SCUOLA DI INGEGNERIA INDUSTRIALE
E DELL'INFORMAZIONE

Course of Aerodynamics of Transport Vehicles

CFD Analysis of Rooftop Cargo Boxes

Alessandro Aponte, 10600576
Giorgio Cesana, 10628987
Alessandro Cornale, 10581686

Professors:
Luigi Vigevano
Paolo Schito

1. Introduction

A well-known problem of long car trips is the cargo space, particularly for the storage of skies and long equipment. This is faced with the implementation of rooftop cargo boxes that provides additional space on top of a vehicle. On the other hand, it causes a decrease of the overall performance of the car, that is highlighted in an increase of drag and consequently in a reduction of fuel efficiency. In addition, a rooftop box increases the overall height and frontal area of a car. Consequently, it can be seen how aerodynamics plays a crucial role in determining the efficiency and stability of the vehicle [7].
The complexity and optimization of a rooftop cargo box is an interesting and often underrated problem, due to the side effects shown above. The purpose of this study is to find the optimized configuration of the rear part of a generic box in order to minimize the drag coefficient and consequently to save fuel.

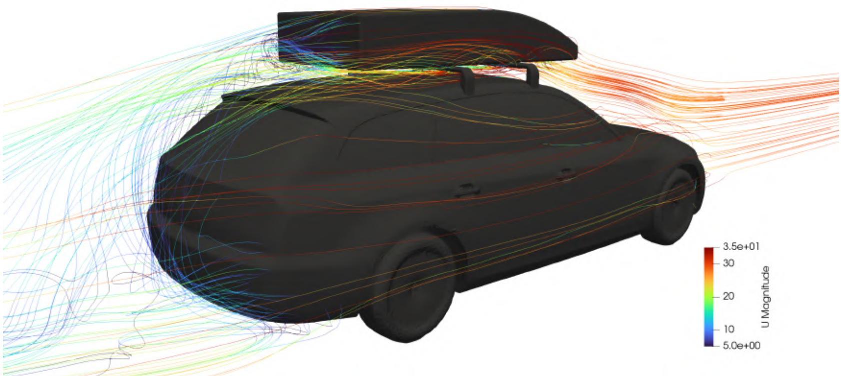


Figure 1: *Car with rooftop cargo box.*

2. Problem and target

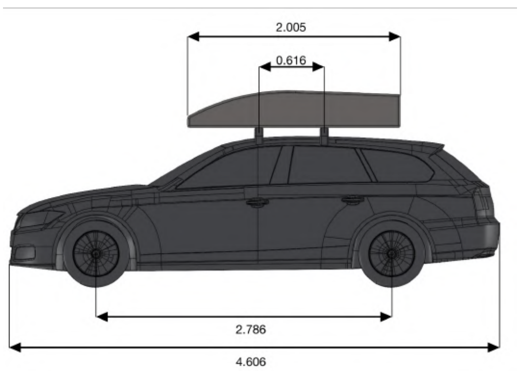
By understanding the principles of aerodynamics, decisions can be made about the design and functionality of a rooftop cargo box. In order to understand the phenomena presented, it is necessary to consider a car model that will be equipped with racks and a cargo box (Fig. 2). Due to the interaction of each element with the others it is fundamental to study the components together.

Figure 2: *DrivAer* model with racks and rooftop cargo box.

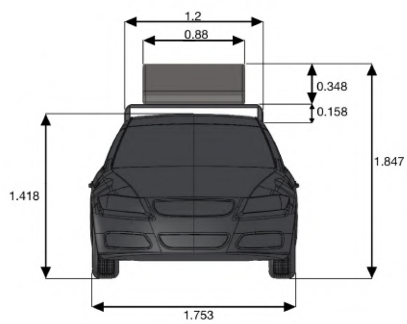
The goal of the study is to simulate configurations of different rear ends of the cargo box, inspired by solutions already developed in car aerodynamics.

3. Geometry

The components of the problem are three: car, racks and rooftop box (Fig. 3). The car comes from the DrivAer model, developed at the Institute of Aerodynamics and Fluid Mechanics at the Technical University of Munich [3] to facilitate aerodynamic investigations of passenger vehicles. It provides a variety of combinations to build the car. In this study, smooth wheels and underbody are considered, while mirrors, exhaust system and everything related to engine and cooling system is neglected. For the rear end, all three configurations available are considered in this study: estateback, notchback and fastback.



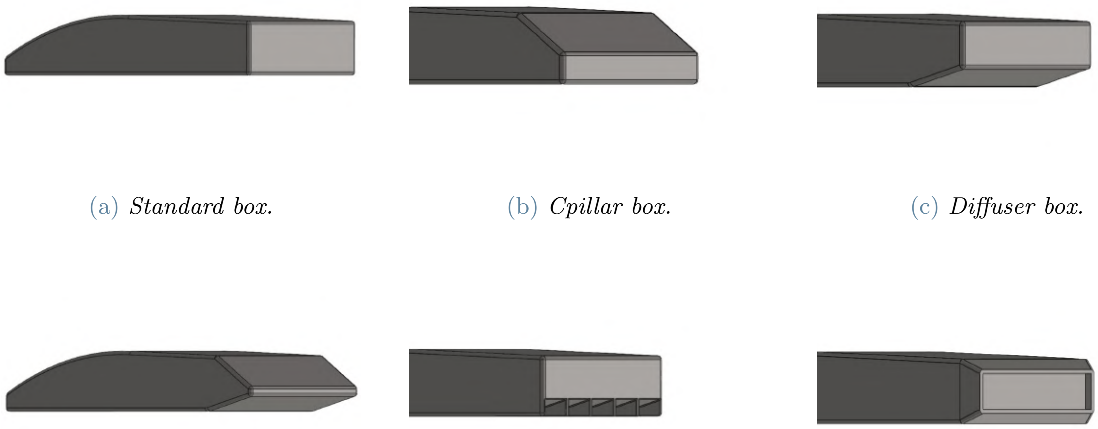
(a) *Lateral dimensions.*



(b) *Front dimensions.*

Figure 3: *Geometries of the model.*

For each rear end layout, racks are built using the software SolidWorks, such as the boxes. The geometry chosen for the racks is fixed, but due to the different shapes, the connection of the rack model to the rooftop is adapted to each car.



(a) *Standard box.* (b) *Cpillar box.* (c) *Diffuser box.*

(d) *Cpillar and diffuser box.* (e) *Diffuser with panels box.* (f) *Boat tail box.*

Figure 4: *Geometries of the rooftop boxes.*

The overall geometry of the box takes inspiration from real boxes. The rear end is built in different configurations that are modeled as typical rear ends of simplified road vehicles:

- a standard configuration with a vertical back (Fig. 4a);
- a configuration with an upper back slant such as the cpillar of cars (Fig. 4b);
- a configuration with a lower back slant such as the diffuser of cars (Fig. 4c);
- a configuration with a combination of the previous two (Fig. 4d);
- a configuration with a lower back slant and vertical plates such as diffusers of race cars (Fig. 4e);
- a configuration with boat tail such as in truck aerodynamics (Fig. 4f).

4. Software and computation

Simulations are performed using the software OpenFOAM [1], a free open source CFD software. There is a relevant number of challenges that are faced when a CFD simulation of a car is performed. Firstly, it is fundamental to accurately model the geometry of the car, in particular the components that interact with the fluid. Another challenge is to define the conditions to set up the simulation, including the velocity of the flow, the speed of the ground, the angular velocity of the wheels and other parameters (Table 1). Finally, it is essential to choose the right turbulence model to appreciate the behavior of turbulence properties.

4.1. Computational limitations

Computational limitations in a 3D CFD simulation are primarily related to the computing resources and time required to run the simulation. Moreover, a 3D car such as the DrivAer model is very demanding due to its complexity (high number of surfaces, curved and irregular shapes, moving wheels, etc.).
These limitations were relevant in this study, in fact the lack of computational power didn't permit to perform better simulations because of:

- Mesh generation: the problem of not being able to generate meshes with more than 3-4 millions of cells was a big issue for the convergence of the simulations;
- Domain size: using a large domain such as the one used in this study requires a lot of computational resources that were not available;
- Simulation time: while trying to perform more demanding simulations, the computational time was too high.

In conclusion, accurate CFD simulations of the models were computationally too demanding for the available resources, consequently we made the choice to build meshes with a smaller number of cells than the required one for a complete study. High-performance computing resources, such as supercomputers or clusters, may be required to run the simulation in a reasonable amount of time.

5. Mesh generation

In CFD, the mesh is a crucial component in the simulation of fluid flows. It splits the domain in smaller and manageable parts called cells or elements (Fig. 5). The mesh has a fundamental role in the accuracy, convergence and efficiency of simulations.

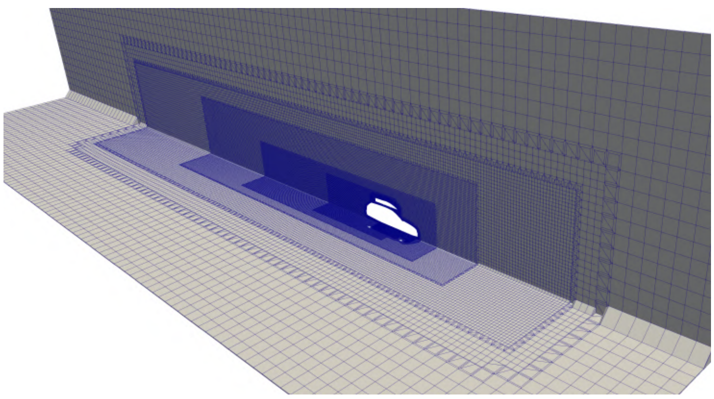
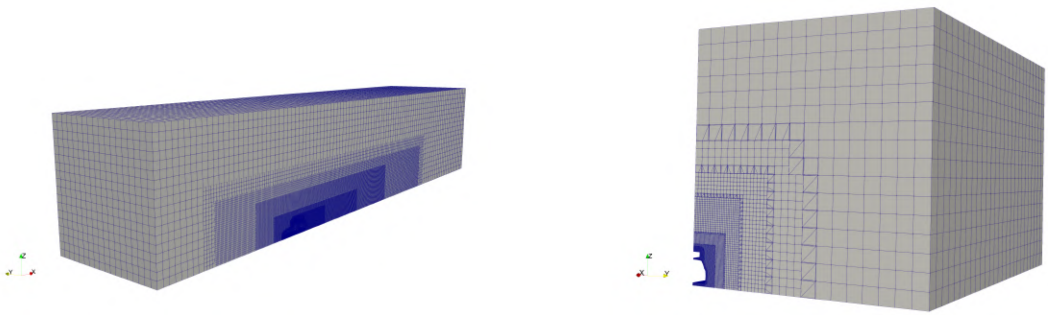


Figure 5: *Mesh generation.*

A 3D simulation usually needs a lot of computational power, so in order to save computational effort the symmetry condition is imposed which splits the domain in half (Fig. 6a) along the centerline of the car. This is possible only if the object

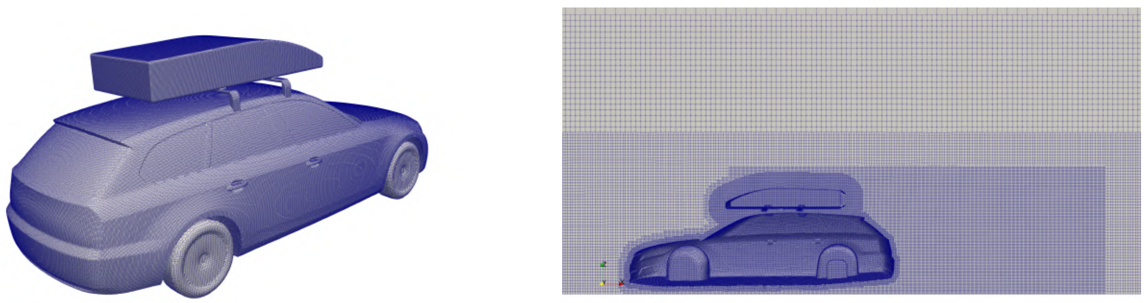
of the study is symmetric along the centerline.
The mesh is created with build-in tools featured in OpenFOAM called SnappyHexMesh and BlockMesh. The dimension of the cells is classified in levels. By increasing the level, the size of the cells splits in half in every direction.



(a) *Lateral view of the mesh.* (b) *Transversal section of the mesh.*

Figure 6: *Sections of the meshed domain.*

In order to better capture the flow features around the body and in the wake region, it is fundamental to decrease the dimension of the cells by giving an increased level of refinement (Fig. 7a). This is possible by defining some refinement regions in terms of boxes into the snappyHexMesh dictionary that decrease the dimension going close to the body (Fig. 6). It is necessary to use a certain number of refinement boxes because an abrupt change of dimensions can cause problems to the simulation, so the size must gradually decrease up to the model.



(a) *Mesh over the body.* (b) *Particular of the distance feature.*

Figure 7: *Sections of the meshed domain.*

To capture the features of the boundary layer attached to the model, the feature addLayer in the snappyHexMesh dictionary allows to generate a very detailed refinement of the region close to the body. Moreover, the command *distance* is used to obtain small enough elements near the model (Fig. 7b). In order to save computational power and to reduce the number of cells, the inner refinement boxes do not capture all the region around the car but only the rear part of the car and a portion of the wake.

6. Simulations performed

The equations solved are RANS equations, consequently it is relevant to choose an appropriate turbulence model in order to close the problem. For this study, the k-ω-SST model is chosen, consequently a value for the turbulent kinetic energy and the turbulent frequency ω is required [4]. Moreover, other boundary conditions are applied (Table 1) such as the rotation of the wheels to achieve the more realistic values of drag possible, in fact it can be seen that coefficients vary a lot between moving and fixed wheels [8].

Parameter	Value
Section of the car [m^2]	1.24
Section of the rooftop cargo box [m^2]	0.16
Total section [m^2]	1.40
Free stream velocity [m/s]	30
Density [Kg/m^3]	1.225
Pressure [Pa]	0
Turbulent kinetic energy [m$^2/s^2$]	0.24
Turbulent frequency [rad/s]	1.78
Angular velocity of wheels[rad/s]	94.64

Table 1: *Boundary conditions.*

6.1. Validation

The validation has the purpose to verify if the models and set ups are correct. It is necessary to refer the validation on existent and already valid data or literature data on the DriveAer model. There is also a publication based on the interaction between the car and rooftop cargo boxes but it is just qualitative, in the sense that there is only an information of the percentage change in drag and lift of some configuration that did not permit us to validate our study.
The validation is being done with the data presented by Gopal Shinde in his conference papaer presented at OSCIC 13 [5]. The configuration used is the estateback one of the DriveAer model with the same set ups and mesh that will be used in the next simulations. From the publication, we compare the value obtained with the expected one of the model with smooth underbody and with ground speed (Table 2).

	Expected value	Measured value	% Error
C$_D$	0.292	0.2898	0.769

Table 2: *Validation of the estateback car simulation.*

6.2. Grid convergence

It is essential to ensure that the solution is independent on the grid size. Therefore, the same problem is solved by using meshes more and more refined and the solutions are compared in order to estimate the error generated from the discretization of the domain.
Four different configurations are provided: very coarse, coarse, medium and fine, with a ratio of 1.25 of refinement between the meshes. The setup is fixed in order to mantain constant the proportion between the reference boxes, the only parameter modified is the dimension of the cells at the boundary of the domain. To obtain a ratio of 1.25, this value is increased each time by 10%. Being the level of the refinement boxes related to this parameter means that also the cells in the refinement regions see a reduction of their dimension.

Mesh refinement	C_D
Very coarse	0.2832
Coarse	0.3095
Medium	0.3270
Fine	0.3348

Table 3: *Grid convergence values.*

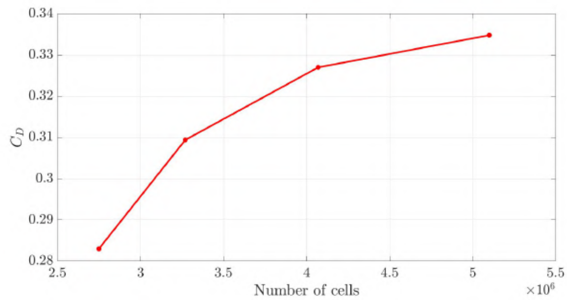


Figure 8: *Mesh Convergence*

An useful output that can be caught from the grid convergence is to choose the best grid that ensure a good balance between number of elements, computational power, accuracy and simulation time. Said that, for the further analysis the mesh refinement chosen is the medium one.

6.3. Results

Validated the problem and checked the convergence of the grid, it is possible to simulate all the configurations of car and rooftop box. Firstly, all the six patterns of the cargo box are studied with the estateback model and the drag coefficient of each combination is evaluated (Table. 4). It is clear that the cpillar cargo box is the most efficient in the estateback case, while a configuration like the diffuser results to be the worst.

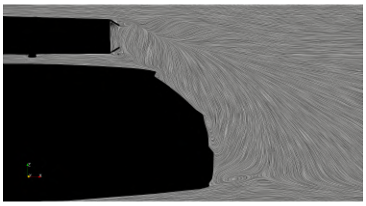
	Standard	Cpillar	Diffuser	Cpillar and diffuser	Diffuser with panels	Boat tail
C_D	0.3270	0.3268	0.3330	0.3294	0.3304	0.3277

Table 4: *Drag coefficients of box configurations with estateback car.*

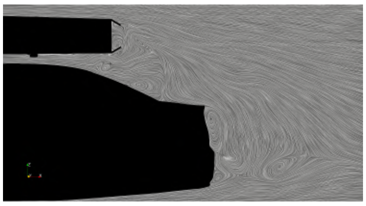
Secondly, the behavior of three boxes of interest are analyzed on the top of the other two configuration of the DrivAer model, fastback and notchback. The configurations chosen are cpillar and boat tail because of their efficiency in the estateback case, but also the diffuser configuration to check its behavior in these two layouts.

C_D	Estateback	Notchback	Fastback
Boat tail	0.3277	0.3255	0.3276
Cpillar	0.3268	0.3220	0.329
Diffuser	0.3330	0.3251	0.3300

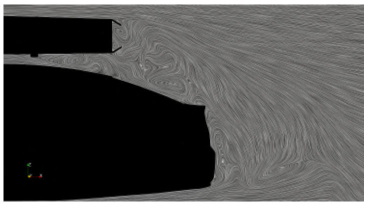
Table 5: *Drag coefficients of three box configurations of interest with all the cars.*



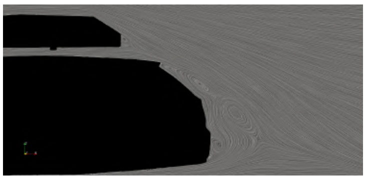
(a) *Estateback with boat tails box*



(b) *Notchback with boat tails box*

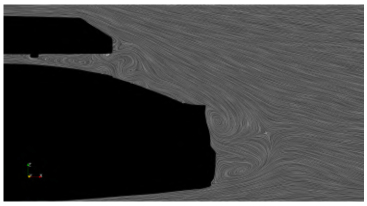


(c) *Fastback with boat tails box*

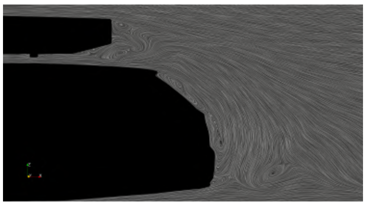


(d) *Estateback with cpillar box*

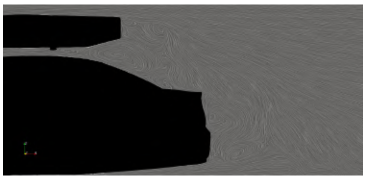
(e) *Notchback with cpillar box*



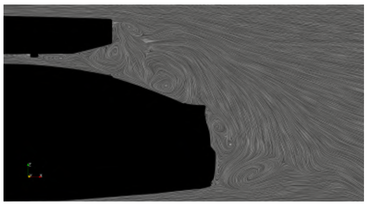
(f) *Fastback with cpillar box*



(g) *Estateback with diffuser box*



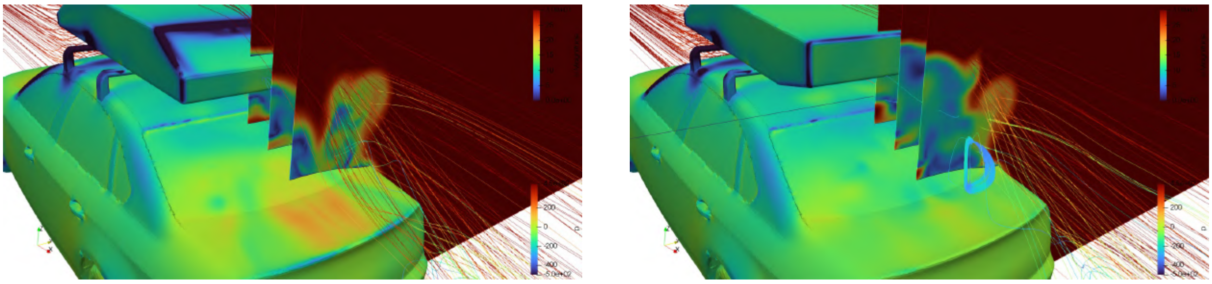
(h) *Notchback with diffuser box*



(i) *Fastback with diffuser box*

Figure 9: *Visualization of cars with cargo boxes.*

The effect of the cpillar configuration can be appreciated in Fig. 10a where it is clear how the wake is pushed down mostly in the center line of the car, helped also by the lateral vortex (Fig. 9d, 9e, 9f). By looking at the values obtained, this has a small effect on the drag reduction.
On the other hand, the effect of the diffuser is to push up the flow Fig. 10b and this is not an optimal configuration because it increases the size of the wake, causing an increase of the drag coefficient (Fig. 9g, 9h, 9i).
The same two effects can be recognized in the boat tail configuration (Fig. 10b), where the flow behavior is a combination of the two generated by the previous layouts of the cargo box (Fig. 9a, 9b, 9c).
It is clear that the different shapes of the rear end combined with the boxes generates different values of the drag coefficient. Firstly, it can be appreciated how the C_D of the notchback and fastback models with cargo boxes are significantly increased with respect to their literature coefficients of the model alone [5]. This



(a) *Slices of cpillar simulation.* (b) *Slices of diffuser simulation.*

Figure 10: *Push down and push up wakes.*

is motivated by the fact that the presence of the cargo box increase the dimension of the wake following the model, with the result of a wake region very similar for all three configurations.

7. Fuel analysis

Vehicles with rooftop boxes are aerodynamically inefficient compared to other ground vehicles due to their larger frontal areas and bluff-body shapes.
The performance of any aerodynamic body does not depend only on their designs but also on how or where the vehicle is operating. Because drag increases with the square of velocity, aerodynamic effects are generally less relevant on a vehicle operating in an urban environment compared to one running a long-haul distribution via highways. To assess vehicle fuel usage accurately, it is necessary to incorporate both vehicle parameters (e.g. dimension, drag coefficient, engine and tyre parameters, etc.) and its operational metrics (e.g. weight, speed, route planning, traffic density, road profile, etc.) [2].
Once obtained the simulation results of the aerodynamic drag of the vehicle, it is needed to establish a mathematical model of the aerodynamic drag and the fuel consumption rate of the vehicle. In this study, it is considered the situation of a vehicle traveling at a constant speed on a straight and well. According to the theory [9], the resulting horizontal force is shown in the following equation:

$$F_t = F_f + F_w = mgf + \frac{C_D A \rho v_r^2}{2} \tag{1}$$

where F_t is the horizontal force (N), F_f is the rolling resistance (N), F_w is the air resistance (N), m is the vehicle mass (kg), g is the acceleration of gravity $(\frac{m}{s^2})$, f is the rolling resistance coefficient, C_D is the aerodynamic drag coefficient, A is the windward area (m^2) , ρ is the air density $(\frac{kg}{m^3})$, and v_r is the driving speed (m/s).

The rolling resistance coefficient, f [6]:

$$f = 0.005 + (\frac{1}{p})(0.01 + 0.0095(\frac{v}{100})^2) \tag{2}$$

where p is the tire pressure (bar) and v is the velocity ($\frac{km}{h}$).
The impact of natural wind on the driving is ignored in the formulation.
The engine power is calculated with the formula:

$$P = \frac{v_a F_t}{\eta_t} \tag{3}$$

where η_t is the transmission system mechanical efficiency and v_a is the velocity $\frac{km}{h}$.
The fuel consumption per one hundred kilometers at constant speed of a car is calculated with the following formula:

$$Q_s = \frac{P_e b}{1.02 v_a \rho g} \tag{4}$$

where b is the fuel consumption ($\frac{g}{kWh}$) and ρ is the fuel density ($\frac{kg}{L}$).

Configuration	Fuel consumption [L/100Km]	% Fuel consumed
Estateback car	4.541	-
Standard	4.900	7.927
Cpillar	4.900	7.914
Diffusre	4.929	8.544
Cpillar and diffuser	4.912	8.178
Diffuser with panels	4.917	8.277
Boat tails	4.904	8.006

Table 6: *Fuel consumptions.*

8. Conclusions

From the results, it is possible to state that the variation of the drag coefficient due to the presence of these types of rooftop cargo box is relevant and consequently it increases the fuel consumption from 7.9% to 8.5% with respect to the estateback model alone. This effect is more relevant by looking at the values obtained for the notchback and fastback cases because of the bigger increase in C_D [5].
It can be seen that the cpillar and the standard configuration are better in terms of fuel consumption for the estateback model (Table 6), as well as for the notchback one (Table 5). On the other hand, the boat tails pattern is the best for the fastback case and so in terms of fuel consumed.
In conclusion, it is clear that for each rear end car model it is necessary to study different types of rooftop cargo box to understand which configuration is the one that maximize the reduction of the drag coefficient.

References

[1] Christopher J. Greenshields. *OpenFOAM User Guide*. 2022.

[2] Zulfaa Mohamed-Kassim and Antonio Filippone. *Fuel savings on a heavy vehicle via aerodynamic drag reduction*. 2010.

[3] Technical University of Munich. *DrivAer model*. 2011.

[4] Stephen B. Pope. *Turbulent flows*. 2000.

[5] Gopal Shinde, Aniruddha Joshi, and Kishor Nikam. *Numerical Investigations of the DrivAer Car Model using Opensource CFD Solver OpenFOAM*. 2013.

[6] Engineering ToolBox. *Rolling Resistance*. 2008.

[7] Luigi Vigevano and Paolo Schito. *Lectures of Aerodynamics of Transport Vehicles*. 2022.

[8] Shibo Wang, Terence Avadiar, Mark C. Thompson, and David Burton. *Effect of moving ground on the aerodynamics of a generic automotive model: The DrivAer-Estate*. 2019.

[9] Zhi-Fa Yang, Shu-Hong Li, Ai-Min Liu, Zhuo Yu, Huan-Jing Zeng, and Shi-Wu Li. *Simulation study on energy saving of passengercar platoons based on DrivAer model*. 2019.