.

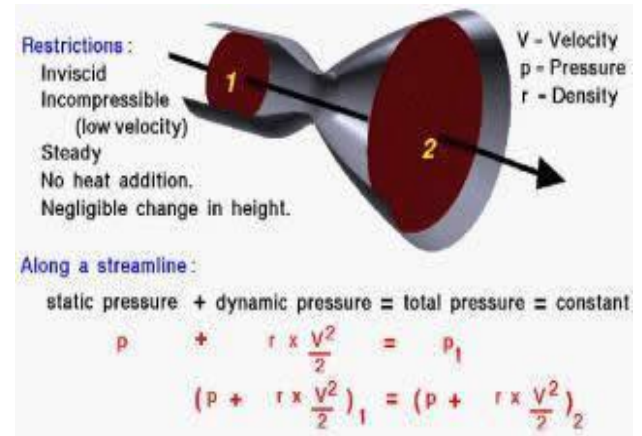


Figure 2: Bernoulli's principle.

Mathematical and pictorial explanation of Bernoulli's Equation as applied to fluid flow through a tube with changing cross-sectional area. As applied to flow along low speed airfoils (i.e. F3 downforce wings), airflow is incompressible and its density remains constant. Bernoulli's equation then reduces to a simple relation between velocity static pressure:

$$\frac{p}{\rho g} + \frac{V^2}{2g} + z = H = \text{constant}$$

\downarrow \downarrow \downarrow \downarrow
 pressure head velocity head static head total head

Eq. (2)

This equation implies that an increase in pressure must be accompanied by a decrease in velocity, and vice versa. Integrating the static pressure along the entire surface of an airfoil gives the total aerodynamic force on a body. Components of lift and drag can be determined by breaking this force down.

In order to discuss lift and downforce, it may be helpful to provide an additional explanation of the relationship that occurs with the above form of Bernoulli's equation. If a fluid flows around an object at different speeds, the slower moving fluid will exert more pressure on the

object than the faster moving fluid. The object will then be forced toward the faster moving fluid. A product of this event is either lift or downforce, each of which is dependent upon the positioning of the wing's longer chord length. Lift occurs when the longer chord length is upward and downforce occurs when it is downward.

4.3.3. Venturi effect

The Venturi effect is the phenomenon that occurs when a fluid that is flowing through a pipe is forced through a narrow section, resulting in a pressure decrease and a velocity increase. The effect is mathematically described through the Bernoulli equation and can be observed in both nature and industry. Many industry applications rely on this effect as they need to be able to predict a fluid's reaction when flowing through constricted piping.

The Venturi effect was named after Italian physicist, Giovanni Battista Venturi, who lived from 1746-1822. He is not only given credit for the effect's discovery, but is also credited with the inventions of the Venturi pump and tube. He later compiled and published many of Galileo's manuscripts and letters after being brought to Leonardo Da Vinci's attention.

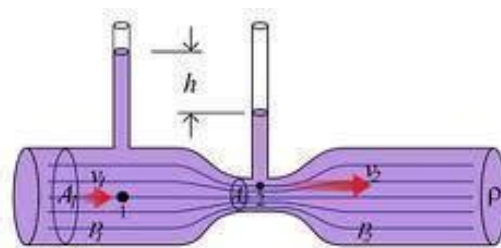


Figure 3: Venturi effect

The Venturi effect is similar to a jet effect, which is similar to the feeling one gets when the thumb is placed at the end of a garden hose with the water turned on. The water's velocity increases when the thumb is placed over the water. The pressure increases over the smaller surface area, however, the narrow flow then creates a vacuum in the water. The fluid's kinetic energy increase results in a pressure decrease, which the physics laws governing fluid dynamics explain. When the fluid reaches a choked flow point, the mass flow decreases, resulting in a decrease in downstream pressure. Bernoulli's equation can be used to calculate the theoretical pressure drop in a system that experiences the Venturi

effect. The equation is as follows: $\frac{\rho}{2} \cdot (v_f^2 - v_i^2)$ where ρ equals fluid density. The formula assumes that the fluid being measured cannot be compressed and maintains a consistent density.

4.3.4. Coanda Effect

A moving stream of fluid in contact with a curved surface will tend to follow the curvature of the surface rather than continue traveling in a straight line.

To get around air stream separation problem in airplane wing construction and in Formula 1, and increase the Coanda effect on wings, dual or more element or slot-gap wings are used, these allow for some of the high pressure flow from (in Formula 1 case) the upper surface of the wing to bleed to the lower surface of the next flap energizing the flow. This increases the speed of the flow under the wing, increasing downforce and reducing the boundary flow separation. If you look at a F1 rear wing few years ago on picture above, you can see this concept taken to the extreme, with multi-element wings creating huge amounts of downforce and little air stream separation even on the flaps with extremely high angle of attack.

The Coanda effect has important applications in various high-lift or high downforce devices on aircraft, or in our area of interest, on the racing car wing, where air moving over the wing can be "bent" using flaps over the curved surface of the top of the wing. The bending of the flow results in its acceleration and as a result of Bernoulli's principle pressure is decreased; aerodynamic lift or downforce is increased.

Notice how unlikely is to have a wing in flight with air flow only on one side. The Coanda effect only works in specific conditions where an isolated jet of fluid (or air) flows across a surface, a situation which is usually man-made. You don't find it much in nature. Just so you know, there is no Coanda lift on an airfoil. Coanda effect helps airstream to stay attached to the wing surface, but Bernoulli principle and difference in pressures are the reason why we have a lift or downforce.

Coanda effect is a balancing act between many factors, among them speed of fluids stream, pressure, molecular attraction, and a centrifugal effect if the surface is curved. Main trouble of the Coanda effect is the airstream becoming turbulent and detaching from the surface, that's how a wing stalls. Pull of surrounding air causes turbulence, drag from the surface and from the ambient air. It's a goal to pull as much as possible ambient air into the airstream, but the drag caused by the difference in velocity between the airstream and the surface is just a loss of energy. If the airstream gets turbulent and stops following the curved surface, there's no more low air pressure, no more thrust.

Since all applications of a Coanda effect involve a fluid object flowing over a solid one, the science behind this effect is known as fluid dynamics. Fluid dynamics represents and study the motion of liquids or gases. Studying this science can lead to many consequential

discoveries like the Coanda effect.

The Coanda effect is used on a modern Formula 1 car everywhere sometime to generate downforce, but sometime not for generating downforce directly, but for guiding and conditioning airflow in one place, as a means of maximizing downforce on other. For example, the rear of a modern Formula 1 car is tightly tapered between the rear wheels, like the neck and shoulders of a coke-bottle. By means of the Coanda effect, the air flowing along the flanks of the sidepods adheres to the contours at the rear, and the airflow here is accelerated, creating lower pressure. In itself, this transverse pressure differential on either side of the car cancels out, and creates no net force. However, the accelerated airflow between the rear wheels and over the top of the diffuser does raise the velocity of the air exiting the diffuser. In addition, bending air away from the rear tires contribute to reducing drag.

4.4. Boundary layer

As an object moves through a fluid, or as a fluid moves past an object, the molecules of the fluid near the object are disturbed and move around the object. Aerodynamic forces are generated between the fluid and the object. The magnitude of these forces depend on the shape of the object, the speed of the object, the mass of the fluid going by the object and on two other important properties of the fluid; the viscosity, or stickiness, and the compressibility, or springiness, of the fluid. To properly model these effects, aerospace engineers use similarity parameters which are ratios of these effects to other forces present in the problem. If two experiments have the same values for the similarity parameters, then the relative importance of the forces are being correctly modeled.

Aerodynamic forces depend in a complex way on the viscosity of the fluid. As the fluid moves past the object, the molecules right next to the surface stick to the surface. The molecules just above the surface are slowed down in their collisions with the molecules sticking to the surface. These molecules in turn slow down the flow just above them. The farther one moves away from the surface, the fewer the collisions affected by the object surface. This creates a thin layer of fluid near the surface in which the velocity changes from zero at the surface to the free stream value away from the surface. Engineers call this layer the boundary layer because it occurs on the boundary of the fluid.

The details of the flow within the boundary layer are very important for many problems in aerodynamics, including wing stall, the skin friction drag on an object, and the heat transfer that occurs in high speed flight.

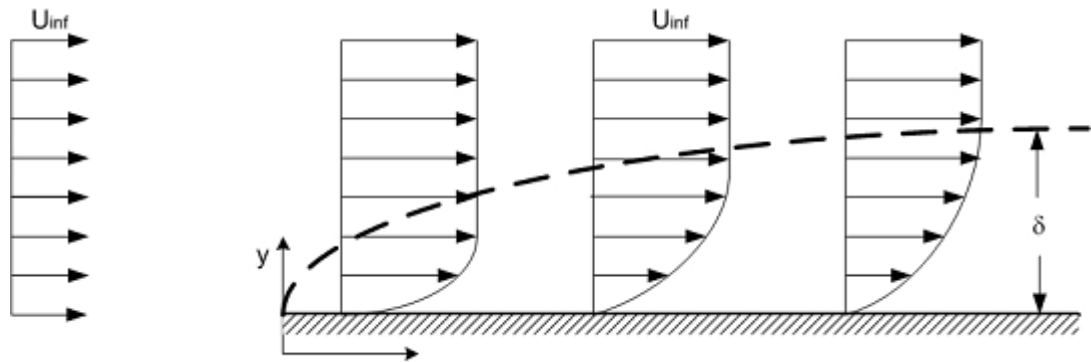


Figure 4: Velocity profile on the boundary layer.

On the figure we show the streamwise velocity variation from free stream to the surface. In reality, the effects are three dimensional. From the conservation of mass in three dimensions, a change in velocity in the streamwise direction causes a change in velocity in the other directions as well. There is a small component of velocity perpendicular to the surface which displaces or moves the flow above it. One can define the thickness of the boundary layer to be the amount of this displacement. The displacement thickness depends on the Reynolds number which is the ratio of inertial (resistant to change or motion) forces to viscous (heavy and gluey) forces and is given by the equation: Reynolds number (Re) equals velocity (V) times density (r) times a characteristic length (l) divided by the viscosity coefficient (mu).

$$Re = \frac{V * r * l}{mu} \quad \text{Eq. (3)}$$

Boundary layers may be either laminar (layered), or turbulent (disordered) depending on the value of the Reynolds number. For lower Reynolds numbers, the boundary layer is laminar and the streamwise velocity changes uniformly as one moves away from the wall, as shown on the left side of the figure.

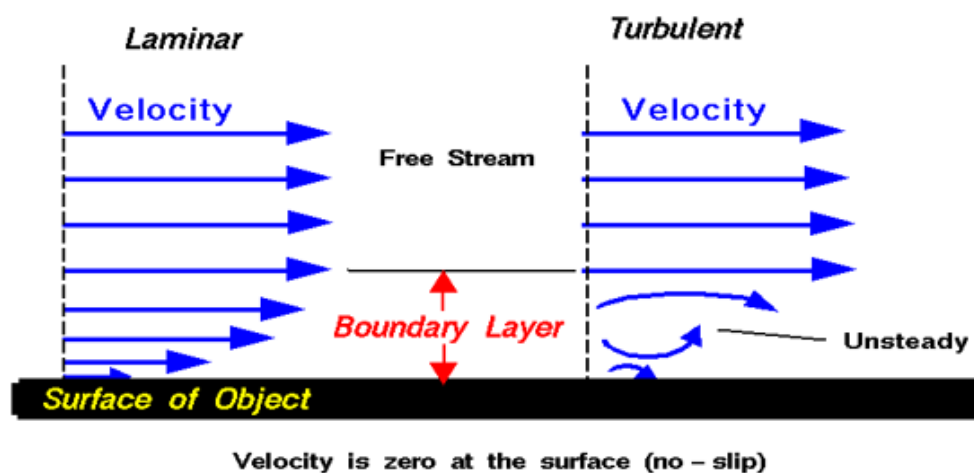


Figure 5: Laminar and turbulent boundary layers

For higher Reynolds numbers, the boundary layer is turbulent and the streamwise velocity is characterized by unsteady (changing with time) swirling flows inside the boundary layer. The external flow reacts to the edge of the boundary layer just as it would to the physical surface of an object. So the boundary layer gives any object an "effective" shape which is usually slightly different from the physical shape. To make things more confusing, the boundary layer may lift off or "separate" from the body and create an effective shape much different from the physical shape. This happens because the flow in the boundary has very low energy (relative to the free stream) and is more easily driven by changes in pressure and we will discuss this on the very next point.

A laminar boundary layer is one where the flow takes place in layers, i.e., each layer slides past the adjacent layers. This is in contrast to Turbulent Boundary Layers shown in Fig.6.2 where there is an intense agitation.

In a laminar boundary layer any exchange of mass or momentum takes place only between adjacent layers on a microscopic scale which is not visible to the eye. Consequently molecular viscosity μ is able to predict the shear stress associated. Laminar boundary layers are found only when the Reynolds numbers are small.

A turbulent boundary layer on the other hand is marked by mixing across several layers of it. The mixing is now on a macroscopic scale. Packets of fluid may be seen moving across. Thus there is an exchange of mass, momentum and energy on a much bigger scale compared to a laminar boundary layer. A turbulent boundary layer forms only at larger Reynolds numbers. The scale of mixing cannot be handled by molecular viscosity alone. Those calculating turbulent flow rely on what is called Turbulence Viscosity or Eddy Viscosity, which has no exact expression. It has to be modelled. Several models have been developed for the purpose.

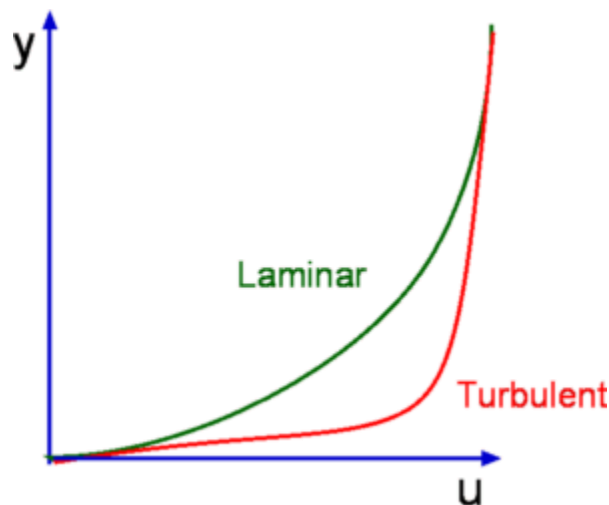


Figure 6: Typical velocity profiles for laminar and turbulent boundary layers

4.4.1. Boundary layer separation

Boundary layer separation is the detachment of a boundary layer from the surface into a broader wake. Boundary layer separation occurs when the portion of the boundary layer closest to the wall or leading edge reverses in flow direction. The separation point is defined as the point between the forward and backward flow, where the shear stress is zero. The overall boundary layer initially thickens suddenly at the separation point and is then forced off the surface by the reversed flow at its bottom.

When the boundary layer separates, its displacement thickness increases sharply, which modifies the outside potential flow and pressure field. In the case of airfoils, the pressure field modification results in an increase in pressure drag, and if severe enough will also result in loss of lift and stall, all of which are undesirable. For internal flows, flow separation produces an increase in the flow losses, and stall-type phenomena such as compressor surge, both undesirable phenomena.

Another effect of boundary layer separation is shedding vortices, known as Kármán Vortex Street. When the vortices begin to shed off the bounded surface they do so at a certain frequency. The shedding of the vortices then could cause vibrations in the structure that they are shedding off. When the frequency of the shedding vortices reaches the resonance frequency of the structure, it could cause serious structural failures.

Boundary layers tend to separate from a solid body when there is an increasing fluid pressure in the direction of the flow--this is known as an adverse pressure gradient in the jargon of fluid mechanics. Increasing the fluid pressure is similar to increasing the potential energy of the fluid, leading to a reduced kinetic energy and a deceleration of the fluid. When this happens the boundary layer thickens, leading to a reduced gradient of the velocity profile with a concomitant decrease in the wall shear stress. For a large enough pressure gradient the shear stress can be reduced to zero, and separation often occurs. The fluid is no longer "pulling" on the wall, and opposing flow can develop which effectively pushes the boundary layer off of the wall. Separation is bound to occur in a sufficiently large adverse pressure gradient. On the other hand, boundary layers like decreasing pressure gradients, which accelerate the fluid and cause the boundary layer to thin.

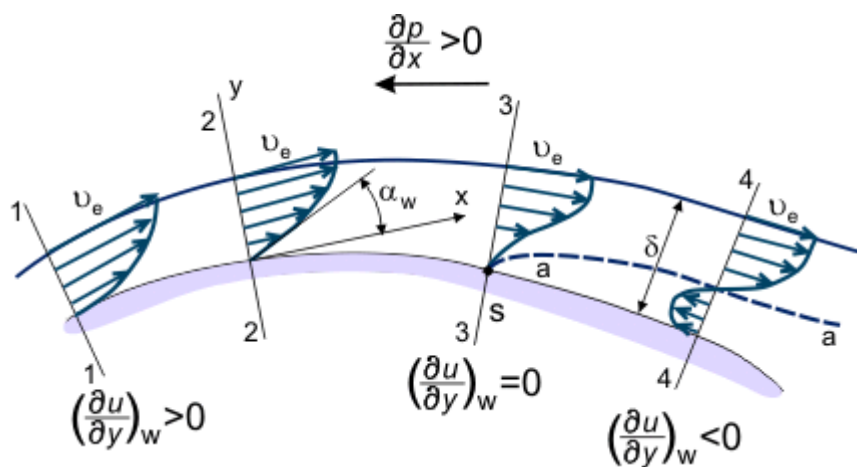


Figure 7: Adverse flow causing the boundary layer separation

Given these considerations, we see that minimizing the pressure drag amounts to preventing or delaying boundary layer separation. Since adverse pressure gradients are the cause of separation, we want to avoid these or at least make the gradients small. Trailing stagnation points are bound to cause problems, so separation can often be delayed by placing the trailing stagnation point at a cusp, so that the fluid leaves the body smoothly. This is known as *streamlining*, and is the preferred shape for airfoils, cars, and fish! Another way of delaying separation is by forcing the boundary layer to become turbulent. The more efficient mixing which occurs in a turbulent boundary layer reduces the boundary layer thickness and increases the wall shear stress, often preventing the separation which would occur for a laminar boundary layer under the same conditions. You can see that there is a trade-off here--the turbulent boundary layer produces a greater drag due to skin friction, but can often *reduce* the pressure drag by preventing, or reducing, boundary layer separation. Since the latter is usually dominant at high Reynolds numbers, various schemes have been invented for producing turbulent boundary layers.

4.5. Vortex

A vortex is a spinning flow of fluid. In particular, vortex is a spiral flow with closed streamlines. All vortices have some special properties. The air (or any fluid) pressure in a vortex is lowest in the center and rises progressively with distance from the center. This is in accordance with Bernoulli's Principle. Two or more vortices that are approximately parallel and circulating in the same direction will merge to form a single vortex. The circulation of the merged vortex will equal the sum of the circulations of the constituent vortices.

Aerodynamically speaking, a Formula 1 car is an interconnected system of vortices and vortex layers. The vorticity is created by viscous shear in thin boundary layers adjacent to the solid surfaces of the car. The downforce generated by a wing is often attributed to the presence of circulation in the airflow around the wing, but the circulation itself is nothing more than the net vorticity in the boundary layers above and below the wing.

When a vortex layer separates from a solid surface, it becomes a free vortex layer, and a



Figure 8: Vortex representation.

separated vortex layer can roll-up into a volume of concentrated vorticity, called a vortex. Vortex have a low pressure core, in some sort of balance with the centrifugal force of the fluid elements spiraling around the vortex on helical trajectories. Oriented in a streamwise direction, such vortices can be particularly useful, both for the direct generation of downforce, and to act as air curtains, sealing off other low pressure areas, for example underbody low pressure area.

Canards, together with vortex generators, generate strong vortices that travel down the sides of the car and act as a barrier. If the canards are positioned correctly, these strong vortices act in way to keep high-pressure air around the car from entering the low-

pressure underbody region, thus maintaining more downforce. If air was allowed to enter the underside, the pressure would inevitably rise, reducing downforce. Therefore, these strong vortices act like a virtual curtain or dam, restricting higher-pressure air around the car's sides from entering the underbody region.

Also, because of vortex high energy, we can use vortices (vortex generators) to prevent early flow separation from aerodynamically incorrect body by energizing boundary layer. There are reported examples of aircraft wings controlling the boundary layer, in which vortex generators successfully delayed flow separation even when the critical Reynolds number is exceeded. Although the purpose of using vortex generators is to control flow separation.

Vortex generators themselves create drag, but they also reduce drag by preventing flow separation at downstream. The overall effect of vortex generators can be calculated by totaling the positive and negative effects, since this effect depends on the shape and size of vortex generators. To select appropriate shape and size of the vortex generator which generates streamwise vortex the most efficiently (with the least drag by itself) is important to achieve our objectives.

Now, the front-wing of a Formula 1 car sees the air first, and therefore sets the conditions for the rest of the car, hence the vortices it generates are particularly important. Front wing vortices are generated by lateral pressure gradients within the front wing assembly, and these exist across the endplate, at the transition between the wing section and the neutral inner-section dictated by regulation, at the inner tips of the front-wing flaps, and at the arched sections in the front-wing. Position and number of vortices is precisely calculated and positioned in relation with the rest of the bodywork downstream, and especially with relation with open rotating wheels. Wrongly calculated and positioned vortex stream can destroy a months of work and millions of dollars.

In Formula 1 for example, bargeboards are used to guide turbulent air from the front wing wake, away from the vital airflow underneath the car. In addition, the lower trailing edge of a bargeboard creates a vortex which travels down the outer lower edge of the sidepod, acting as a skirt or dam, helping to seal the lower pressure area under the car. With such techniques we can see continued utility of ground effect (explained further on in the project) in Formula 1.

4.6. Aerodynamic forces

Two are the aerodynamic force that will be discussed in this chapter and that will be widely analyzed after the simulation with a Computational Fluid Dynamics program. This two forces are the result of the interaction between the fluid particles and the solid object and need to

be well understood now in order to be able to comprehend the way a diffuser works and why is it so important nowadays on racing cars.

4.6.1. Downforce

Motor sports are all about maximum performance, to be the fastest is the absolute. There is nothing else.

To be faster you need power, but there is a limit to how much power you can put on the ground. To increase this limit, force to ground must be applied on the wheels. Increasing weight can do this, but weight makes handling worse and require more power. So we need some virtual weight, we call it downforce and get it from airflow around the car. A wing can make a plane fly, but if we put it upside down, it can make a car NOT fly

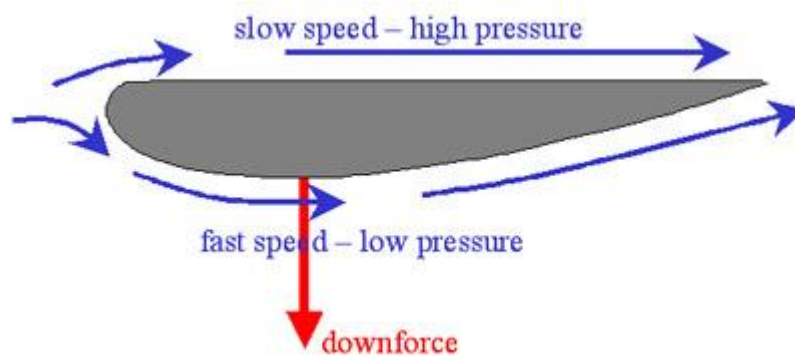


Figure 9: Downforce mechanism

Typically the term "lift" is used when talking about any kind of aerodynamically induced force acting on a surface. This is then given an indicator, either "positive lift" (up) or "negative lift" (down) as to its direction. In aerodynamics of ground racing (cars, bikes, etc.) the term "lift" is generally avoided as its meaning is almost always implied as positive, i.e., lifting the vehicle off the track. The term "downforce", therefore, should always be implied as negative force, i.e., pushing the vehicle to the road.

Both the drag force and the downforce are proportional to the square of the velocity of a car.

The downforce is given by:

$$F_L = \frac{1}{2} C_L \rho A U^2 \quad \text{Eq. (4)}$$

Where:

F_L - Aerodynamic downforce

C_L - Coefficient of lift

ρ - Air density

A - Surface area

U - Object velocity

Cl is determined by the exact shape of the car and its angle of attack

The desire to further increase the tire adhesion led the major revolution in racing car design, the use of negative lift or 'downforce'. Since the tires lateral adhesion is roughly proportional to the downloading on it, or the friction between tire and road, adding aerodynamic downforce to the weight component improves the adhesion allowing a car to travel faster through a corner by increasing the vertical force on the tires, thus creating more grip. Downforce also allows the tires to transmit a greater thrust force without wheel spin, increasing the maximum possible acceleration. Without aerodynamic downforce to increase grip, modern racing cars have so much power that they would be able to spin the wheels even at speeds of more than 160 km/h.

Downforce has to be balanced between front and rear, left and right. We can easily achieve the balance between left and right by simple symmetry, so it will not be discussed. Front and rear is a different thing. Flow in the front greatly affects flow in the back of the car, and vice versa.

Downforce must be adjusted according to racing track and behavior of the car. Too much front downforce induce oversteer. Too much back downforce induce understeer. Varying downforce you can resolve the problems with oversteering or understeering car. Normally, at the price of adding drag.

The success of these features relies primarily on the appropriate and efficient harnessing of drag and downforce - both of which are ruled by physical principles explained by Bernoulli's equation. Though Bernoulli's principle is a major source of lift or downforce in an aircraft or racing car wing, Coanda effect plays an even larger role in producing lift.

Diffusers are, after wings, the most commonly seen devices to generate downforce in the rear portion of the racing vehicle. In them, we use the Bernoulli equation, much in the same way that we do with a Venturi tube. In a Venturi, we can see clearly that pressure and velocity squared are inversely proportional, so diffusers can help to reduce the pressure of the flow under the car by increasing its velocity.

4.6.2. Drag force

Drag is the aerodynamic resistance experienced as a solid object travels through the air. One form of drag occurs as air particles pass over a car's surfaces and the layers of particles closest to the surface adhere. It's known as Boundary layer drag or Skin Friction Drag. Skin friction drag is caused by the actual contact of the air particles against the surface of the moving object. The layer above these attached particles slides over them, but is consequently slowed down by the non-moving particles on the surface as explained in section XXX. The layers above this slowed layer move faster. As the layers get further away from the surface, they slow less and less until they flow at the free-stream speed. The area of slow speed, called the boundary layer, appears on every surface, and causes one of the three types of drag.

The force required to shift the molecules of air out of the way creates a second type of drag, Form Drag. Due to this phenomenon, the smaller the frontal area of a vehicle, the smaller the area of molecules that must be shifted, and thus the less energy required to push through the air. With less engine effort being taken up in the moving air, more will go into moving the car along the track, and for a given engine power, the car will travel faster.

The drag force given by this type of drag is:

$$F_D = \frac{1}{2} \rho A C_D v^2 \quad \text{Eq. (5)}$$

F_D - Aerodynamic drag

C_D - Coefficient of drag

D - Air density

A - Frontal area

V - Object velocity

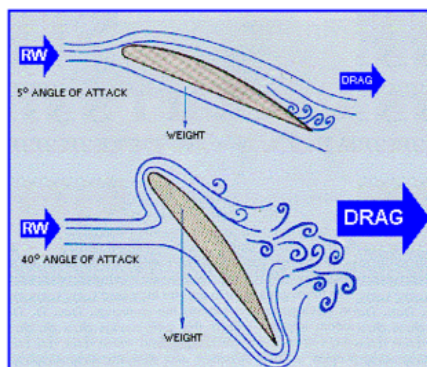


Figure 10: Drag mechanism.

Where C_L is the coefficient of lift determined by the exact shape of the car and its angle of attack.

Form drag and pressure drag are virtually the same type of drag. The separation of air creates turbulence and results in pockets of low and high pressure that leave a wake behind the airplane, car or airfoil (thus the name pressure drag). This opposes forward motion and is a component of the total drag. Streamlining the moving object will reduce form drag, and parts of a

racing car that do not lend themselves to streamlining are enclosed in covers called fairings.

Another factor that plays a role in aerodynamic efficiency is the shape of the car's surfaces. The shape over which air molecules must flow determines how easily the molecules can be shifted. Air prefers to follow at surface rather than to separate from one. The term "separation" refers to the smooth flow of air as it closely hugs the surface of the wing then suddenly breaking free of the surface and creating a chaotic flow. Interestingly, researchers of aerodynamics have found the 'teardrop' shape, round at the front and pointed at the back, to be most efficient at propelling through air while providing a suitable surface for the

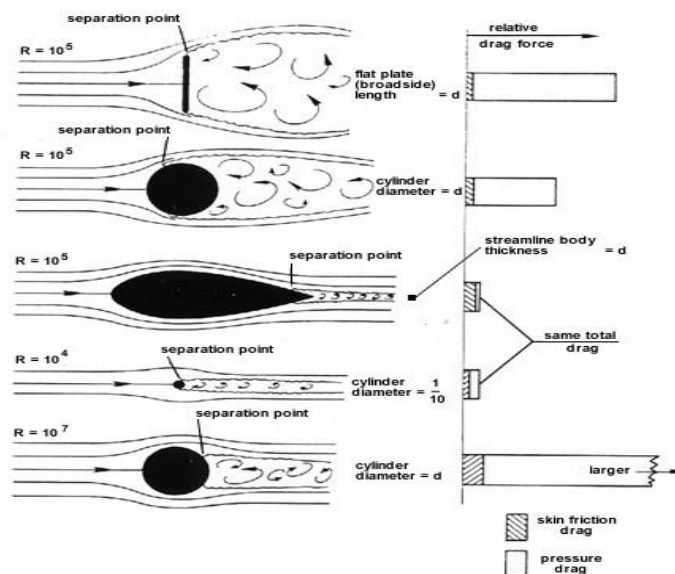


Figure 11: Different shape efficiencies.

air to easily move across. With this shape there is little or no separation. It is important to note that sharp frontal areas, rounded ends, sharp curves or sudden directional changes in a shape should be avoided since they tend to cause separation, which increases drag.

Another type of drag is Induced Drag. It is noted as such because it is caused by or "induced" by the lift on the wings. Induced drag is an unfavorable and unavoidable byproduct of lift (or downforce). You can't do much about induced drag, since you wouldn't have "lift" without it. It occurs on wings of standard or inverted position. In fact, the potential of displaying induced drag exists for all bodies that exhibit opposite pressures on their top and bottom surfaces. Being that air (or any fluid) prefers to move from high to low-pressure regions, air from low-pressure regions has a tendency to curl downward around the ends of a F1 or F3 car wings, for example. It travels down from the high-pressure region to the low-pressure region on the bottom of the wing (opposite direction in case of airplane wings) and collides with moving low-pressure air. Wing tip vortices are a result of this situation. Looking from the tail of the airplane, the vortices will circulate counterclockwise from the right wing tip and clockwise from the left wing tip because on airplane wing high pressure area is

below the wing. In case of racing car, high pressure area is on the top of the wing, and vortices will circulate in opposite direction. The greater the size of the vortices, the greater the induced drag.

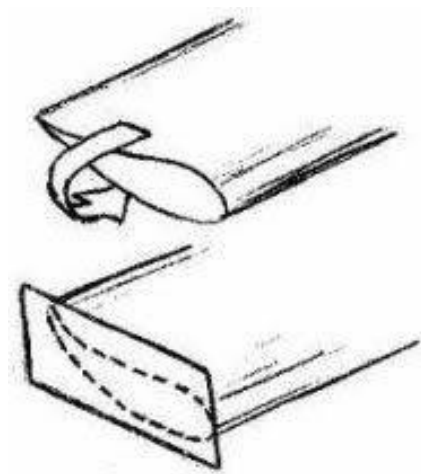


Figure 12: Vortices on a wing

These vortices occur on both airplane wings and F3 car wings even though end plates may be used to reduce this type of drag. It should be noted that the kinetic energy of these turbulent air spirals acts in a direction that is negative relative to the direction of travel intended. In the case of induced drag on F1 cars, the engine must compensate for the losses created by this drag.

A rectangular wing produces much more severe wing tip vortices than a tapered or elliptical wing, therefore many modern airplane wings are tapered. Typically, straight wings produce between 5–15% more induced drag than an elliptical wing. Some early aircrafts and some sport

car wings and spoilers have fins mounted on the tips of the wing which served as endplates. More recent aircraft have wing tip mounted winglets or wing fences to oppose the formation of vortices. Designs such as winglet, wing fence, modified wing tip, etc all reduce induced drag. But there is not a system invented yet to prevent it completely.

5. Formula 3 Introduction

Formula Three, also called Formula 3 or F3, is a class of open-wheel formula racing. The various championships held in Europe, Australia, South America and Asia form an important step for many prospective Formula One drivers. Formula Three has traditionally been regarded as the first major stepping stone for F1 hopefuls – it is typically the first point in a driver's career at which most drivers in the series are aiming at professional careers in racing rather than being amateurs and enthusiasts. F3 is not cheap, but is regarded as a key investment in a young driver's future career. Success in F3 can lead directly to higher formula series such as a GP2 seat, or even a Formula One test or race seat.

There has never been a World Championship for Formula Three. In the 1970s and into the 1980s the European Formula Three Championship and British Formula 3 Championship (once one series had emerged from the competing British series in the 1970s) were the most prominent, with a number of future Formula One champions coming from them. France, Germany, and Italy also had important Formula Three series, but interest in these was originally subsidiary to national formulae – Formula Renault in France and Formula Super Vee in Germany. These nations eventually drifted towards Formula Three. The Italian series tended to attract older drivers who moved straight across from karting whereas in other nations drivers typically graduated to F3 after a couple of years in minor categories. The European series died out in the mid-1980s and the national series became correspondingly more important. For 2003, French and German F3, both suffering from a lack of competitive entrants, merged to recreate the Formula 3 Euro Series.

Brazil's SudAm Formula Three Championship, which now has the most powerful engine of all Formula Three series, was known for producing excellent drivers who polished their skills in the British Formula 3 championship. Perhaps the most curious of all was the small All-Japan Formula Three Championship. Although few drivers spent a significant amount of time there, future stars such as Ralf Schumacher and Jacques Villeneuve scored victories there. An Asian series was established in 2001 and grew to produce past A1 drivers for Indonesia and Australia.

6. Diffuser

The role of the diffuser is to expand the flow from underneath the car to the rear, in turn produce a pressure potential, which will accelerate the flow underneath the car resulting in reduced pressure and as such, a desired increased downforce generation. The Formula 3 diffuser consists of three main channel sections running underneath the car as seen in Figure 13, from the regulations it may also be noted the underbody ground planes can be

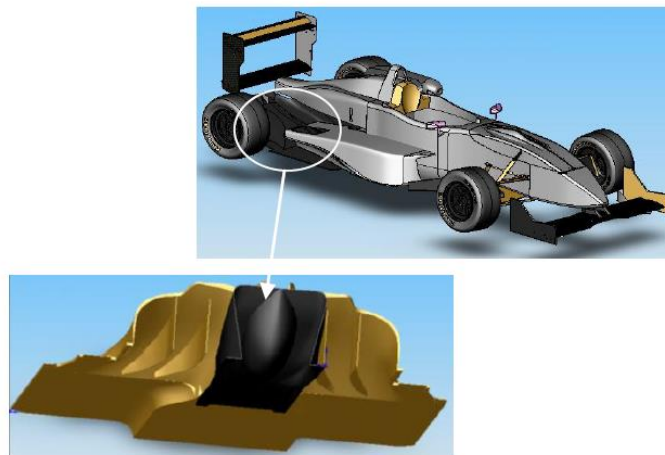


Figure 13: Formula 3 Diffuser.

based off two levels, the center section and the two side channels where the sidepods are located above.

One of the major aspects of the design of the diffuser is the ramp angle or curvature of the diffuser and length. Han [1] investigated the effects of the flow over the rear end of a car using a simple rectangular prism bluff body for comparison and looked into all aspects of the rear end, such as boat angle, ramp angle and backlight angle. He concluded that by

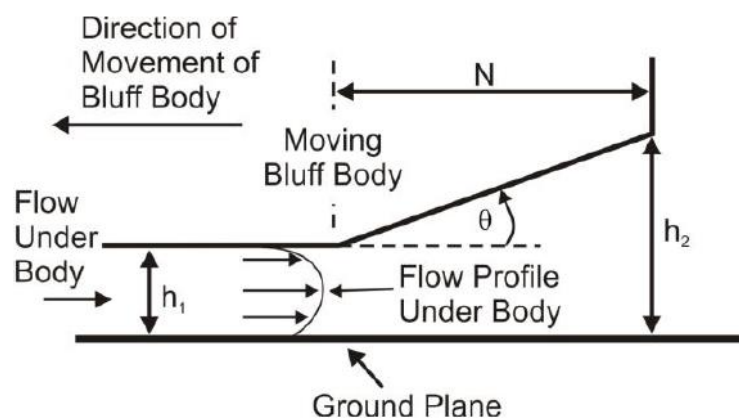


Figure 14: Diffuser's characteristic parameters and geometry.

comparing each angle separately, that the ideal ramp angle for the diffuser should be 17.8 degrees. Figure 2 shows the key aspects of the diffuser; ride height, h_1 , outlet height, h_2 , diffuser length, N , and ramp angle θ .

Cooper [2] has performed two separate investigations on the performance and optimization of the diffuser of an automotive underbody. Cooper looked primarily into the flow and performance (lift and drag) values produced by the diffuser at varying ride heights and ramp angles for two different diffuser lengths. The simulation was done with the use of a wind tunnel employing a moving belt to simulate the moving ground plane. This design made use of a simplified bluff body design, which has been seen to be an ideal comparison benchmark. The results concluded that from the wind tunnel a ramp angle of 9.64 degrees generated the most downforce, while the CFD results showed the optimal value to lie near 13 degrees which is similar to that used by Indy Light cars. Orbit: The University of Sydney undergraduate research journal Page 20.

It was found during testing that the diffuser actually acted as a pump to generate downforce over the underbody flow path. This was not deemed to be the only identifiable fluid mechanical mechanism affecting the flow path around the diffuser. The three main aspects were; "ground effect", "underbody upsweep" and "diffuser pumping". [3]

Ground Effect plays a role when an object is used in the vicinity of a moving ground plane. Flow asymmetry is developed from the flow accelerating as it travels underneath the body due to ground constraint as a result the static pressure underneath the body is reduced which provides the resulting downforce. This would otherwise increase indefinitely with increased ground proximity if not for that real flows are inviscid. Fluid viscosity is of minimal concern for larger ride heights, however this becomes a dominating factor with reduced ride height due to the restricted area underneath the body for which the flow to travel.

Underbody Upsweep refers to the upsweep of the upsweep at the rear. This is typically cambered in shape, similar to the upper surface of an airfoil. Due to the direction of this camber, a resulting downward directed lift force will result during flow interaction.

Diffuser Pumping refers to the increasing cross-sectional area over the diffuser length, which can be used to increase the flow rate through a system via pressure potential. As the ratio of the inlet to outlet area becomes increasingly greater than unity, this generates greater pressure recovery that, due to the base pressure remaining constant will increasingly depress the base pressure at the inlet. The diffuser acts to reduce the underbody pressure due to the expansion resulting in increased flow rate under the body. This increase results in further decrease in underbody pressure, which produces the „pumping down“ or downforce generated. At very low ride heights, the flow rate under the body is reduced so downforce generated is also restricted.

6.1. Performance parameters of a diffuser:

There are several key elements of a diffuser geometry, which ascertain the performance that will result. The pressure recovery coefficient (C_p) is one of these, which relates the pressure at the inlet and outlet of the diffuser section.

$$\overline{C}_p = \frac{(p_2 - p_1)}{\frac{1}{2} \rho \overline{U}_1^2} \quad \text{Eq. (6)}$$

Where U_1 is the area averaged inlet velocity, p_1 is the diffuser inlet static pressure and p_2 the static pressure at the diffuser outlet plane. From idealized full expansion of 1-D flow assuming no losses the pressure coefficient is found by:

$$C_{pi} = 1 - \frac{1}{AR^2} \quad \text{Eq. (7)}$$

Where the area ratio (AR) is a relation between the inlet and outlet heights of the diffuser section. The area ratio for an asymmetric body can therefore be stated as:

$$AR = 1 + \left(\frac{N}{h_1} \right) \tan \theta \quad \text{Eq. (8)}$$

Where N is the diffuser length, h_1 the ride height and θ the diffuser ramp angle. This shows the relation between geometric parameters of the diffuser and enables a realization that vehicles with a greater ride height will possess a smaller area ratio for a given diffuser ramp angle compared to that of a lower ride height.

6.2. Downforce mechanism on a diffuser

Lift coefficient values are the primary outcome result that will govern the performance improvement of the diffuser along with drag coefficient. Following the expressions for both lift coefficient of the bluff body and the streamwise-distance-averaged, mean-effective

pressure coefficients are:

$$C_L = \frac{L}{H} [\overline{C}_{pl} - \overline{C}_{pu}] \quad \text{Eq. (9)}$$

$$\overline{C}_{pi} \equiv \frac{1}{x_i} \int_0^{x_i} C_p(x) dx \quad \text{Eq. (10)}$$

Where l and u are the lower and upper surfaces, and L and H the length and height of the body. Therefore it can be noted that the difference between the upper and lower surface pressures is the main concern in which to increase downforce on the body. For all tests cases then, it will be attempted to maintain upper surface pressure values with only variances to lower surface pressure by means of underbody geometry and clearance variance. Since that downforce is denoted as a negative lift coefficient, it is desired that C_{pl} be made as negative as possible. Furthermore, C_{pl} can be broken up into two components:

$$\overline{C}_{pl} = \left(1 - \frac{N}{L}\right) \overline{C}_{pf} + \frac{N}{L} \overline{C}_{pd} \quad \text{Eq. (11)}$$

Where f refers to the underbody surface upstream of the diffuser (including frontal radius) and d refers to the diffuser length N .

It has been noted that the force behind downforce generation with the diffuser is the pressure recovery performance. The mean effective pressure coefficient can be determined from the fact that the axial pressure distribution in a subsonic diffuser has a characteristic non-linear shape that can be established. The equation for the mean effective pressure (C_{pd}) for asymmetric, plane-walled, underbody diffusers in viscid, incompressible, one-

$$\overline{C}_{pd} = \frac{(C_{p2} - C_{p1})}{(1 - C_{p1})} \quad \text{Eq. (12)}$$

$$C_{p2} = \left(\frac{p_2 - p_\infty}{q_\infty} \right) \quad \text{Eq. (13)}$$

dimensional flow to be [3]:

Where C_{p2} is the pressure coefficient at the diffuser exit.

$$\overline{C_p} = \frac{(C_{p2} - C_{p1})}{(1 - C_{p1})} \quad \text{Eq. (14)}$$

And C_p is the overall pressure recovery coefficient.

It can be deemed that equation (14) is suitable in determining the non-linear behavior of the pressure distribution in underbody diffusers.

7. Flow simulation program

As we have been talking all along the project the core of it is the flow analysis of the Formula 3 diffuser with a simulation program. In this section we intend to present, explain and clarify all aspects and concepts regarding the program that will be used to carry out this task. The reader will be able to understand the governing equations that the solver of the program uses to obtain the solution along with the method applied to solve the mentioned equations but also very important, the different steps to follow in order to carry out a complete CFX simulation.

7.1. ANSYS – Computational Fluid Dynamics

Computational fluid dynamics, usually abbreviated as CFD, is a branch of fluid mechanics that uses numerical analysis and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved.

The fundamental basis of almost all CFD problems are the Navier–Stokes equations, which define many single-phase (gas or liquid, but not both) fluid flows. These equations can be simplified by removing terms describing viscous actions to yield the Euler equations. Further simplification, by removing terms describing vorticity yields the full potential equations. Finally, for small perturbations in subsonic and supersonic flows (not transonic or hypersonic) these equations can be linearized to yield the linearized potential equations.

The use of CFD has become increasingly prominent in calculating the solution of complex fluid dynamics problems in recent years, due in part to the increasing capability of computers to handle larger computational loads. The physical characteristics of fluid motion can be described through several fundamental mathematical equations, mentioned above, typically expressed as partial differential equations. These fundamental equations are known as the governing equations. To solve these governing equations, finite differencing methods are applied to the equations to discretize the equations in such a way that they can be expressed as an algebraic approximation that are calculated over a number of varying locations in the flow. To solve these discretized governing equations over a computational domain containing fluid and solid bodies, the domain must be separated into small elements, creating what is known as a mesh or grid. Finite differencing allows for higher order approximations on these grids, thus resulting in a greater level of accuracy in the solution.

To briefly discuss the process of obtaining finite difference approximations for the governing equations, a simple example for calculating the first-order derivative of an arbitrary flow field variable, $f(x)$, will be discussed. The function $f(x)$ is analytical, and as a result, $f(x+\Delta x)$ can

$$\begin{aligned} f(x + \Delta x) &= f(x) + (\Delta x) \frac{\partial f}{\partial x} + \frac{(\Delta x)^2}{2!} \frac{\partial^2 f}{\partial x^2} + \frac{(\Delta x)^3}{3!} \frac{\partial^3 f}{\partial x^3} + \dots \\ &= f(x) + \sum_{n=1}^{\infty} \frac{(\Delta x)^n}{n!} \frac{\partial^n f}{\partial x^n} \end{aligned} \quad \text{Eq. (15)}$$

be expanded through the use of a Taylor series

From this, the equation for $\frac{\partial f}{\partial x}$ can be found to be:

$$\frac{\partial f}{\partial x} = \frac{f(x + \Delta x) - f(x)}{\Delta x} - \frac{\Delta x}{2!} \frac{\partial^2 f}{\partial x^2} - \frac{(\Delta x)^2}{3!} \frac{\partial^3 f}{\partial x^3} + \dots \quad \text{Eq. (16)}$$

To obtain the first-order approximation for $\frac{\partial f}{\partial x}$, all terms with factors of Δx and higher are summed into a representative function, $H(\Delta x)$. This results in the first-order forward approximation for $\frac{\partial f}{\partial x}$.

$$\frac{\partial f}{\partial x} = \frac{f(x + \Delta x) - f(x)}{\Delta x} + H(\Delta x) \quad \text{Eq. (17)}$$

Similarly, second-order and higher-order approximations are obtained by included factors of Δx^2 and higher, respectively, in this equation. With a basic introduction to finite difference approximations, the governing equations that describe fluid flow will be discussed in the next section.

7.2. CFX – Software

In order to carry out the flow simulation for the diffuser we will use CFX, which is a high performance, general-purpose fluid dynamics program that engineers have applied to solve wide-ranging fluid flow problems for over 20 years. At the heart of CFX is its advanced solver technology, the key to achieving reliable and accurate solutions quickly and robustly.

The modern, highly parallelized solver is the foundation for an abundant choice of physical models that capture virtually any type of phenomena related to fluid flow.

The set of equations solved by CFX ANSYS are the unsteady Navier-Stokes equations in their conservation form, as shown below:

Coordinates: (x,y,z)	Time: t	Density: ρ	Pressure: p	Reynolds Number: Re
Velocity Components: (u,v,w)	Stress: τ	Heat Flux: q	Prandtl Number: Pr	

Continuity:
$$\frac{\partial \rho}{\partial t} + \frac{\partial(\rho u)}{\partial x} + \frac{\partial(\rho v)}{\partial y} + \frac{\partial(\rho w)}{\partial z} = 0$$

X – Momentum:
$$\frac{\partial(\rho u)}{\partial t} + \frac{\partial(\rho u^2)}{\partial x} + \frac{\partial(\rho uv)}{\partial y} + \frac{\partial(\rho uw)}{\partial z} = -\frac{\partial p}{\partial x} + \frac{1}{Re_r} \left[\frac{\partial \tau_{xx}}{\partial x} + \frac{\partial \tau_{xy}}{\partial y} + \frac{\partial \tau_{xz}}{\partial z} \right]$$

Y – Momentum:
$$\frac{\partial(\rho v)}{\partial t} + \frac{\partial(\rho uv)}{\partial x} + \frac{\partial(\rho v^2)}{\partial y} + \frac{\partial(\rho vw)}{\partial z} = -\frac{\partial p}{\partial y} + \frac{1}{Re_r} \left[\frac{\partial \tau_{xy}}{\partial x} + \frac{\partial \tau_{yy}}{\partial y} + \frac{\partial \tau_{yz}}{\partial z} \right]$$

Z – Momentum:
$$\frac{\partial(\rho w)}{\partial t} + \frac{\partial(\rho uw)}{\partial x} + \frac{\partial(\rho vw)}{\partial y} + \frac{\partial(\rho w^2)}{\partial z} = -\frac{\partial p}{\partial z} + \frac{1}{Re_r} \left[\frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial \tau_{zz}}{\partial z} \right]$$

Total Energy – Et:
$$\begin{aligned} \frac{\partial(E_T)}{\partial t} + \frac{\partial(uE_T)}{\partial x} + \frac{\partial(vE_T)}{\partial y} + \frac{\partial(wE_T)}{\partial z} = & -\frac{\partial(up)}{\partial x} - \frac{\partial(vp)}{\partial y} - \frac{\partial(wp)}{\partial z} \\ & + \frac{1}{Re_r} \left[\frac{\partial}{\partial x}(u\tau_{xx} + v\tau_{xy} + w\tau_{xz}) + \frac{\partial}{\partial y}(u\tau_{xy} + v\tau_{yy} + w\tau_{yz}) + \frac{\partial}{\partial z}(u\tau_{xz} + v\tau_{yz} + w\tau_{zz}) \right] \\ & - \frac{1}{Re_r Pr_r} \left[\frac{\partial q_x}{\partial x} + \frac{\partial q_y}{\partial y} + \frac{\partial q_z}{\partial z} \right] \end{aligned}$$

Figure 15: Unsteady Navier - Stokes equations

These equations describe how the velocity, pressure, temperature, and density of a moving fluid are related. The equations were derived independently by G.G. Stokes, in England, and M. Navier, in France, in the early 1800's. The equations are extensions of the Euler Equations and include the effects of viscosity on the flow.

The Navier-Stokes equations consists of a time-dependent continuity equation for conservation of mass, three time-dependent conservation of momentum equations and a time-dependent conservation of energy equation. There are four independent variables in the problem, the x, y, and z spatial coordinates of some domain, and the time t. There are six dependent variables; the pressure p, density r, and temperature T (which is contained in the energy equation through the total energy Et) and three components of the velocity vector; the u component is in the x direction, the v component is in the y direction, and the w component is in the z direction, All of the dependent variables are functions of all four independent variables.

The terms on the left hand side of the momentum equations are called the convection terms of the equations. Convection is a physical process that occurs in a flow of gas in which some property is transported by the ordered motion of the flow. The terms on the right hand side of the momentum equations that are multiplied by the inverse Reynolds number are called the diffusion terms. Diffusion is a physical process that occurs in a flow of gas in which some property is transported by the random motion of the molecules of the gas. Diffusion is related to the stress tensor and to the viscosity of the gas. Turbulence, and the generation of boundary layers, are the result of diffusion in the flow. The Euler equations contain only the convection terms of the Navier-Stokes equations and cannot, therefore, model boundary layers. There is a special simplification of the Navier-Stokes equations that describe boundary layer flows.

7.3. CFX process

The process of performing a single CFD simulation is split into four main components:

7.3.1. Creating the Geometry/Mesh

This interactive process is the first pre-processing stage. The objective is to produce a mesh for input to the physics pre-processor. Before a mesh can be produced, a closed geometric solid is required. The geometry and mesh can be created in the Meshing application or any of the other geometry/mesh creation tools. The basic steps involve:

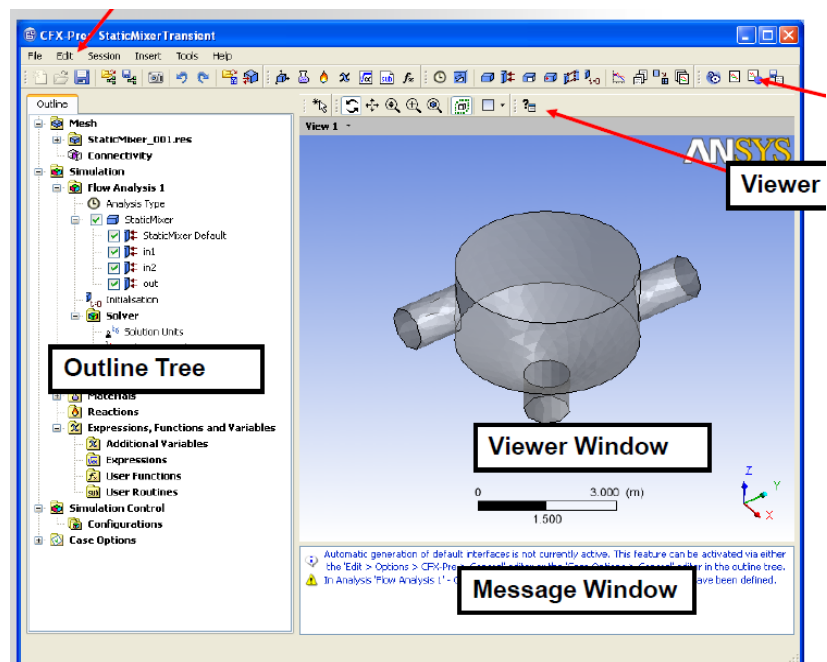


Figure 16: CFX Preprocessor display.

1. Defining the geometry of the region of interest.
2. Creating regions of fluid flow, solid regions and surface boundary names.
3. Setting properties for the mesh.

This pre-processing stage is now highly automated. In CFX, geometry can be imported from most major CAD packages using native format, and the mesh of control volumes is generated automatically.

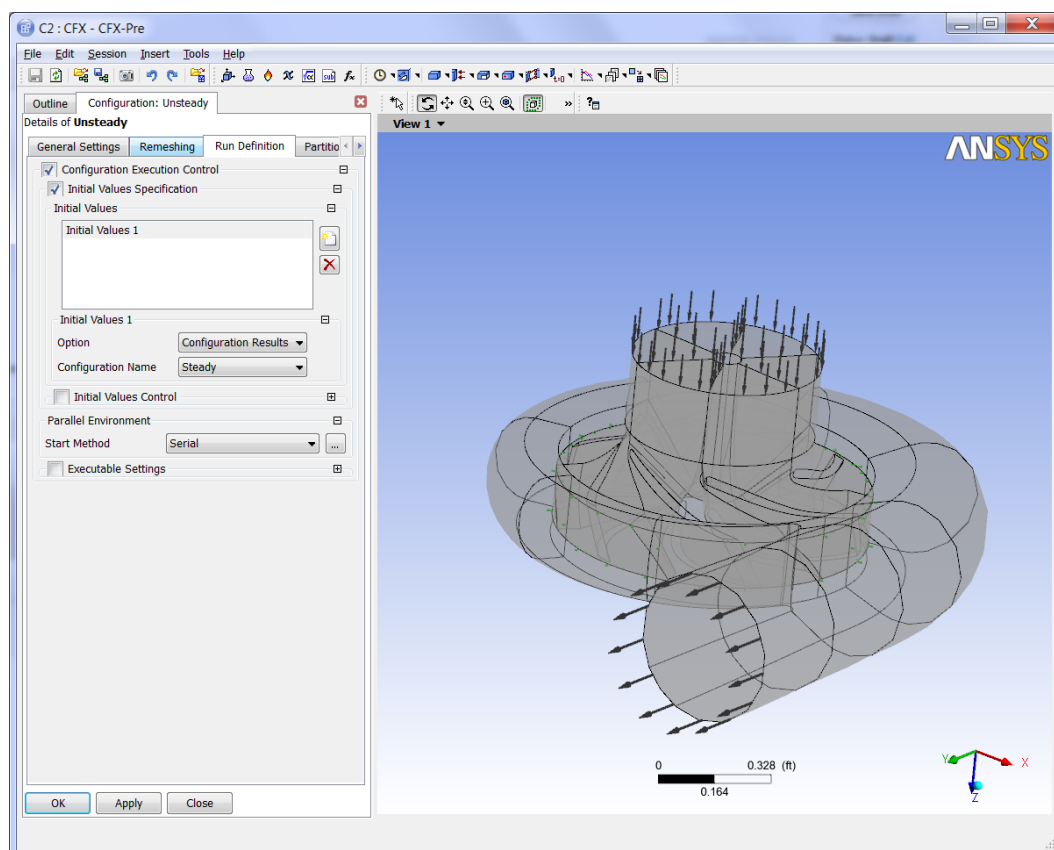


Figure 17: CFX Preprocessor display.

7.3.2. Defining the Physics of the Model

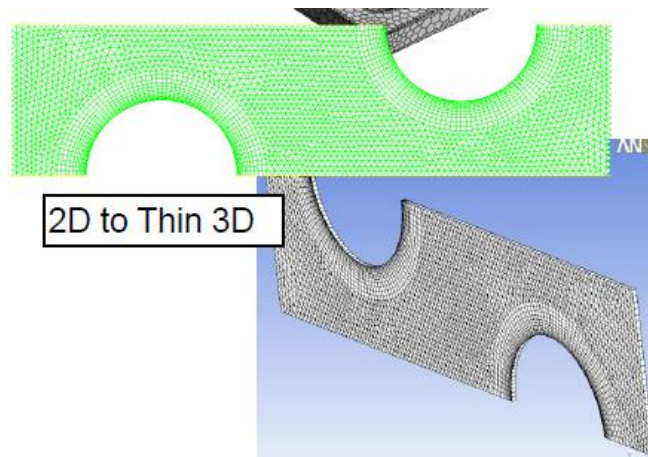


Figure 18: Mesh for 3D control volume.

This interactive process is the second pre-processing stage and is used to create input required by the Solver. The mesh files are loaded into the physics pre-processor, CFX-Pre. The physical models that are to be included in the simulation are selected. Fluid properties and boundary conditions are specified.

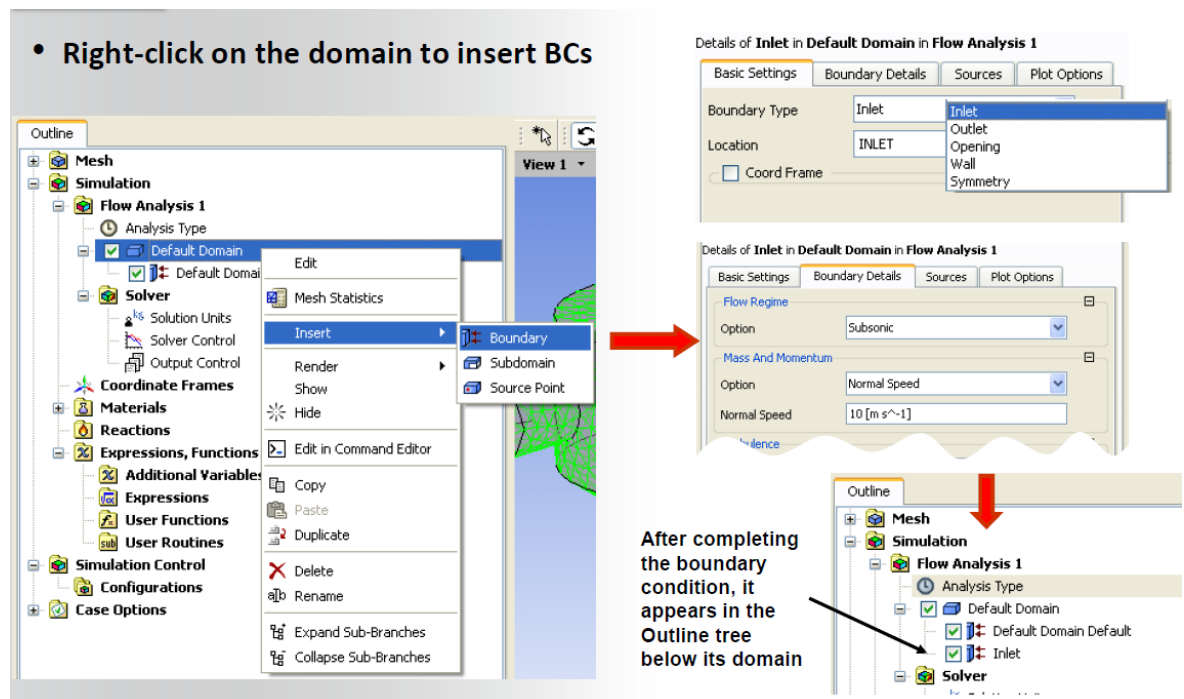


Figure 19: Figure on how to create boundary conditions.

7.3.3. Solving the CFD Problem

The component that solves the CFD problem is called the Solver. It produces the required results in a non-interactive/batch process. A CFD problem is solved as follows:

1. The partial differential equations are integrated over all the control volumes in the region of interest. This is equivalent to applying a basic conservation law (for example, for mass or momentum) to each control volume.
2. These integral equations are converted to a system of algebraic equations by generating a set of approximations for the terms in the integral equations.
3. The algebraic equations are solved iteratively.

An iterative approach is required because of the nonlinear nature of the equations, and as the solution approaches the exact solution, it is said to converge. For each iteration, an error, or residual, is reported as a measure of the overall conservation of the flow properties.

How close the final solution is to the exact solution depends on a number of factors, including the size and shape of the control volumes and the size of the final residuals. Complex physical processes, such as combustion and turbulence, are often modeled using empirical relationships.

The approximations inherent in these models also contribute to differences between the CFD solution and the real flow. The solution process requires no user interaction and is, therefore, usually carried out as a batch process. The solver produces a results file that is then passed to the post-processor.

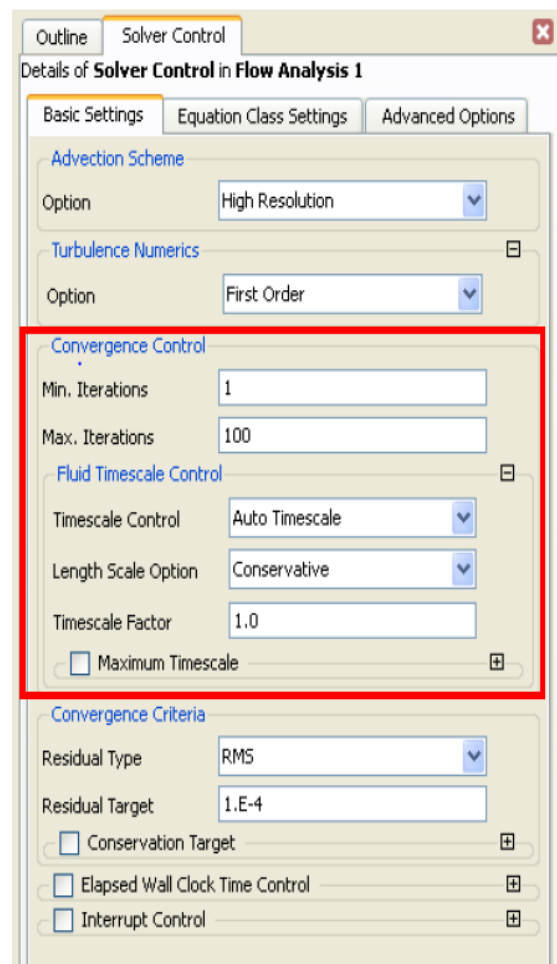


Figure 20: Solver control and settings.

7.3.4. Visualizing the Results in the Post-processor

The post-processor is the component used to analyze, visualize and present the results interactively. Post-processing includes anything from obtaining point values to complex animated sequences. Examples of some important features of post-processors are:

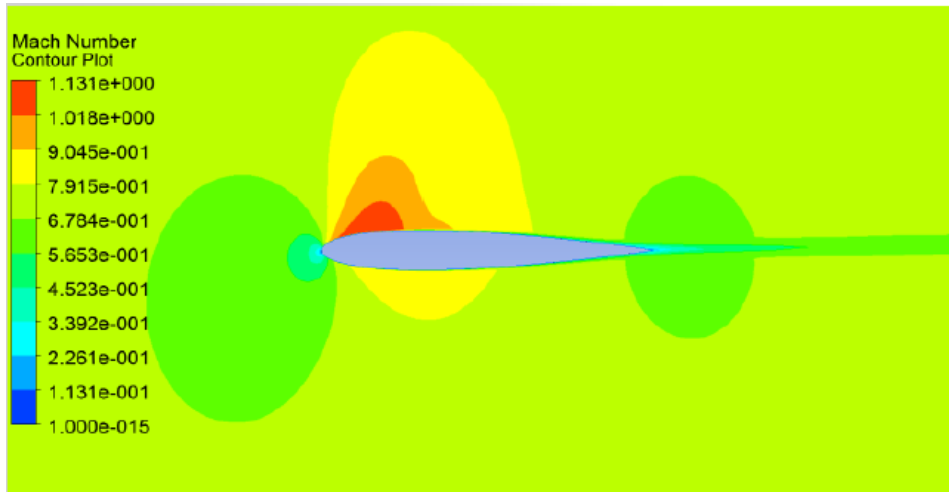


Figure 21: Example of contour plot on an airfoil for the Mach number, Postprocessor.

- Visualization of the geometry and control volumes
- Vector plots showing the direction and magnitude of the flow
- Visualization of the variation of scalar variables (variables that have only magnitude, not direction, such as temperature, pressure and speed) through the domain
- Quantitative numerical calculations
- Animation
- Charts showing graphical plots of variables
- Hardcopy and online output.

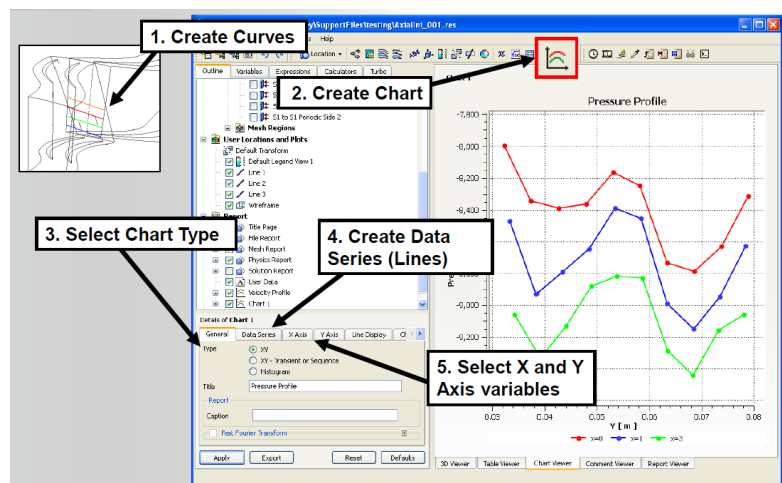


Figure 22: How to create graphs at Postprocessor.

8. CFX Simulation – Flow analysis of the diffuser

After defining and learning about all the aerodynamic concepts related to a diffuser function it is time to simulate the air flow through it. In this chapter we will use the CFX software previously described to analyze the air flow through the diffuser, taking also into account the air that enters under the car through the sides of it (double diffuser). The first step will be to create the geometry, for that, Solid works has been the computer program used. After that, five different steps have to be done in order to get to accurate results with CFX and be able to obtain accurate and precise conclusions.

During this chapter we will carry out different simulations with different geometries, all of them created one after the other with the sole purpose of improving the diffuser efficiency: gaining down force while reducing drag force and therefore obtaining a low Drag Coefficient and a high Lift Coefficient (negative).

8.1. First Analysis

8.1.1. Geometry

This first geometry was designed and inspired after looking at multiple pictures and 3D documents on the internet. It was created knowing it wouldn't be the definitive but in order to make an idea of the air flow through the diffuser and in which way improvements had to be assessed. That way we could learn about the basic mistakes or geometry problems of a diffuser and be more prepared to solve them.

A picture of this first geometry can be seen below:

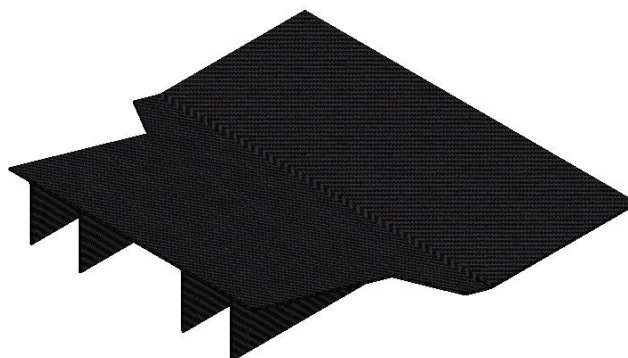


Figure 23: First diffuser design

8.1.2. CFX Analysis

8.1.2.1. Geometry

The first step in CFX is to create the geometry to analyze. What we have to comment here is that in a software of air flow analysis the geometry we have to create is that one corresponding to the air flow. This means that with Solid Works, after having designed the diffuser, we have to obtain the negative figure of it, representing this the air flowing under the car and the geometry that will be imported to CFX. It is also called control volume.

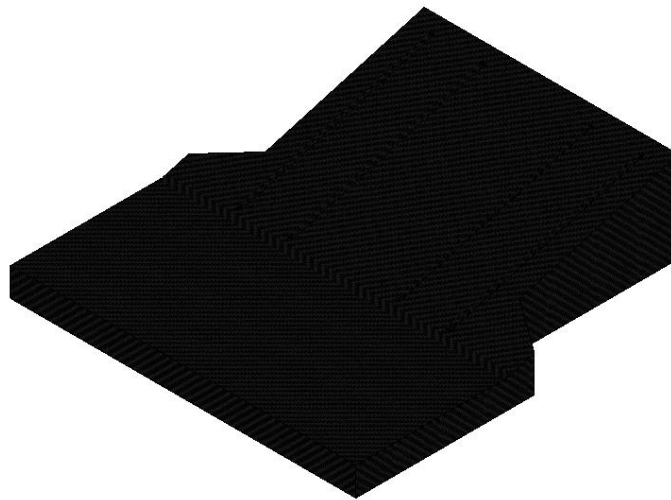


Figure 24: Negative model of the diffuser

8.1.2.2. Mesh

The next step in CFX is to define the mesh that we will use to run the simulation, it is very important to choose a proper mesh because otherwise we can obtain biased results. In this simulation we have applied the following special characteristics to the default mesh:

Inflation: This tool will let us properly analyze the boundary layer on the diffuser by having a graduated set of layers (10 in our case) of parallel elements as they get closer to the wall of the control volume (the lower wall of the diffuser). As already said, it will let us analyze the boundary layer and its separation in case it happens with high precision.

In order to know how many layers to select for the inflation we made a simple

calculation for the boundary layer thickness. Using the formula used for flat plates (eq. 18) and previously calculating the Reynolds (eq. 19) we obtained a result of 1,53e6 for the Reynolds number and a boundary layer thickness of 10 mm. Seeing how small this parameter could be we considered that the inflation needed more than 5 layers, the default value of CFX. We would need very close elements next to the wall in order to analyze the boundary layer.

$$\delta/x = 0.16/(Re)^{(1/7)} \quad \text{Eq. (18)}$$

$$Re = \rho \cdot x \cdot u / \mu \quad \text{Eq. (19)}$$

Element size: We have set the element size to 10 mm, this was done with the purpose of obtaining the maximum accuracy trying not to exceed the computational capacity of the program. This means, looking for the balance between

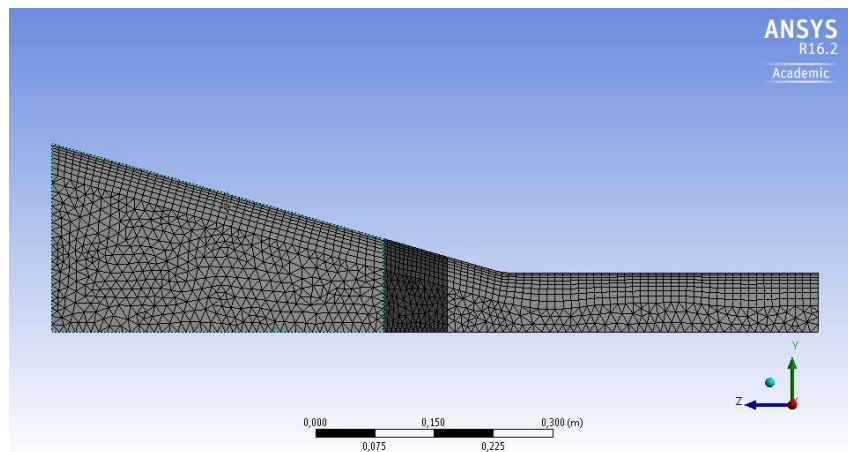


Figure 24: Mesh.

result's accuracy and computational program efficiency. Further on this section of the project we will see how a not so good mesh can change the result considerably.

Element size on curvature: This default set-up option has been turned off because it was creating great number of elements at some unnecessary parts of the control volume consuming computational capacity uselessly.

8.1.2.3. Boundary conditions

This is also a very important step of the simulation. Here is where we define the conditions of the air flow so that it simulates and represents the real situation the more precisely possible.

The boundary conditions defined for this simulation are as follow:

Inlet: As shown in the figure below, we set the entry of the air through the frontal part of the control volume. The air speed is set to 40 m/s considered as an average of the speed that a Formula 3 car develops during a race. The material is air with a density of 1,18 g/cm³ at the temperature of 25 °C.

Outlet: Also as shown in the figure we set the exit of the air through the back part of the control volume. The parameter to define in this boundary condition is the relative pressure, which takes a value of 0 Pa or 1 atm.

Wall: This kind of boundary condition has been applied to two different parts of the diffuser. The kind of wall is non-slippy. These parts of the geometry represent the wall of the diffuser itself, being the lateral wall the part that represents the function of the double diffuser and then the tires at the back.

Moving wall: this boundary condition is set to the lower part of the control volume. As we already know, in terms of aerodynamics, it is the same to consider that the car is moving at 140 m/s and the ground is static than considering the air is moving at 140 m/s and the car is static. Following this rule we have to take into account that both ground and air move always at the same speed, so if we have set that the air is moving at 140 m/s we have to do the same for the ground. This is what we are doing applying this boundary condition. We give to the wall boundary condition the property of moving wall, and we set a speed of 140 m/s for this all in the same direction as the air flow.

8.1.2.4. Run

Once we have all the parameters and conditions ready for the simulation is time to run it. We run a normal simulation, stationary state with a limit of 100 iterations. The turbulence model chosen for the simulation is SST (Shear Stress Transport). The reasons to choose this model are because it gathers the positive characteristics of both the k-Epsilon model and k- ω model. This results in a good analysis close to the wall where the boundary layer separation can happen but also far from this points. This matter will be discussed further on the project with more detail.

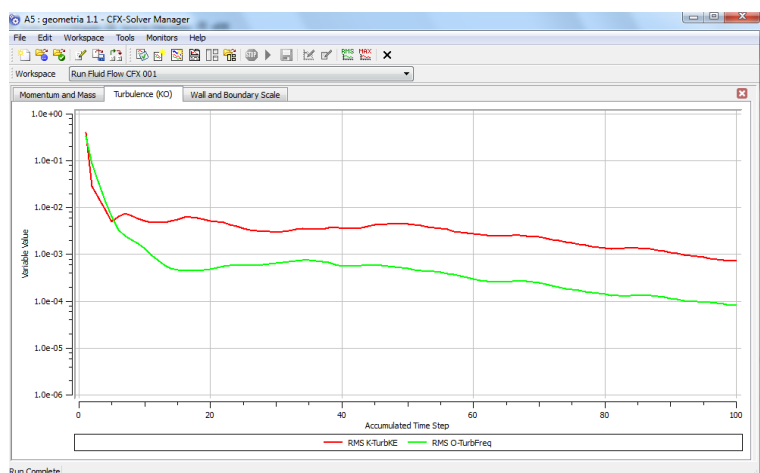


Figure 25: Residuals for turbulence model.

During the run we monitor the residuals for the mass and momentum and also for the turbulence model. This parameters are the ones that indicates us if the simulation has been successful or should be repeated considering new settings. Residuals have to be steady and lower than 0.001 at the end of the simulation.

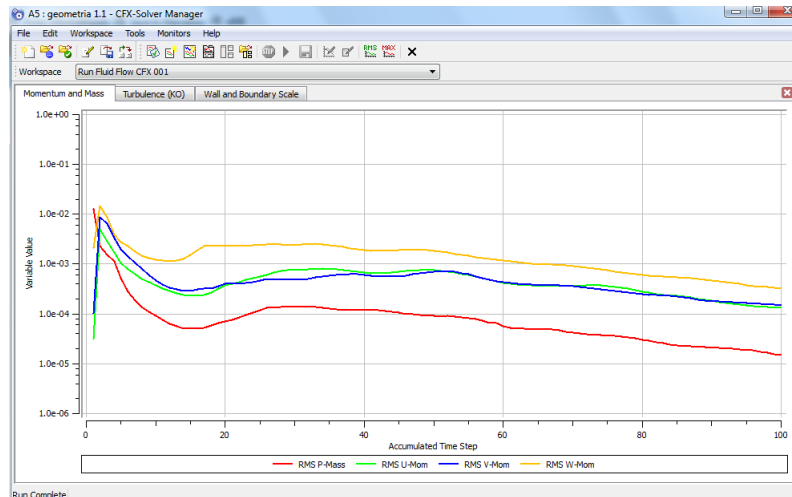


Figure 26: Residuals for mass and momentum

8.1.2.5. Results

In this final step of the simulation we will use contour plots, vector plots or other tools of the software such as streamlines to determine how good the simulation has been and how should we proceed in order to improve the efficiency of the diffuser.

The first results obtained show a downforce of 143.61 N and a drag force of 22.02 N. In terms of drag force the result obtained is good, giving a value for the Drag Coefficient of 0.126. On the other hand, the value for the downforce is to low, giving a Lift coefficient of 0.197, much less than what we aim to achieve.

In order to find out an explanation for this results we will use a streamline plot and a vector plot in different parts of the diffuser to see how the air flowed through it.

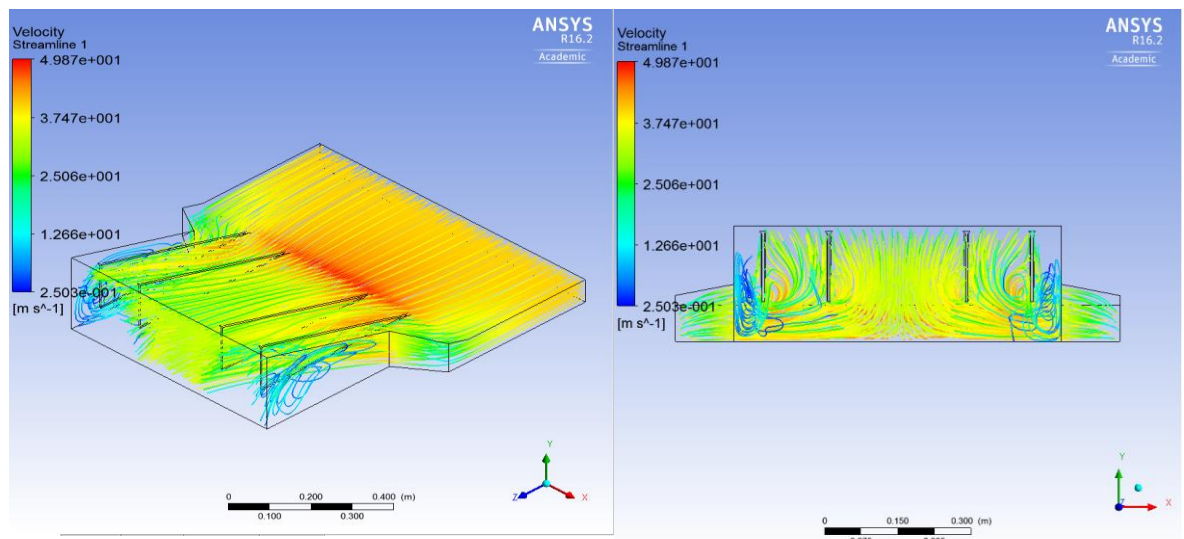


Figure 27: Streamline.

These two pictures of the streamline created at the post processing show us how the air is not being properly lead to the back of the diffuser. It is not kept attached to the lateral walls and creating turbulence at the back. This is probably due to the straight wall designed representing the double diffuser and the rest of air flowing under the car, we will have to curve this part of the diffuser and see if this problem can be reduced or totally eliminated.

Taking a look at the vector plots (Figure 28) we can identify this very same problem. There is a heavy turbulence created at the back of the diffuser on the lateral area.

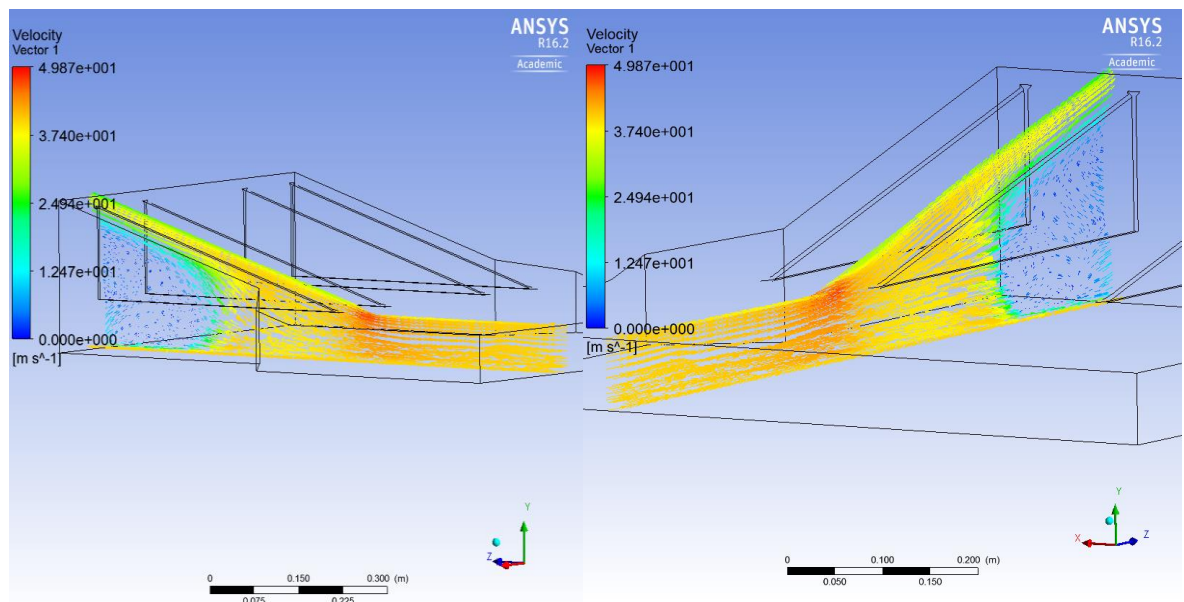


Figure 2: Vector plot showing the effect of whirlwinds.

Finally we can see a picture of a contour plot for the pressure on the wall of the diffuser. The pressure gradient is not low, but it is not created in the appropriate parts of the diffuser. The lowest pressures should be found on the angle and the higher pressures should be found at the back of the diffuser, which is not what happens in this simulation showing once again this design should be modified in order to get good results.

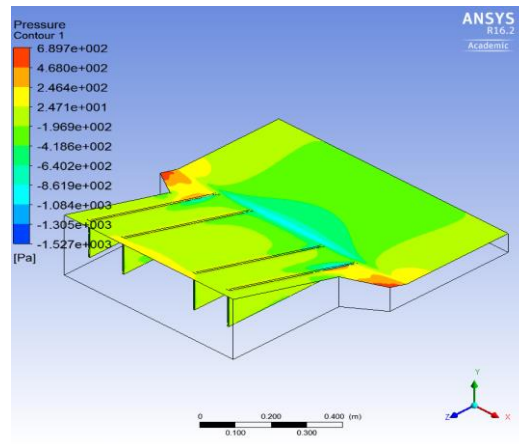


Figure 3: Contour plot of the pressure.

8.2. Second Analysis

8.2.1. Geometry

As the final results showed, the first geometry that was design was very basic and will take various steps till the diffuser offers a good efficiency.

The new design that we can see in the figure to the right has been carried out following the



Figure 30: 2nd geometry.

conclusions obtained after the first simulation. We expect to improve the double diffuser effect avoiding whirlwinds that create a heavy turbulence at the outlet of the diffuser.

Although it may seem a little change on the design the one taken, we insist on the importance of taking small steps so that we can appreciate the effect of every one of them, if we change the design of the diffuser abruptly we won't be able to distinguish how the changes have exactly affected the behavior of the diffuser and thus the air flow.

Also I would like to clarify that from now on the changes shown in the geometry will be in the

negative model of the diffuser. It will save us time of design and at the end what we would need to do to obtain the final geometry is to build a mold from this piece.

8.2.2. Mesh, boundary conditions and run.

For the following simulations the mesh, boundary conditions and settings for the run will be the same as in the first simulation. This means in the next analysis there will not be a point to comment this but we have to take into account it has been done for each simulation.

8.2.3. Results

Commenting once again on the forces acting on the diffuser we have some significant changes compared to the previous simulations. The down force has reach a value of 333.25 N which is considerably high (34 kg of force) obtaining a new value for the Lift Coefficient of 0.457 which is also good for a Formula 3 car. However, the drag force has increased a little bit reaching a value of 28.28 resulting in a Drag coefficient of 0.162. This is probably caused, as we will see next, by a better attachment of the air flowing throw the diffuser that has also helped to obtain a much higher down force and a better distribution of the pressure on the diffuser although this last matter still has to be improved. Areas of high pressure

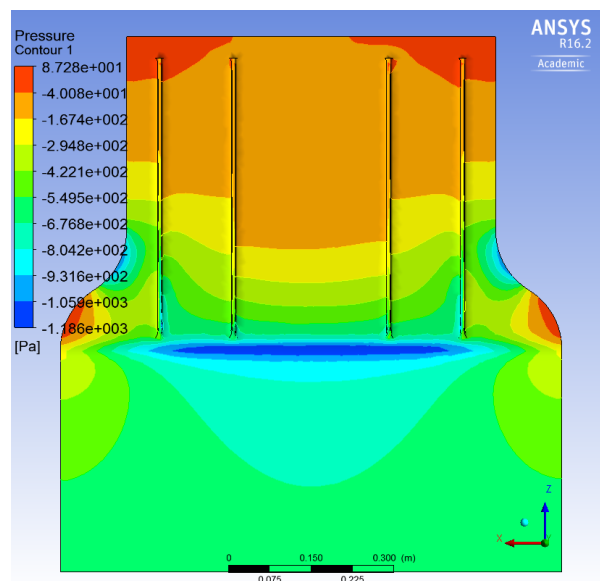


Figure 4: Contour plot of the pressure on the diffuser

should only be found at the back of the diffuser, something we are still not achieving despite the curvature introduced in the lateral as we can see in figure 31. This high pressure at those points means the air is creating turbulence or being highly slowed down so we are not succeeding on bringing the air flowing next to the car into the diffuser. The value for the maximum difference of pressure in the diffuser is 1273.28 Pa.

It also appears, judging by the vector plots in figure 32 that with this new geometry we have eliminated the turbulence that the air coming from the side of the car was causing on the

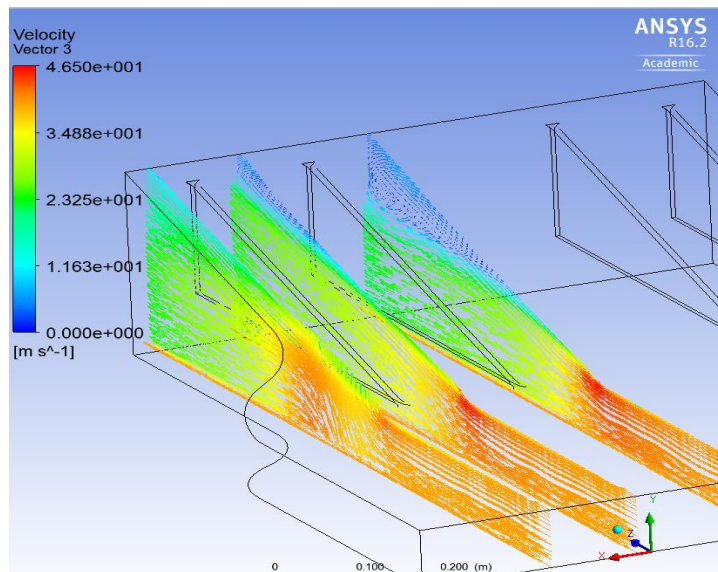


Figure 32: Vector plot on three different planes

first design although now a new problem has risen, there is a boundary layer separation at the back of the diffuser, being more notable on the center of it.

Figure 33, a streamline plot taken from the inlet and another one from the outlet show how in the middle back part of the diffuser there is a great turbulence that doesn't allow us to obtain greater efficiency on the diffuser because of the boundary layer separation, our next design of the diffuser will be taken in order to try to solve this problem.

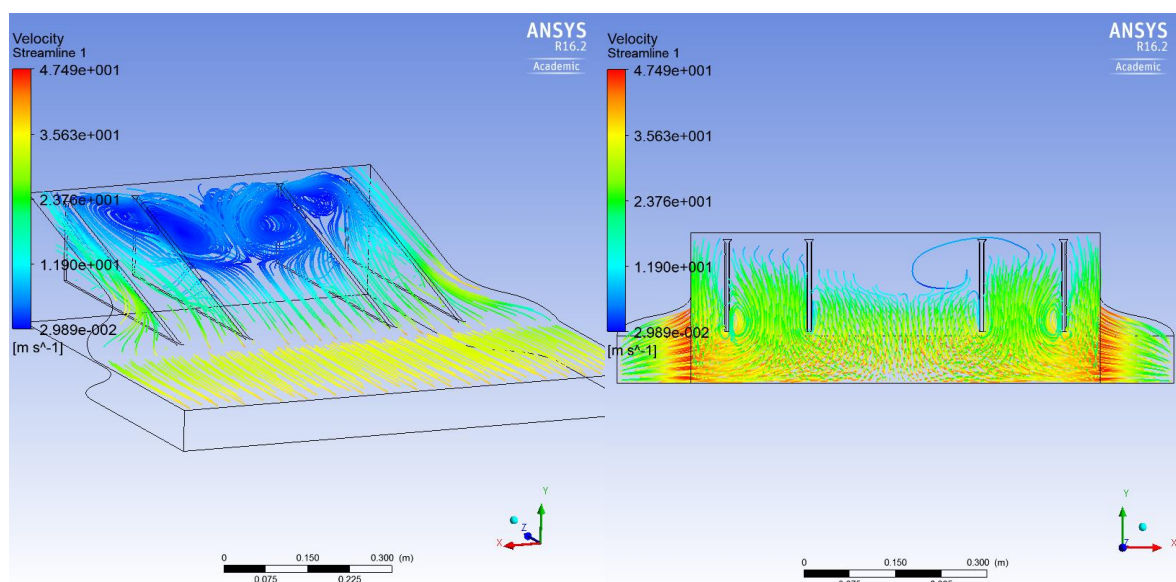


Figure 33: Streamline taken from the inlet and wall of the diffuser.

8.3. Third Analysis

8.3.1. Geometry

After examining the results of the second simulation a new design of the diffuser was introduced. As commented on the section above we needed to reduce the turbulence in the middle back part of the diffuser and, after a deep research of photos and different designs of formula 3 diffusers of the past years we came up with the following design. Once again we are working on the negative piece of the diffuser.

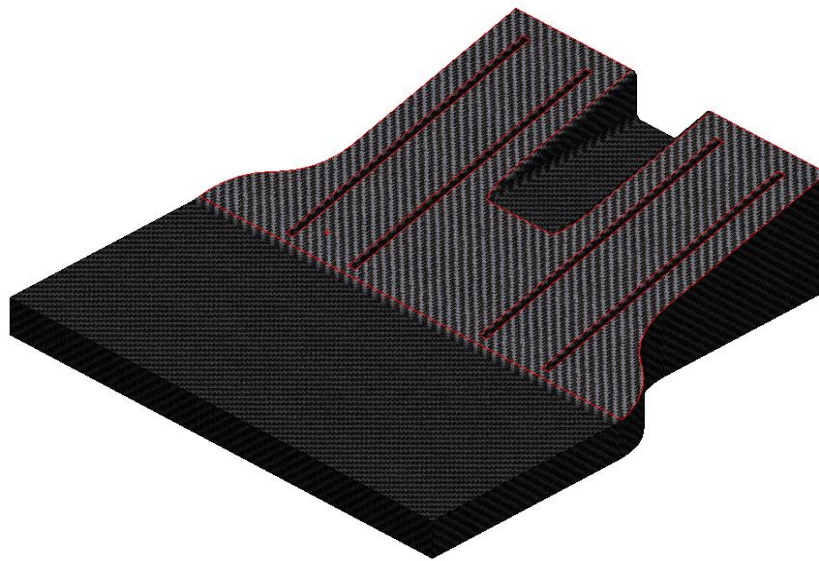


Figure 34: 3rd Design

What we are trying with this new design is to stop the boundary layer from detaching on the middle of the diffuser and consequently reduce the drag force on the diffuser.

8.3.2. Results

After a third analysis of a third design we can say we still achieve the little goals we set and we keep on improving, however, this time, we didn't succeed at a 100% but first let's comment on the value of the forces obtained. The down force in this simulation has been of 329.17 (33.59 kg), a little bit lower than in the previous simulation and a Lift Coefficient also a little bit lower reaching a value of 0.451. But we have succeeded on reducing the drag force, it presents now the magnitude of 19.85, we have to take into account the fact that

now the drag area is lower so the fact of improving the drag force could have been a result of reducing the area but fortunately, as we can assure by the new value of the Drag coefficient (0.123), this is not the case and the improvement is notable.

If we take a look now at the figure shown below, a contour plot of the pressure, we can see we have obtained a similar pressure maximum difference, 1260.98 Pa, but with a more uniform distribution of the pressure at the back. However we still need to correct the curvature on the laterals which will be the next step.

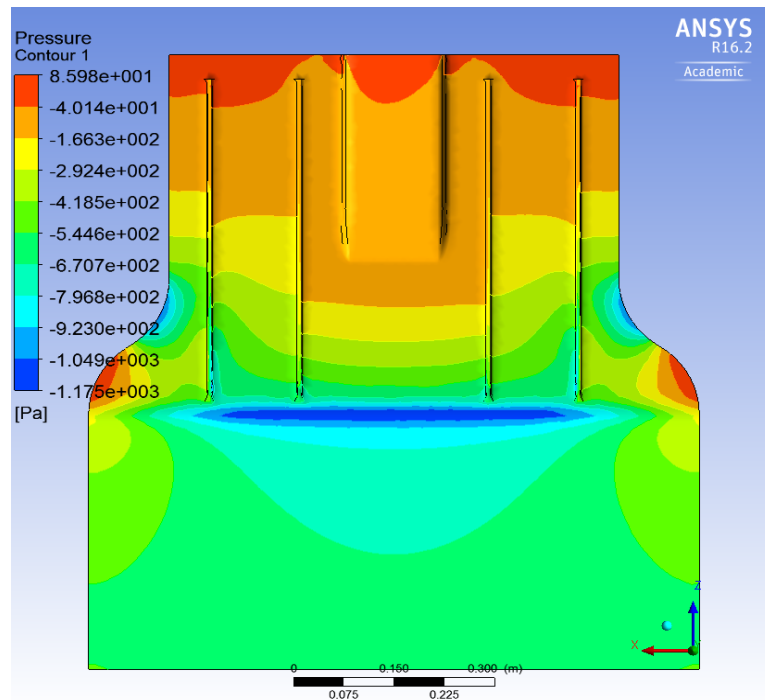


Figure 35: Contour plot of the pressure.

This better distribution of the pressure has been achieved thanks to the new design, as we can see in the vector plots (Figure 36), we have clearly reduced the turbulence in the middle part of the diffuser and there is no boundary layer separation except for two parts that can be better seen in the following figure, a couple of vector plots.

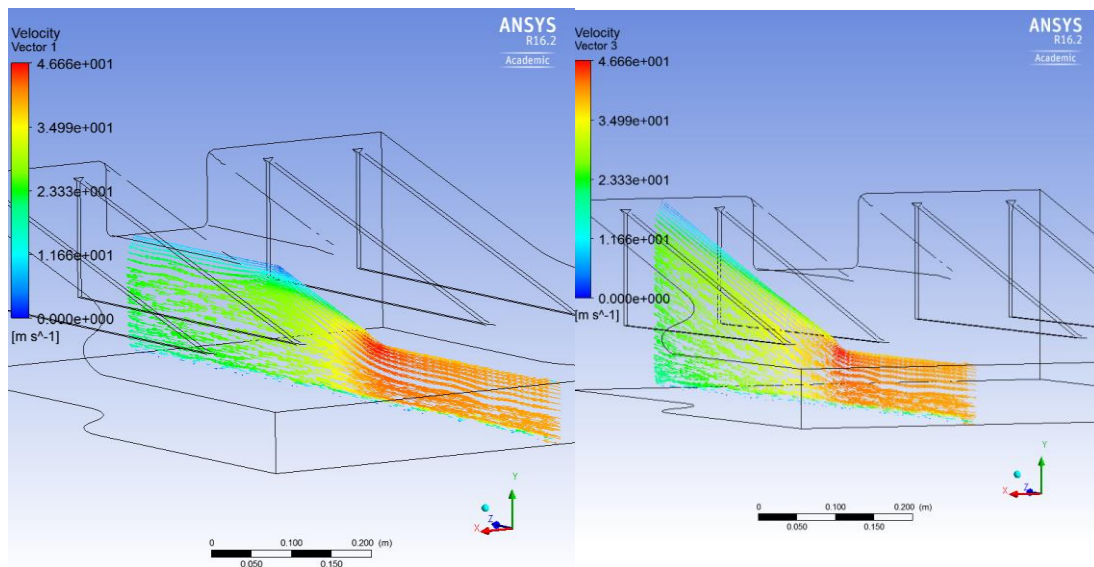


Figure 36: Vector plots showing no boundary layer separation at this areas of the diffuser.

Next to the new element introduced in the diffuser we can still find a boundary layer separation, the detachment is produced further at the wall but it is also our aim to avoid this situation at the whole diffuser.

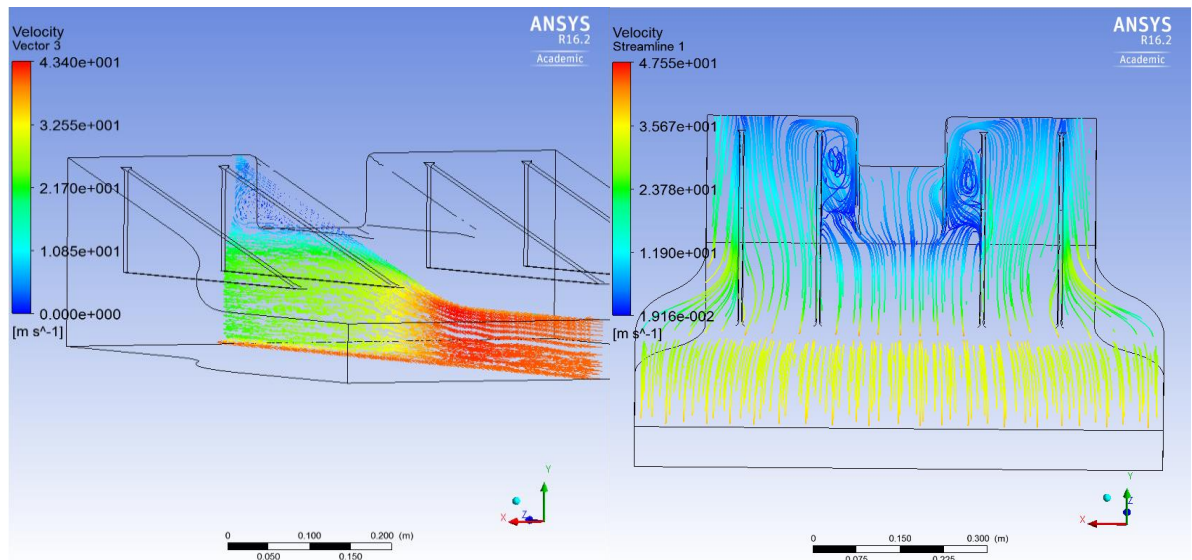


Figure 37: Vector plot and streamline evidencing this effect.

8.4. Fourth Analysis

8.4.1. Geometry

As commented on the previous point, the aim of this fourth analysis was to have a good pressure distribution on the diffuser, meaning this we have achieved the purpose of recirculating the air flowing next to the car into the diffuser without creating turbulence. For that we have simply smoothed the curvature on the lateral, hoping the air will follow the geometry and enter the diffuser.



Figure 5: 4th Design

The new geometry can be seen in Figure 38. With this new geometry we expect to gain some extra downforce while maintaining or even reducing a bit the drag force. We also expect to improve the Lift coefficient of the diffuser.

8.4.2. Results

Once again we will commence by commenting on the results of the forces and coefficients. As expected, this new design has let us to higher down force on the diffuser, although it is not a big change, the value of 379.27 N (38,7 Kg) makes a great significance in the diffuser efficiency and also allows the Lift coefficient to reach a value higher than 0.5, to be more precise of 0.521. In terms of drag force we have also obtained a not so expected improvement, we have reduced its value to 14.8 which give us a Drag Coefficient value of 0.092, the lowest one we have obtained so far.

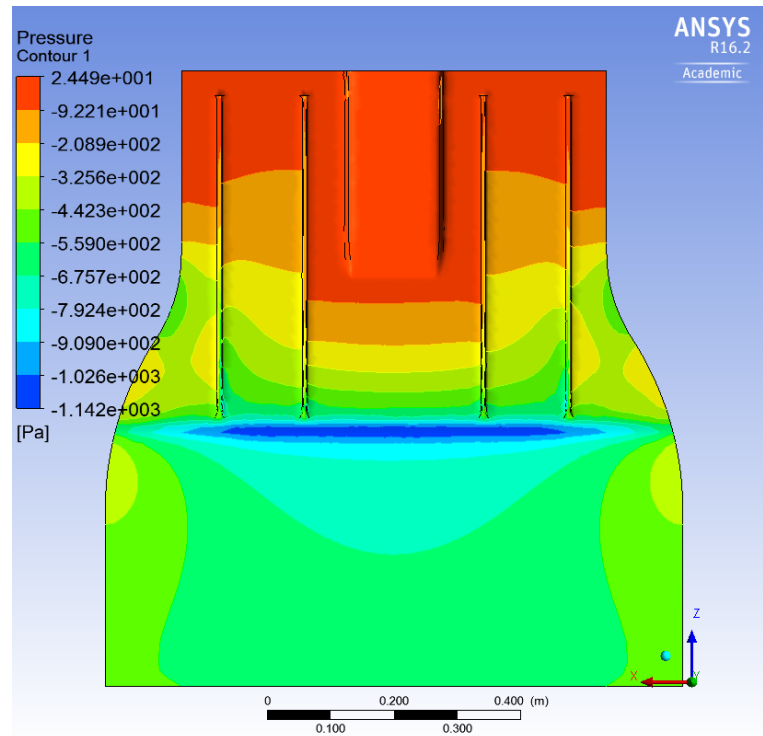


Figure 39: Contour plot of the pressure.

The main objective of this fourth simulation and the one that is responsible for this better results was to have a good pressure distribution on the diffuser. The next contour plot, in figure 39, shows how this has been possible. We have succeeded to eliminate the areas of high pressure at the laterals and there is a greater area of high pressure at the back of the diffuser. The area of low pressure and greater velocity is concentrated in the angle, where it should be. The pressure maximum difference, despite its absolute value is a bit lower than in previous simulation, 1166.49 Pa, it keeps its magnitude and considering now the air is flowing properly.

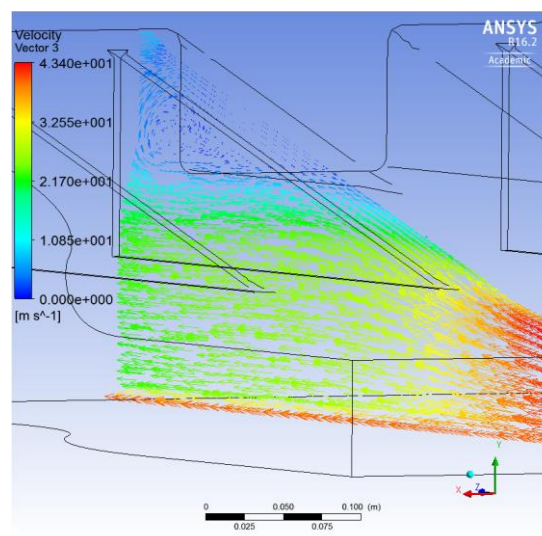


Figure 40: Vector plot showing the boundary layer detachment.

Nevertheless, we also have to comment on those things that are still a problem for the diffuser and have to be improved. Despite it was not the main purpose of this new design, we still have a boundary layer separation at the back of the diffuser next to the central element, as shown in the figure below.

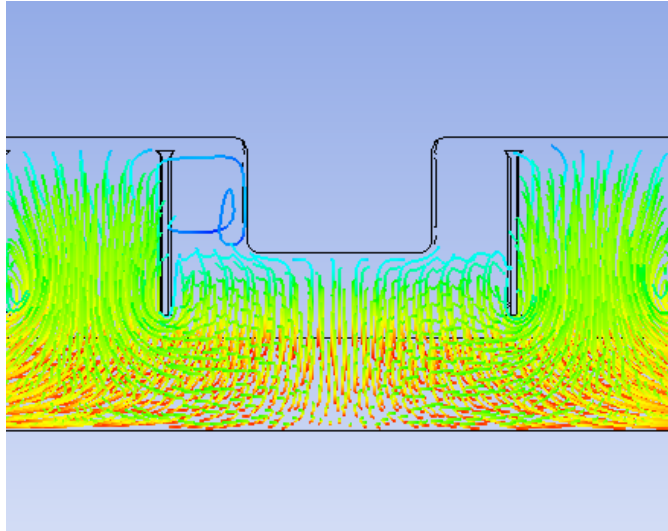


Figure 41: Streamline born in the inlet and seen from the back.

The streamline certifies this problem. A streamline created from the inlet and seen from the back (figure 41) makes clear the boundary detachment as there are no lines flowing in that area. If we look at a streamline born from the wall (figure 42) we will clearly see all the lines representing the turbulence due to the boundary layer separation creating an adverse flow.

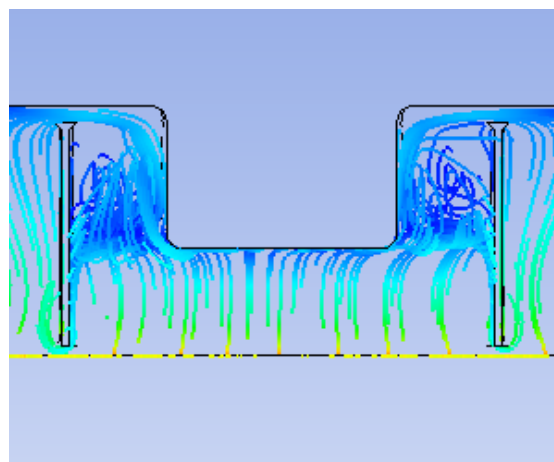


Figure 42: Streamline seen from the back and born from the wall.

The fifth and final geometry will be pointed in the direction of eliminating this turbulence and boundary detachment.

8.5. Fifth Analysis

8.5.1. Geometry

In order to achieve this last goal it has been decided to soften the transition between the flat part (bottom of the car) and the diffuser slope. See figure below:

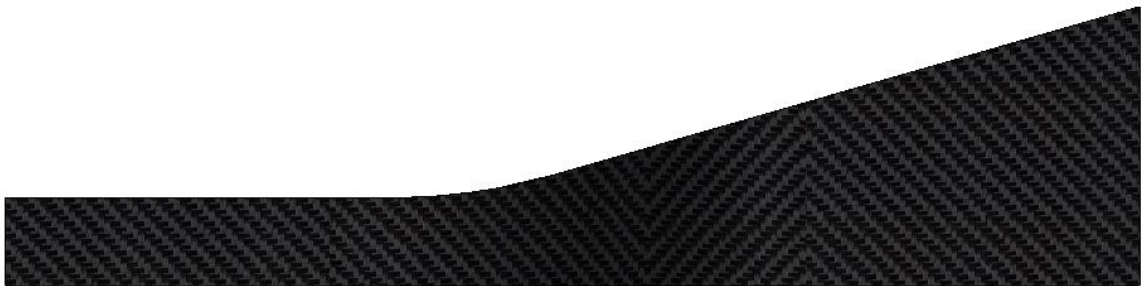


Figure 43: Lateral view of the 5th geometry.

It is believed that with this new design the air will not find it so difficult to attach to the wall of the diffuser since the curvature is not so abrupt and this way we will be able to avoid any boundary layer separation.

8.5.2. Results

Unfortunately this time we cannot say we have achieved the goals of this fifth simulation and design. As you can see in the vector plot (Figure 44) we still have a boundary layer separation in the same area as before, even more notable. We have not succeeded to solve that problem.

In terms of down force and drag we have obtained a very similar results compared to the prior simulation. Actually, the drag force and drag coefficient values are almost the same. The down force is a bit lower and so is the Lift Coefficient ($F = 351.88 \text{ N}$, $C_l = 0.483$)

So it seems clear that the problem causing that boundary layer separation is not the angle abrupt curvature and in order to improve this matter other consideration should be taken into account. This possible solutions will be comment on the next point.

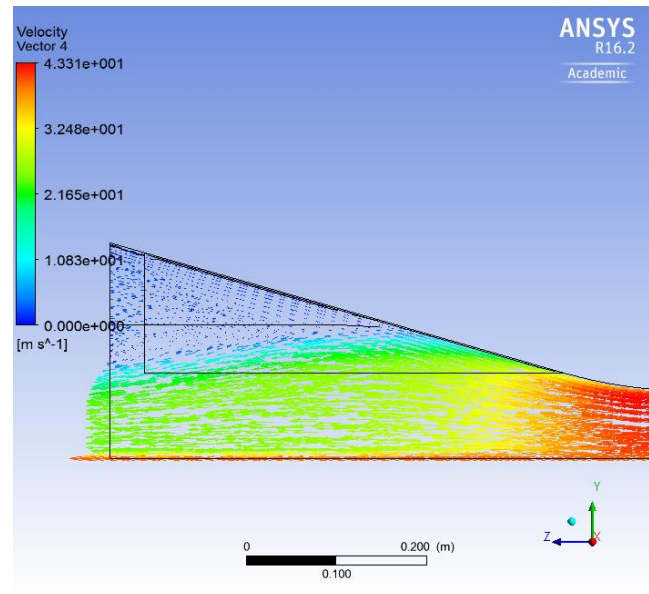


Figure 44: Vector plot showing boundary layer separation and adverse flow.

8.6. Possible ways to further improve on the design of this diffuser.

After five different simulations and geometries we have succeed to complete a great improvement on the diffuser efficiency but it has also been clear we have not reached the perfect design. On possible way to try to improve the diffuser would be to change the angle, although this is not believed to be the problem of the diffuser since the research on this matter was very extensive it exists the possibility that for this specific design another angle would have helped to improve its efficiency.

Another possibility that could have been tried out is to have designed curved wings instead of the straight ones always used for this diffuser. This may have helped to take better advantage of the effect of the double diffuser.

We could have also tried to simulate the different designs for different velocities, being able to identify at what speeds work better each component of the diffuser and thus modifying its elements to achieve a higher performance and efficiency.

And finally modification of the central piece of the diffuser, making it a little bit less squared could also be the solution to the small turbulence still found next to it.

8.7. The importance of the mesh and Turbulence Model.

It was also considered essential in this project to justify the use of a certain turbulence model instead of the two other existing and why should we mesh as precisely and refined as it was done before taking any final conclusion.

8.7.1. Mesh importance

In order to show how important the mesh is to obtain accurate results it was decided to use the geometry of the fifth simulation and mesh it with the default mesh, no inflation was applied, a bigger element size was chosen (30 mm, three times bigger) and the optional refined mesh on the curvature was left active. First of all they were both checked for orthogonal Quality and skewness statistic parameters to make sure that they were both okay to run a simulation. But although the present similar values for this two parameters the difference will be notable on the results.

A vector plot was taken at the post processing stage to highlight the difference of results with difference meshes. We can see that according to this simulation there is no boundary layer separation. The lack of an inflation that allows a good analysis of the air flow close to the walls could have lead us to believe that the diffuser design would have no flow detachment and that it did not need any further improvement regarding that matter.

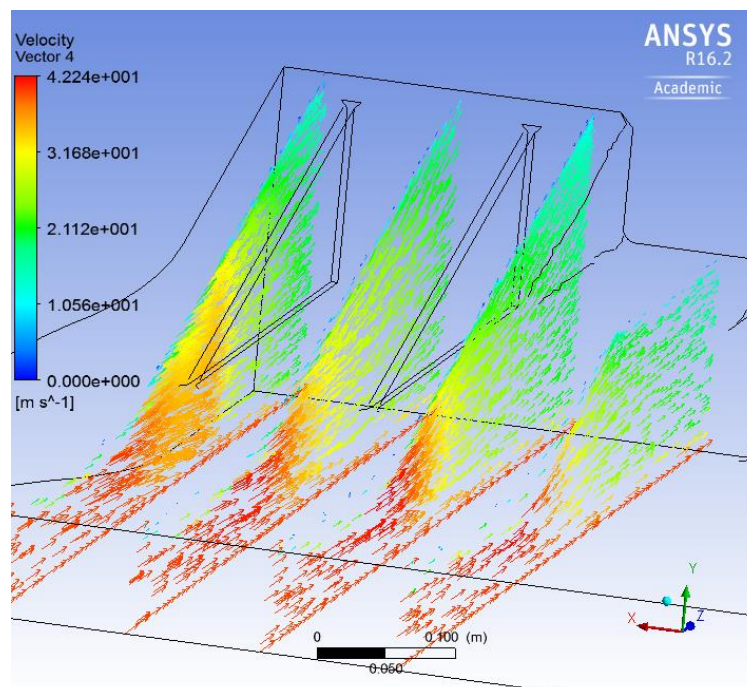


Figure 45: Vector plot on different planes.

Another fact that makes clear that this mesh is not sufficiently good is the momentum and mass residuals. As we have previously commented, in order to have carried out a good simulation these values have to be stable at the end of it, something that did not happen in this case as shown in figure 46.

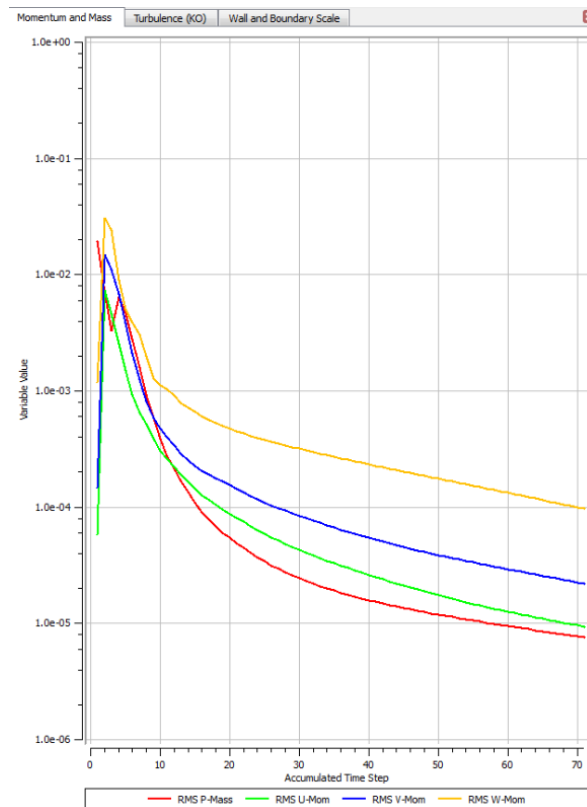


Figure 46: Residuals for mass and momentum.

8.7.2. Turbulence models

As explained at the beginning of this chapter of the project, the turbulence model used to carry out the simulation was SST (Shear Stress Transport), but there were two other possible models to be chosen: K-Epsilon and K- ω

Let's see now the pros and cons of each model and discuss about the choice made:

K-Epsilon Model

- Pros: Robust. Widely used despite the known limitations of the model. Easy to implement. Computationally cheap. Valid for fully turbulent flows only.

Suitable for initial iterations, initial screening of alternative designs, and parametric studies.

- Cons: Performs poorly for complex flows involving severe pressure gradient, separation and strong streamline curvature. Most disturbing weakness is lack of sensitivity to adverse pressure gradients; another shortcoming is numerical stiffness when equations are integrated through the viscous sublayer which are treated with damping functions that have stability issues

K- ω Model

- Pros: Superior performance for wall-bounded boundary layer, free shear, and low Reynolds number flows. Suitable for complex boundary layer flows under adverse pressure gradient and separation (external aerodynamics and turbomachinery). Can be used for transitional flows (though tends to predict early transition).
- Cons: Separation is typically predicted to be excessive and early. Requires mesh resolution near the wall. [4]

SST Model

Shear Stress Transport (SST) is a variant of the standard k- ω model. Combines the original Wilcox k- ω model for use near walls and the standard k- ϵ model away from walls using a blending function, and the eddy viscosity formulation is modified to account for the transport effects of the principle turbulent shear stress, so it basically benefits from both previous models explained. [5]

- Pros: Offers similar benefits as standard k- ω . The SST model accounts for the transport of turbulent shear stress and gives highly accurate predictions of the onset and the amount of flow separation under adverse pressure gradients. SST is recommended for high accuracy boundary layer simulations.
- Cons: Dependency on wall distance makes this less suitable for free shear flows compared to standard k- ω . Requires mesh resolution near the wall.

After exposing all the characteristics of this three different models it is pretty clear that for the case that concerns us, the more accurate and precise model is SST. A model that gathers the advantages of the two other models offering a very good simulation close and far from the wall, thus representing the effect of the boundary layer separation very close to reality and reducing to a few its computational limitations.

In order to make this choice even more convincing it was decided to run a simulation with the last geometry designed changing the turbulence model to K-Epsilon to prove how the results change and the cons of this model are more than obvious.

First of all, let's take a look at the next figures that highlight one of the greater differences of using this method. Look where the boundary layer separation takes place in the middle part of the diffuser, just after the angle, creating a heavy adverse flow in the center of the diffuser. But this is not the only place where we can find a detachment of the boundary layer using this turbulence model, if we take a look at figure 47 we can clearly appreciate how the air does not keep attached to the upper wall of the control volume at the lateral area of the diffuser.

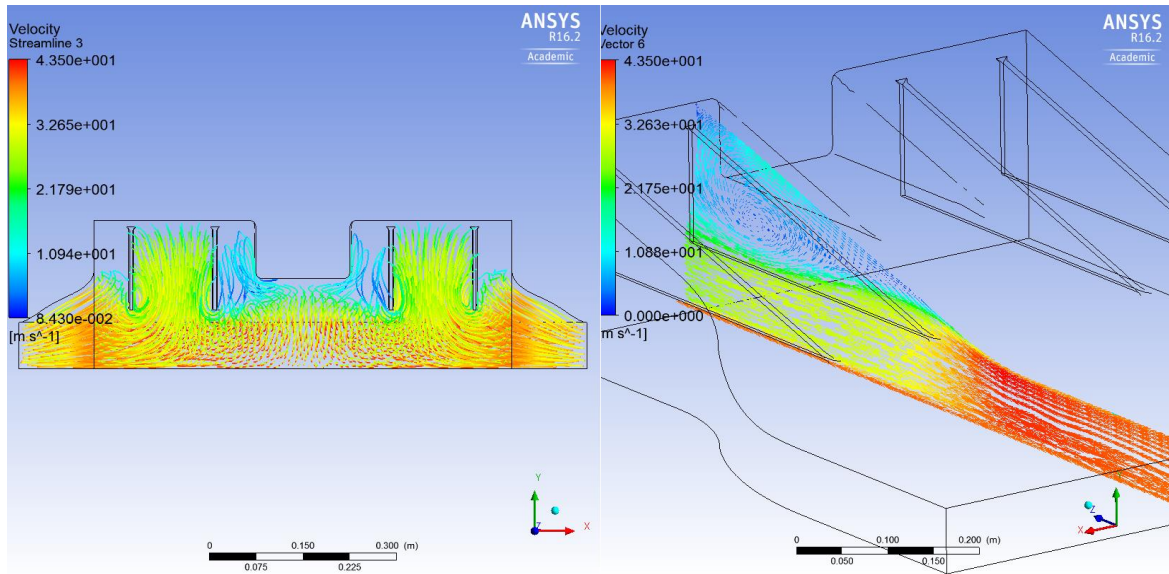


Figure 47: Streamline and vector plot showing boundary layer detachment.

Another difference that we can find is the pressure distribution and pressure gradient. The first one mentioned it is not uniform and the second one is low due to the turbulence created under the diffuser. On top of that and as a consequence of the prior differences the value of the downforce is lower and the drag force is higher which result in a decrease of the diffuser's efficiency according to this simulation

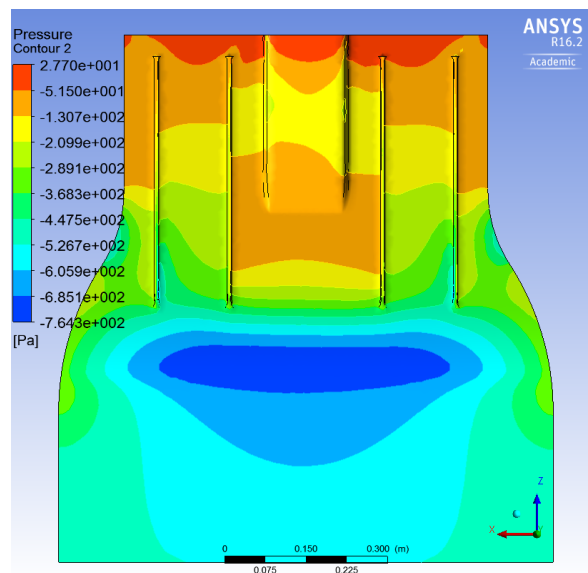


Figure 48: Contour plot of the pressure.

Basically this simulation, compared to the one carried out with the SST model, just shows all the weakness of the K-Epsilon model and lets us insist on the importance of a good research and gathering of information before using a key element of a simulation such as the turbulence model, that could completely change the final results.

8.8. Comparative analysis and numerical conclusions

After five simulations we believe it's appropriate to gather all the numerical information in a table that will help us see more clearly which has been the more efficient geometry tested and hence the one to build and be used by a racing car.

	Down Force	Drag Force	Lift Coeff.	Drag Coeff.	Difference between Max. and Min. pressures on the diffuser.
1 st Simulation	143.61 N	22.02 N	-0.197	0.126	Not significant
2 nd Simulation	333.25 N	28.28 N	-0.457	0.162	1273.28 Pa
3 rd Simulation	329.17 N	19.85 N	-0.451	0.123	1260.98 Pa
4 th Simulation	379.27 N	14.81 N	-0.521	0.092	1166.49 Pa
5 th Simulation	351.88 N	16,75 N	-0.483	0.104	868.71 Pa

Table 1: Numerical results for forces, coefficients and pressure difference on the five simulations.

Simply looking at the numbers and having in mind the goal and purpose of a diffuser it is logic to conclude that the final geometry to be used should be between the fourth and the fifth. The values that the fourth design presents are slightly better and considerably better than the other geometries. Moreover, in terms of building it and assembling it into the car is probably better than the fifth, since normally the space under the racing cars is small and there is no place for very smooth curvatures like the one applied for the fifth geometry.

So after this study, the geometry to be assembled in a formula 3 racing car would be the fourth, but of course the correct way to proceed would be to continue with the study of the diffuser until it's efficiency reaches higher standards, a job that is out of the reach of this project.

Finally we have to compare these results to the ones that an actual formula 3 car can obtain. The downforce generated by the diffuser of these racing cars is between 70 and 80 kg, reaching normally values of the lift coefficient of 0.7 roughly speaking. It is clear we

haven't been able to reach such high and effective values, but I consider this study, assuming the knowledge and tools at our hand, has had a great outcome. We have obtained, in a very limited study, results close to reality and that would make a great difference on a racing car in terms of high speed cornering.

9. Environmental Impact

Due to the nature of this study it has been considered that the environmental impact study must be focused on the savings that account for the use of CFD tests instead of using the wind tunnel or measuring on the track while racing with the car.

The automobile competitions have captured the attention of some environmental groups for the simple reason of consuming fuel and thus contaminating for purely recreational or show purposes without control of the emissions of CO₂ to the atmosphere or the levels of fuel consumption of the racing cars. Keep in mind that we have to take into account this lack of commitment with the environment from the racing competition during the stage of development of the car since this project is based in a job carried out during the preseason.

The energy savings that resulted from CFD study towards the same study in real conditions with a wind tunnel testing, circuit and fuel transport are considered and discussed below:

- Impact of the use of wind tunnel:

A wind tunnel as the ones used in Formula 1 models restricted to 60% of their original size and with a limited speed of 150km/h, has a consumption of 2MW/h, regardless of the electronic measuring equipment consumption.



Figure 49: Red bull's wind tunnel.

This means that with this study we are heavily reducing the power consumption, although the price of a kW of power is not as expensive as the price of 1 l of fuel, it is the great amount of power that we use that turns into a great saving the realization of this project.

According to different studies on how much does it cost to run a F1 team, the average expense on wind tunnel testing of a team of F1 is 16 M. It might not be such high in Formula 3, but we can assume an expense of 10 M. With that number in mind, we can easily make an idea of how many energy is consumed during those tests.

The money invested to build a wind tunnel can reach a value of 150 M. but formula 3 teams don't usually own a wind tunnel, they rent it.

- Impact of testing at the circuit:

In order to estimate the impact of all the factors that can be considered to evaluate the impact of the preseason testing of a racing team we would have to make some assumptions on the price of the gasoil, the consumption of the racing cars or the km travelled.

What is important to be aware of is that by carrying a study on a computer we can save the expenses of the trucks transporting al the required material to the circuit plus the fuel needed for this trucks. We would also save all the fuel consumed by the racing car, tires, spare parts needed, and the transport of all the team to the testing area.

All this previously mentioned can be considered as a big reduction on the emission of contaminating and harmful gases for the atmosphere and the consumption on non-renewable sources.

10. Budget of the project

To make the economic analysis of the project the costs of four different items have been taken into account. These studies are carried out with a computer without using consumables because the costs associated with the manufacture of the designed diffuser are not considered.

The item of Human Resource includes the costs related to the staff that has been involved in the project, which are basically the author of the project and the external assistance received from professors of the faculty. Keep in mind that the cost associated with each phase of the process varies, therefore in this item are included: the time spent collecting the information, the time spent on processing and analyzing the information gathered, the time spent on designing in SolidWorks, the time spent testing with ANSYS CFX and finally the time devoted to the drafting of all documentation. It is assumed that the engineer has worked an average of five days a week at the rate of 2 hours a day, and that external assistance has been one hour a week. The economic cost of this time has been evaluated assuming the cost of an engineer working as an intern, ranging from 6 to 10 € / h depending on the process and a cost of 25 € / h on average for the help of professors from the faculty.

Regarding the item of Software, we must consider the licenses of SolidWorks and ANSYS design and simulation softwares and Microsoft Office for preparing the report. Consider licenses within a period of six months, coinciding with the duration of the project.

The personal computer means an expense of 800 € with a useful life of five years. Depreciation in half a year is therefore 80 €.

Referring to the power consumption of the personal computer (Power 1KW), consider the price of electricity of 0.15 € / kWh, with a use of 500 hours.

We must also count the industrial profit tax by the author of the project, 15% considered and finally account for VAT of 21%.

Below we can find table 2, a summary of the project's budget breakdown and the hours spent on each task.

Human Resources	Cost [€/h]	Invested Time [h]		Total [€]
Information gathering	6	50		300
Analysis of the information	6	55		330
Solid Works design	8	25		200
CFX Simulation	8	85		680
Report editing	8	75		600
External assistance	25	25		625
Total		315		2.735 [1]
Software	Price [€]	License rights [years]	Time spent [years]	Total [€]
SolidWorks	6000	2	0,5	1.500
Ansys CFX	10000	2	0,5	2500
Microsoft office	100	1	0,5	50
Total				4050 [2]
Machinery	Price [€]	Useful life [years]	Time spent [years]	Total [€]
Personal Computer	1000	5	0,5	100 [3]
Electric consumption	Cost [€/kWh]	Power [kW]	Time spent [h]	Total [€]

Personal Computer	0,15	1	315	47,25 [4]
Total expense				Total [€]
Before benefits and taxes	[1]+[2]+[3]+[4]			6932,25 [5]
Industrial Benefit	15% de [5]			1039,84 [6]
After industrial benefit	[5] + [6]			7972,09 [7]
IVA	21% de [7]			1674,14 [8]
After taxes	[7]+[8]			9646,23 €

Table 2: Table for total costs.

In the graphic below we can see in a different way how the different items of the project contribute to the total expense. The most significant items are Human Resources and Software. The latter is actually the higher cost, licenses for very powerful and resourceful programs are very expensive but they are clearly worth paying for when we compare the cost to the one generated by an air tunnel.

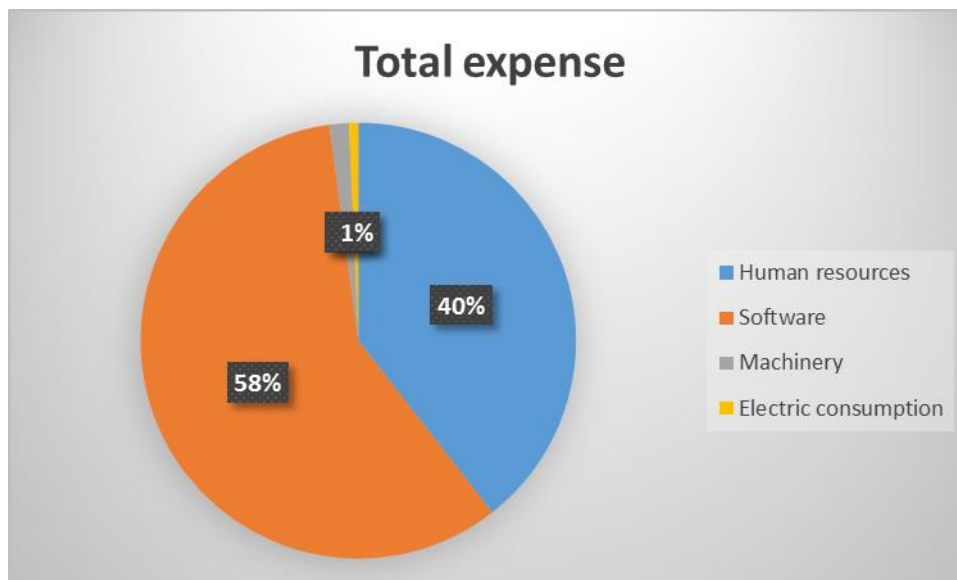


Figure 50: Graph for each item of the project cost.

11. Project Planning

This project has been carried out since the beginning of February until the end of August of 2016. Figure XX shows the Gantt diagram in which we can see the most important stages representing the realization of the project and also its duration has been indicated.

The important stages in Table 3, match the sections of the project. These are the sections that were considered most relevant and interesting to study when a brainstorming, programming and project's reach evaluation was done at the beginning of the semester, when the topic to study was chosen.

These sections have been carried out generally in order, however, in some cases, two or more sections could have been carried out at the same time. The work was structured so that, as the project was evolving, we were able to understand every step taken and every conclusion determined.

Finally, consider the time spent by drafting documentation. This has been carried out throughout the semester and once objectives were being fulfilled and sections were being finished.

Activity	Months													
	February		March		April		May		June		July		August	
Brainstorming														
Introduction														
Aerodynamics														
Formula 3 Introduction														
Diffuser														
Flow Simulation Program														
CFX Analysis														
Environmental impact														
Budget of the project														
Conclusion														
Report writing														

Table 3: Project planning.

Conclusions

After days of hard work invested on this project it is time now to discuss and present all the learnings and knowledge obtain through the realization of this work.

Investigating the world of aerodynamics has made me realize how complex it can be to design any aerodynamic element of a car, the numerous hours of work behind those cars we watch on TV and enjoy watching them fight for the victory.

The smallest detail in the design of any aerodynamic piece can mean the world when it comes to racing, a better design than your opponents can make your car a winner. It might be difficult to believe how something that we are not even able to see can make such a great difference on the track but aerodynamics is a complex science, that needs of great effort and research in order to be successful.

Thanks to this project we have also been able to see how all the concepts learned in courses such as Fluid Mechanics are extensively used in big competitions like Formula 3, that those things that may seem very basic sometimes at class are the cornerstones to a great science.

During this project we have also come to realize the power and usefulness of a simulation software such as ANSYS CFX or any other branch of ANSYS.

It is also important to highlight the results after having carried out 5 different simulations. This study presents a 5 steps evolution of an initial diffuser design in which clue parameters and learning have been determined.

The buyer of this study will benefit of a very developed diffuser design with a great efficiency. For a very low price compared with the normal amount of money a racing team would pay to develop a diffuser we have designed one that is able to create close to 40 kg of downforce generating almost no drag force on the car. The study will also allow the buyer to learn how to take into account the double diffusor effect, take a measure of how important can be the effect of the air flowing through the laterals of the car being recirculated under the car and expelled at the back.

He will also learn about the key parameters of a diffuser, how to modify them to make the diffuser more efficient and the car more competitive. In only five different designs it has been possible to reach values close to professional designs. This has been a limited study with the goal of presenting the great work that can be done with Simulation softwares, but for further studies with more time invested on designing and testing much more efficient

diffusers could be developed.

This design presented some limitations in terms of working hours but also in terms of design. In further studies a new design of both the double diffuser and the diffuser itself could be carried out. They both would be done separately, getting into more detail in the design of both pieces which would for sure generate better results. The design on this study wanted to show how both elements of a racing car contribute to the down force generation, however, this brought some limitations to the study: Normally the outlet area of a diffuser is bigger, the air also expands to the sides besides of the vertical direction, this generates a greater pressure gradient that turns into higher downforce. The control volume considered was small, but actually this diffuser built on a racing car would generate a greater gradient pressure and therefore a greater down force. Also the air normally flowing next to the car is more than the one considered in this study, another reason why we think that our diffuser can present in real life a greater efficiency than the one numerically stated in this project.

Acknowledgements

First of all I would like to thank my professor Enric Trillas for accepting and giving me the opportunity to carry out a project of my choice and the support and assistance given along the semester whenever it has been requested.

In second place I would like to thank all the person that have also helped me during this last steps of the bachelor's degree and namely this project. Thanks to Roger Lloret and Esteve Baraut for all the advices and assistance offered that has kept me going in this project and has made the path a lot easier.

I would also like to thank all those professors I have had during this career and that devote their life so that the future generations can have a brilliant professional life. Thank for all the lessons taught and all the things I have learned that would make of me a more prepared engineer.

Finally I would also like to acknowledge the support of my family that is always of great importance in every decision I take and face.

Bibliography

Bibliographic References

- [1] Han, T. Hammond Jr., D.C. and Sagi, C.J. Optimisation of Bluff-Body Rear End Shape for Minimum Drag in Ground Proximity, Vehicle Aerodynamics, PT-49, Edited by Dr. V. Sumantran and Dr. G. Sovran, pp-341,
- [2] Cooper, K. R. Bertenyi, T. Dutil, G. Syms, J. Sovran, G. The Aerodynamic Performance of Automotive Underbody Diffusers, SAE 980030, 1998
- [3] Sovran, G. Klomp, E.D. – Experimentally Determined Optimum Geometries for Rectilinear Diffusers with Rectangular Conical or Annular Cross-Section, Synopsium on Fluid Mechanics of Internal Flow, Elsevier Publishing, Amsterdam, 1967
- [4] F. R. Menter, “Zonal Two Equation k-w Turbulence Models for Aerodynamic Flows,” AIAA Paper #93-2906, 24th Fluid Dynamics Conference, July 1993; F. R. Menter, “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications,” AIAA Journal, vol. 32, no. 8, pp. 1598-1605, 1994
- [5] F. R. Menter, “Zonal Two Equation k-w Turbulence Models for Aerodynamic Flows,” AIAA Paper #93-2906, 24th Fluid Dynamics Conference, July 1993; F. R. Menter, “Two-Equation Eddy-Viscosity Turbulence Models for Engineering Applications,” AIAA Journal, vol. 32, no. 8, pp. 1598-1605, 1994

Complementary bibliography

- [6] Appendix J, Article 275 2003 – Formula 3 Technical Regulations, Federation Internationale De L'Automobile.
- [7] Jimenez, J. Perspective in Fluid Dynamics: A Collective Introduction to Current Research, Chapter 5 „Turbulence”, Cambridge University Press, 2003.
- [8] Aroussi, A. Measurement and Prediction of Recirculating Flows in a “T” Shaped Cavity, Computational Fluid Dynamics, G. de Vahl Davis and C. Fletcher, 1988, p229
- [9] Saif-Deen A, Journal articles by Fluent Software Users (1999) Running RAMPANT: Computational Fluid Dynamics In Formula Design,

- [10] Prada y Nogueira, Isaac. El túnel de viento como herramienta de ensayo aerodinámico en la Fórmula 1. Madrid. 2006.
- [11] Professor K. P. Garry. *An Introduction to Boundary Layers*. Cranfield. 2014
- [12] Frank M. White. *Mecánica de Fluidos*. Madrid: McGRAW-HILL, 1990
- [13] Tu, J., Yeoh, G.H., Liu, C., *Computational Fluid Dynamics: A Practical Approach*, Elsevier, Oxford, UK, 2008.
- [14] Nunn, R., *Intermediate Fluid Mechanics*, Hemisphere Publishing Corporation, New York, NY, 1989, pp. 167-178.
- [15] Launder, B.E., Spalding, D.B., "The Numerical Computation of Turbulent Flows", *Computer Methods in Applied Mechanics and Engineering*, Volume 3, Issue 2, 1974, pp. 269-289.
- [16] ANSYS 12.0: ANSYS FLUENT Theory Guide"