



Final Project

External Aerodynamics of a Car Using SimFlow

ME412 - Introduction to Computational Fluid Dynamics

Meliha Delalić – Department of Mechanical Engineering

Prof.Dr.Muhamed Hadžiabdić

Sarajevo, January 2025.

Table of Contents

1. Project Objectives.....	3
2. Introduction.....	4
3. METHODOLOGY - Preprocessing	6
3.1. Importing Geometry.....	6
3.2. Generating Mesh.....	7
3.3. Turbulence Model.....	8
3.4. Boundary Conditions	8
3.5. Solver and Flow Assumptions	9
4. METHODOLOGY – Processing.....	10
5. METHODOLOGY – Postprocessing.....	13
5.1. Velocity and Pressure Fields	13
5.2. Drag and Lift Coefficients.....	13
6. Results.....	18
6.1. Velocity and Pressure Fields	18
6.2. Drag and Lift Forces and Coefficients	21
7. Discussion	25
8. Conclusion.....	26
9. Tables, Figures and Pictures	27
10. Literature and Sources.....	28

1. Project Objectives

In this paper, we are going to explore the physics behind external aerodynamics of a car, as well as to analyze such simulation done in SimFlow computational software.

The main objective of this project is to analyze the external flow around the car geometry and see how it affects the flow in some critical regions where flow separation could occur. We will discuss parameters that can reduce undesirable effects such as wake region and drag force. Therefore, velocity and pressure fields and their visualization is very important for identifying critical regions. It is also necessary to obtain the drag and lift coefficients that help us understand the overall performance and efficiency of the vehicle.

The postprocessing will be done in Paraview as it is intergated within Simflow, and provides a solid interpretation of results.

2. Introduction

Automotive industry relies heavily on using computational tools in almost every stage of manufacturing process. In a time when climate change and its impacts are exponentially growing, achieving a high-speed car comes with an additional challenge – to reduce the gas exhaustion and still maintain good performance of the vehicle.

The two critical fields for analysis in car aerodynamics are velocity and pressure fields. These allow us to recognize critical regions such as high pressure area in the front, and also separation region on the back of the car where low pressure bubble forms. These can easily be visualized by flow streamlines that follow the shape of the car and eventually separate at sharp edges, again creating low pressure zones.

Drag force is the main factor influencing the speed of the car as it acts in the opposite direction of the air flow and pulls it back. To resist this force, car therefore requires more energy, resulting in the increased fuel consumption. That is where optimizing aerodynamics of the car comes into play: careful design geometry and critical areas can significantly reduce the drag force and therefore increase its efficiency which results in reduction of fuel consumption. During the last century, testing cars and the flows was done almost entirely in wind tunnels, when computational power was yet to be established. Flow simulations finally became available to industries in the 1970s, with McDonnell Douglas and IBM being first companies to use it.¹ Nowadays, using CFD for testing car performance is an irreplaceable step in the manufacturing workflow because it offers a cost-effective solution for analysis and optimization.

Managing lift force is another challenge that must be addressed, particularly in case of supercars, such as race cars, that are designed for high speeds and agility. When a car achieves high speeds, lift force is generated as an undesirable effect which must be counteracted by a rear wing placed on the back of the car. In critical moments, the rear wing redirects the airflow upward, creating a downward force that pushes the car back to the road, enabling stability and safe turning at sharp angles.

We know that both drag and lift forces depend heavily on the geometry of the car, its surface roughness, and also the fluid properties such as viscosity and density. However, area of attack is what matters the most for both forces since we are dealing with a bulk body such as car. That means that in our analysis, the pressure plays the major role in generating both drag and lift forces, as shear stress is more significant for long and slender areas of the car, which makes it a smaller fraction of the drag and lift forces.

¹ *The History of Computational Fluid Dynamics* | Resolved Analytics. (n.d.). Resolved Analytics.
<https://www.resolvedanalytics.com/cfd/history-of-cfd#:~:text=The%20first%20truly%20practical%20CFD,relatively%20short%20amount%20of%20time.>

Therefore, we can define drag and lift forces and their components due to pressure and shear stress as:

$$F_D = F_{D,pressure} + F_{D,shear}$$

$$F_L = F_{L,pressure} + F_{L,shear}$$

Though shear stress components are expected to be negligible, they can not be fully excluded from the analysis as they still contribute to the flow development in near-wall region, where the velocity boundary layer is not fully developed.

Overall performance of the car is evaluated by drag and lift coefficients (c_D and c_L) which depend on the forces we discussed above, but also on the upstream velocity which represents velocity of the air sufficiently far from the car where the flow is unaffected by the velocity boundary layer. These coefficients are calculated by the following expressions:

$$c_D = \frac{F_D}{\frac{1}{2} \rho U_\infty^2 A_{frontal}}$$

where c_D is the drag coefficient, ρ is the fluid density, U_∞ is upstream velocity and $A_{frontal}$ is the frontal area attacked by drag force. Similarly, the expression for lift coefficient is as follows:

$$c_L = \frac{F_L}{\frac{1}{2} \rho U_\infty^2 A_{planform}}$$

The main difference is that now we are using the planform area of the car, which is perpendicular to the vertical flow.

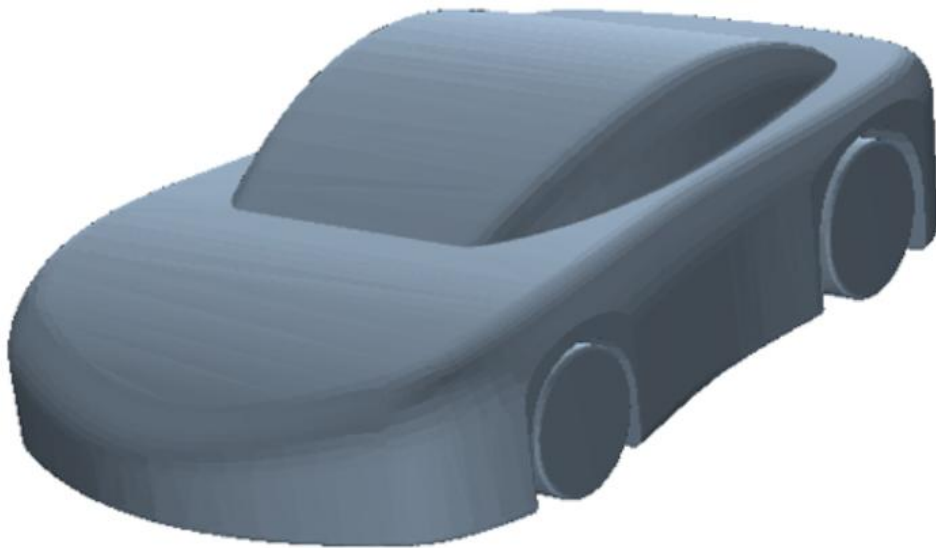
In the following sections, we will explain the methodology of the test case including the preprocessing setup, as well as the postprocessing needed to visualize the velocity and pressure field, flow streamlines, and most importantly, how to calculate the drag and lift coefficients.

3. METHODOLOGY - Preprocessing

In the preprocessing phase, several steps like generating mesh, setting boundary conditions and solver setup must be correctly done to ensure valid results in the postprocessing phase.

3.1. Importing Geometry

To start this test case, we first import the car geometry .stl file in SimFlow.



Picture 1 – Car geometry.

In the layout section of this file, we can find the dimensions of the car body to be:

$$h = 2.2142 \text{ m}$$

$$w = 1.0436 \text{ m}$$

$$l = 3.5676 \text{ m}$$

Next we will create the the car and base mesh in SimFlow.

3.2. Generating Mesh

To capture appropriately the flow behaviour around the car, we need to adjust the mesh near its surface to be more fine than in other regions. In simflow, the refinement for car body is set to:

$$MIN = 1 \text{ and } MAX = 3$$

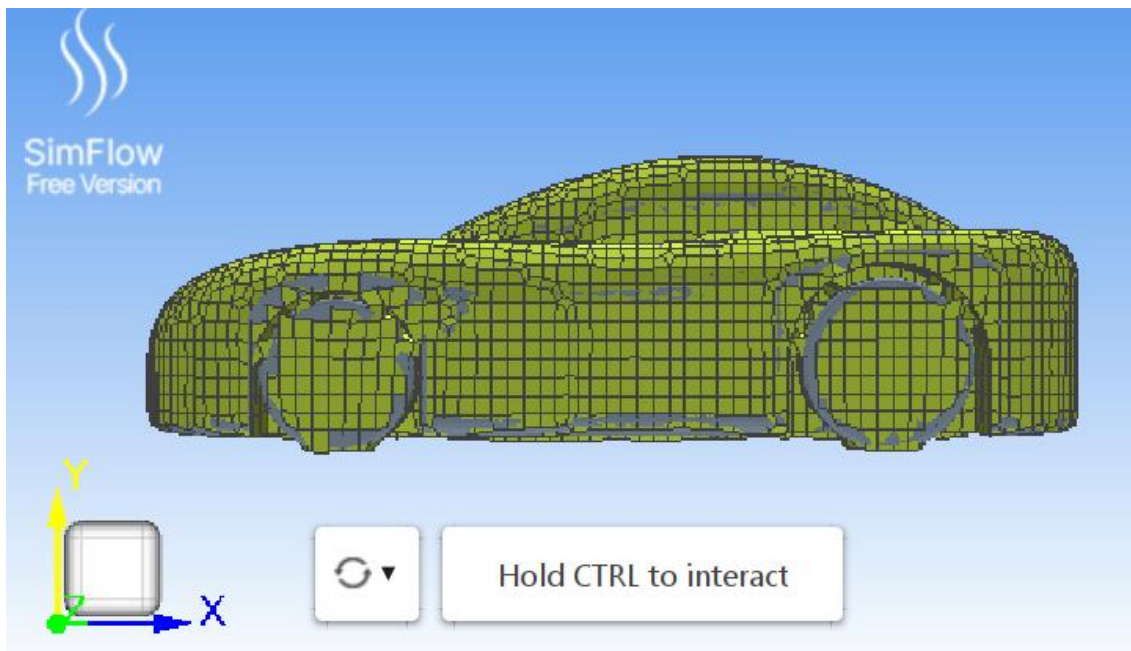
The refinement for base mesh is set to:

$$MIN: (-3,0,0) \text{ and } MAX: (10,5,6)$$

Also, division of the base mesh is:

$$DIVISION: (30,10,12)$$

By ensuring that division is small enough, we can create a mesh that is sufficiently fine which leads to smooth visualization and more accurate results.



Picture 2 – Car mesh.

We can see how the mesh is more fine in the front and also in the back of the car. This will ensure that we capture high pressure zones and separation bubble in the back of the car.

3.3. Turbulence Model

For this case, we are using the standard $k - \omega$ SST turbulence model which is commonly used for aerodynamics application. This model is also based on RANS, and is similar to the standard $k - \varepsilon$ model. In addition to the usual continuity equation, this model also solves two PDE's with the variables being k (turbulent kinetic energy) and ω which is the rate of dissipation per unit turbulent kinetic energy. So, the relationship between the dissipation rate ε , and ω is described by:

$$\varepsilon = C_\mu k \omega$$

where $C_\mu = 0.09$. This model is applied in cases where the flow is influenced by walls, or similarly constrained regions where it offers more accurate results. It is however, more computationally expensive than $k - \varepsilon$ model for many external flow problems.

3.4. Boundary Conditions

To define the boundary conditions for this case, we first need to name each side of the base mesh accordingly. Then, specific boundary conditions will be assigned.

Base mesh side	Name	Boundary Condition	Turbulence
-X	Inlet	$U = 20 \frac{m}{s}$	$k = 5e - 03$ $\omega = 1e - 03 m$
+X	Outlet		
-Y	Bottom	$U = 20 \frac{m}{s}$	
+Y	Top	<i>Slip wall condition</i>	<i>0 gradient for k and ω</i>
-Z	Symmetry	<i>Symmetry</i>	
+Z	Right	<i>Slip wall condition</i>	<i>0 gradient for k and ω</i>

Table 1 – Defining boundary conditions.

To simulate the movement of the car on the road, we impose initial velocity in x-direction on the bottom of the base mesh, and also at its inlet. Then, *Slip wall condition*

is necessary to simulate the warping of air around the car body. *Symmetry* is imposed on the car so that we reduce the computational cost, so we will only have results for one half of the car. These can easily be mirrored for a full picture.

Next, we also defined the turbulence parameters, turbulent intensity (k), and mixing length (ω).

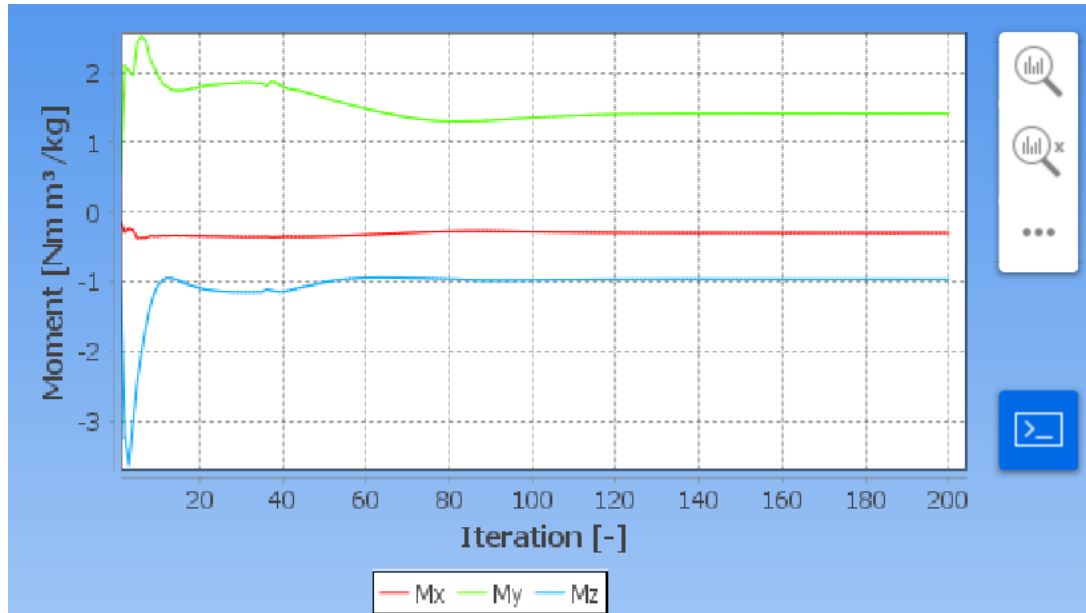
3.5. Solver and Flow Assumptions

Finally, the only remaining step is to set the solver to *Simple* and run the simulation. Number of iterations in this case was set to 200 and the conditions for the flow conditions that are turned on are *Steady State*, *Incompressible* flow.

Thanks to the fact that we set the constant speed of the car, we can assume that the flow behaviour will also remain constant during the motion as there are not any additional changes applied to disrupt the initial conditions. As for incompressibility of the flow, it is proved that for all flows with $Mach < 0.3$, the flow is considered incompressible because there are not any large scale changes of air density.

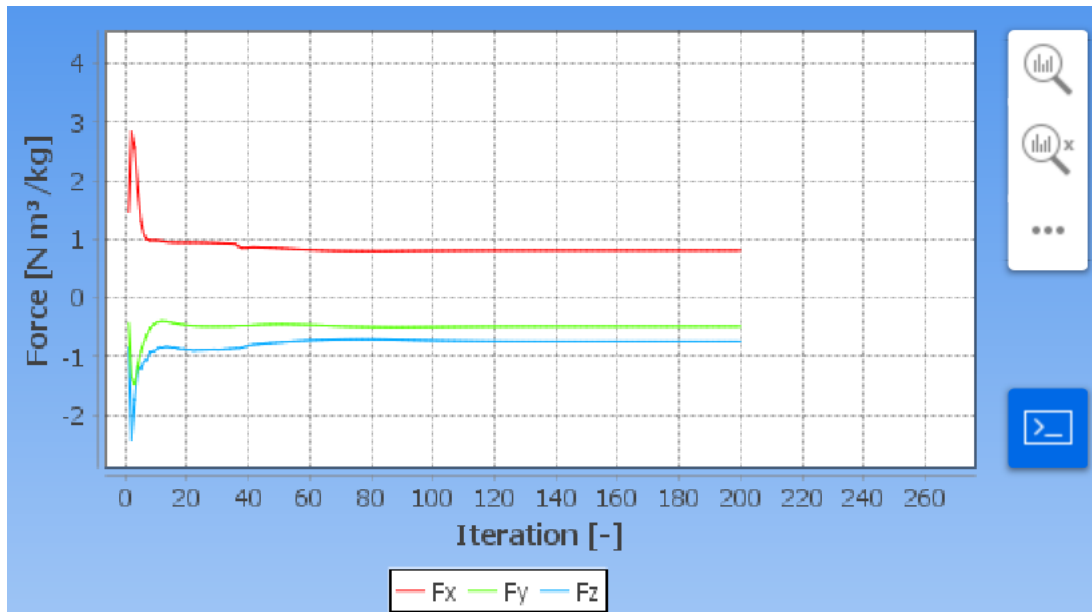
4. METHODOLOGY – Processing

In this section, we will demonstrate the behaviour of moment, force and residual during the time of solving the case.



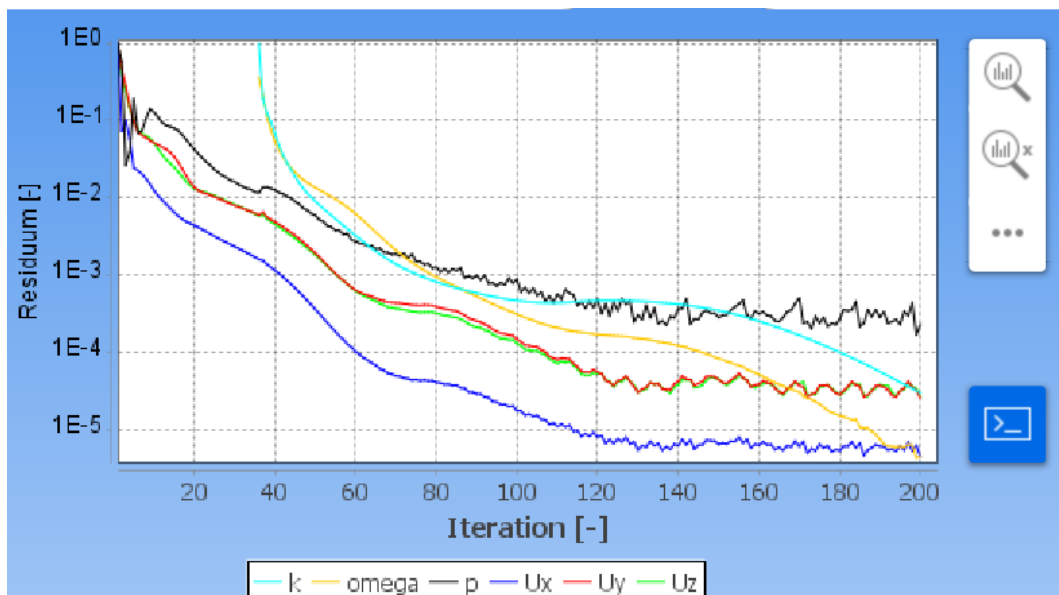
Plot 1 – Moment components during the simulation processing.

Moment in x-direction continuously falls around zero meaning that there are no pitching tendencies produced by the flow at the inlet of the domain. However, the moment component in z and y-directions seem to be mirrors of each other indicating how the flow in these directions adjusts to the boundary conditions at the right and top sides at the start of simulation. As the time passes, these moment also stabilize, leading to no further disruptions of the flow and it also does not push the car to rotate around z-axis (0 yaw angle).



Plot 2 – Force components variations.

Regarding the forces, at the start of running the simulation, the x component is at a positive peak where the changes due to boundary conditions are first experienced by the solver. Going further to the right, the force value stabilizes below 1. As for the y and z components, these peak in the negative half at the start, indicating how they act in the opposite direction and they are influenced by the initial turbulence in the flow field.



Plot 3 – Residuals variations during the simulation.

In the plot 3, we can see how errors are gradually reduced for each property during the solving process. This gradual decrease is present up to the values of close to $1e - 3$ for

the turbulent dissipation ω , and velocity field converges at the residual values below $1e - 4$. The pressure field is observed to have the smallest error at the last iterations, reaching values of below $1e - 5$. This indicates that the pressure field achieves highest level of accuracy. However, k has the most stable residual variation, converging with the same error as the y-velocity component.

Next we will go through the postprocessing analysis done in Paraview where the simulation results are visualized and calculated.

5. METHODOLOGY – Postprocessing

In this section, we will explain the steps and tools used to visualize velocity and pressure fields, and also steps and tools required for calculating drag and lift coefficients.

5.1. Velocity and Pressure Fields

To visualize the velocity and pressure fields, as well as their streamlines, we apply the filter called StreamTracer. To make sure that the streamlines are visualized in the entire area, integration type is „Both“. Also, for the visual enhancement, in the display section of this filter, we also set the opacity and specular lightning to maximum. Next, we rescale the data to custom range, in order to capture the regions with lower and higher intensities of velocity and pressure.

5.2. Drag and Lift Coefficients

Calculating drag and lift coefficients requires calculation of the drag and lift forces, value of the air density and upstream velocity, as well as the areas of attack in both cases.

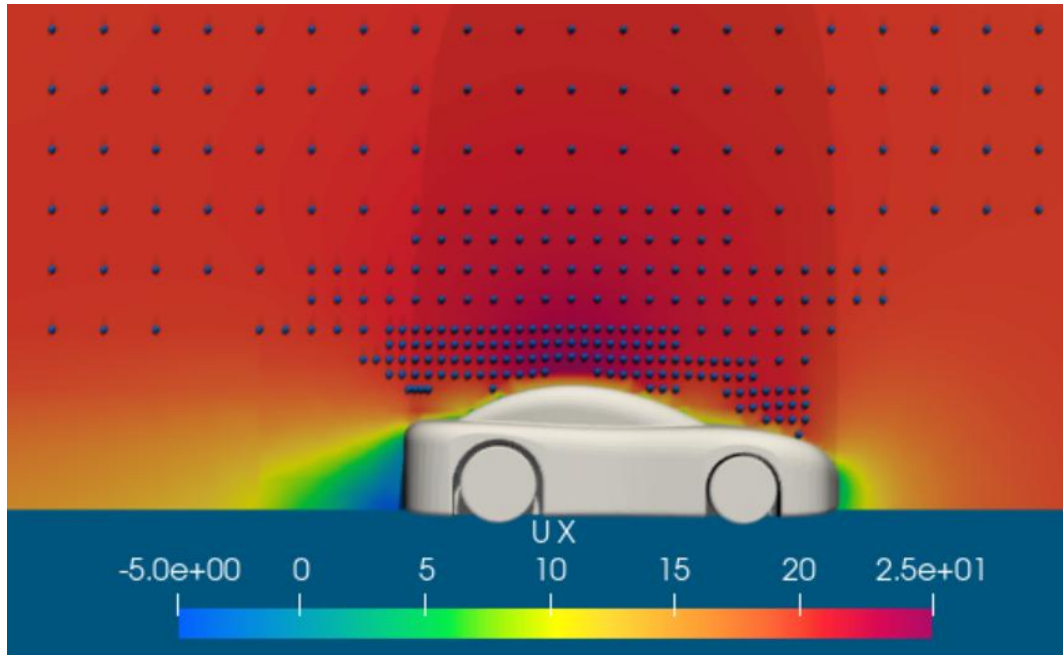
Air Density (ρ)

We take the value of air density at $T = 20^\circ\text{C}$ to be $1.204 \frac{\text{kg}}{\text{m}^3}$.

Turbulent dynamic viscosity (μ_T)

In Paraview, one of the resulting parameters in the data array is kinematic turbulent viscosity. It is necessary that we calculate the dynamic viscosity for obtaining shear stress components of forces.

$$\mu_T = \rho \nu = 1.204 \frac{\text{kg}}{\text{m}^3} * 1.5 * 10^{-5} \frac{\text{m}^2}{\text{s}} = 1.806 * 10^{-5} \frac{\text{kg}}{\text{m} * \text{s}}$$



Picture 3 – Using Find Data to read the upstream velocity value in x-direction.

Upstream Velocity (U_{∞})

We know that the inlet velocity was set to $20 \frac{m}{s}$. We can therefore assume that this will also be the upstream in the controlled conditions. We can also use the tool *Find Data* in Paraview which allows us to read the exact values of velocity in all directions at any point of the region.

Frontal Area ($A_{frontal}$)

Area of attack for drag force is called frontal area and it is calculated as:

$$A_{frontal} = h * w = 1.0436 \text{ m} * 2.2142 = 2.3107 \text{ m}^2$$

Planform Area ($A_{planform}$)

$$A_{planform} = l * w = 3.5676 \text{ m} * 1.0436 \text{ m} = 3.7231 \text{ m}^2$$

We see that for drag force, the frontal area is perpendicular to the fluid flow in x-direction, and in the case of lift force, it is the planform area (x-z plane) that is perpendicular to flow in y-direction.

Drag and Lift Forces

We again recall that drag and lift forces depend on pressure and shear stress. In case of drag force, we will calculate these component relative to x-direction, while for lift, the normals and velocity components are in y-direction. Other than directions, the steps for extracting these forces are fairly same.

$$F_D = F_{D,pressure} + F_{D,shear}$$

$$F_L = F_{L,pressure} + F_{L,shear}$$

We start by opening the file .foam in Paraview and then apply *Cell Data to Point Data* filter to it. This is done because throughout the postprocessing and analysis, we are only dealing with point data, so they must be converted from the volume data. Next step is to apply *Extract Surface* filter which will isolate the surface of the car. Then, we apply *Generate Surface Normals* so that all normal vectors are computed for every surface. This way we can have normal vectors in all three directions which ensures that forces can be integrated over the car's surface. After this, we can apply the calculator filter for each force component.

Pressure components of drag and lift forces is found by:

$$F_{D,pressure} = -Normals_x * p$$

$$F_{L,pressure} = -Normal_y * p$$

Negative sign is placed to ensure we get the positive values for forces, otherwise they would be negative as the surface normals are directed outwards and the forces are acting towards the car's body.

Shear stress components of drag and lift forces are calculated by:

$$F_{D,shear} = -\tau_x * Normals_x = -\mu_T \frac{dU}{dx} * Normals_x$$

$$F_{L,shear} = -\tau_y * Normals_y = -\mu_T \frac{dV}{dy} * Normals_y$$

After all the four components are calculated, we can get the total drag and lift forces and then use them for calculating the drag and shear coefficients by the formulas:

$$c_D = \frac{F_D}{\frac{1}{2} \rho U_\infty^2 A_{frontal}}$$

$$c_L = \frac{F_L}{\frac{1}{2} \rho U_\infty^2 A_{planform}}$$

It is important to note that we used *Integrate Variables* to sum up all the point contributions of force components and therefore of the total drag and lift force as well.

Final values of the c_D and c_L are read from the spreadsheet viewer.

Point ID	c_drag	Fd_pressure	Fd_shear	Fd_total		
0	0.181721	101.112	-1.85561e-44	101.112	-36.4205	501.368

Picture 4 – Calculation steps for c_D

Point ID	c_lift	Fl_pressure	Fl_shear	Fl_total		
0	0.218012	195.453	-6.55024e-45	195.453	-36.4205	501.368

Picture 5 – Calculation steps for c_L

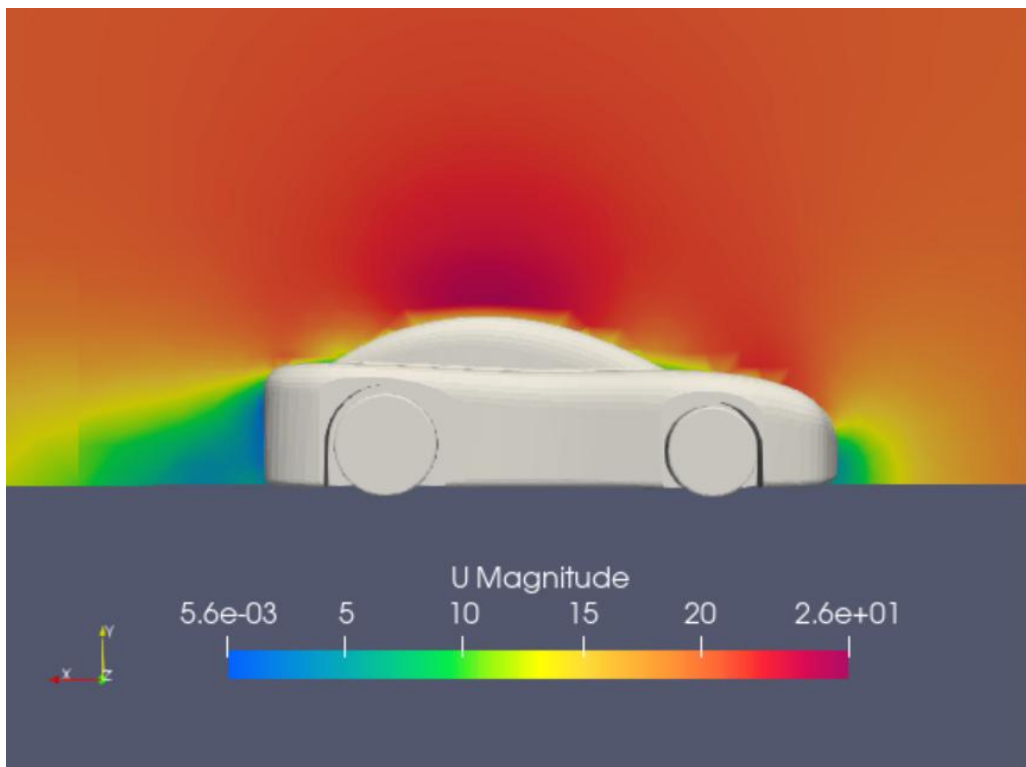
Resulting pressure force component is positive, and so is the total resulting force. However, shear stress force component is negative since it represent the friction force that opposes the car motion.

In the next chapter, we will present the resulting visualizations and table values.

6. Results

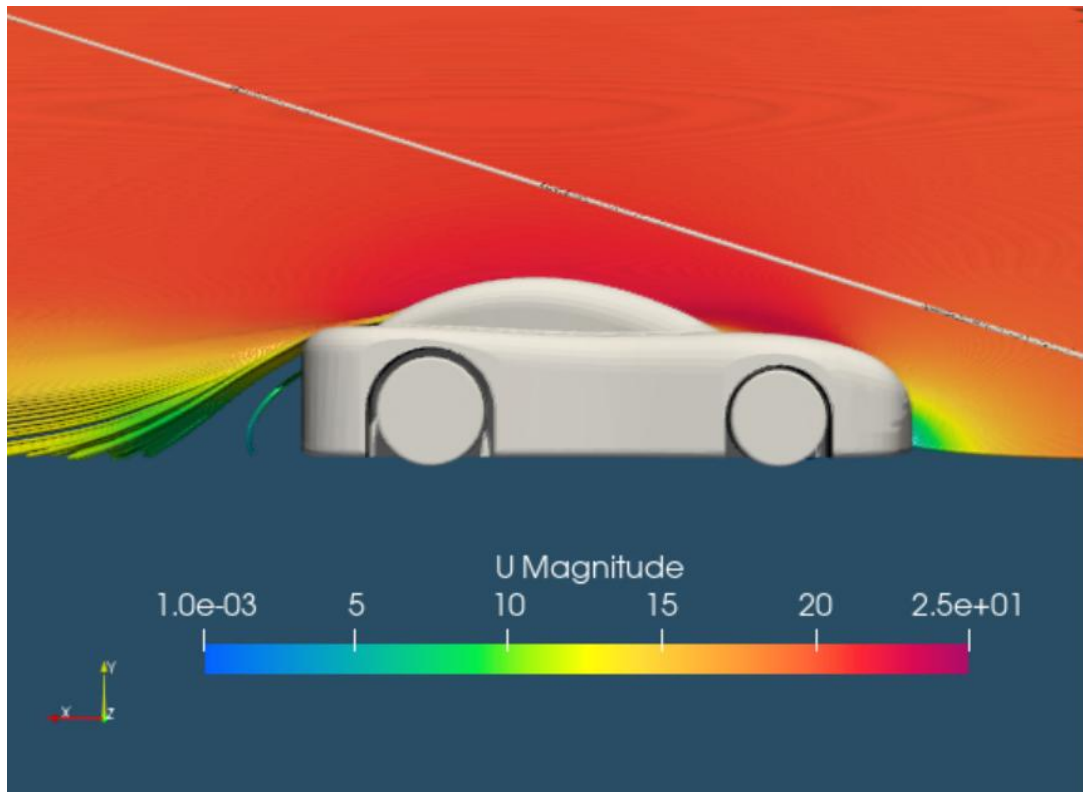
After all the steps in preprocessing phase are done, we can see that in Paraview, resulting properties are found in DataArray such as k , ω , velocity, pressure, turbulent kinematic viscosity and also some positional parameters. Using this data and the right tools, we can produce significant visualizations.

6.1. Velocity and Pressure Fields



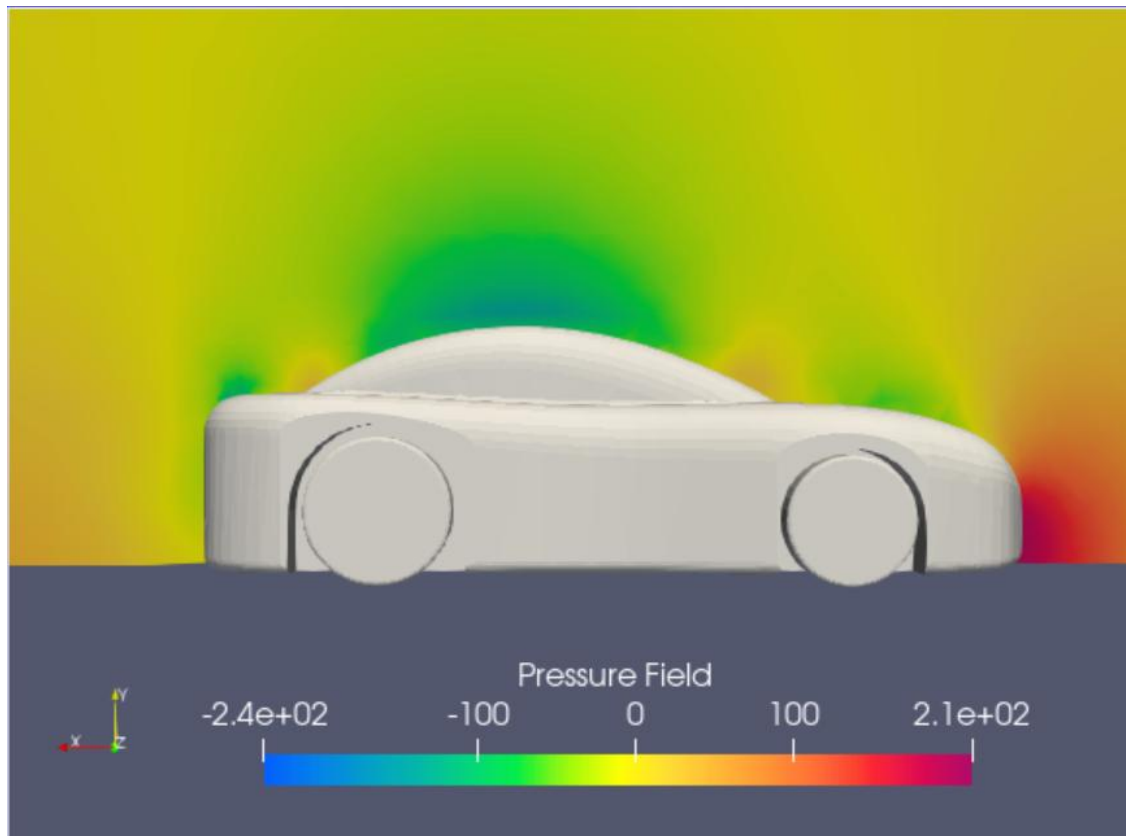
Picture 5 – Velocity field.

The highest velocity magnitudes are found right at the top of the car where it reaches values up to $26 \frac{m}{s}$. After this point, the flow starts to separate and trip behind the car, creating a low-pressure zone that acts as a vacuum and tries to suck the car from the back. In this region, also known as separation bubble, the velocity reaches its smallest values that go as far as to $0.0036 \frac{m}{s}$. The high range of velocity values indicates the presence of flow separation.



Picture 6 – Velocity field streamlines.

In the picture above, the wake region is even more enhanced thanks to the velocity streamlines that are slightly curled towards the car, showing how air trips back and swirls due to dissipation of turbulent eddies. In this region, turbulent kinetic energy of the smallest eddies is converted to heat through shear forces.

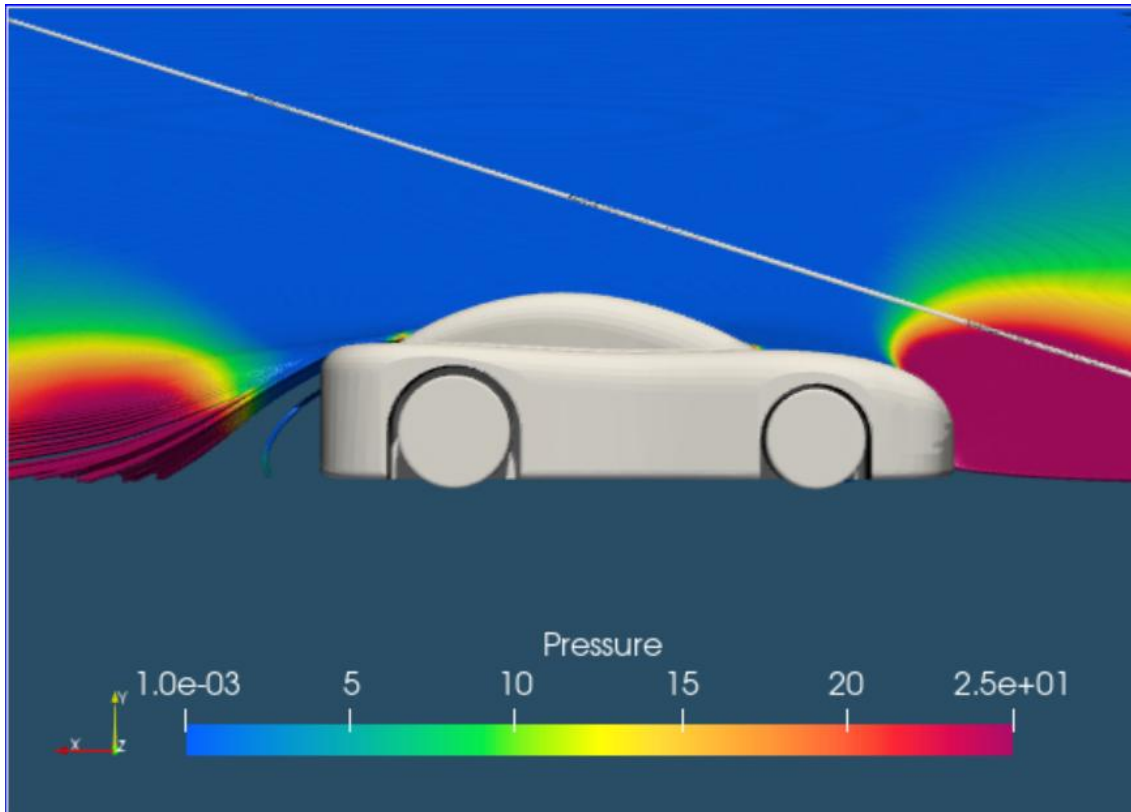


Picture 7 – Pressure field.

As for the pressure field, it clearly complements the velocity field which means that at the regions of lower velocity, pressure will be high. Maximum pressure is seen right at the front of the car where the high speed air strikes the car first. As the flow faces any edges in the car's shape, it will inevitably lead to some amount of flow separation which creates low pressure zones. The convexity of the car's ceiling is responsible for low pressure zone at its top, since the flow separates at the point of meeting with a steep front windshield.

Pressure in the wake region reaches near-zero values, showing how drag force due to pressure is present.

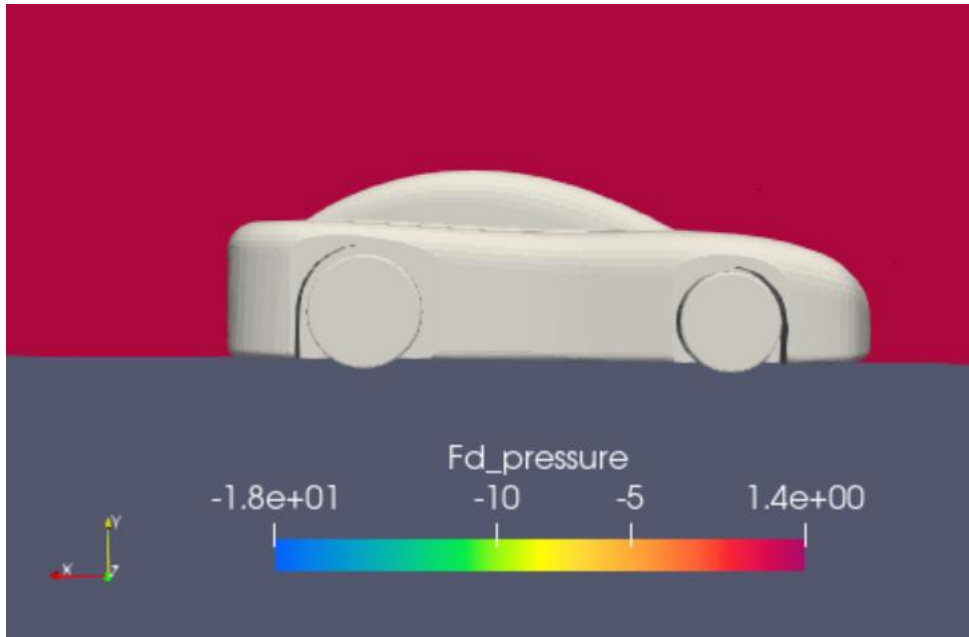
As for the pressure streamlines seen below, it is seen how pressure streamlines trip behind the car, however they soon reattach to the original flow and reach high pressure values. Also, the front of the car is exposed to high pressure, again visualized by the streamlines.



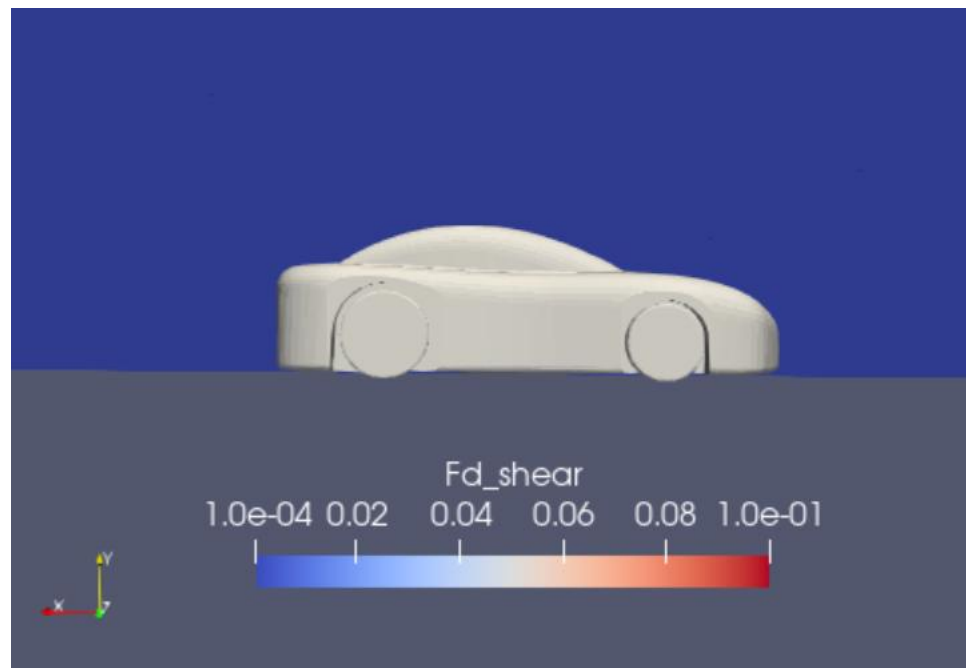
Picture 8 – Pressure field streamlines.

6.2. Drag and Lift Forces and Coefficients

The resulting drag and lift forces and their components due to pressure and shear stress are shown below:

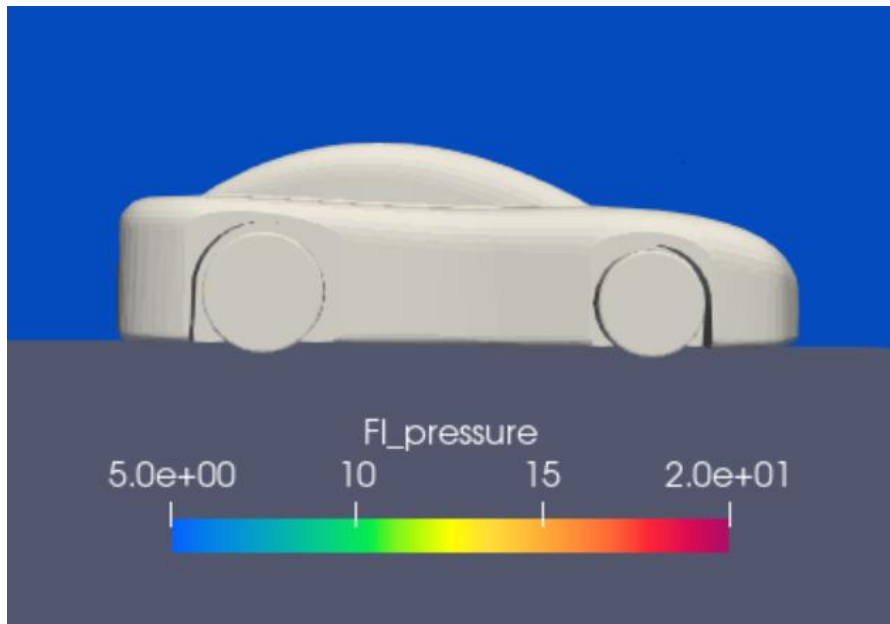


Picture 9 – Drag force component due to pressure.

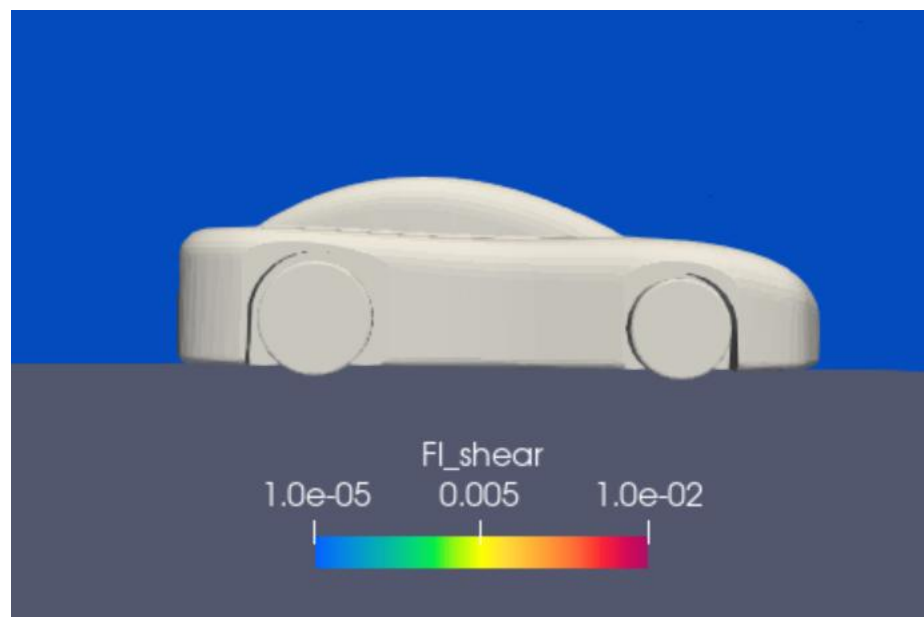


Picture 10 – Drag force component due to shear stress.

The point values of drag force due to pressure reaches maximum value of 14 Pa , and drag force due to shear stress sits on the lower end of the range. That is because for bulk bodies such as a car, drag force is mostly due to pressure, but still, shear stress component of drag force must be accounted for.



Picture 11 – Lift force component due to pressure.



Picture 12 – Lift force component due to shear stress.

Now we can see that, both lift force components lay at the lower end of the range. This is because lift force is not generally as impactful as drag force in case of car aerodynamics. But, when it comes to special type of cars, such as supercars and race cars, lift force is necessary for keeping the car on the ground.

The resulting force components and total forces after integrating them over the surface are shown in the tables:

Drag Force		Lift Force	
$F_{D,pressure}$	$F_{D,shear}$	$F_{L,pressure}$	$F_{L,shear}$
101.112	$-1.855e - 44$	195.453	$-6.550e - 45$

Table 2 – Drag and lift force components

$F_{D,total}$	$F_{L,total}$	c_D	c_L
101.112	195.453	0.181721	0.218012

Table 3 – Total values of drag and lift forces, drag and lift coefficients.

**All the forces are in Newtons.*

7. Discussion

To get a better grasp of our results, let us analyze the standard drag coefficient values for different types of cars. A typical passenger car has c_D ranging somewhere between 0.25 – 0.33. Value of 0.2 and lower is considered amazing performance and for sport cars, manufacturers aim to reduce the drag coefficient even below 0.2, by optimizing the slender shape of the car, and its frontal area design.

Streamlined cars optimized for low drag typically have large lift coefficients which can be said about most EV's. Usual street cars have lift coefficients between 0.15 – 0.20. However, it is considered ideal to have near-zero lift. As for supercars, their goal is to have negative lift coefficient with values up to -0.5 , which means a sufficient downward force is generated to keep it on the ground.²



Picture 13 – BMW M3 with $c_d = 0.33$ and $c_l = 0.08$.

When compared to the standard drag coefficients of existing cars, our CFD result of $c_D = 0.181$ may seem unrealistic since achieving such low drag coefficient is exceptionally challenging in reality. We must, however, note that the turbulence model we used is particularly sensitive to mesh quality. Coarse mesh may fail to resolve the near wall conditions, leading to lower drag force which in the end results in lower than expected drag coefficient. Since SimFlow had limitations in its free version for adjusting

² Occamsracers, A. (2024, April 4). *Thinking in aerodynamic coefficients*. Occam's Racer. <https://occamsracers.com/2024/03/22/thinking-in-aerodynamic-coefficients/>

mesh refinement, we had to work on the mesh quality within the prescribed limits. That is why, instead of setting the mesh refinement to be $MIN = 3$ and $MAX = 5$, we had to decrease it and set to $MIN = 1$ and $MAX = 3$.

On the other hand, lift coefficient value of $c_L = 0.218$ may not be completely unrealistic. Despite the error present in c_D , it is very likely that still it does not go above the current c_L value. It is common that low drag cars have higher lift coefficients, but given that the mesh was not refined enough and the pressure was not resolved fully in critical regions, we must admit some level of error.

Overall the results for drag and lift coefficients seem respectable enough for the current setup conditions, and more accurate values could be obtained by having high quality meshing.

8. Conclusion

Analyzing external aerodynamics of a car can offer a good insight in its overall performance and fuel consumption requirements. Observing the velocity and pressure fields allows us to detect wake region that generates drag force due to low pressure. Another aspect for optimizing the flow conditions is reducing the frontal area which can lead to a decrease in the drag force ($F_D = \frac{1}{2} \rho U_\infty^2 A$).

Simulation resulted in unusually low drag coefficient ($c_D = 0.181$), though this can be explained by poor mesh quality. Higher lift coefficient ($c_L = 0.218$) pairs well with lower drag coefficient, however due to coarse mesh, it must be off by some degree just like c_D . To obtain more accurate results in the future, mesh refinement must be sufficiently increased.

9. Tables, Figures and Pictures

Table 1 – Defining boundary conditions.

Table 2 – Drag and lift force components

Table 3 – Total values of drag and lift forces, drag and lift coefficients.

Plot 1 – Moment components during the simulation processing.

Plot 2 – Force components variations.

Plot 3 – Residuals variations during the simulation.

Picture 1 – Car geometry in the .stl file.

Picture 2 – Car mesh.

Picture 3 – Using Find Data to read the upstream velocity value in x-direction.

Picture 4 – Calculation steps for c_D

Picture 5 – Velocity field.

Picture 6 – Velocity field streamlines.

Picture 7 – Pressure field.

Picture 8 – Pressure field streamlines.

Picture 9 – Drag force component due to pressure.

Picture 10 – Drag force component due to shear stress.

Picture 11 – Lift force component due to pressure.

Picture 12 – Lift force component due to shear stress.

Picture 13 – BMW M3 with $c_d = 0.33$ and $c_l = 0.08$.

10. Literature and Sources

Fluid Mechanics Fundamentals and Applications by Yunus A.Çengel and John M.Cimbala, second edition.

The History of Computational Fluid Dynamics / Resolved Analytics. (n.d.). Resolved Analytics. <https://www.resolvedanalytics.com/cfd/history-of-cfd#:~:text=The%20first%20truly%20practical%20CFD,relatively%20short%20amount%20of%20time>

Compressible Flow vs Incompressible Flow in Fluid Mechanics. (2023, August 11). SimScale. <https://www.simscale.com/docs/simwiki/cfd-computational-fluid-dynamics/compressible-flow-vs-incompressible-flow/>

Occamsracers, A. (2024, April 4). *Thinking in aerodynamic coefficients.* Occam's Racer. <https://occamsracers.com/2024/03/22/thinking-in-aerodynamic-coefficients/>

Software used:

- SimFlow 5.0. 30
- Paraview 12.5.0

Link to the SimFlow tutorial on car aerodynamics case:

Car CFD Simulation Software. (n.d.).

<https://help.sim-flow.com/tutorials/car>