MITO CFD ANALYSIS REPORTS 1 & 2

Car Body Design and Aerodynamics – Academic Year 24/25

Politecnico di Torino

Massone Tommaso Francesco s347793

Exercise 1

Introduction

The aim of this first Lab experience is to familiarize with the simulation software. In order to do it, a simulation of a given geometry was performed. The conditions of the test had been settled by the guide and the geometry given is the Alfa Romeo Mito. After the simulations, the results had to be analysed.
This first analysis focusses on evaluating the aerodynamic performance of the car. The simulation was conducted under specific test conditions in order to determine the Drag Coefficient (Cd) and Lift Coefficient (Cl)

The goals are:

- Learn basics of CFD workflow
- Learn how to setup and run a CFD simulation
- Learn how to analyse aerodynamic results
- Learn how present your results in efficient way

The test was performed considering:

- Standard production vehicle
- Constant speed of 38.89 m/s or 140km/h
- Closed air intake

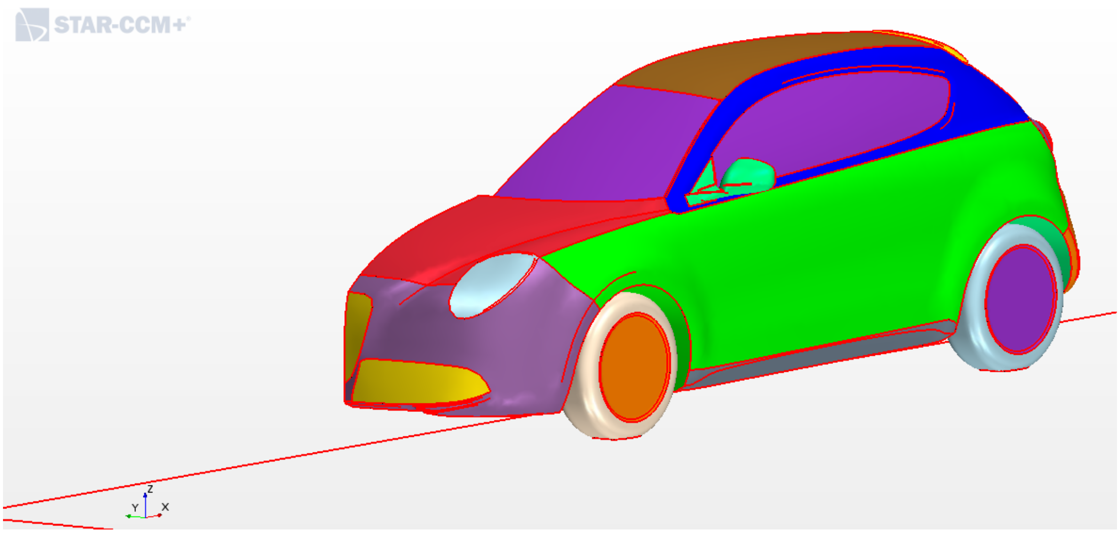

Figure 1 – Alfa Romeo Mito Geometry 1

Setup surface and volume mesh

In order to obtain reliable results, the set-up of mesh is a key step. The density of the mesh is a delicate balance between precision of the data and computational time, which is also influenced by the processing power of the computer used. Therefore, a denser mesh is applied when greater accuracy is required, such as for key aerodynamic components such as the spoiler, side mirrors, diffuser, and pillars. On the other hand, less dense mesh is employed for other parts of the vehicle and the wind tunnel to optimize computational efficiency.

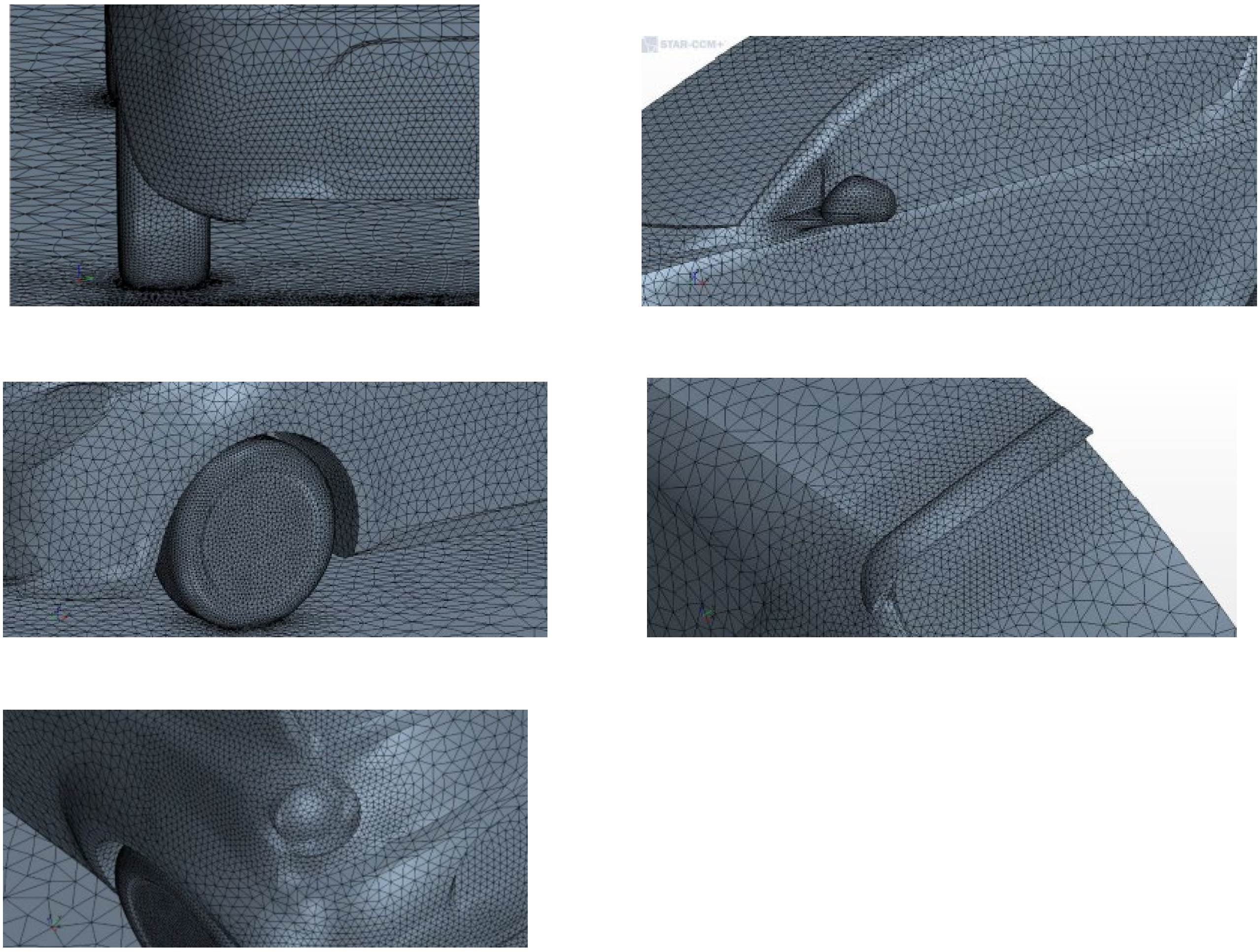

Figure 2 – Different aerodynamic surfaces with a different mesh density

To the mesh near the wall, a *High Y+* approach was used, which estimates the quantities in the first cells.

The parameters used for the base mesh (surface and volume) are:

- Base size = 25 mm
- Target Surface Size = 200%
- Minimum Surface Size = 100%
- Mesh density = 0.7

The parameters used for the aerodynamic surfaces mesh are:

- Target Surface Size = 75%
- Minimum Surface Size = 37.5%

The base prism layer was defined as absolute value:

- Prism layer near wall thickness = 2mm
- Numbers of layers = 3
- Total layer thickness = 10 mm

After finishing the mesh set up, the next step involves defining the physics of the simulation by attributing the relevant parameters. With these preparations finalized, the simulation can be executed. A stopping benchmark of 1000 iterations is established in order to determine when the simulation should conclude, the more iterations be applied, the more results converge.

Results

The Residual Analysis is a powerful tool to analyse the quality of the simulation. Due to the great reduction of the residual value, we can consider the results as reliable.

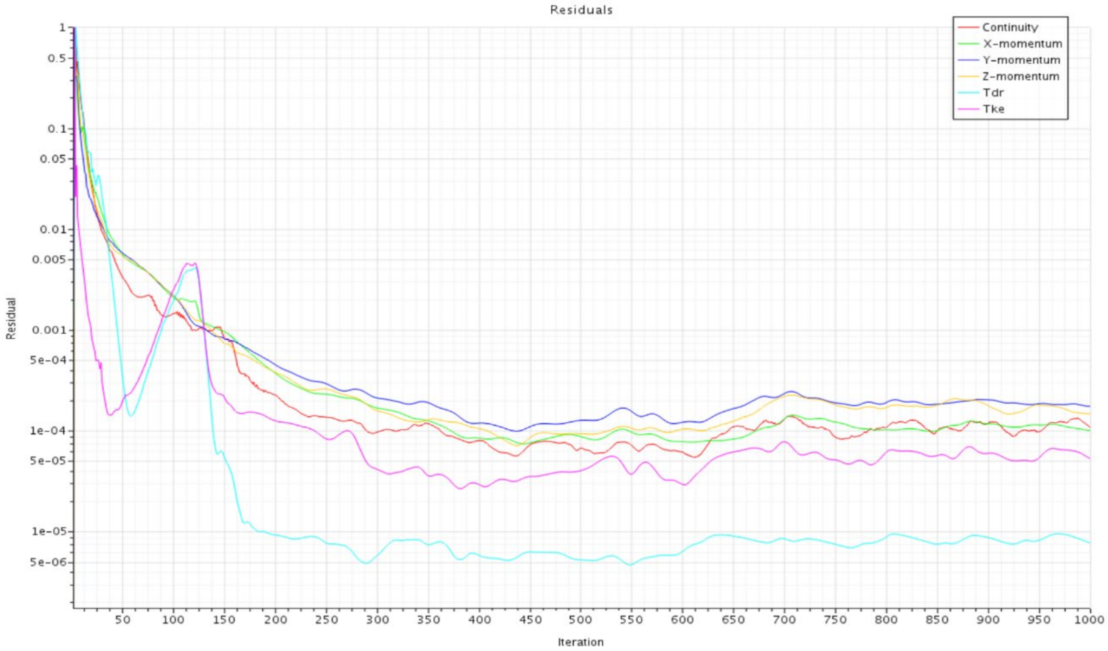

Figure 3 – Residuals

The other values obtained by the simulations are the Drag coefficient Cd and the Lift coefficient Cl

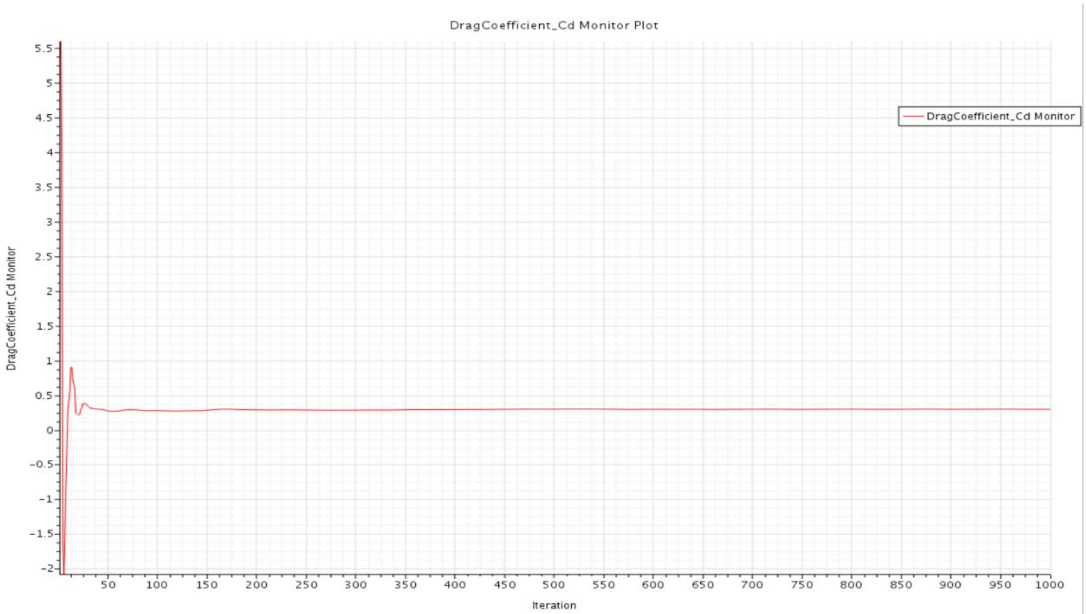

Figure 4 – Cd results

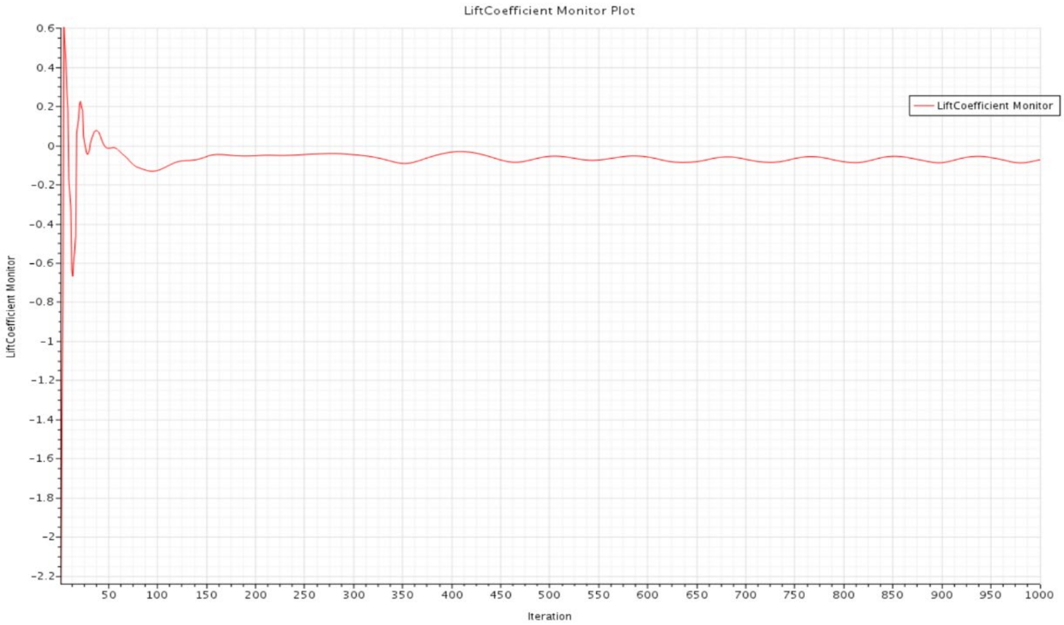

Figure 5 – Cl results

The values obtained from this simulation are:

- *Cd = 0.299*
- *Cl = - 0.077*

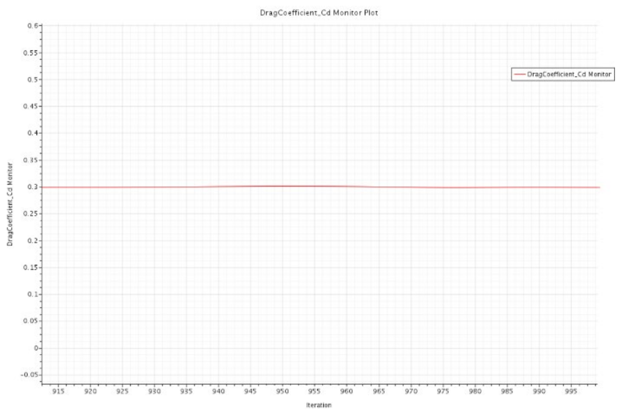

Figure 6 – Cd zoom results

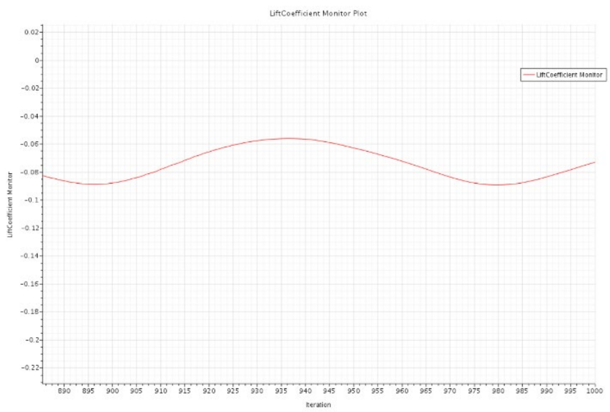

Figure 7 – Cl zoom results

Whereas the experimental values obtained from the wind tunnel are the following:

- *Cd = 0.29*
- *Cl = 0.1*

Conclusion

To sum up, it is clear that the values of the Drag coefficient are relatively consistent, meanwhile those of the Lift coefficient show several variances. In the simulation, the presence of a negative Lift coefficient suggests that the vehicle performs better aerodynamically than it does, which can be instinctive explained for certain reasons.
Generally, efforts aim to minimize both the Drag and Lift coefficients. In particular, a negative Lift coefficient in the simulation indicates that the vehicle generates downforce, enhancing its stability and performance. Therefore, we can say that during the simulation the vehicle performed better than in real life. This difference can be solved improving the fidelity of the model, that in some parts is simplified.
To align the simulation more closely to the wind tunnel results, the following solutions can be considered:

- *Higher-quality meshing:* Increasing mesh density, either for the entire vehicle and wind tunnel or focusing on key aerodynamic parts, can improve the precision. However, this will require more computational time and power. A gradual increase in density can strike a balance between accuracy and resource-use.

- *Increasing iterations:* Running more iterations may help the Drag and Lift coefficients converge more closely to experimental values.

- *Convergence analysis:* A detailed examination of residual convergence is crucial, especially since higher mesh density may affect convergence criteria.

In conclusion, during the simulation the vehicle performed better than in real life. This difference can be solved improving the fidelity of the model, that in some parts is simplified interfacing with pros and cons.

Exercise 2

Introduction

The aim of this second Lab experience is to analyse the difference in performance that can affect our model according to a change in its geometry.
The change is the presence of a front wheel spat.

The goal of this exercise is to analyse

- Differences in Cd between the 2 configurations
- Visualization of the differences through scalar and vector fields

The conditions of the tests are the same than the first Lab

- Standard production vehicle
- Constant speed of 38.89 m/s or 140km/h
- Closed air intake

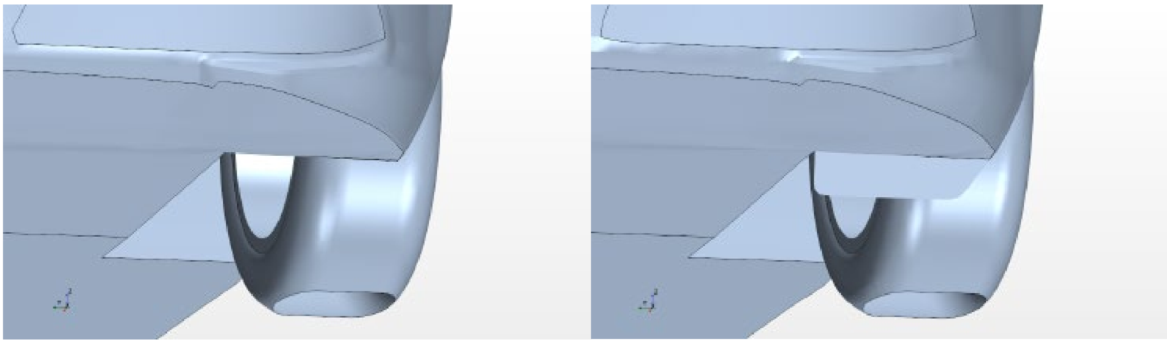

Figure 1 – Differences between the 2 configurations

Post-Processing

Both models were imported into the software, where the geometries, surface mesh, and volume mesh were meticulously inspected. New scenes were created in order to compute and analyse various field functions, visualize the flow fields, and evaluate the vehicle's aerodynamic behaviour.

Surface Pressure Coefficient Cp

The surface pressure coefficient is initially analysed to understand the pressure distribution on the vehicle's surface. This metric is essential for identifying areas that generate drag or lift and spotting regions of flow separation.

In comparing the 2 configurations, it is evident that the pressure distribution is different. Focusing on the front wheel, the area of the wheel's surface where Cp > 0 is smaller in the configuration with the wheel spat compared to the one without it. This occurs because the air encounters the wheel spat before reaching the wheel itself.

Since Cp > 0 at the vehicle's front correlates with drag generation, it can be concluded that the wheel with the spat configuration produces less drag. This reduction is due to the lower air pressure creating up in front of the wheel.

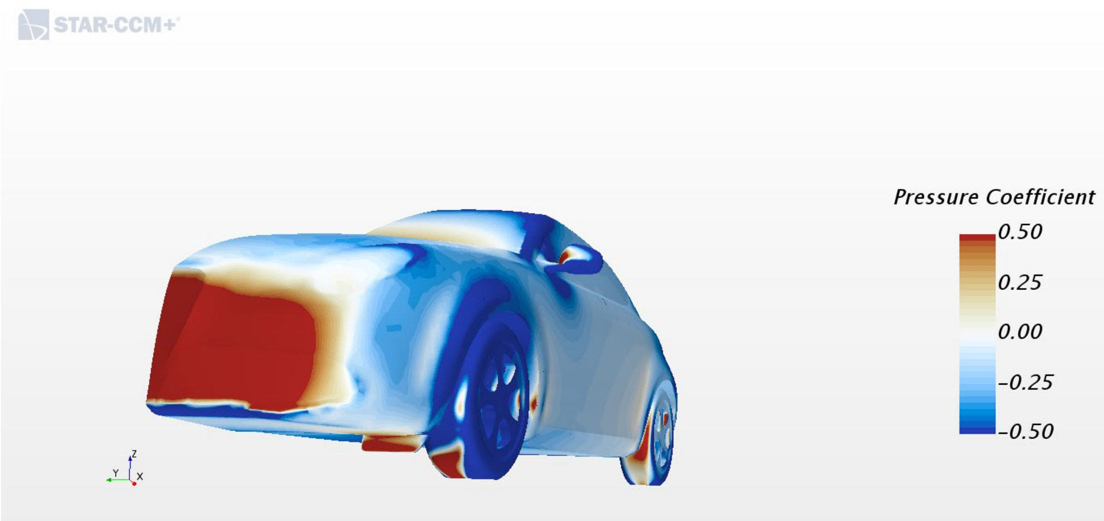

Figure 2 –Pressure distribution with wheel spats

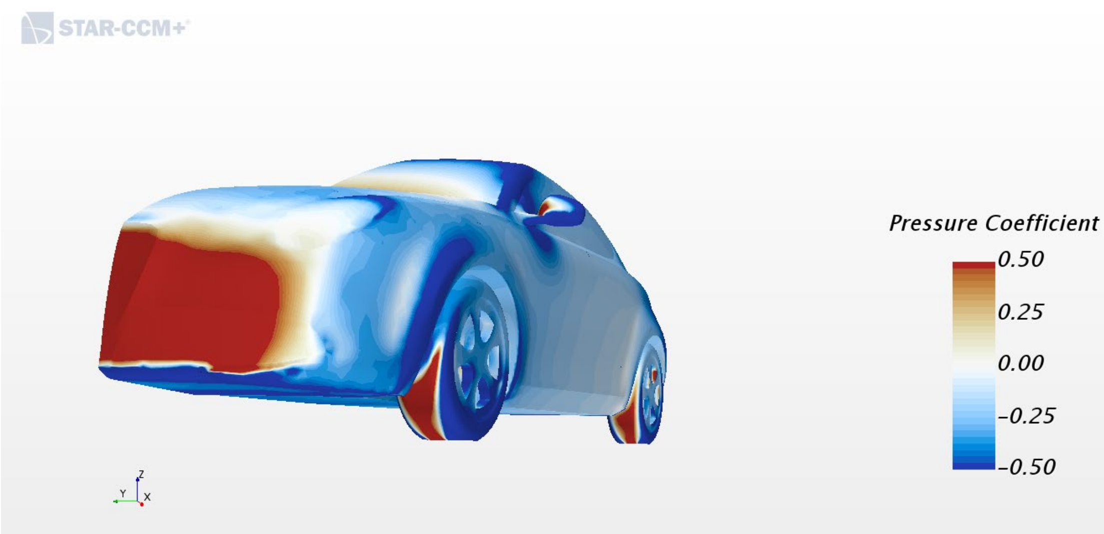

Figure 3 – Pressure distribution without wheel spats

Skin Friction Coefficient Cf

Wall shear stress is a key factor to understand how the flow interacts with the vehicle's surface. While it is not directly linked to drag generation, it plays a significant role in the overall aerodynamic performance of the vehicle.

The primary difference between the 2 configurations come to light in the wake downstream of the front wheel. This wake, typically marked by turbulent airflow and vortices, is where flow separation occurs. High Cf values indicate that the flow remains attached to the vehicle's surface, whereas low Cf values correspond to regions of flow separation.

In the configuration with the wheel spat, the region downstream of the wheel shows a higher friction coefficient, signifying reduced turbulence. This indicates better flow attachment around the vehicle in the wheel spat configuration.

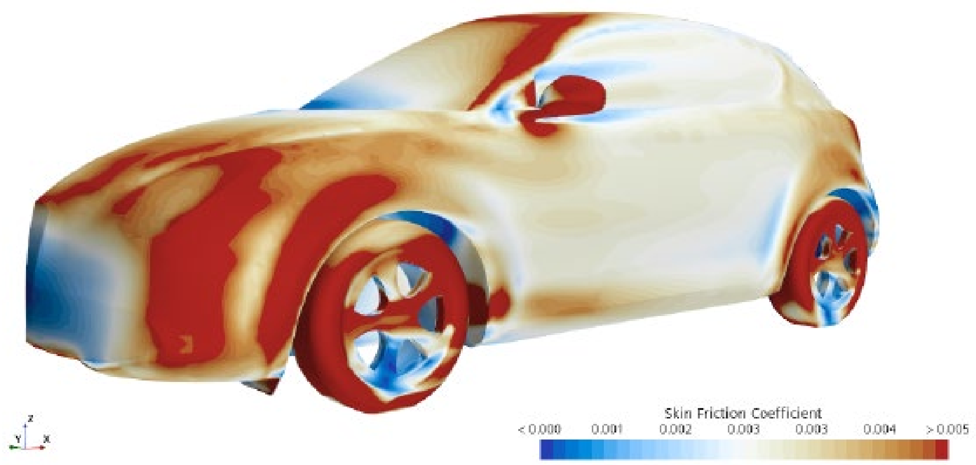

Figure 4 – Skin friction with wheel spats

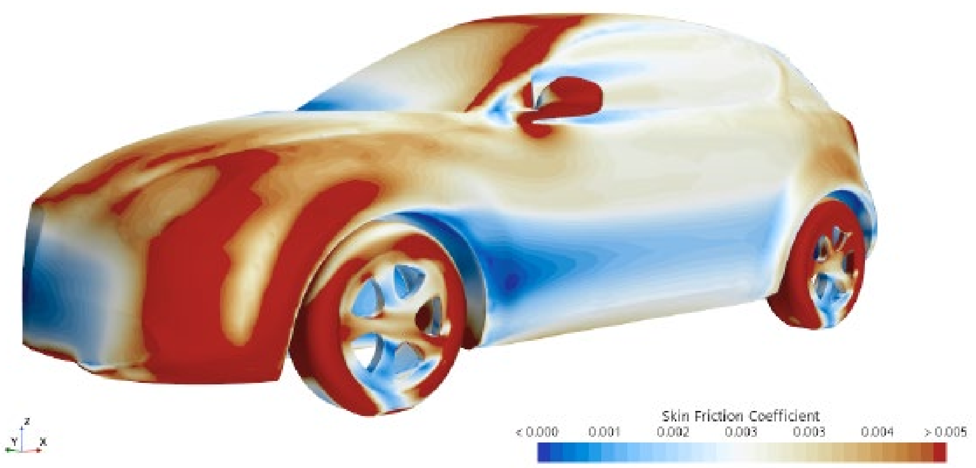

Figure 5 – Skin friction without wheel spats

Streamlines

Streamlines are used to visualize the direction and behaviour of the fluid flow.

A key relationship to remember is that regions with the highest static pressure coefficient (Cp) correspond to areas of the lowest velocity, a phenomenon clearly observable at the front of the vehicle.

The flow paths differ between the 2 configurations. In the case of the wheel spat, the high velocity flow is directed outward, away from the wheel, showing how the spat influences the aerodynamics by altering the flow trajectory.

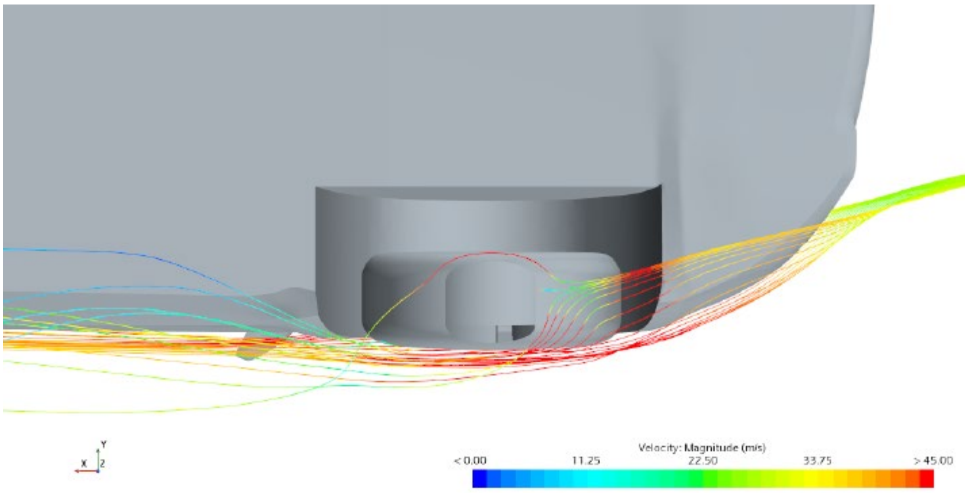

Figure 6 – Streamlines with wheel spats

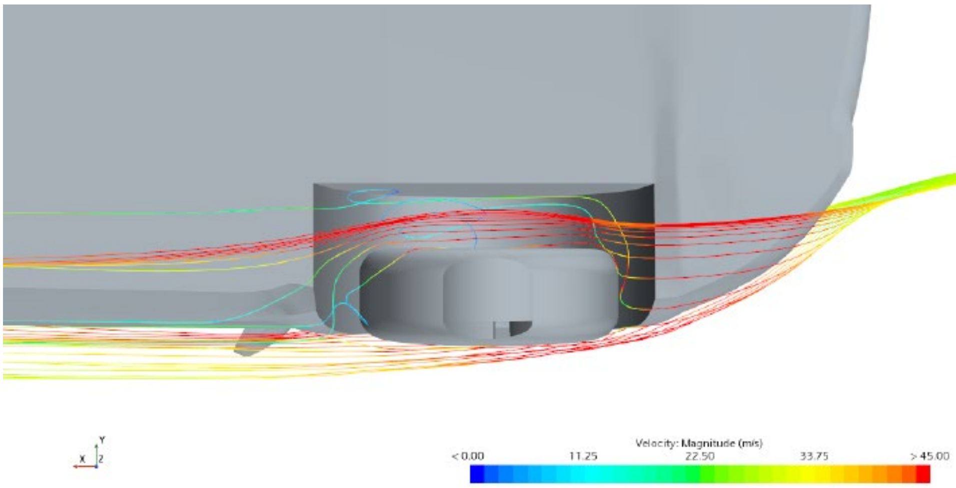

Figure 7 – Streamlines without wheel spats

Total Pressure Coefficient Cptot

The total pressure coefficient is a valuable tool for analysing the wake regions of the vehicle.

Thanks to the skin friction coefficient plot, the most significant difference between the 2 configurations is the presence of a wake downstream of the front wheel in the first case, where no wheel spat is used. In contrast, this wake is absent in the configuration with the wheel spat, highlighting its impact on reducing turbulence.

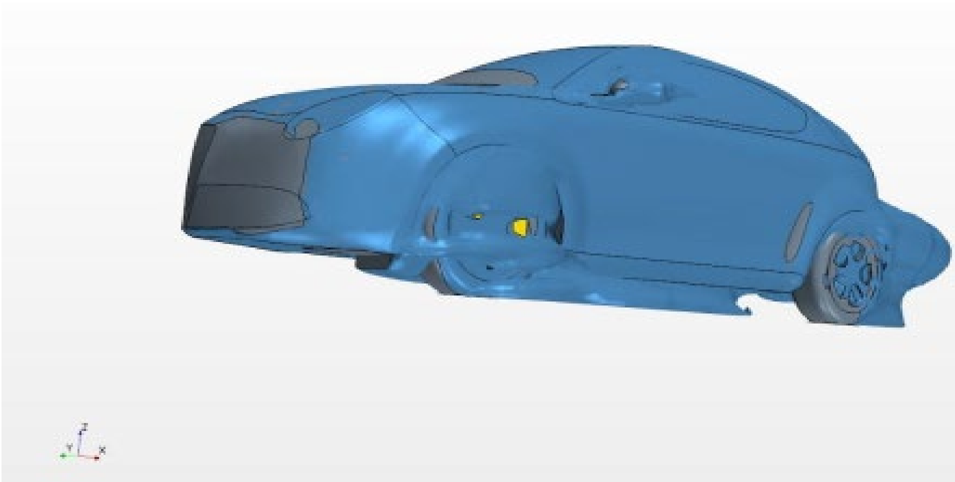

Figure 8 - Cptot with wheel spats

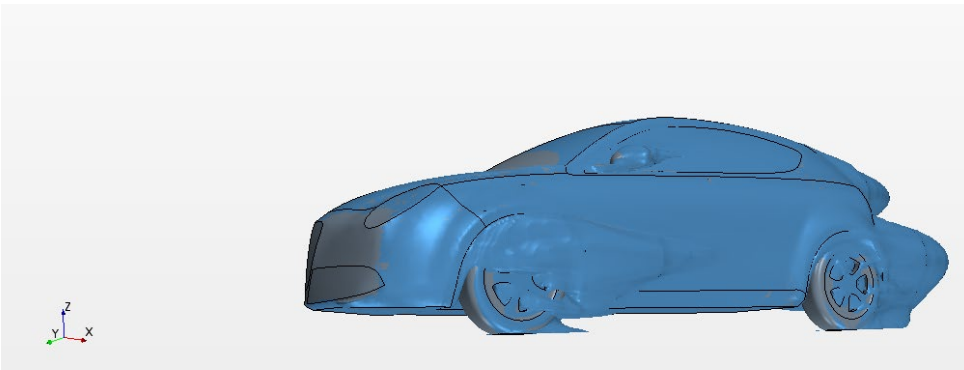

Figure 9 – Cptot without wheel spats

Drag Coefficient outline

Another useful tool is the display of the accumulated force along the vehicle, in this way it is possible to see the differences in every part of the vehicle.

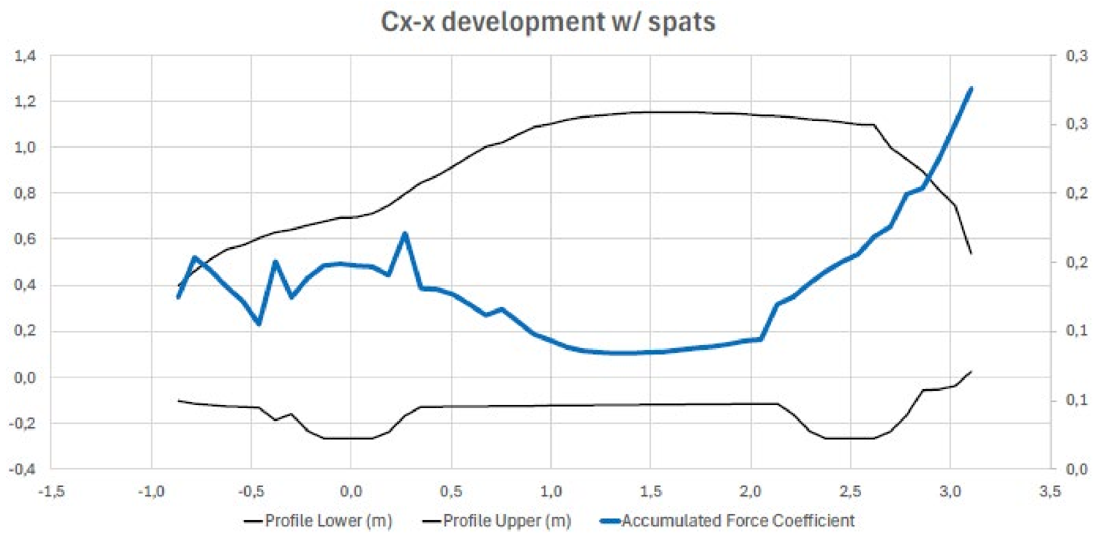

Figure 10 – Cd with wheel spats

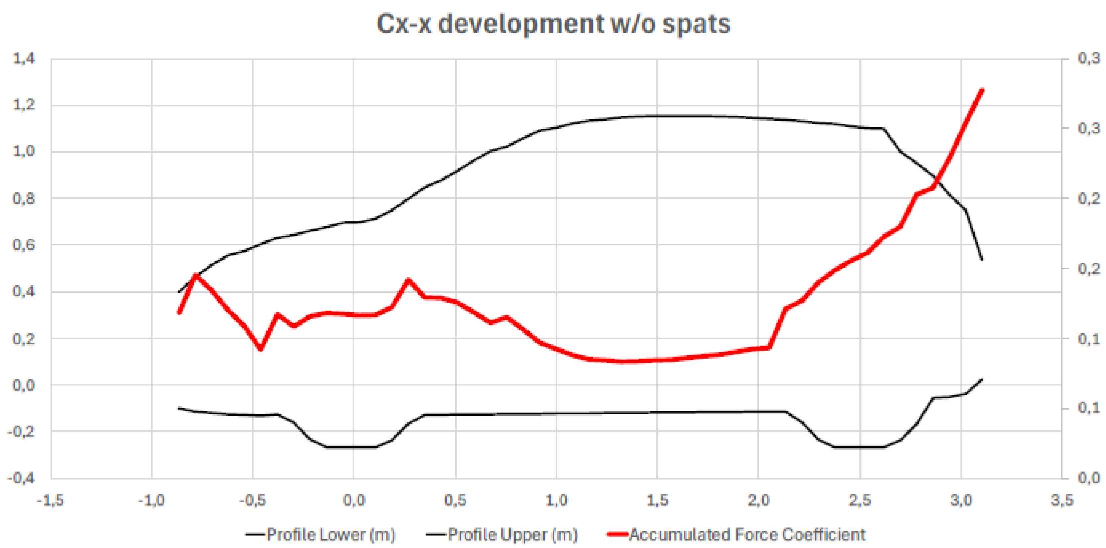

Figure 11 – Cd without wheel spats

The spat introduces an increase of drag, but the low-pressure zone around the wheels and the better attached air flow along the length of the vehicle bring to a recover and a loss in terms of drag.

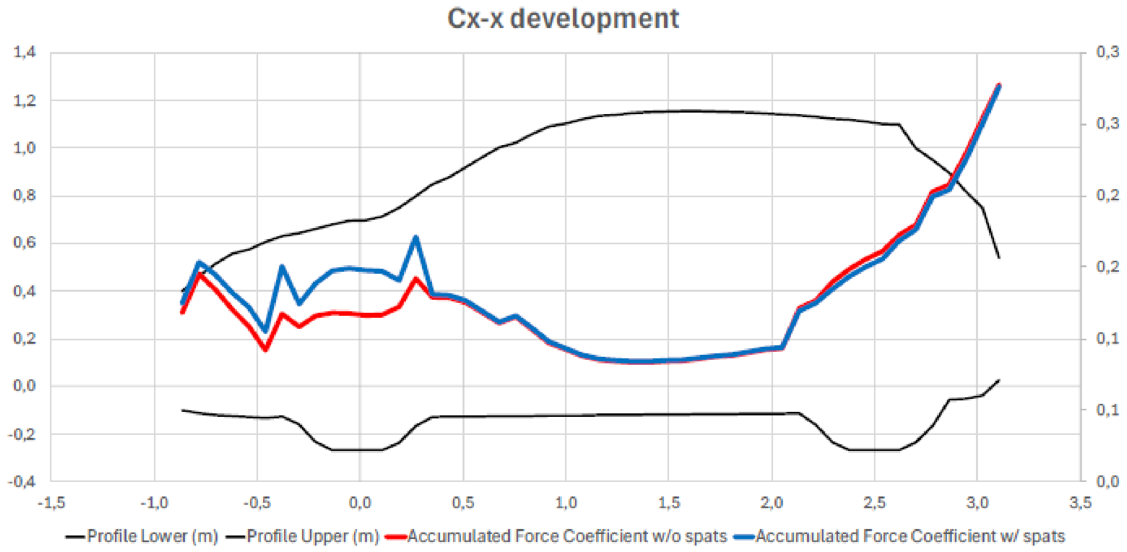

Figure 12 – Overlap of the 2 Cd graphs

Conclusion

The configuration with wheel spats can guarantee better performances than the original one in terms of drag and flow path. Its advantage can be seen only in a global analysis, because locally it caused an increase of Drag coefficient.