ESCUELA TÉCNICA SUPERIOR DE INGENIERÍA (ICAI)

GRADO EN INGENIERÍA ELECTROMECÁNICA

Especialidad Mecánica

STUDY AND IMPROVEMENTS IN AERODYNAMICS OF A TOURISM

Autor: Rodrigo Sánchez de Rojas Lozano

Director: Luis Manuel Mochón Castro

Madrid

Agosto 2018

AUTORIZACIÓN PARA LA DIGITALIZACIÓN, DEPÓSITO Y DIVULGACIÓN EN RED DE PROYECTOS FIN DE GRADO, FIN DE MÁSTER, TESINAS O MEMORIAS DE BACHILLERATO

1º. Declaración de la autoría y acreditación de la misma.

El autor D. Rodrigo Sanchez de Rojas Lozano

DECLARA ser el titular de los derechos de propiedad intelectual de la obra: Study and improvements in aerodynamics for tourism.

que ésta es una obra original, y que ostenta la condición de autor en el sentido que otorga la Ley de Propiedad Intelectual.

2º. Objeto y fines de la cesión.

Con el fin de dar la máxima difusión a la obra citada a través del Repositorio institucional de la Universidad, el autor **CEDE** a la Universidad Pontificia Comillas, de forma gratuita y no exclusiva, por el máximo plazo legal y con ámbito universal, los derechos de digitalización, de archivo, de reproducción, de distribución y de comunicación pública, incluido el derecho de puesta a disposición electrónica, tal y como se describen en la Ley de Propiedad Intelectual. El derecho de transformación se cede a los únicos efectos de lo dispuesto en la letra a) del apartado siguiente.

3º. Condiciones de la cesión y acceso

Sin perjuicio de la titularidad de la obra, que sigue correspondiendo a su autor, la cesión de derechos contemplada en esta licencia habilita para:

a) Transformarla con el fin de adaptarla a cualquier tecnología que permita incorporarla a internet y hacerla accesible; incorporar metadatos para realizar el registro de la obra e incorporar "marcas de agua" o cualquier otro sistema de seguridad o de protección.
b) Reproducirla en un soporte digital para su incorporación a una base de datos electrónica, incluyendo el derecho de reproducir y almacenar la obra en servidores, a los efectos de garantizar su seguridad, conservación y preservar el formato.
c) Comunicarla, por defecto, a través de un archivo institucional abierto, accesible de modo libre y gratuito a través de internet.
d) Cualquier otra forma de acceso (restringido, embargado, cerrado) deberá solicitarse expresamente y obedecer a causas justificadas.
e) Asignar por defecto a estos trabajos una licencia Creative Commons.
f) Asignar por defecto a estos trabajos un HANDLE (URL *persistente)*.

4º. Derechos del autor.

El autor, en tanto que titular de una obra tiene derecho a:

a) Que la Universidad identifique claramente su nombre como autor de la misma
b) Comunicar y dar publicidad a la obra en la versión que ceda y en otras posteriores a través de cualquier medio.
c) Solicitar la retirada de la obra del repositorio por causa justificada.
d) Recibir notificación fehaciente de cualquier reclamación que puedan formular terceras personas en relación con la obra y, en particular, de reclamaciones relativas a los derechos de propiedad intelectual sobre ella.

5º. Deberes del autor.

El autor se compromete a:

a) Garantizar que el compromiso que adquiere mediante el presente escrito no infringe ningún derecho de terceros, ya sean de propiedad industrial, intelectual o cualquier otro.
b) Garantizar que el contenido de las obras no atenta contra los derechos al honor, a la intimidad y a la imagen de terceros.
c) Asumir toda reclamación o responsabilidad, incluyendo las indemnizaciones por daños, que pudieran ejercitarse contra la Universidad por terceros que vieran infringidos sus derechos e

intereses a causa de la cesión.

d) Asumir la responsabilidad en el caso de que las instituciones fueran condenadas por infracción de derechos derivada de las obras objeto de la cesión.

6º. Fines y funcionamiento del Repositorio Institucional.

La obra se pondrá a disposición de los usuarios para que hagan de ella un uso justo y respetuoso con los derechos del autor, según lo permitido por la legislación aplicable, y con fines de estudio, investigación, o cualquier otro fin lícito. Con dicha finalidad, la Universidad asume los siguientes deberes y se reserva las siguientes facultades:

- La Universidad informará a los usuarios del archivo sobre los usos permitidos, y no garantiza ni asume responsabilidad alguna por otras formas en que los usuarios hagan un uso posterior de las obras no conforme con la legislación vigente. El uso posterior, más allá de la copia privada, requerirá que se cite la fuente y se reconozca la autoría, que no se obtenga beneficio comercial, y que no se realicen obras derivadas.
- La Universidad no revisará el contenido de las obras, que en todo caso permanecerá bajo la responsabilidad exclusive del autor y no estará obligada a ejercitar acciones legales en nombre del autor en el supuesto de infracciones a derechos de propiedad intelectual derivados del depósito y archivo de las obras. El autor renuncia a cualquier reclamación frente a la Universidad por las formas no ajustadas a la legislación vigente en que los usuarios hagan uso de las obras.
- La Universidad adoptará las medidas necesarias para la preservación de la obra en un futuro.
- La Universidad se reserva la facultad de retirar la obra, previa notificación al autor, en supuestos suficientemente justificados, o en caso de reclamaciones de terceros.

Madrid, a 30 de AGOSTO de 2018

ACEPTA

Fdo...

Motivos para solicitar el acceso restringido, cerrado o embargado del trabajo en el Repositorio Institucional:

ESTUDIO Y MEJORAS EN LA AERODINAMICA DE UN TURISMO

Autor: Sánchez de Rojas Lozano, Rodrigo

Directores: Mochón Castro, Luis Manuel

Entidad Colaboradora: ICAI - Universidad Pontificia Comillas

Partiendo del diseño de un prototipo en tres dimensiones de un vehículo antiguo, es posible hacer un estudio del comportamiento aerodinámico del mismo. Una vez realizado dicho estudio, se podrá implementar nuevos dispositivos y piezas que nos permitan alcanzar una mejora de los coeficientes aerodinámicos del modelo original. Finalmente, mediante un proceso de simulación por medio de softwares de CFD, se han obtenido los resultados aerodinámicos del modelo final, pudiendo observar mejoras con respecto al prototipo antiguo.

Declaro, bajo mi responsabilidad, que el Proyecto presentado con el título STUDY AND IMPROVEMENTS IN AERODYNAMICS OF A TOURISM en la ETS de Ingeniería - ICAI de la Universidad Pontificia Comillas en el curso académico 2017/2018 es de mi autoría, original e inédito y no ha sido presentado con anterioridad a otros efectos. El Proyecto no es plagio de otro, ni total ni parcialmente y la información que ha sido tomada de otros documentos está debidamente referenciada.

Fdo.: RODRIGO SÁNCHEZ DE ROJAS LOZANO Fecha: 30/ 08/ 2018

Autorizada la entrega del proyecto

EL DIRECTOR DEL PROYECTO

Fdo.: LUIS MANUEL MOCHÓN CASTRO Fecha: 30 / 08 / 2018

STUDY AND IMPROVEMENTS IN AERODYNAMICS OF A TOURISM

Author: Sánchez de Rojas Lozano, Rodrigo

Directors: Mochón Castro, Luis Manuel

Collaborating Entity: ICAI - Universidad Pontificia Comillas

Starting from the design of a prototype in three dimensions of an old vehicle, it is possible to make a study of the aerodynamic behavior of it. Once this study is done, new devices and parts can be implemented that allow us to achieve an improvement of the aerodynamic coefficients of the original model. Finally, through a simulation process through CFD software, the aerodynamic results of the final model have been obtained, being able to observe improvements and obtain conclusions, with respect to the old prototype.

Contents

1. Introduction

1.1 Background

Formerly, there was nothing related to aerodynamics in the earliest car designs. Those shapes did not have anything similar with a teardrop form, which was considered the best aerodynamic reference taken from nature. On the contrary, the former cars reminded us of a horse carriage. However, as long as those cars were not fast, it did not matter. It was not until the beginning of the 20th century when car designers start thinking about a better way to travel through the air.

In 1920´s a German designer and, in 1930´s an American designer, decided to create two car models taking into account the shape of the chassis in order to reach a low resistance against the wind. Unfortunately, these new designs coincided with the Second World War and they had to wait around 20 years to start selling those models. By that time, the world had changed. Everything was evolving towards manufacturing vehicles in big scale. This fact, together with a good period of low prices of fuel, left apart from the awareness of aerodynamics.

Finally, a sharp rise of the fuel price in the 1970´s forced designers to rethink the motor weights and the car bodies. As a result, it arose more compact cars and the return to explore new designs that optimized the aerodynamic. Most of the biggest advances come from racing. The engineers experimented with being aware of the fact that the airflow could help their cars to go faster and to be more stable at higher speeds.

Nowadays, the aerodynamic has become “invisible”. The car designs have focused on channelling the airflow in some way within the car and the shape does not depend that much on fluid forms.

1.2 Main purpose

This project was chosen in order to know what factors determine a body car design. As it was mentioned above, in the Early's 20^{th} century, the aerodynamics was not important as it is in the present. Applying the fluid dynamics principles learned during the degree and some new knowledge about aerodynamics, the purpose of this work have been a deep study of the effect of loads caused by the wind on an old car, and then try to get an advantage of them with some modifications on the original body.

1.3 Scope of work

For achieving that purpose, it was necessary to develop a work plan which mainly consisted in the following steps:

- Find information about aerodynamic devices implemented in cars, mainly from racing models.
- Design of the old vehicle BMW E34 through a CAD software called SolidWorks
- Aerodynamic measures in order to know which parts of the vehicle needed changes. This was developed with the CFD software Fluent ®
- Design of the parts that were going to be implemented in the original model.
- New flow simulations to check if the new design obtained better coefficients than the former car.

2. Theoretical fundaments

2.1 Basic principles

This section will be focused on stablishing what theoretical principles are going to be needed during the project. Therefore, any type of result obtained in section 3 and later is based on theorems and equations developed in this part. Not only that, all technical terms and scientific vocabulary used throughout this report will be also explained.

2.1.1 Bernoulli Equation

It was stated in 1738 in a textbook by Daniel Bernoulli and a complete derivation of the equation was given by Leonhard Euler. This equation has some restriction as all fluid are viscous and thus all flows have friction to some extent. Thus, for using it correctly, one must confine it to regions of the flow which are nearly frictionless (White, 2003). Besides, some assumptions must be considered to derive this equation:

- The fluid must be homogenous.
- Incompressible fluid without friction.
- There cannot be either energy losses or gains or heat transfer.
- The fluid must flow along a streamline.

Thereby, considering that a fluid particle is moving from a region which cross-sectional area A_1 to another A_2. The volume of fluid is constant ($V = A\,x = cte$). There are changes in the kinetic energy and potential energy which are defined in the following way:

$$\Delta E_c = \frac{1}{2} m (v_2^2 - v_1^2) \tag{1}$$

$$\Delta E_p = mg(h_2 - h_1) \tag{2}$$

Where m is the mass of the volume of fluid considered and which density will be:

$$\rho = \frac{m}{\Delta V} \tag{3}$$

Besides, each point is suffering a force due to the pressure:

$$F_1 = P_1 A_1 \tag{4}$$

$$F_2 = P_2 A_2 \tag{5}$$

Then, the work done on the fluid portion 1 and 2 can be expressed as:

$$W_1 = F_1 d = F_1 x_1 = P_1 A_1 x_1 \tag{6}$$

$$W_2 = F_2 d = -F_2 x_2 = -P_2 A_2 x_2 \tag{7}$$

$$W_{total} = W_1 + W_2 = P_1 A_1 x_1 - P_2 A_2 x_2 = P_1 V_1 - P_2 V_2 = P_1 \frac{\rho}{m} - P_2 \frac{\rho}{m} \tag{8}$$

Finally, applying the principle of energy conservation, the Bernoulli formula is reached.

$$\Delta E_c + \Delta E_p = W_{Total} \tag{9}$$

$$\frac{1}{2} m(v_2^2 - v_1^2) + mg(h_2 - h_1) = P_1 \frac{m}{\rho} - P_2 \frac{m}{\rho} \tag{9.1}$$

$$(P_2 - P_1) + \frac{1}{2}(v_2^2 - v_1^2) + \rho g(h_2 - h_1) = 0 \tag{9.2}$$

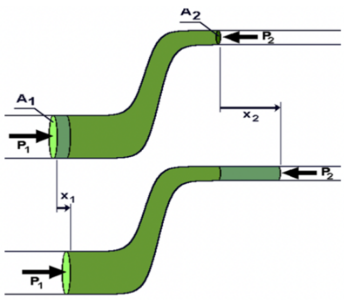


Figure 1 - Scheme for understanding the Bernoulli equation deduction

As a result of the equation (9.2), it is stated when the velocity of the flow increases, the pressure decreases at the same time.

2.1.2 Continuity Equation

When the law of mas conservation is written for a differential control volume, it is obtained the continuity equation. The general differential expression for this equation is:

$$\frac{\partial \rho}{\partial t} + \vec{\nabla}(\rho \vec{V}) = 0 \tag{10.1}$$

In our case, where the fluid is considered incompressible, the formula 10 can be rewritten as follows:

$$\vec{\nabla} \cdot \vec{V} = 0 \tag{10.2}$$

$$\frac{\partial \mathrm{u}}{\partial \mathrm{x}} + \frac{\partial \mathrm{v}}{\partial \mathrm{y}} + \frac{\partial \mathrm{w}}{\partial \mathrm{z}} = 0 \tag{10.3}$$

2.1.3 Navier-Stokes Equation

The Navier-Stokes equations are a set of partial derivative and non-linear equations which describes the fluid movement. These equations govern any phenomenon in which a Newtonian fluid is involved (Mora, 2016). A fluid is Newtonian when its viscosity can be considered constant. Thus, the Navier-Stokes equations are the base for the fluid mechanics in a high level. They are second order and unstable equations. Therefore, it is currently impossible to resolve them for all geometries. Hence, turbulent models in Computational fluid dynamics are needed in order to obtain a good approximation of the solution. Considering our fluid is incompressible and isothermal, the equations result as follow:

$$\rho \frac{D\vec{v}}{dt} = \rho \left(\frac{\partial \vec{v}}{\partial t} + (\vec{v} \ \cdot \nabla) \ \vec{v} \right) = -\nabla \vec{p} + \rho \vec{g} + \mu \nabla^2 \vec{v} \tag{11}$$

The formulas mentioned above, (10.3) and (11) are useful when computing the velocity and pressure field and boundary conditions known.

2.1.4 External and internal flow

When talking about aerodynamics, the main study is how cars in movement are subjected to forces caused by a mass of air which passes through them. Thus, it will be called aerodynamic resistance to the set of forces that acts in the opposite direction to the movement. In this sense, there are two ways for classify the flow.

On one hand, what is called internal flow. It is due to the air that passes through the interior of the body. This type is mainly used for motor refrigeration. However, it is very harmful to aerodynamics as it caused a big resistance against the movement of the car. Nowadays the research has focus on ways to channel this flow.

On the other hand, it is found the external flow. Since it is the main cause of the aerodynamic resistance, this project will focus on modifying the body of the vehicle with the aim of directing this air current so that the aerodynamics forces will be as less as possible.

Figure 2 - External flow over a car body

2.1.5 Laminar and Turbulent flows

When the flow velocity exceeds a certain value, the nature of it becomes very complex. This value is known as Reynolds number and it is an important characteristic when talking about fluid dynamics. This number is defined as the relation between inertial forces and viscous forces present in a fluid. Therefore, the formula of the Reynolds number is:

$$Re = \frac{\rho v D}{\mu} \tag{12}$$

The critical value mentioned above is set in $5 \cdot 10^5$ and it determinates when the fluid is, what is called laminar for lower values, or turbulent for higher values.

- Laminar flow. In this case, the flow is well organized and stratified. It is moving in parallels plates without mixing between each other. From the previous formula, it is deduced that this type of flow is found at low velocities and high viscosity.
- Turbulent flow. In contrast to the laminar flow, in this regime the flow is moving at high velocities and low viscosity, turning into turbulent. As a result, it is found a chaotic profile of the particles. Besides, they are moving to form small swirls called vortex.

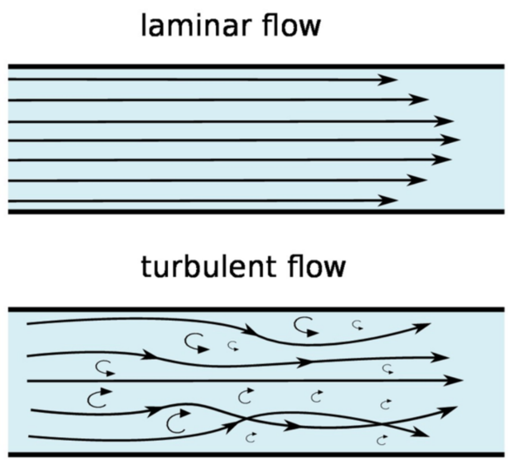


Figure 3 - Laminar and Turbulent velocity profiles

To conclude this section, it is necessary to remark that the fact that the flow becomes turbulent has a big influence in the aerodynamic resistance, reaching higher values of them as the flow becomes more turbulent.

2.1.6 The boundary Layer

"The boundary layer is a thin layer of viscous fluid close to the solid surface of a wall in contact with a moving stream in which the flow velocity varies from zero at the wall up to the free stream velocity" (Epifanov, 2014). Therefore, it makes sense to make a separate study between the fluid close to the surface and away from it. The concept of boundary layer was first suggested by Ludwig Prandtl in 1904.

The fact in which the velocity at the surface is zero is called the non-slip condition. Besides, this condition will lead to a reduction of the velocity resulting in a shear friction on the surface of the body. That is why is necessary to measure accurately the effect of the boundary layer, as the shear friction mentioned above will lead a force acting against the movement of the vehicle.

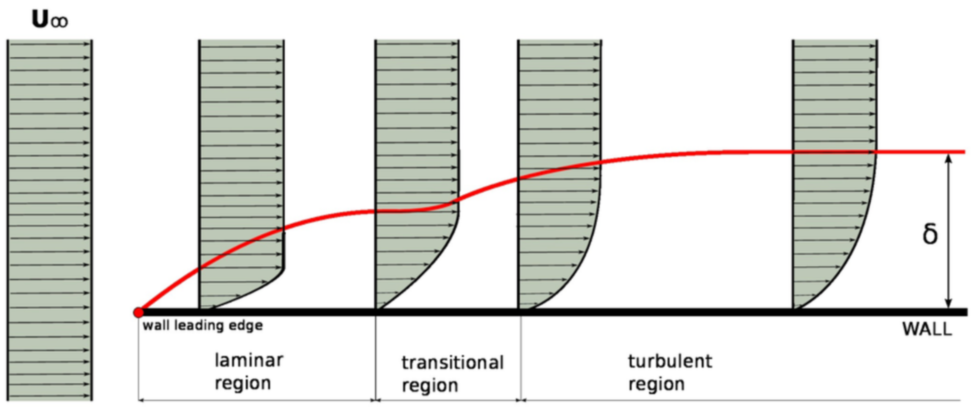


Figure 4 - Velocity profile within the boundary layer

The thickness of the boundary layer is denoted as δ and it is measured perpendicular to the surface from the bottom up to the point where the velocity is 99% of the free stream velocity. Throughout the surface, the thickness of the boundary layer is increasing, reaching considerable values at the end of the surfaces. Moreover, as it is shown in the picture, the boundary layer can be laminar, transitional (when it is mixed the laminar and flow regime) and turbulent.

2.1.7 Aerodynamic loads

When it was mentioned above the word aerodynamic resistance, it was referred to the forces that act directly to the car due to the air influence. These forces are very important in this project, as the main scope has been to take an advantage of them for achieving stability and less fuel consumption in high speeds.

Thus, it is distinguished by two types of aerodynamic loads mainly. On one hand, it is found what is called the drag force. It is defined as the force that acts in the opposite direction of the car movement. Therefore, it has a direct influence on the car consumption and must be reduced as much as possible.

On the other hand, the lift force. This force acts perpendicular to the car direction. The most important application of the lift is in aeronautics, since it is the reason why the planes are able to fly. However, what this project is looking for is the opposite than in planes. The lift in vehicle aerodynamics pursues to use the lift in order to make the vehicle goes as close as possible to the ground in order to reach stability.

The drag force can be expressed mathematically through the following formula:

$$F_D = C_D \left(\frac{\rho v^2}{2}\right) A \tag{13}$$

Where

- C_D is the drag coefficient. It is non-dimensional, and it depends on the geometry and the position respect to the fluid flow.
- A is the projected area of the body car in the flow direction.

In the same way, it is defined the lift force.

$$F_L = C_L \left(\frac{\rho v^2}{2}\right) A \tag{15}$$

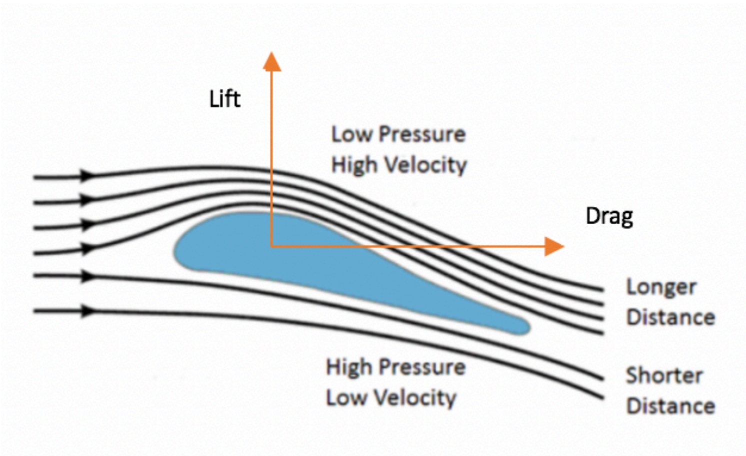


Figure 5 - Drag and Lift forces

The forces mentioned above are illustrated in *Figure 5.* There, it is possible to deduct, applying the Bernoulli's equation, the direction of the forces. Where the streamlines have a longer distance, it is found a low pressure and high velocity zone. Similarly, where the streamlines have a shorter distance, it is found a high pressure and low velocity zone. As a result, the lift force will be in ascending direction. Besides, as the fluid flow goes from the left to the right-hand side, the drag force will go in the same direction.

2.1.8 Spatial discretization

The issue focused on this section it is a way to reach a numerical approximation solution of the Navier-Stokes equations. As it was mentioned above, these equations describe the behaviour of the fluid in movement. In order to reach that solution, the CFD software uses computational grids, which will be called mesh. As a result, three types of mesh are founded:

- Structured meshing

The main advantage of this type of mesh is the way it is organized. It allows accessing to the next cell easily. On the contrary, it has problems to adapt to complex geometries. The structured grid can be represented in a cartesian or curvilinear system.

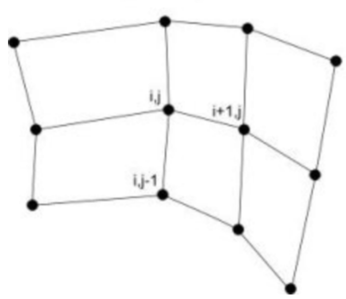


Figure 6 - Structured mesh

- Unstructured meshing

The second type of grids are the unstructured. This type is suitable for complex geometries, since they can be adapted to the surface easily. It is based on triangles, in the two dimensions, and tetrahedrons in the three-dimensional case. However, it has a big disadvantage, it takes a lot of computational memory comparing to the structured grids.

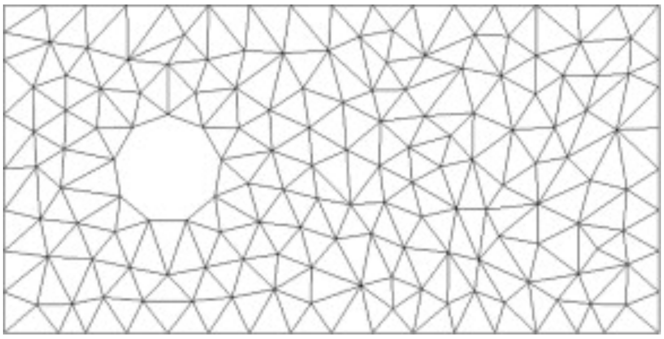


Figure 7 - Unstructured mesh

- Hybrid meshing

Finally, the third type of mesh is the hybrid meshing. It is simply a mix of the previous two into a single one. In some cases, it is the most useful. Since in the same geometry, there are parts which need a more accurate measure than others, a structured mesh is needed close to the surface. While in the rest of the space, an unstructured grid is enough.

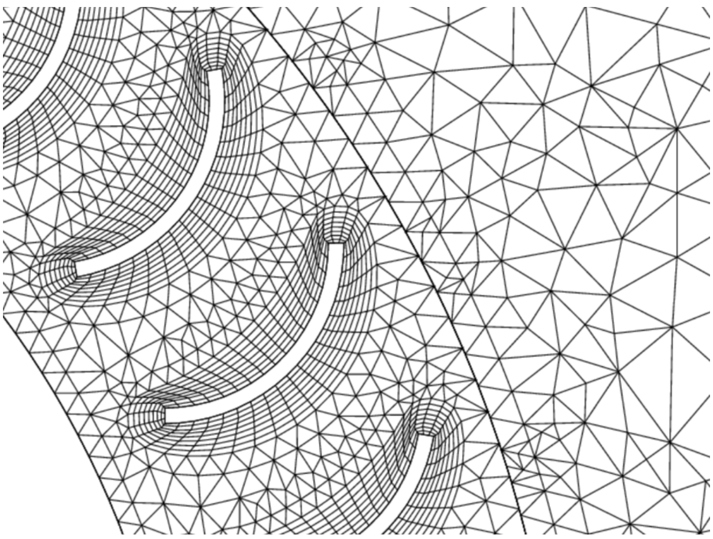


Figure 8 – Example of a hybrid mesh

2.2 Aerodynamic devices

During a long process of researching, it has been identified the most important parts in a car body that need changes or need to be added in order to improve the aerodynamic of it. Within all modifiable parts, this project will be focus on: adding a rear spoiler, modifying the front part, adding thickness at the bottom and finally, a diffuser at the back.

2.2.1 The spoiler

The term spoiler refers to a device taken from aeronautics. It is simply used for producing aerodynamic load, in other words, it is used for reducing the lift force producing a downforce. Apart from this, spoilers are used to delay the swirl zone created downstream, which leads to an increase in the drag force. The farther this zone is created; the less drag force the car will suffer. Spoilers are normally located at the rear part of the car, but, in some racing vehicles could also be located at the sides.

The cross-section of a spoiler is usually an aerofoil. When talking about spoilers, there are some important parts and terminology that needs to be specified. First of all, the leading edge. It is the part of the aerofoil which hits the air first. At the back, it is found the trailing edge. The straight line that joins the leading with the trailing edge is called chord line, and the distance measured in that line is the chord. The mean camber line joins the leading and the trailing edge equidistantly between the upper and lower surface. Finally, the angle of attack. It is one of the most important parts of a spoiler, since it determinates the amount of lift or drag added to the whole car. It is the angle measured between the wind direction and the chord line. (skybrary, 2015)

The picture below shows the cross section of an aerofoil used in aeronautics. Once again, in automobilism this aerofoil is rotated 180 degrees in order to obtained negative values of the lift force, thus it is possible to reach more stability.

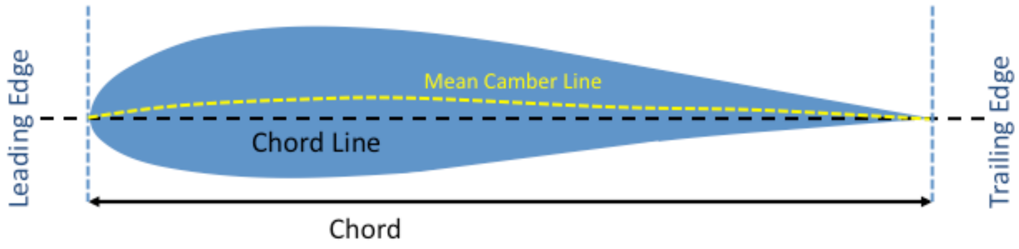


Figure 9 - Aerofoil cross section terminology

Additionally, there exist several types of spoilers used in car design. Depending on the number of aerofoils that it contains, they are classified in:

- Spoilers with one element

In this type, the aerodynamic load is enhancing as the angle of attack is increasing. There is one limit where the quotient between the lift and drag coefficient starts decreasing, this limit has been considered the optimal point.

- Spoilers with two elements

When more aerodynamic load is needed, spoilers with two elements are implemented. They are conformed simply with a spoiler with one element plus another one called flap. In this way, the surface is increased. Moreover, the flap has another function, which is delay the detachment of the boundary layer. *Figure 12* shows a real example of a spoiler with two elements.

Figure 10 – Spoiler with two elements

- Spoiler with several elements

Similarly, comparing with the spoilers mentioned right above, it is found the spoiler with more than two elements, which purpose is almost the same. The difference only resides in the number of flaps.

To conclude, it is important to emphasize that the first type of spoiler is commonly used in tourisms. Whilst the rest of them are frequently used in racing, where higher speeds are reached and a good finish as well as aesthetic are not that important as in tourisms.

2.2.2 Front part

When a deep focus on the aerodynamic in a car body is made, the first part in contact with the air that is observed is the front part. Here, it is formed, what is called a stagnation point. It means that the velocity in that point is close to zero. According to Bernoulli´s equation, this point has a high dynamic pressure. Consequently, the front part requires more attention when designing the body car shape. Depending on what shape it has, less or more drag will be obtained. That is why, in the front part of a vehicle, a smooth transition is needed.

Although in this project has not been included as a change, in some cases a little flat surface is added parallel to the floor in order to reach more negative lift. It is normally added at the bottom part of the bumper.

2.2.3 Flat floor and "Ground Effect"

Due to mechanic elements located at the bottom part of the vehicles, turbulent zones and irregular flow is produced. Hence, it is preferable cover these zones in order to avoid turbulent flow and, consequently, an increasing of the drag force.

However, the reason mentioned above, it is not the most important why the body car design tends to be increased at the bottom part. The closer the vehicle is to the ground, the lower pressure this area has, since less air goes through it. This phenomenon is called "ground effect". Once again, this effect is translating into negative lift force, since in the upper part of the car the pressure is higher than at the bottom. *Figure 13* shows a scheme of how this effect results in downforce.

Figure 11 - "Ground effect"

As it is observed, the major part of the air goes through the upper part of the car, obtaining a zone of high pressure.

2.2.4 Diffuser

The implementation of a diffuser is a practice that is implemented along with the ground effect. It is located at the rear part and it usually has a curved shape where the trailing part is located higher than the leading part. The diffuser ends at the end of the car.

Furthermore, the main scope of the diffuser is to expand the air that comes from the front and travels down the bottom with high pressure and low velocity. This is important as the air that comes from the upper part with high velocity and low pressure is mixed with the air from the bottom. The pressure difference can lead to turbulence and, also, it results in increasing drag force.

In some cases, a small plane is added perpendicular to the air direction at the bottom entrance in order to reduce the section. This implementation is done in order to reach higher values for the difference between the entrance and exit area.

As a result, the main objective of the diffuser is to delay as much as possible the detachment of the fluid with the car body. In other words, to produce the turbulent zone as further as possible.

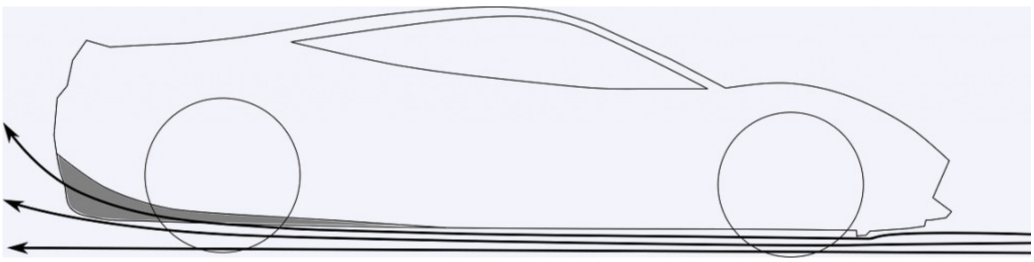


Figure 12 - Streamlines path with a diffuser action implemented

2.3 Working tools

2.3.1 Wind tunnel

When aerodynamics starts being an important characteristic in car design, only one tool was available for testing, wind tunnels. To begin with the definition, a wind tunnel is a tool used in engineering in order to study the effect of the wind around bodies. This tool is based in simulating under which type of conditions that, in this case a car, will work in. In a wind tunnel, the object will remain stationary, while it is the air that moves through it. In some cases, it is common to use a model with a reduced scale in order to save money when the original body has certain dimensions, as it can happen with an airplane.

There are two types of wind tunnels depending on how the flow is produced and how the air is moving within the tunnel. These types are shown in *Figure 9* and *Figure 10.* Moreover, within them, it is found some with only one fan to produce the air flow, and others have more than one to reach the necessary conditions.

Finally, it is important to know that the use of a wind tunnel to test aerodynamic properties is really expensive and no everyone has access to them.

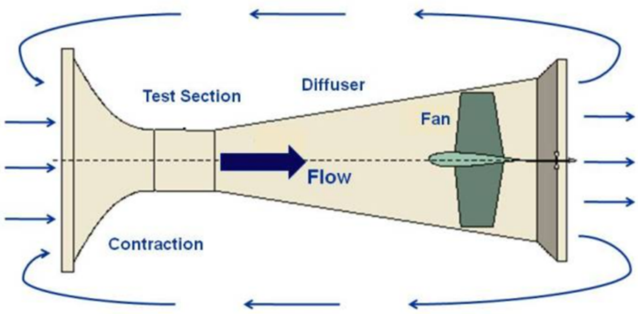


Figure 13 - Open Return Wind Tunnel

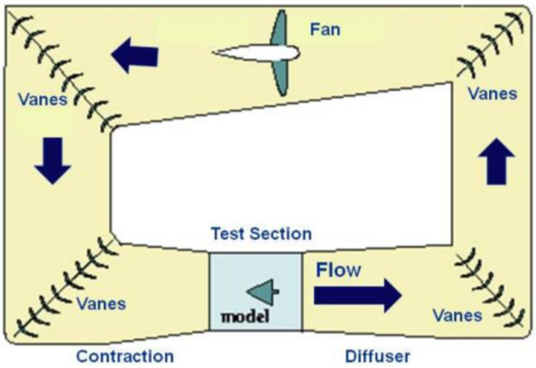


Figure 14 - Closed Return Wind tunnel

2.3.2 CFD simulations

As it has been mentioned above, the use of wind tunnels is not available for everyone. This, together with the cost that the use supposes, has arisen the necessity to developing computer softwares. Besides, they are able to provide results closed to the ones obtained in a wind tunnel. These softwares received the name of Computational Fluid Dynamics (CFD).

However, the use of one of them does not exclude the other one. In some cases, first, it is done some test with a real scale with CFD to predict the aerodynamic of the real body. Afterwards it is possible to create the real model in real scale for testing it in the wind tunnel. This way to proceed has the advantage of saving money, since there is no need to build models in small scales for testing it in the wind tunnel.

There are many CFD softwares available in the market, some more accurate than others. For this research, it has been impossible to build models for testing in wind tunnels. Therefore, it had been chosen ANSYS® Fluent because it is one of the most accurate when modelling a turbulent regime is needed.

2.3.3 CAD software

The Computer aided design (CAD) is, nowadays, an important range of tools in engineering design. It is divided into programs of two-dimensional design and three-dimensional modelling. The CAD softwares let us design any type of models for building them afterwards.

For this research, it had been chosen the SolidWorks ® software.

3. Stages of the Project

First of all, this section will be focused on developing the steps that had been followed during the project. To begin with, the model selection. It will be followed by a three-dimensional design for being able to study the main aerodynamic characteristics. Then, some implementations will be added to conform a new model. The last step of the project will be focused on obtaining final results in order to see what improvements have been reached.

3.1 Vehicle selection

The vehicle, which was selected for making this research, was the BMW M5 E34. This model was released to the marked between 1992 and 1995. The reason why this vehicle was chosen instead of another one was simply that for doing an aerodynamic study it was necessary to choose an old vehicle. Nowadays, the aerodynamic is really considered when talking about car design. In this sense, it is necessary to select a vehicle where it is possible to make changes in order to improve its aerodynamic characteristics.

Figure 15 - BMW M5 E34 (1995) model

Hence, the project starts with some technical characteristics provided by the manufacturer. These characteristics will lead to know if the initial results make sense and also, it will allow obtaining conclusions about, for instance, fuel consumption among other things. Between all the features facilitated by BMW the following are highlighted as the most important:

- Length: 4750 mm
- Width: 1752 mm
- Height: 1413 mm
- Weight: 1724 Kg
- Maximum torque: 400 Nm
- Maximum engine power: 340 horsepower / 250kW
- Drag coefficient: 0.33
- Fuel consumption:
    - Urban conditions: 7.47 l/100 km
    - Extra-urban conditions: 8.37 l/100 km
    - Mix-conditions: 18.04 l/100 km

3.2 Vehicle design

As it was mentioned above, the vehicle design was made through the three-dimensional design software SolidWorls ®. This process followed some steps in order to obtain the final result.

Thus, the first step was finding photos (from internet) of every angles of views: front, top and lateral view. In this way, it was possible to import these pictures to start drawing the sketch. The whole vehicle was built by several planes, as it is shown in the following pictures.

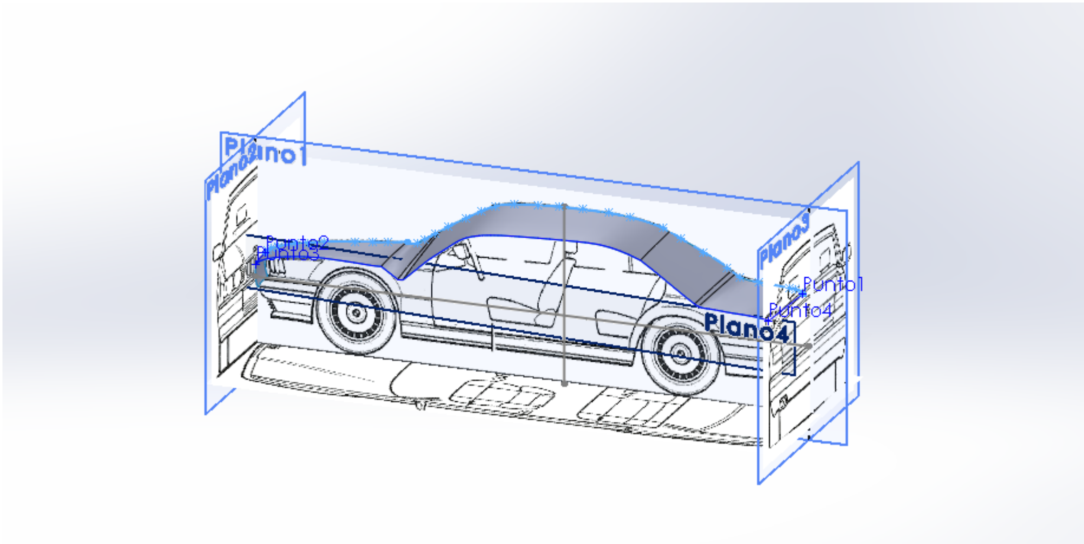


Figure 16 - Sketch process 1

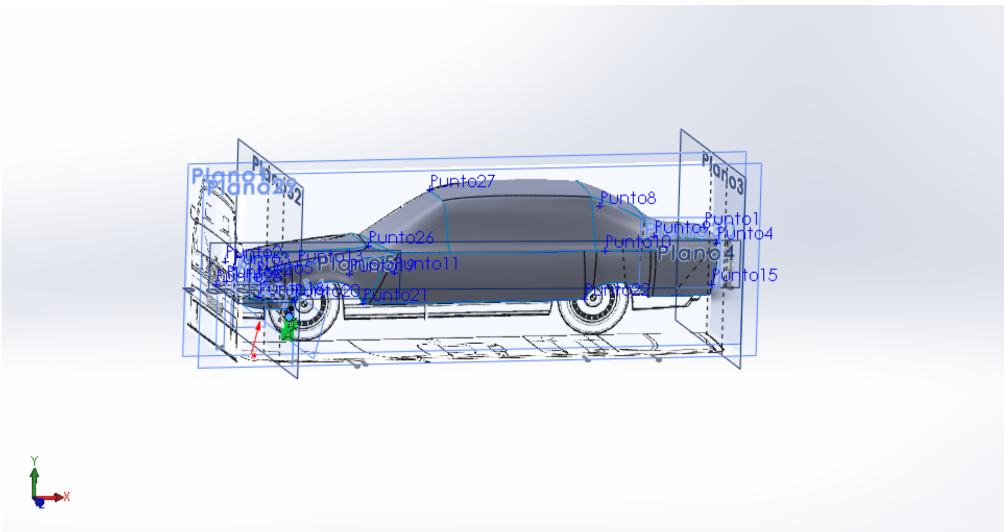


Figure 17 - Sketch process 2

When all of the surfaces where drawn, the last step consisted in joining all of them into one, forming a kind of shell. Only in this way, it was possible to create a unique solid in order to be able to export the model to the desired extension. Before creating the solid, it was tried to export the shell to make the simulations with it. The problem found was that the program didn't recognise the model and the air pass through it without experiment any deviation from the original path. As a result, the final solid model obtained is shown in *Figure 18.*

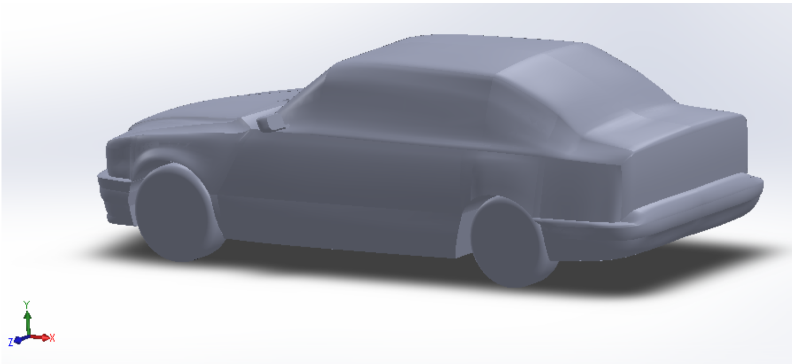


Figure 18- BMW E34 1995 prototype

The first model created for this research, was totally different from the one used finally. It was more squared with steeper vertices. The decision to change and create one more similar to the real car, came from trying to obtain a more accurate model where the results were more reliable.

3.2.1 Old vehicle simulation

When talking about vehicle simulation, it is referred to a process of introducing the model designed into a software charged with performing the flow analysis. Therefore, this program will create an air current which will pass over the car body in order to obtain the aerodynamic behaviour. As it was mentioned in section 2.3.2, the software chosen was ANSYS ® fluent as being this program one of the most accurate ones when the turbulent regime is studied.

Hence, it is necessary to set up several settings and conditions in order to obtained good results in the simulation. These steps are one of the most important and must be done carefully, since they will allow getting as close as possible to the conditions given in reality.

First, the CAD model must be exported into a determined extension which is. IGES. Then, it will possible to import the geometry in ANSYS. Once the geometry had been imported, it was time to create an enclosure for developing the mesh.

In the beginning, a mesh of the full car was created. Afterwards, after realising that the software takes a lot of time on this process, it was decided to create a mesh of half of the model, as it was dealing with a symmetric model. Thus, a hybrid mesh was chosen. The reason why it is that close to the surface, a regular mesh was needed in order to capture the development of the boundary layer, this is called inflation of the mesh. Far from the car surface, an unstructured mesh was enough. An inflation was also done at the bottom part, as it simulates the road where the boundary layer it is important to study. Finally, another important consideration when creating the mesh is the wheel zones. There, the air must not pass. Further, the wheel is not round at all because it must. Moreover, the plane of the end of the mesh will intersect part of the wheel. As a result, the mesh obtained is shown in the figures below.

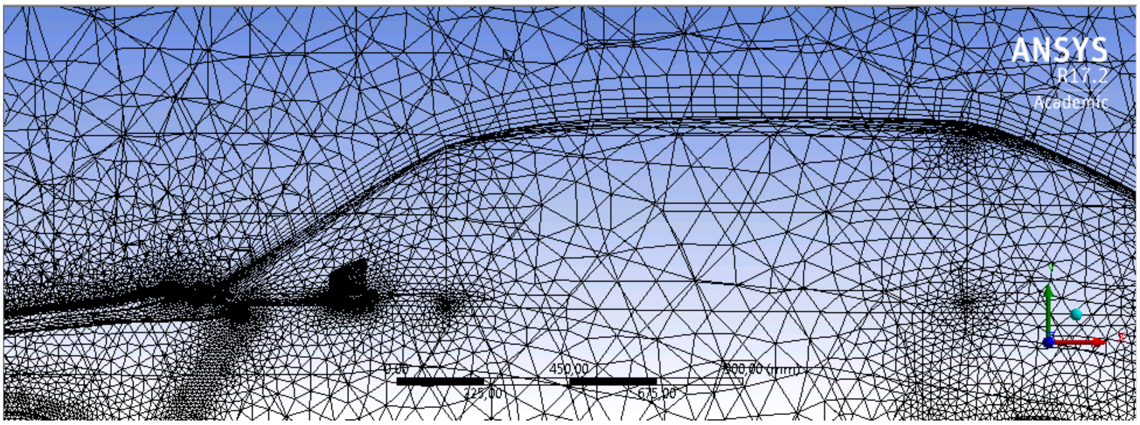


Figure 19 - Inflation detail near the car surface

The second step after shaping the mesh it is configuring theoretical models and the boundary conditions to be used in the flow simulation. First, it will be set the turbulent models. This will be used in order to obtain a good approximation of the Navier Stokes equation result. The energy equation is not considered as temperature changes and heat transfer are not involved in this kind of problems. Hence, the viscous model used was, what is called "k-epsilon", following the ANSYS user´s guide. There, it says that this model is suitable for complex problems such as strong changes in the direction of the flow lines, rapid flow analysis, turbulence swirl effect and circular flow problems (ANSYS, 2009). Then, the software provides a list of parameters to set up, as it is shown in *Figure 21*. Most of them were left as a default except the Near-Wall Treatment in which the Non-equilibrium Wall function option was selected as it performs better the adverse pressure gradient, flow separation and reattachment.

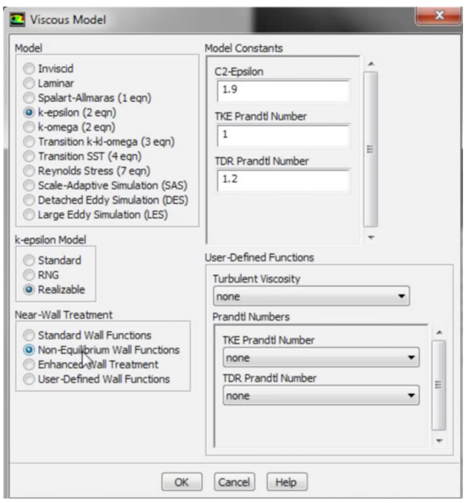


Figure 20 - Viscous model configuration

Secondly, it is necessary to set the boundary conditions to control the flow behaviour. As it is a tourism and not a racing vehicle, the velocity inlet was 80 m/s. The higher the speed is, the more advantages the aerodynamic devices takes, since the flow remains attached to the car surface longer. Besides, the car surfaces and the floor which simulates the road, were set with a no-slip condition, which means that the flow velocity there is zero. Once again, ANSYS user´s guide recommends a range between 1% and 5% for external flow velocity intensity. Thus, it was set at 1% at the inlet, where the air is calmed, and 5% at the outlet of the enclosure.

In order to obtain the drag and lift coefficients, it had been needed to introduce the projected area in one of the parameters. When dealing with such a complex surface, this area cannot be calculated by hand. Thus, ANSYS fluent has already implemented this function. The projected area was 2.004 m^2.

Finally, the last step to set up after running the iterations for solving the aerodynamic problem is the solution methods. Pressure and velocity were solved as coupled. Gradient and Pressure set up was left as default within the spatial discretization box. Nevertheless, for the first iterations, Momentum, Turbulent Kinetic energy, and Turbulent dissipation Rate were solved with first order upwind approximation. The reason was because it converges much faster to the solution but less accurately. Afterward, the last iterations before convergence were obtained with a second order upwind approximation, in order to obtain a better solution.

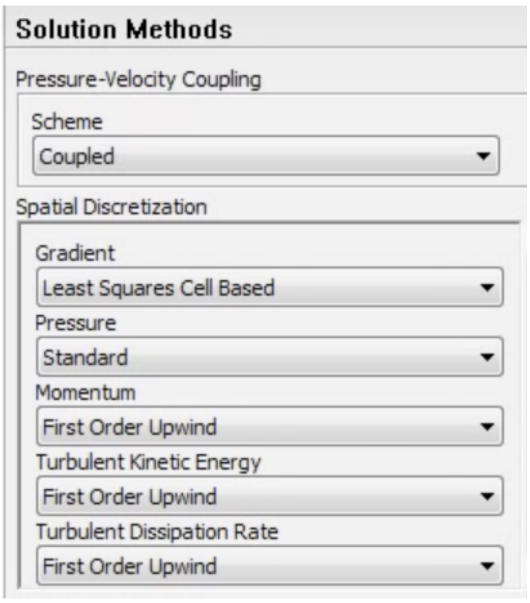


Figure 21 - Solution methods box for first iterations

All of these configurations will be only mentioned once, but they will be applicable for all the simulations made during this research.

As a result, after establishing all the parameters which involved, the solutions obtained were:

Drag coefficient	Lift Coefficient
0.349	0.360

Table 1 - Drag and lift old design coefficients

The drag coefficient obtained differs a bit from the one provided by the manufacturer, since the model was created from drawings. Therefore, it was obtained something close to the real model but not equal.

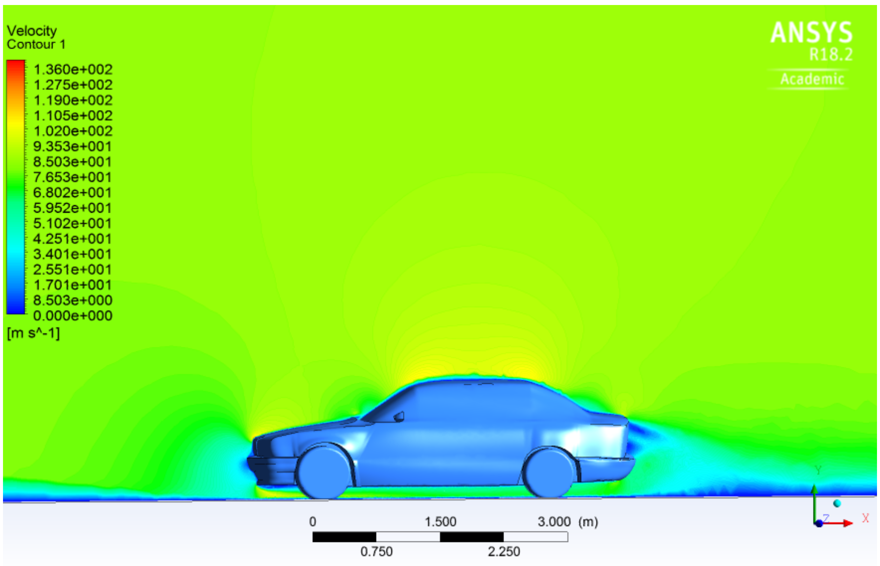


Figure 22 – Velocity distribution in the symmetry plane

In the figure above, it is shown the velocity distribution in a plane which cuts the car in two parts (symmetry plane). It can be appreciated the no-slip conditions at the surface and the development of the boundary layer. Moreover, at the rear part, it is observed how the flow with high velocity from the upper part is mixed with the flow from the bottom, giving place to a swirl zone. This zone is observed clearer in *Figure 25.* Also, it is observed a disorderly velocity distribution at the bottom. Thus, this zone will be changed in order to take advantage of the floor effect mentioned in section 2.2.3.

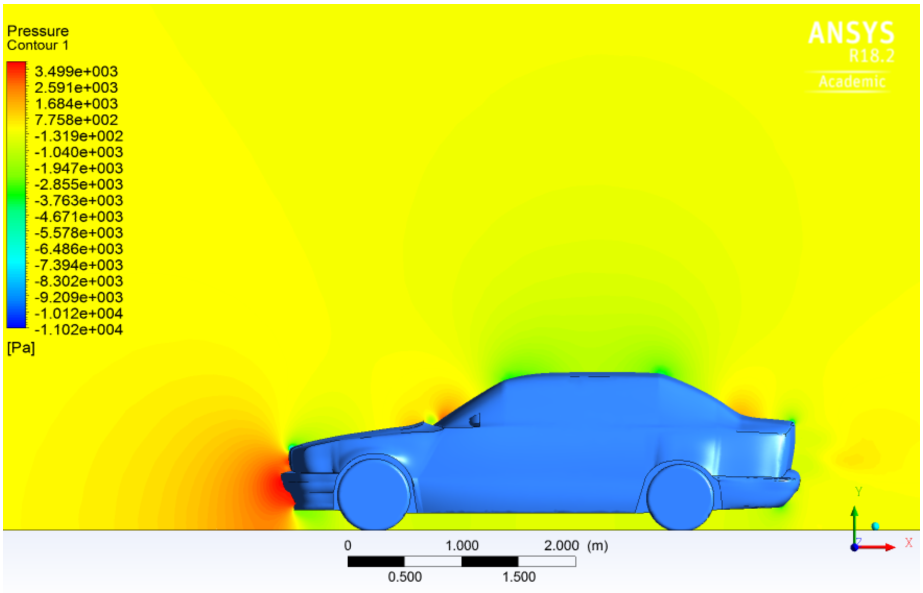


Figure 23 – Pressure distribution in the symmetry plane

From the pressure distribution over the car, it can be deducted that the most critical zone is located at the front bumper. Hence, this part will need to be modified afterwards.

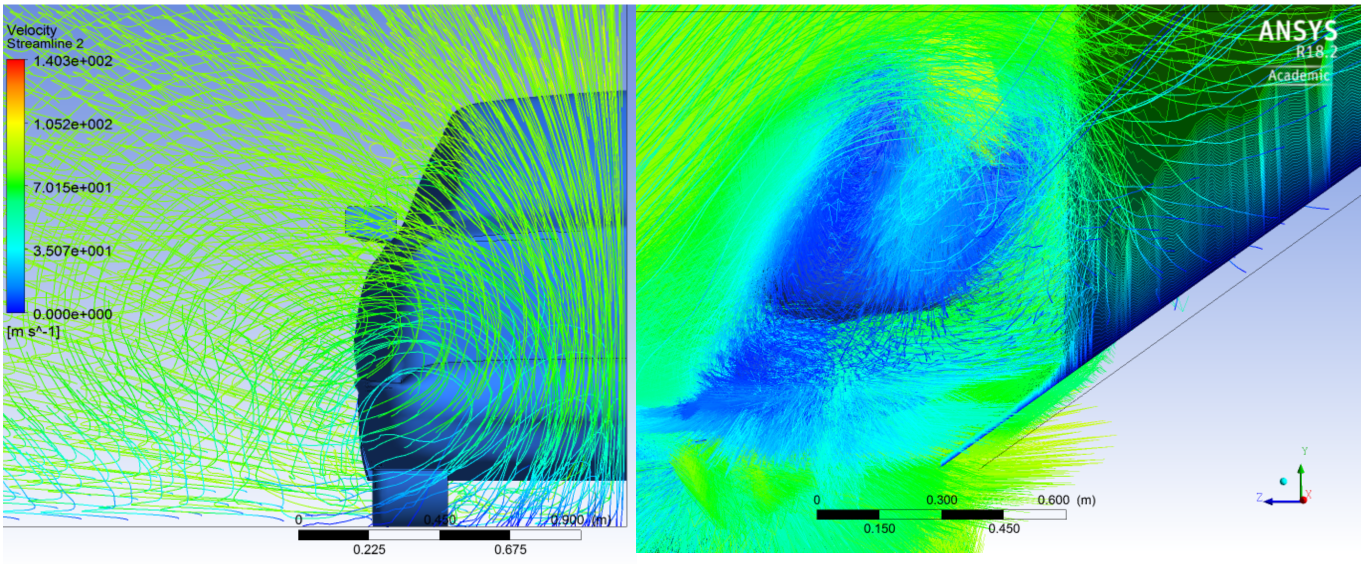


Figure 25 - Velocity streamlines and velocity vectors viewed from the rear part

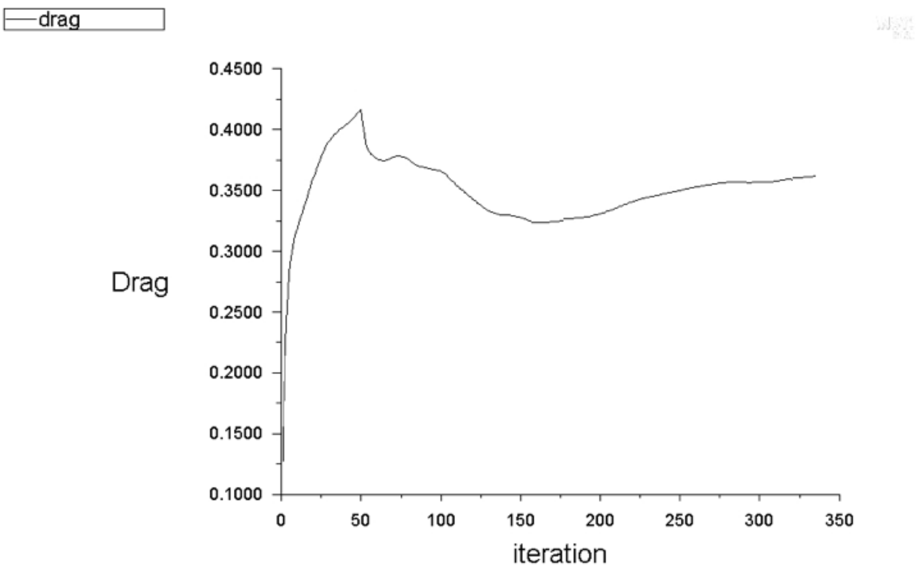


Figure 24 – Drag coefficient result convergence

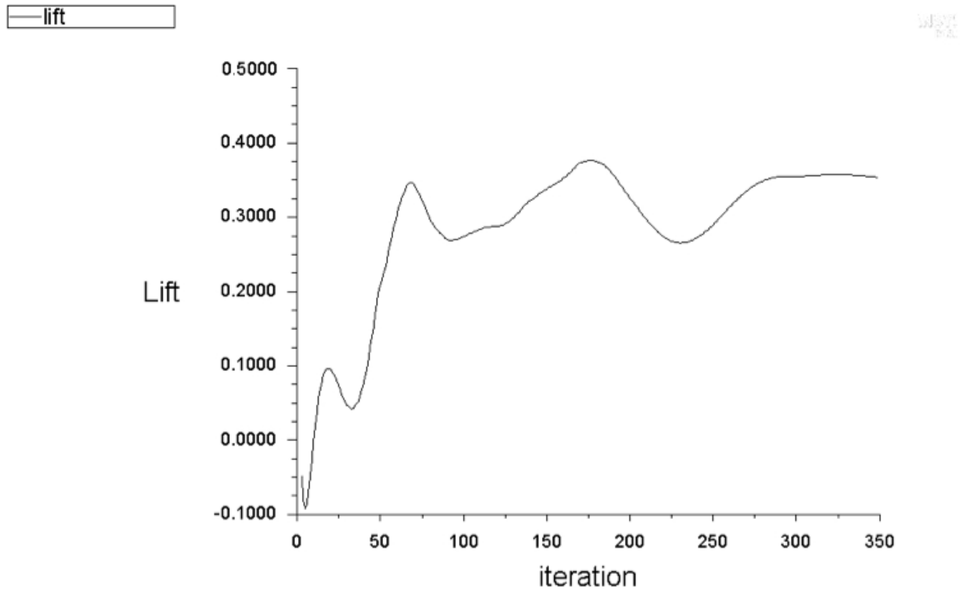


Figure 25 – Lift coefficient result convergence

3.3 Current normative

The modifications proposed in this project can be considered either as implementations for new models fabricated by BMW, or as changes made in a mechanical workshop. In the first case, there is no more problem about normative than the one to apply in tourism design. However, if it is the case where some devices are added to a model already produced, a specific regulation will be applied. Therefore, the two cases will be considered in this report.

On one hand, if the modifications are introduced by the car designer as a new model, the current normative for tourism will be fulfilled, as it was met in the old model BMW E34 1995. The reason why it is because the changes introduced do not affect that much of the original model. The height, length and the space available for travellers are not be affected.

On the other hand, if the project is understood as modifications that can be made by a mechanic in a workshop, then there are several regulations that must be considered.

Dealing with an M1 vehicle category, the applicable regulation will be: Regulation UE 1230/2012 and the Manual of vehicle reforms established by the ministry of industry.

Firstly, the Regulation UE 1230/2012 Annex I establish the maximum authorized dimensions: Length: 12 m; Height: 4 m; Width: 2.55 m.

Secondly, the Manual of vehicle reforms established by the ministry of industry, Part 8.52. Here, the regulation for incorporations, modifications or uninstallations of elements on the outside of the vehicle is set.

Affected system	Reference
External projections	74/483/CEE
Frontal collision	96/79/CE

Table 2- Regulatory acts

Necessary Documentation				
Technical project	Final certificate of work	Compliance report	Workshop certificate	Additional documentation
YES	YES	YES	YES	NO

Table 3 - Necessary Documentation

3.4 Vehicle modifications

This section of the report will be focused on what modifications can be implemented in the old car model. Hence, the theory from section 2.2 (Aerodynamic devices) will be the basis to demonstrate with what objective these changes had been made.

Thus, three new devices will be added with the purpose of reducing the drag and lift coefficients. Starting from the rear part, a spoiler will be incorporated. Subsequently, it will be tried to soften the bumper. Finally, the bottom part will be modified.

3.4.1 Rear part. The spoiler

The profile chosen for spoiler designing was taken from aeronautics discipline and it is called NACA profiles (National Advisory Committee for Aeronautics). The reason why is because this aerofoil type has its shape characteristics already determined. It is, therefore, from the name of the profile, the main dimensions are deducted.

Hence, the name is based on the word NACA following by four digits.

- The first digit goes from 0 – 9.5 %. It determines the maximum camber
- The second digit goes from 0 – 90%. It describes the maximum camber position.
- Third and fourth digit goes from 0 – 40%. It represents the thickness of the aerofoil.

As beforehand, it does not be known which profile will give more aerodynamic load (downforce) without increasing the drag force, three profiles were chosen for being tested. In all of them, the thickness remained constant and it changes its other parameters. In this way, the three types selected were:

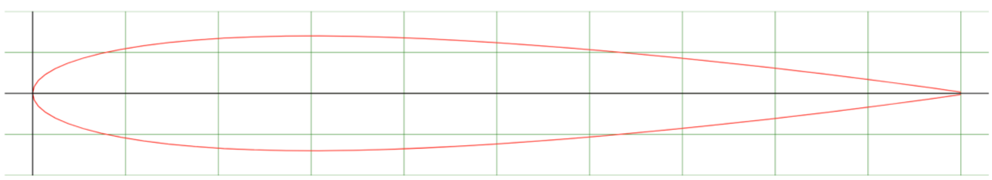


Figure 28 – NACA 0014 aerofoil profile

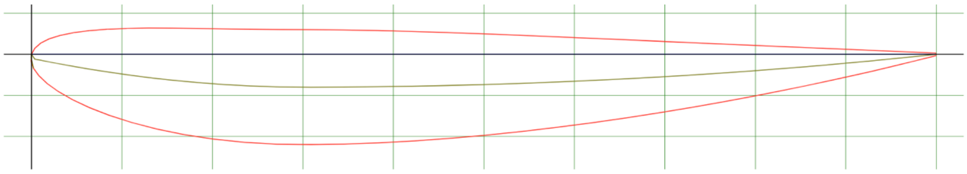


Figure 29 – NACA 4314 aerofoil profile

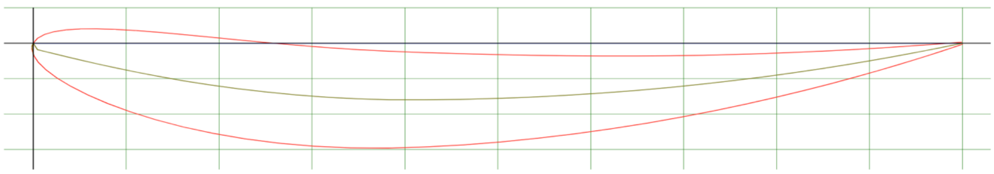


Figure 30 – NACA 8414 aerofoil profile

For selecting the best profile, the three of them were testing for several angles of attack. In this sense, it will be chosen the one which has the higher ratio. This ratio was defined as a quotient between lift and drag coefficients. All of the aerofoils were simulated as it was done previously. As a result, the following data table was obtained.

NACA 0014			
Angle of Attack	CL	CD	CL/CD
2	1.18	0.14	8.25
4	2.22	0.16	13.46
6	2,9	0.21	13,7
8	3.57	0.22	16,22

NACA 4314			
Angle of Attack	CL	CD	CL/CD
4	3.83	0.22	17.4
6	4.31	0.23	18.64
8	4.40	0.26	18.54
NACA 8414			
Angle of Attack	CL	CD	CL/CD
4	5.20	0.35	14.85
6	5.60	0.37	15.13
8	5.74	0.43	13.17

Table 4 - Data table aerofoil profiles

It is concluded from the table, that the most suitable profile to be implemented is the NACA 4314. This one has the highest ratio for 6 degrees of angle of attack. As a result, the spoiler was designed with this shape. The plane with the spoiler dimensions is found in Annex I.

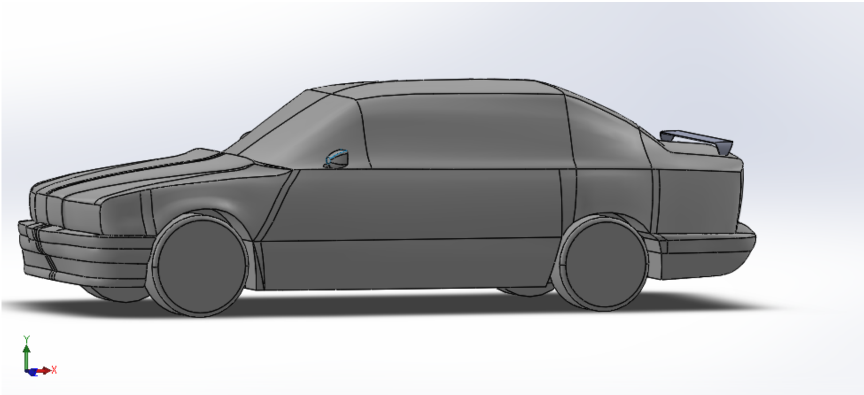


Figure 26 – Car design with spoiler implemented

Next step is to simulate the airflow current over the car body through ANSYS® in order to prove if the fact that adding a spoiler has any advantages over the lift and drag coefficients. The projected area was not be increased, but adding a spoiler supposes a greater resistance to the air. Accordingly, the results expected were a small increase in drag force and a decrease of lift force. After running the simulation with the set up mentioned above, the solutions were as it was expected.

Drag coefficient	Lift Coefficient
0.356	0.214

Table 5 - Drag and lift coefficients with spoiler

Thus, the drag force suffered an increase of 2%. Nevertheless, the decreased lift force was 40% less, as a result of creating a down force. This leads to obtaining a force from the wind which will bring a greater grip to the road and therefore a greater stability in high speeds. Additionally, the spoiler was located as far back as possible in the trunk area, allowing the swirl zone to be produced further. That is why the increase in drag was not too high.

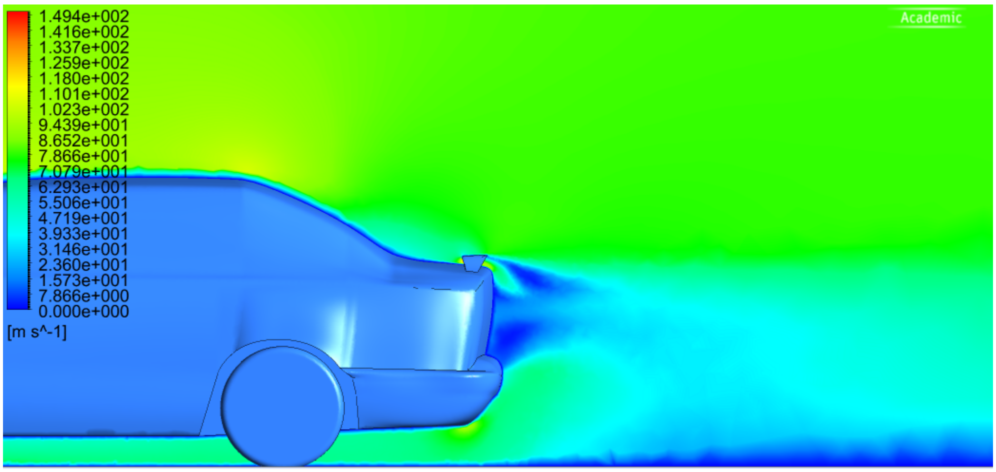


Figure 27 – Effect of the spoiler in the velocity field

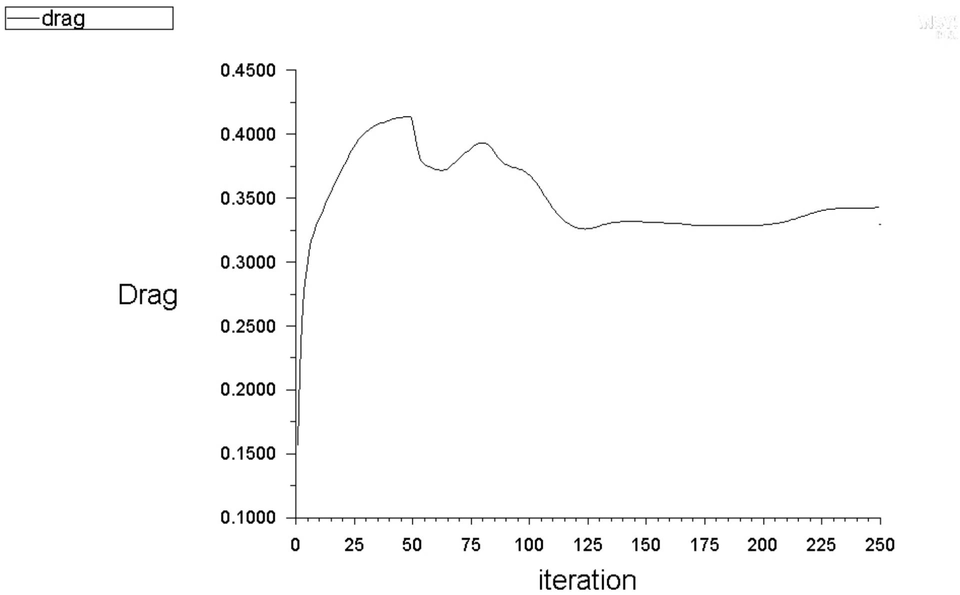


Figure 28 – Drag coefficient result convergence of the car with spoiler

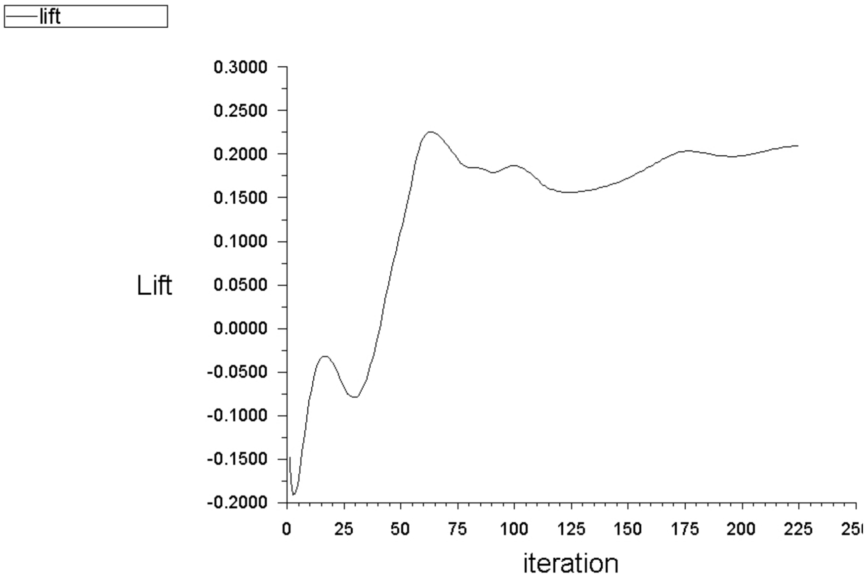


Figure 29 – Lift coefficient result convergence of the car with spoiler

Hence, these graphics above support the results shown in *Table 5*. In the first one, convergence is considering to be reached at iteration 230 with a value of 0.356. However, the second graphic does not converge at this iteration number, but it is considered 0.214 to be a good approximation to the result.

3.4.2 Front part. Soften the front

Secondly, the next change implemented in the model was located at the front. Previously, when the original model was designed, it was already observed that the bumper needed changes. The hypothesis was confirmed after the first simulation, where the results showed a very high-pressure build-up. The main reason why was that several faces were designed with a flat form orientated against the wind direction.

Thereby, this car area was selected for being changed in order to improve the aerodynamic performance. Thus, what it was done was to design a new bumper in soften way. With that, the airflow will not suffer a dramatic change of direction.

All the design was done respecting the shape of the lines which came from the car bonnet, since these lines are highly characteristics in the models of the brand chosen. Therefore, the new front area took the following form:

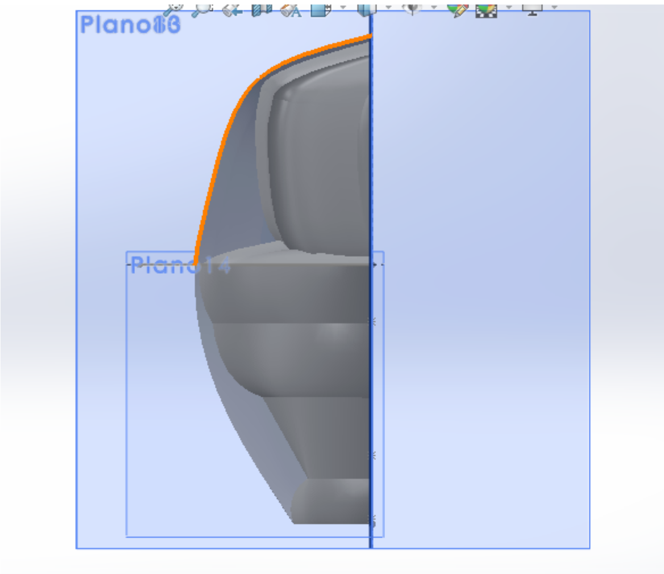


Figure 30 – Front part design

Figure 31 – Final bumper shape

According to the methodology followed in all the research the next step is to check if the changes have reached any reduction of the drag and lift coefficients. The expected results were, on the one hand, an important reduction of drag coefficient, as the resistance against the wind supposed to be lower. On the other hand, a small reduction in lift force was expected, since this new implementation, did not change the pressure distribution at the top and bottom part. Accordingly, the results obtained are shown in

Drag coefficient	Lift Coefficient
0.319	0.204

Table 6 - Drag and lift coefficients of the model with spoiler and soften font part

The results above were obtained from the simulations made with ANSYS ® Fluent. This software, configured as it was done previously, provide the following graphics of convergence for 350 iterations.

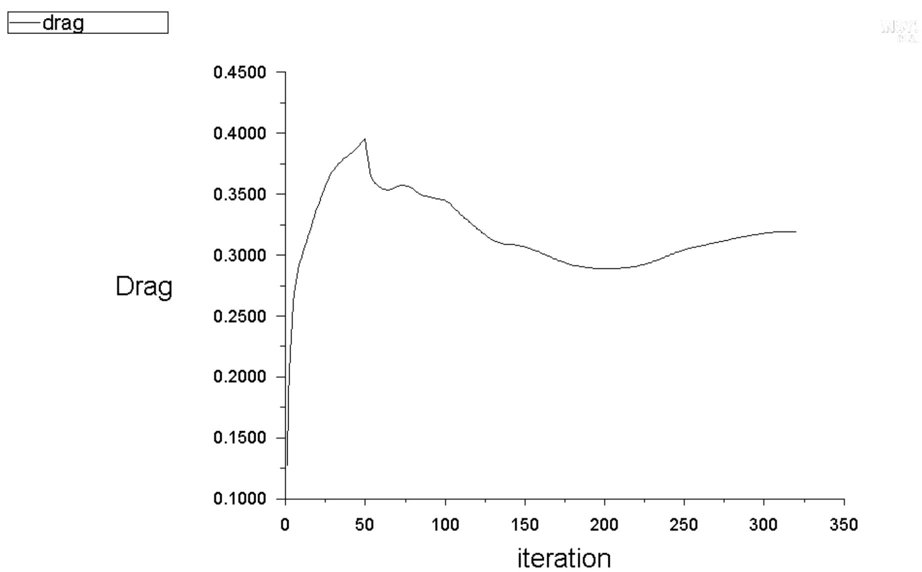


Figure 32 – Drag coefficient result convergence of the car with spoiler and soften front part

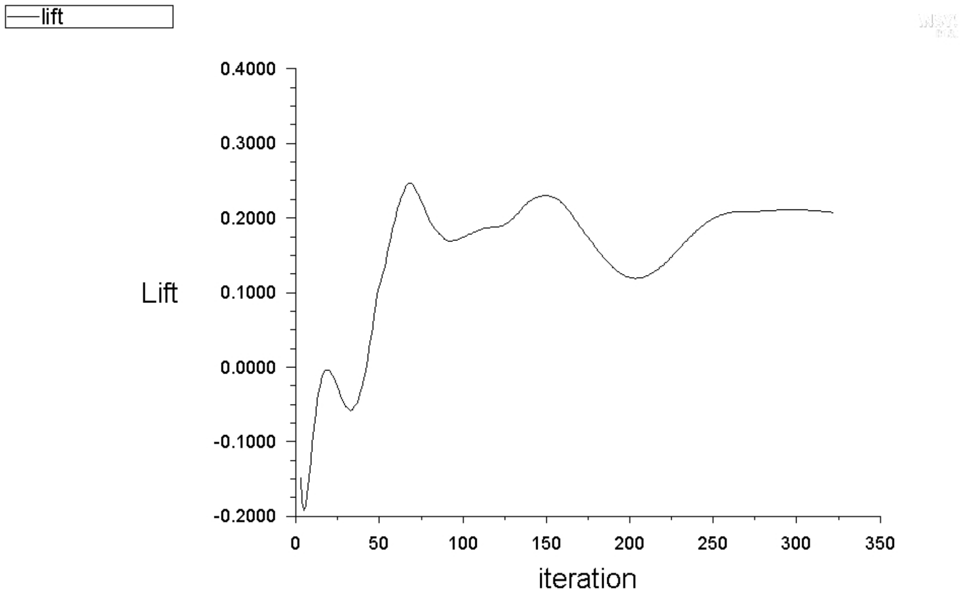


Figure 33 – Lift coefficient result convergence of the car with spoiler and soften front part

In addition, both the pressure distribution and velocity distribution have changed. For instance, it was achieved $3.84 \cdot 10^3\ Pa$ at the point of maximum pressure. This value, compared to the pressure obtained at the same point in the old model, it is almost the same. Nevertheless, the global distribution had decreased. Furthermore, the velocity streamlines do not suffer a steep change of direction as it happened with the old bumper design. These two conclusions can be deducted from *Figure 39* and *Figure 40.*

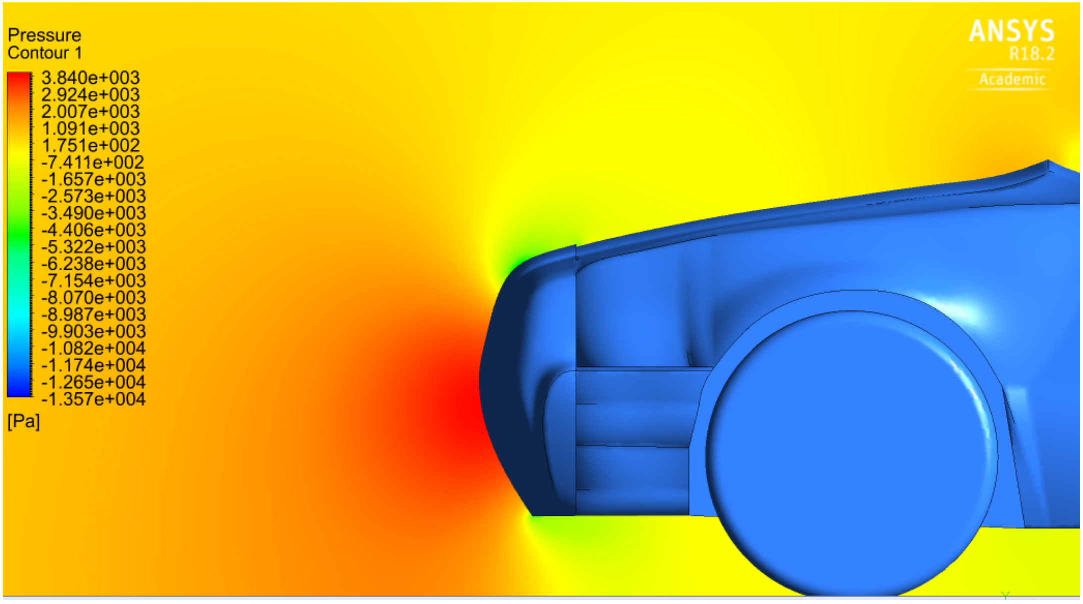


Figure 34 – Pressure contour plot of the bumper new design

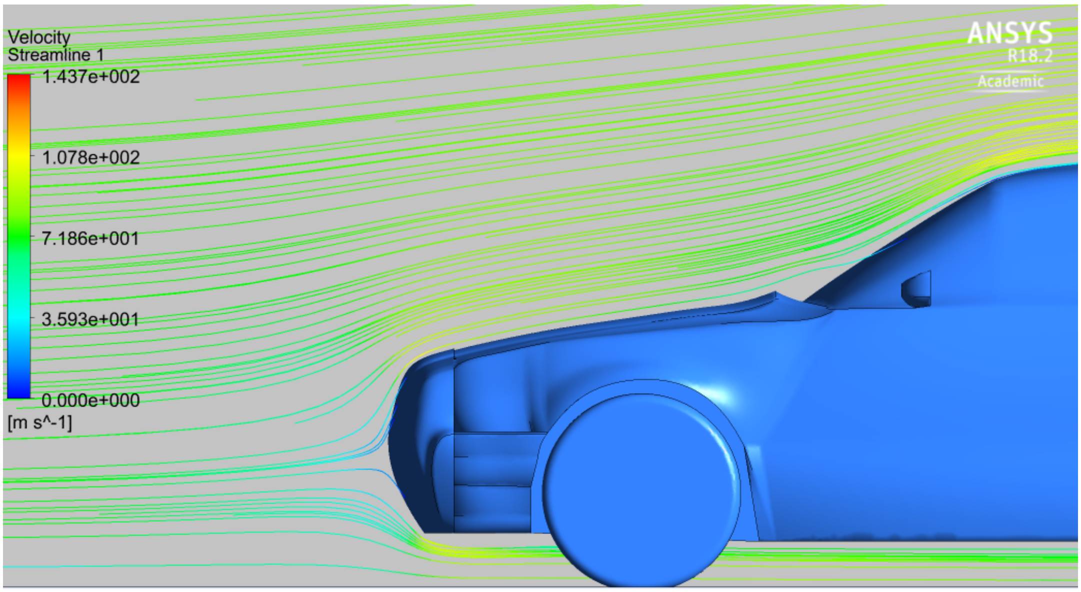


Figure 35 – Velocity streamlines of the bumper new design

3.4.3 Bottom part. Flat floor and diffusers

Lastly, the bottom part modification was orientated to reduce the lift coefficient by taking advantages of the ground effect. For this purpose, what it was done was to increase the thickness of the lower part. This enables to reduce the amount of air which goes through this area. The fact that reduce the entrance area, will lead to making the airflow to travel slower and it will increase the pressure at the same time.

Simultaneously, it was pursued a reducing of the drag coefficient. For achieving it, the exit area was increased in order to accelerate the fluid. Flat plates were added at the rear part for orientated the air, obtaining a more organized flow. This implementation has the same advantages as adding a spoiler, in terms of reducing drag force. As a result, the swirl zone was reduced, as well as displaced.

Hence, the final design of the piece to be assembled to the car took the following shape.

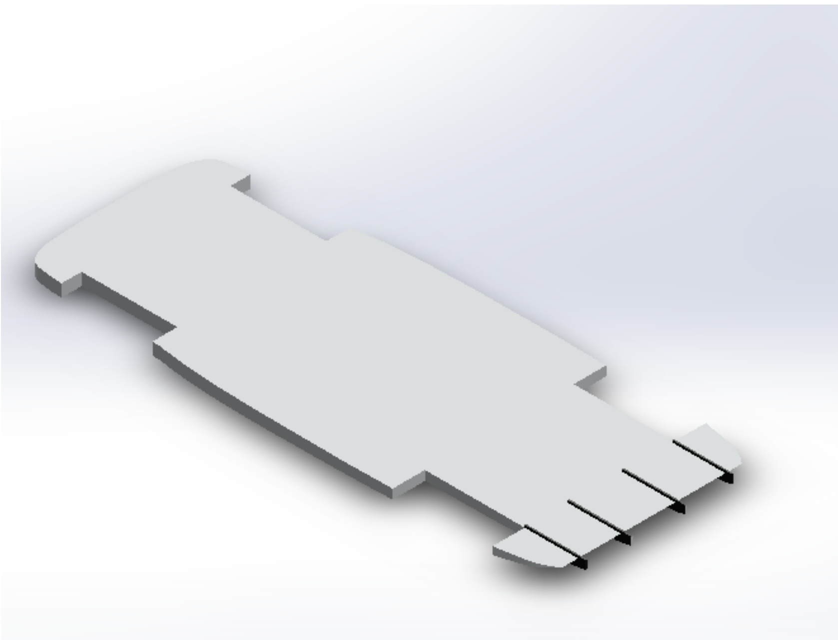


Figure 36 – Bottom part design

The last simulation was made with the car equipped with the full modifications. Therefore, the coefficients obtained were the final results of this improvement. It is observed now a reduction on the lift as a consequence of creating a down force. This force comes from the difference between the upper pressure distribution comparing to the lower pressure distribution.

Drag coefficient	Lift Coefficient
0.304	0.184

Table 7- Final coefficient results

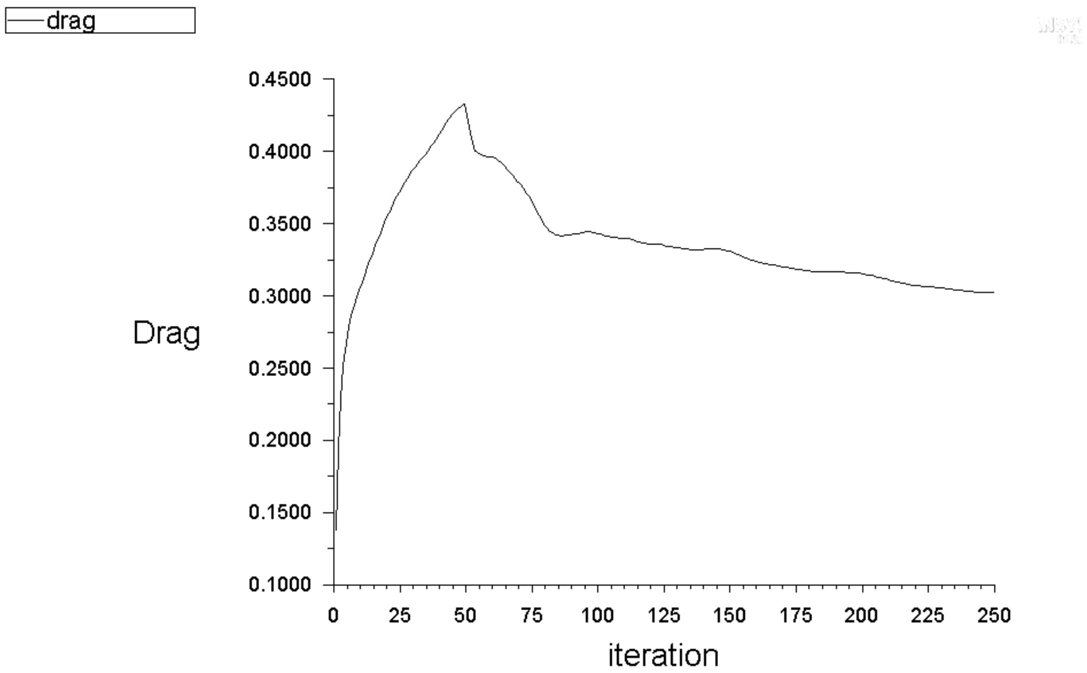


Figure 37 – Drag coefficient graphic with all modifications implemented

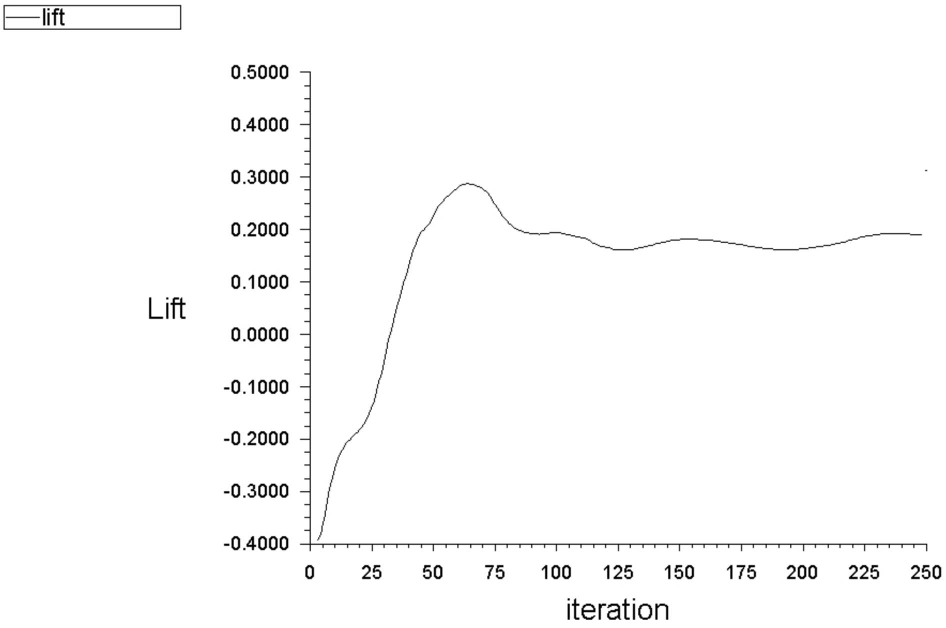


Figure 38 – Lift coefficient graphic with all modifications implemented

The displacement of the swirl zone and the reduction of the velocity is clearly viewed in the following picture. Furthermore, this zone has experimented a reduction in its intensity.

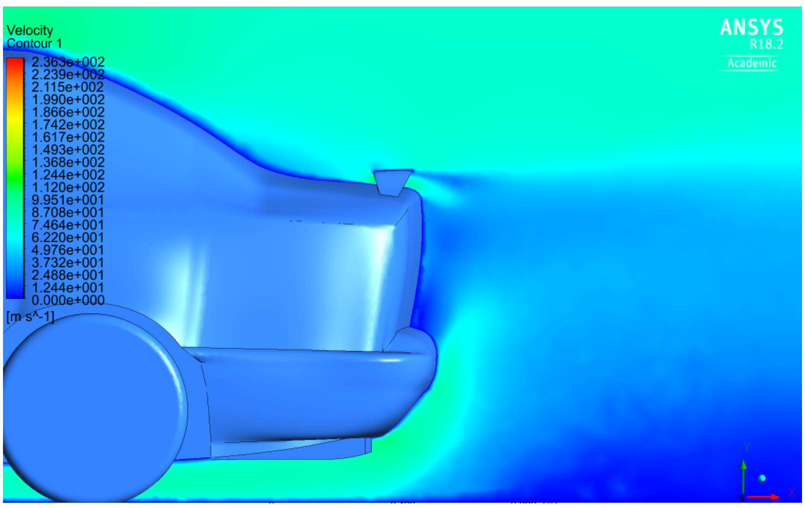


Figure 39 – Velocity field of the model with all modifications implemented

3.5 Comparative between the old model and the new one

In summary, it is been implemented all modifications that it was pretended. Therefore, from a model BMW E35 built in 1995, it has been obtained an improved prototype aerodynamically speaking. In other words, it has been possible to reduce the drag and lift coefficients.

Thereby, it was started from determined coefficients. They were obtained from a prototype measured which design seemed as much as possible to the real model. Those coefficients were 0.349 for the drag and 0.360 for the lift.

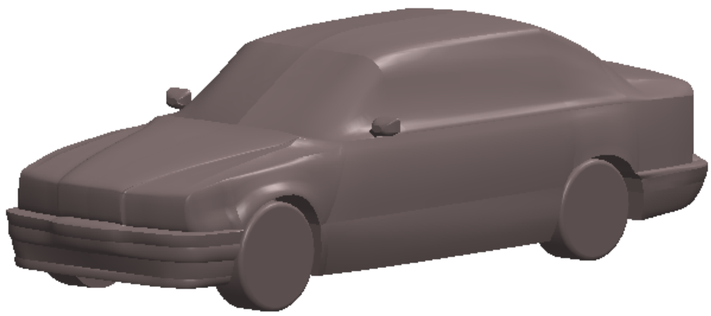


Figure 40– BMW E34 1995 starting model

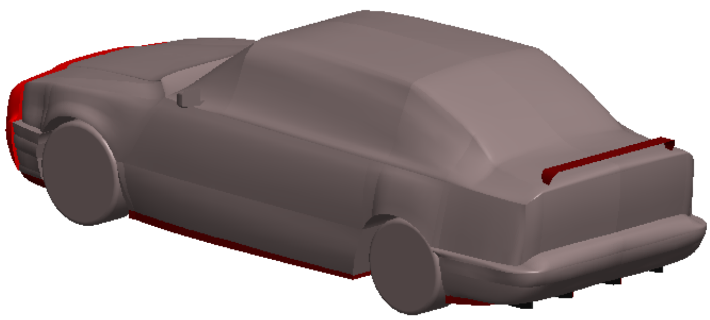


Figure 41 – BMW E34 1995 final model

Accordingly, the final coefficients were 0.304 for drag and 0.184 for lift. Through several changes as a spoiler implementation, soften bumper and the use of a diffuser action at the bottom, a reduction of 12.3 % and 48.8 % respectively was achieved.

4. Conclusions

Due to the fact that the reduction of the coefficients mentioned in section 3.5, it is possible to describe some advantages that can be deducted from it. As it was shown in the second part of this report, the resistance offered by the air can be used for positive purposes.

Hence, the main advantage that can be obtained by reducing the lift force, in other words creating a downforce, it is gain stability. The reason why it is simple: the more force to the ground is created, the greater grip of the tires to the road is get.

For instance, a numerical example is shown above. From the case where the velocity of 80 km/h the lift force is computed in the two cases:

- Lift force in the old prototype:

$$F_L = C_L \left(\frac{\rho v^2}{2}\right) A = 0.360 \left(\frac{1.205 \cdot 22.22^2}{2}\right) 2 = 214\ N$$

- Lift force in the new design:

$$F_L = C_L \left(\frac{\rho v^2}{2}\right) A = 0.184 \left(\frac{1.205 \cdot 22.22^2}{2}\right) 2 = 109\ N$$

In this case, the reduction of the lift force is equivalent to introducing 11 kg inside the car. This value increases with the square of the velocity. Thereby, the aerodynamic advantages are higher at high speed.

In order to see clearly the effect of reducing drag force, the speed is going to raise up to 150 km/h. The coefficient used in this numerical example is considered to be the same as the one computed at 80 km/h, since the variation of it is almost negligible. Thus, the reduction of the force that the car engine must overcome will be:

$$\Delta F_D = C_{D_1}\left(\frac{\rho v^2}{2}\right)A_1 - C_{D_2}\left(\frac{\rho v^2}{2}\right)A_2 =$$

$$= 0.349\left(\frac{1.205 \cdot 41.67^2}{2}\right)2 - 0.304\left(\frac{1.205 \cdot 41.67^2}{2}\right)2.04 = \ 81.44\ N$$

Thus, the main advantage to obtain from reducing the drag force is to reduce fuel consumption, as the resistance against the wind is reduced with the square of the velocity.

5. Future researches.

After years of research, the aerodynamic shapes of the body cars are very studied. Nowadays, the lowest drag coefficient is 0.21 and reaching a lower result is very complicated. Hence, mechanical engineers are working hard in new ways of improving the aerodynamic.

Current trends are moving towards the channelling of the air flow inside the vehicle. This air can be tapped for refrigerating the passenger area and also, all part of the engine.

Figure 42 - Current aerodynamic trends

6. List of tables and figures

- List of tables

- List of figures

7. Bibliography

ANSYS. (2009). *ANSYS FLUENT 12.0 user´s guide* .

Epifanov, V. (2014). Boundary Layer. *Thermopedia*.

Europea, L. c. (n.d.). *Normativa UE 1230/2012 Anexo I.*

George, P. E. (n.d.). *History of Aerodynamic Car design.*

McBeath, S. (2005). *Aerodinamica del automovil de competicion.* CEAC.

Ministerio de Economia, I. y. (n.d.). *Manual de Reforma de Vehículos* .

Mora, X. (2016). Las Ecuaciones de Navier-Stokes. *Metode Science Studies Journal*, 67-73.

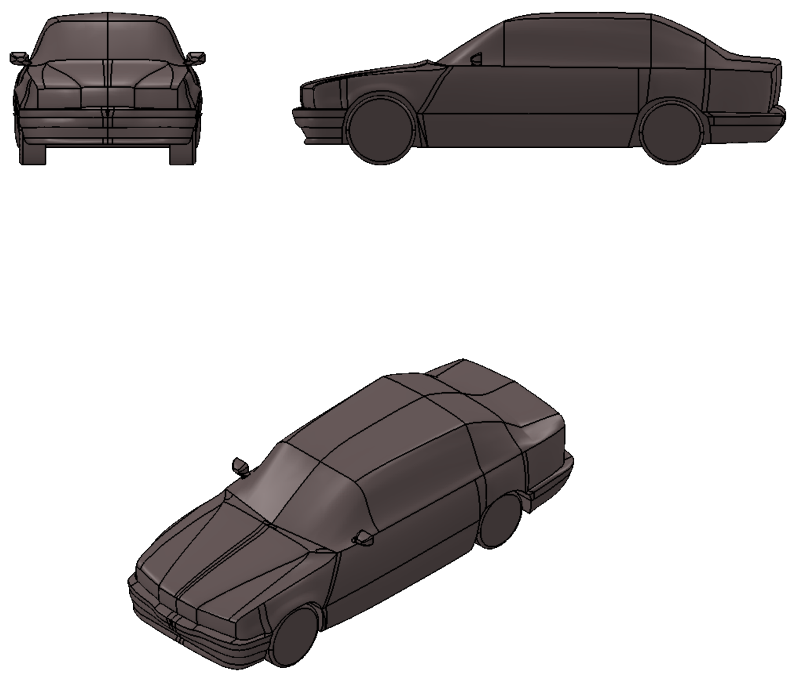
Ortego, L. M. (2013, Marzo 31). La aerodinamica y la eficiencia: un vistazo a la historia.

skybrary. (2015). *skybrary.* Retrieved from CAST: https://www.skybrary.aero/index.php/AP4ATCO_-_Aerofoil_Terminology#References

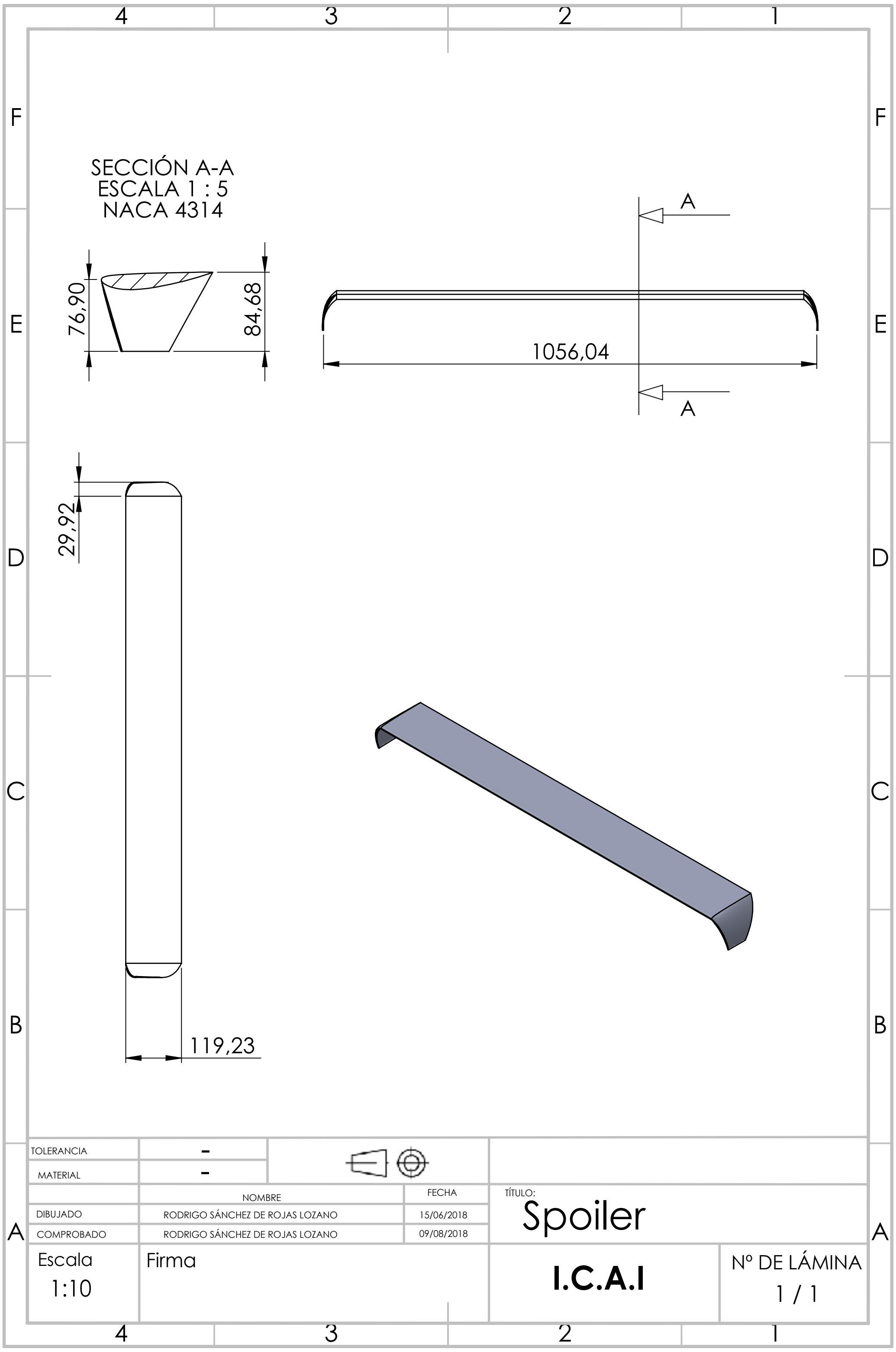
Ventura, E. E. (2014). *Demostracion de la ecuacion de Bernoulli.*

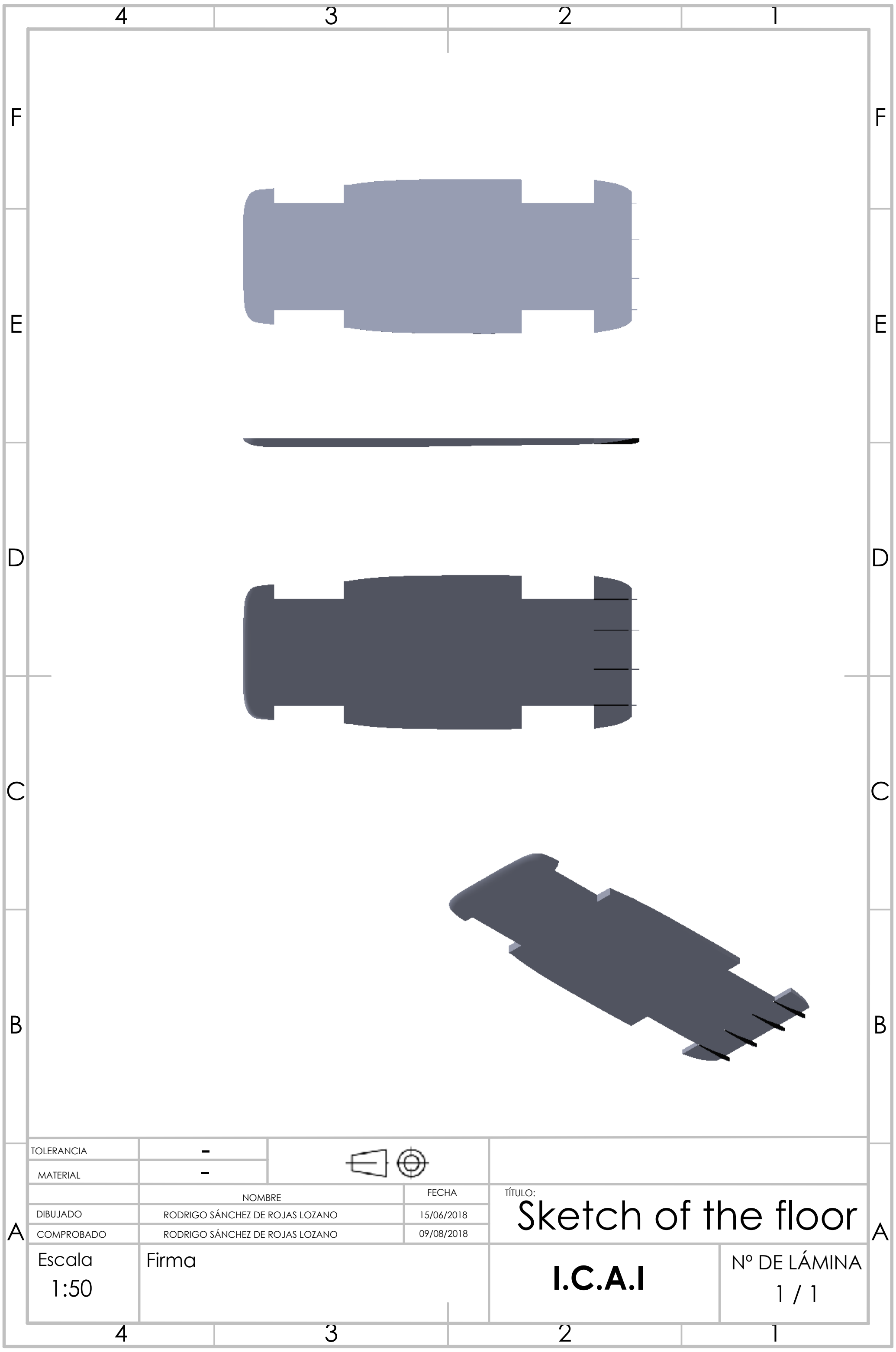
White, F. M. (2003). *Fluid Mechanics.* McGraw-Hill Series in Mechanical Engineering.

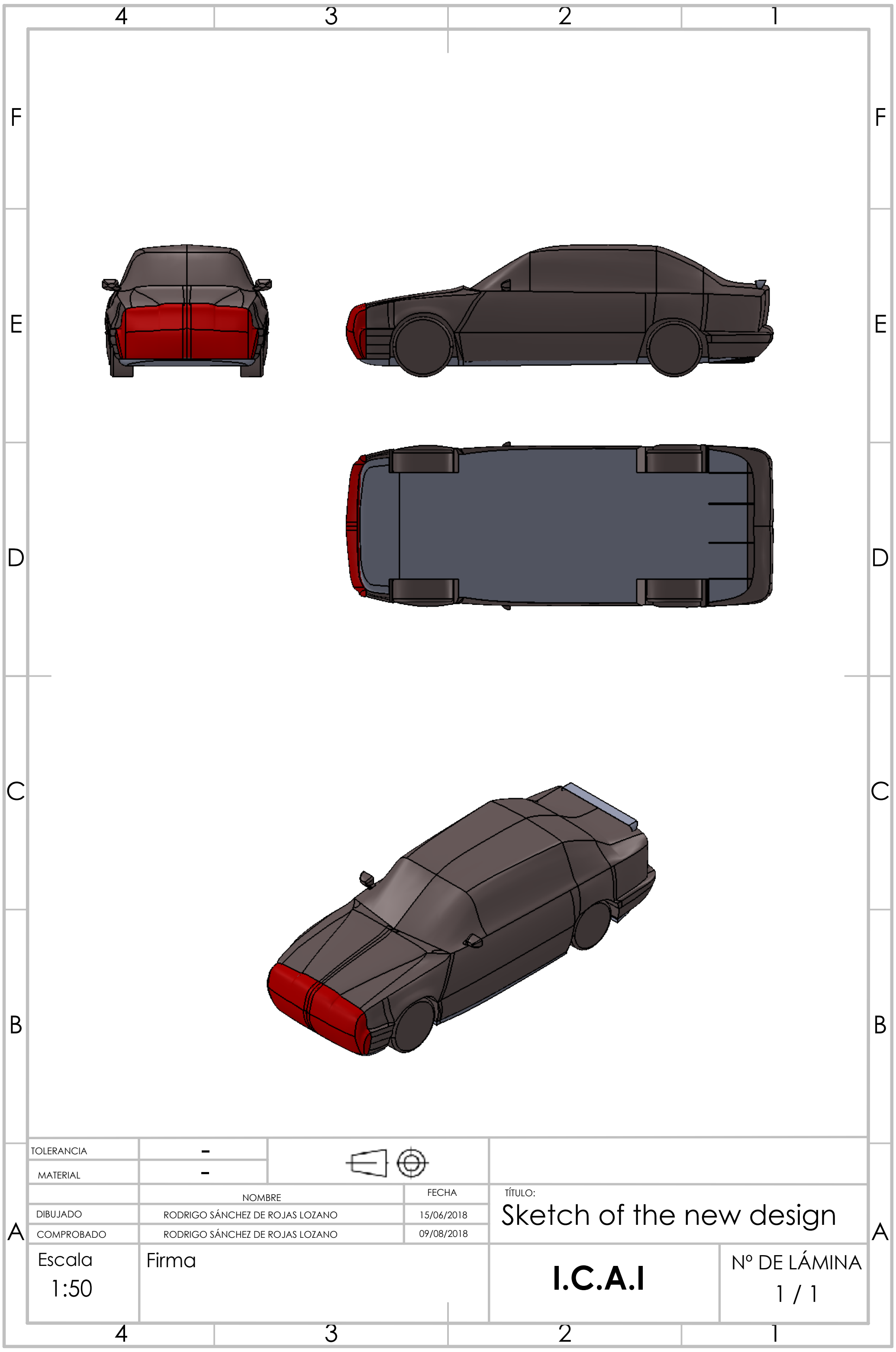
Annex A: Plans and sketches



TOLERANCIA	-		
MATERIAL	-		
	NOMBRE	FECHA	TÍTULO:
DIBUJADO	RODRIGO SÁNCHEZ DE ROJAS LOZANO	15/06/2018	Sketch of the old design
COMPROBADO	RODRIGO SÁNCHEZ DE ROJAS LOZANO	09/08/2018	
Escala 1:50	Firma	I.C.A.I	Nº DE LÁMINA 1 / 1


SECCIÓN A-A
ESCALA 1 : 5
NACA 4314
76,90
84,68
A
A
1056,04
29,92
119,23
TOLERANCIA -
MATERIAL -
NOMBRE
FECHA
DIBUJADO RODRIGO SÁNCHEZ DE ROJAS LOZANO 15/06/2018
COMPROBADO RODRIGO SÁNCHEZ DE ROJAS LOZANO 09/08/2018
TÍTULO:
Spoiler
Escala
1:10
Firma
I.C.A.I
Nº DE LÁMINA
1 / 1


TOLERANCIA -
MATERIAL -
NOMBRE
FECHA
DIBUJADO RODRIGO SÁNCHEZ DE ROJAS LOZANO 15/06/2018
COMPROBADO RODRIGO SÁNCHEZ DE ROJAS LOZANO 09/08/2018
TÍTULO:
Sketch of the floor
Escala 1:50
Firma
I.C.A.I
Nº DE LÁMINA 1 / 1



TOLERANCIA -
MATERIAL -
NOMBRE
FECHA
DIBUJADO RODRIGO SÁNCHEZ DE ROJAS LOZANO 15/06/2018
COMPROBADO RODRIGO SÁNCHEZ DE ROJAS LOZANO 09/08/2018
TÍTULO:
Sketch of the new design
Escala 1:50
Firma
I.C.A.I
Nº DE LÁMINA 1 / 1