COBEM 2021 26th International Congress of Mechanical Engineering
ABCM Associação Brasileira de Engenharia e Ciências Mecânicas
November 22-26, 2021, Virtual Congress, Brazil

COB-2021-1689
CFD-BASED MULTI-OBJECTIVE AERODYNAMIC OPTIMIZATION OF A CAR REAR DIFFUSER

Denis Ramon dos Santos Peixoto
Giaffone Racing. Dr. Tomás Sepe St., 677. Cotia/SP – Brazil. 06711-270
Department of Mechanical Engineering, Escola Politécnica, University of São Paulo. Av. Prof. Mello Moraes, 2231, São Paulo/SP – Brazil. 05508-030
denisrdsp@usp.br

Bruno Souza Carmo
Department of Mechanical Engineering, Escola Politécnica, University of São Paulo. Av. Prof. Mello Moraes, 2231, São Paulo/SP – Brazil. 05508-030
bruno.carmo@usp.br

Abstract. *The lift coefficient plays an essential role in high-performance cars like race cars to improve the overall downforce and consequently maximize the car grip. Higher grip provides more safety during car handling and increase the lateral acceleration of the vehicle during the corners, which is a factor that contributes decisively to reduce the overall lap time. By reducing the drag coefficient, it is possible to achieve higher top speed and improve the car's fuel efficiency. In this work, computational fluid dynamics (CFD) is employed as a tool for the aerodynamic optimization of the geometry of a car rear diffuser, aiming to minimize drag and lift coefficients. Due to the ground effect, the rear diffuser can generate a high downforce level without a substantial drag penalty. The three-dimensional CFD simulations were performed for half-model, taking advantage of the longitudinal symmetry. We solved the flow considering steady state and employed the k-omega shear stress turbulence (SST) model. Ground effect was considered, and some simplifications were made to the geometry to make the numerical simulations possible, the most important of them was to neglect the wheels. A mesh convergence analysis was carried out, employing the grid convergence index methodology. Lift and drag coefficients were optimized with respect to the length and maximum height of the diffuser, and the meta-heuristic FAST method was employed as the optimization tool. A pareto front was obtained, and one of the designs was chosen to have its flow field analysed and compared to the data obtained from simulations of the flow around the car without the diffuser. That optimal design chosen exhibited an increase of 164% in the absolute value of the lift coefficient (downforce), with a penalty of 16% in the drag coefficient, when compared to the car without the diffuser. The behaviour of flow quantities in the wake and distribution along the car surface could be correlated to the change of the force coefficients, leading to some physical insights in the complex flow that happens around the car.*

Keywords: *Vehicle aerodynamics, Computational fluid dynamics, Drag, Lift, Multi-Objective Optimization*

1. INTRODUCTION

The study of the flow around a car is primarily concerned with the forces that the flow exerts on the body, that is, the lift, drag and side forces. The purpose of studying this flow during the automotive design process is to reduce the drag force and lift force, which are important for handling characteristics. Other factors affecting the external aerodynamics must also be considered, such as the requirements for cooling of the engine, transmission and brakes (Gillieron, 1999); the flow within the passenger compartment area; and the prevention of dirt build-up on the windows of the automobile (Raghu, 1999). These considerations, however, are more important to passenger cars. An additional branch of external automotive aerodynamics covers racing vehicles. Here, the main concern is to provide a large amount of downforce (negative lift) while not generating excessive drag.

The flow features around both passenger and racing cars are very complex. There are large regions of transition, separated flow, vortex and wake formation. Allowing considerations of such complex flow characteristics to be integrated into the design of a new vehicle requires the understanding of, and the ability to predict, the flow field early in the development program. This can be done using a variety of methods: wind tunnel or water tank tests using scale models, or Computational Fluid Dynamics (CFD). Scale models were expensive to build, in both time and cost, but recently, with the evolution of the 3d printing technology, the cost and prototyping time have become much smaller. The initial costs involved in building a wind tunnel are considerable and once built, they still require upkeep and maintenance costs. By contrast, CFD requires a smaller initial financial outlay and, with the increasingly widespread use of CAD models, involves less work, time, and cost to produce a flow analysis for each vehicle. However, CFD is not simple to use and there are several problems like high computational cost, CAD model accuracy, turbulent models and boundary conditions, that can prevent engineers from using it as a reliable tool to develop new automobiles.

Engineers in the field of aerodynamics have been using CFD for a long time. Traditionally, CFD is used to optimize car shapes in terms of lift and drag, and study salient flow features. External flow analysis over car geometry is always an interesting part of research in aerodynamic field. The study is usually performed employing experiments in wind tunnels as well as computational simulations. Due to the cost of experimental studies, and the evolution of computational power in the last years, CFD has been gradually becoming a viable tool in vehicle design and in aerodynamic fields, particularly when the results are validated against wind tunnel tests. The aerodynamics forces on road vehicles are the result of complex interactions between flow separation and the dynamic behaviour of the released vortex wake. Drag is caused due to the pressure difference between the frontal and the rear end of the vehicle. It can be reduced by modifying the design of the vehicle, leading to a change of the air flow around it. We also look into design and aerodynamics of a vehicle in order to maintain better control for steering and braking. It is very often necessary to generate downforce to improve traction and thus cornering abilities. Lift can be dangerous for an automobile, especially at high speeds. So, in order to maintain control for steering and braking, cars are designed so that the automobile exerts a downward force as its speed increases. However, increasing this downward force increases drag, which in turn, limits the top speed and increases fuel consumption. Hence, these two forces must be carefully balanced.

The main role of the aerodynamics applied to race cars is to generate a substantial level of downforce causing the minimum drag. In addition, the aerodynamic balance (relationship between front and rear axle downforce) is very important considering all speed conditions. The complex flow features associated to individual components are intertwined and difficult to separate. Therefore, a clear understanding of the physics of the flow associated to the aerodynamic components is a prerequisite to obtain a global understanding of the flow, and eventually a better vehicle geometry. Race cars have many components that provide better aerodynamic performance for the vehicle, like inverted wings, diffuser, and vortex generators, each one with a specific feature. The components that are more efficient are those which generate ground effect and provide less drag. While drag reduction is a very important research field, the downforce generation also has an important role in lap time reduction, security and race car balance.

The main objective of the present work is to propose a geometry optimization of a car model in order to improve the aerodynamic efficiency, that is minimize drag and lift. More specifically, we will focus on the design of a rear diffuser. The study of rear diffusers considering ground effect has been the theme of other computational studies found in the literature. Computational simulation of diffuser flow in ground effect was conducted as part of the research of (Cooper et al. 1998). The 3D model with 9.17° and 13.5° diffusers was simulated as a symmetric half-model and without the side plates. RANS simulation was performed and the $k - \omega$ turbulence model was used. Fine near-wall grid spacing allowed resolution to the diverging wall. Adequate lift and pressure predictions were obtained for the 9.17° diffuser; however, the simulation was less successful for the 13.5° diffuser. The results of these and similar computations for different diffuser lengths were conducted for use in their analytical model (Cooper at al. 2000). Details of the solutions were not presented; however, the results were utilized in providing certain input data for the model. The model calculated the total underbody mean-effective pressure coefficient from a correlation based upon the CFD data for different diffuser lengths and on the experimental data. Predictions of the underbody mean-effective pressure coefficient calculated for diffusers of various lengths in proportion to model length were given for several area ratio parameters. The authors provided a useful insight into the design of underbody diffusers, concluding an optimum area ratio parameter of approximately (AR=)1–2 and a diffuser of approximately half the length of the vehicle itself**.** Jowsey (2013) presented an interesting experimental (wind tunnel tests) study focusing on the performance of plane and multi-channel diffusers using force, pressure and PIV measurements. Later, Ehirim et al. (2019) presented a review of ground-effect diffuser aerodynamics, aiming at creating a systematic understanding of the physics that influence the performance of this device.

In this work, we will estimate the lift and drag through three-dimensional steady state numerical simulations for a symmetrical half model, with ground effect considered in the boundary conditions. The URANS (Unsteady Reynolds Averaged Navier-Stokes) approach was chosen, employing the $\mathrm{k} - \omega$ SST model. The optimization will be performed using the FAST algorithm. Details of the methodology are given in section 2, the numerical setup is presented in section 3, the results are shown and discussed in section 4, and in section 5 we draw final conclusions about this work.

2. METHODOLOGY

A very widely used approach for turbulent flow simulation in industry for engineers is a set of time-averaged equations, which is called Reynolds-Averaged Navier-Stokes (RANS) equations. The time-averaged solution for turbulent flow was introduced by Reynolds. This concept is based on replacing all the fluctuant variables such as velocity and pressure with time-averaged part and a fluctuating part. The results are widely reported in the literature (see, for instance, Ferziger, 1997), which are the equations with the addition of a term which corresponds to the Reynolds stresses. These are then modelled in the RANS approach, through a term that includes a turbulent viscosity, which is function of space, and is calculated through the solution of transport equations related to turbulent kinetic energy and dissipation rate. In this work, we employ the *k-ω* SST turbulence model. Regarding the numerical solution of the equations, we employed the finite volume method (Ferziger, 1997), implemented in the Ansys Fluent 2020R1 commercial software.

A very important step in this thesis is the Multi-Objective Optimization of the car geometry. This analysis should be robust, accurate and unexpensive enough to make it possible to achieve a satisfactory result in a viable time, considering the CFD simulation time spent to simulate a car. In this work we employed the FAST NSGA-II algorithm. FAST is an optimization algorithm combining real and Response Surface Method (RSM) based (virtual) optimization strategies. Both real and virtual optimization are performed by one of the evolutionary or heuristic algorithms for solving single and multi-objective problems (Rigone, 2014). This algorithm achieved a satisfactory result quicker and needed less simulations to form a Pareto frontier in preliminary tests and also in other studies reported in the literature (Nogueira & Carmo, 2018).

In order to accomplish the objectives proposed for this thesis, appropriate computational tools had to be chosen, taking into account the computational resources available and the synergy of the different disciplines involved. In this chapter, the methodology to use these tools in an accurate way will be presented. To achieve the goals, it was necessary to perform CFD and Optimization analysis, as well as the CAD geometry generation and parametrization. The software suites used were SolidWorks 2019 for CAD design and geometry treatment of the car model; ANSYS DesignModeler to prepare the CAD model for CFD simulation; ANSYS Meshing as integrated tool in the workbench was chosen to create an accurate mesh; for the CFD simulation ANSYS Fluent was used; finally modeFRONTIER was chosen to work integrated with ANSYS Fluent to automate the CFD analyses and perform the optimization studies.

Figure 1 shows the interaction between the pieces of software. It consists of an optimization loop where the modeFRONTIER controls the ANSYS Workbench, changing the geometric characteristics of the car (rear diffuser dimensions) to achieve the best performance (Minimize Lift and Drag).

Figure 1 Loop of optimization diagram (Software's interaction)

3. NUMERICAL SETUP

To perform a CFD simulation with a car we used a "smooth" car geometry, the 3d cad model was obtained from a site that provides 3d model for download (https://grabcad.com/). Due to the need to perform a quicker and accurate simulation during the optimization, some simplifications were made, like removing wheels and mirrors. These components would certainly be important for the aerodynamics of the car, and the wheels in particular would affect the flow under the car. However, we would have to consider the rotation of the wheels in the simulation and the contact with the ground and modelling these aspects can be very challenging. Although this is an important and interesting area of research, we decided to leave it out of the scope of this work to focus our effort on the development and test of the optimization methodology on a simpler, yet representative, numerical model. This is a full-scale model with the following geometric features: 4710 mm long, 909 mm wide and 1247 mm high. A single body domain of air was created surrounding the car walls. After subtracting it from the air enclosure, the body was suspended 60 mm from the ground. In rear diffuser studies the distance to the ground plays a very important role, so many studies were made to test different distances to the ground, as the focus of this work was to optimize the car geometry, we just used 60 mm for the ground clearance (Figure 2). The domain dimensions were $8L$x$3L$x$2.5L$ in the streamwise (y), spanwise (x) and stream-normal (z) directions, respectively, with L being the car length. This domain was chosen to make time simulation time not so long, without losing accuracy. The coordinate system adopted was positive *"x"* in the longitudinal direction of the car, positive *"y"* in vertical direction (pointing to the top of the body) and positive *"z"* in the lateral direction (pointing to opposite side of the *xy* symmetrical plane). The center of the coordinate system was placed at the middle of the car model ($x = 0$ middle of the model, $y = 0$ symmetry plane, $z = 0$ ground plane). As the car and flow are symmetrical, we considered only a half model and created a symmetry plane (*xy* plane) that cuts through the entire domain. Figure 3 shows the entire domain.

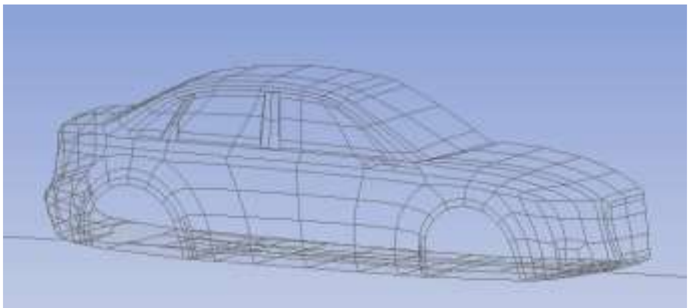

Figure 2. Full scale car model.

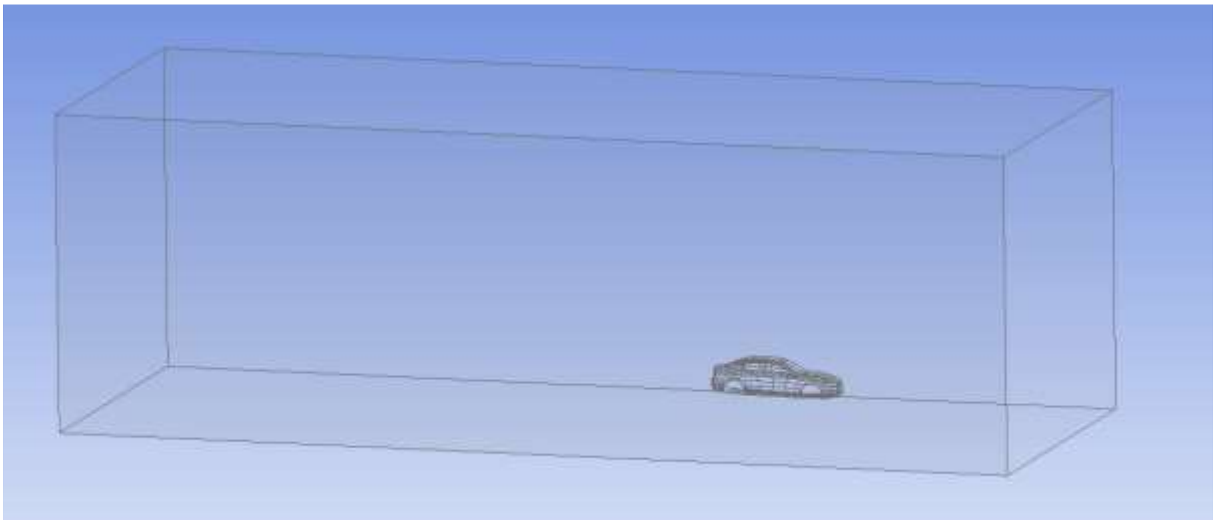

Figure 3. Numerical simulation domain.

For the domain discretization and grid refinement, we used internal boxes created around the vehicle and in the wake region to explicitly control mesh size. The grid presents hexahedral elements close to the car walls to capture the high normal gradients at the boundary layers. Figure 4 shows the mesh employed in the simulations.

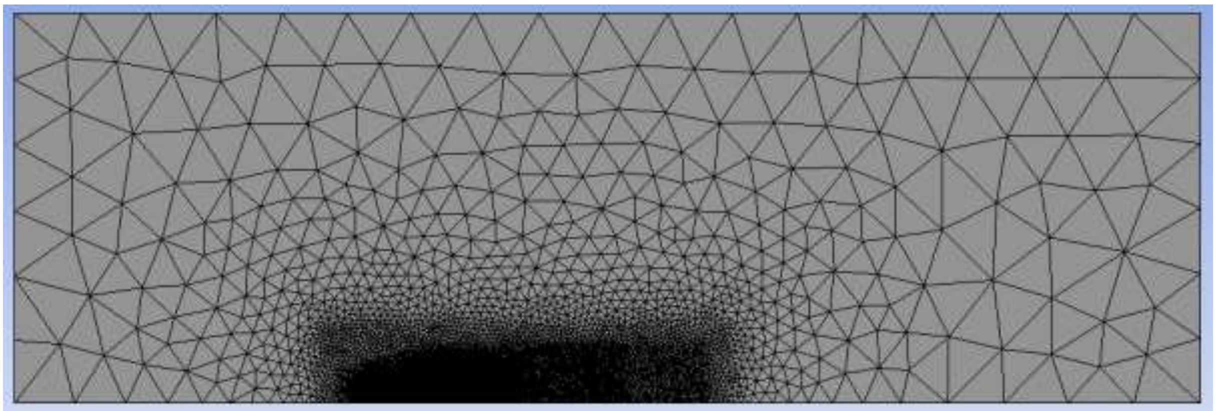

Figure 4. General view of the mesh used for the simulations. The higher refinement around the car and in the wake is noticeable.

With respect to the flow solver solution methods, the pressure-velocity coupling scheme selected was the Coupled algorithm. The spatial discretization schemes employed are summarized in Table 1. Numerical simulations were performed to check the influence of different spatial discretization's schemes on the numerical solution, and changes were insignificant. The final selection was based on specific recommendations from literature (Lanfrit, 2005). The boundary conditions were selected in order to emulate a wind tunnel experiment, and they are listed in Table 2.

Table 1. Spatial discretization schemes selected for the numerical analyses.

Gradient	Least Squares Cell Based
Pressure	Second Order
Momentum	Second Order Upwind
Turbulent Kinetic Energy	Second Order Upwind
Specific Dissipation Rate	Second Order Upwind
Transient Formulation	Second Order Implicit

Table 2. Boundary conditions settings for the numerical analyses

Wind tunnel inlet	Velocity inlet (40 m/s)
Wind tunnel outlet	Pressure outlet
Wind tunnel top and side	Symmetry
Wind tunnel bottom	Velocity inlet (40 m/s)
Car Body	Wall (no-slip condition)
Turbulent intensity at the inlet (%)	5%
Turbulent viscosity ratio at the inlet	10

It was necessary to create a parametrized geometry to establish the optimization loop. The parametrized geometry was created in ANSYS DesignModeler. The parametrized dimensions are show in Figure 5, these dimensions are the height and length of the rear diffuser. For this case one vertical fin was added to avoid vortex generation, and three-dimensional turbulent flow across the diffuser, keeping the streamlines in rear car direction increasing the rear diffuser efficiency (Figure 6).

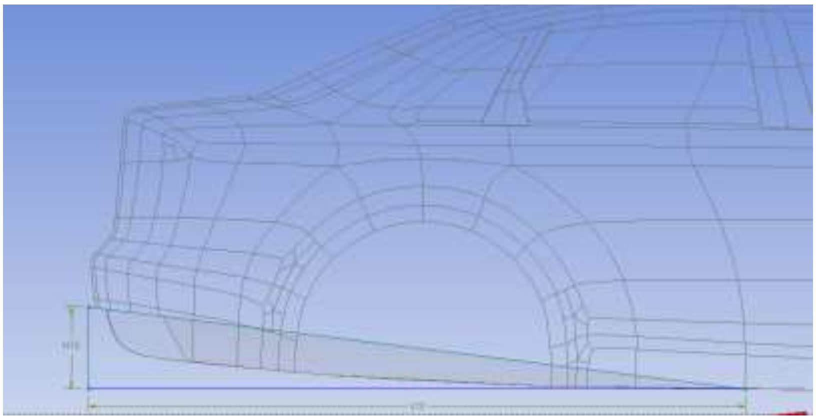

Figure 5. Car rear diffuser parametrized dimensions.

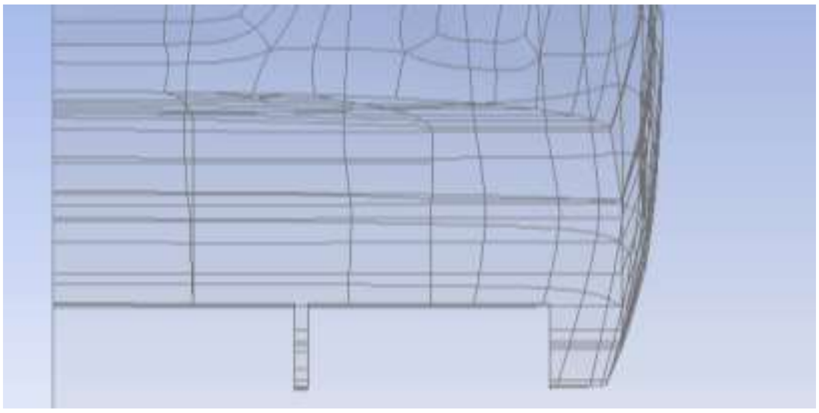

Figure 6. Vertical fin added to the rear diffuser.

3.1 Grid convergence

We conducted preliminary studies to make sure that we have an appropriate mesh, considering accuracy and computational time. To ensure that, we performed a Grid Convergence Index (GCI) study (Roache, 1998) considering three successively finer grids. Table 3 shows the results of the three simulations, for C_D and C_L coefficients associated to each grid size. The constant refinement ratio used was $r = 2$; that is, the next finer mesh should have half of the grid space of the previous one. The third column show the grid size normalized to be used for the GCI study.

Table 3 Car geometry GCI study results.

Grid	# Elements	Grid Spacing (normalized)	C_D results	C_L results
A	$8x10^6$	1	0.123	-0.367
B	$4x10^6$	2	0.124	-0.370
C	$2x10^6$	4	0.128	-0.418

The results from Table 3 were used to make to different studies, one considering C_D and other considering C_L results. The two results were compared to understand how accurate the simulations were for both parameters. Using a safety factor of 1.25 to account for the fact that only three meshes were used to estimate the order of convergence, we concluded that the converged C_D was 0.122 with an error band of 0.272%, and the converged C_L was -0.366 with an error band of 0.105%.

From the results obtained in the GCI study we considered Grid 2 the best option. This option gives an acceptable error and reasonable computational time.

4. RESULTS

The optimization loop was implemented using modeFRONTIER and Ansys Workbench. The aim of this simulation was test different combinations of height and length of the rear diffuser in order to get the minimum values of C_D and C_L. The interval of height of the diffuser was 0.2 – 0.4 m and the length of the diffuser could vary from 0.85 to 2.2 m. These ranges were chosen based on the original geometry and on a previous study conducted using the Ahmed body, and also taking into account geometry restrictions that a real car have, that can cause some interference in systems like powertrain and suspension, or the need to redesign these systems depending on the priority of aerodynamic under the other systems. We started the simulation with 5 DOE (Design of experiments). We ran the loop for 10 generations, and each generation had 5 individuals. Therefore, we performed 50 simulations in the optimisation study. Figure 7 shows the optimization loop results.

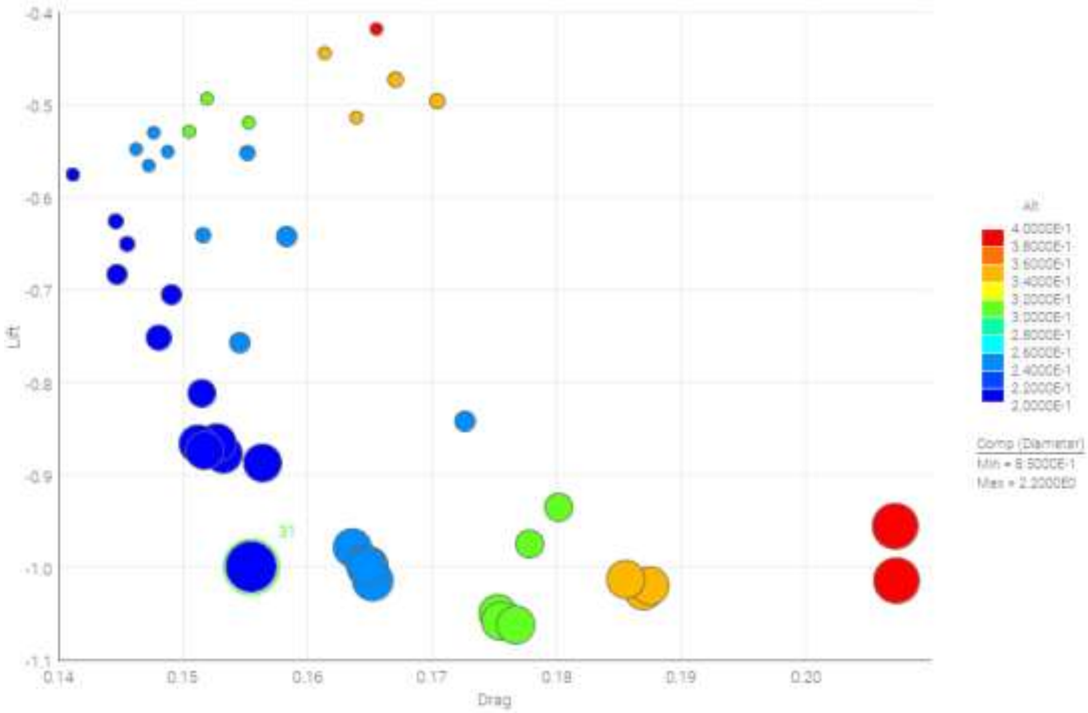

Figure 7. Optimization results obtained with the FAST algorithm applied to rear diffuser dimensions applied to Car Geometry. This is a 4D bubble chart, vertical axis represents lift coefficient, horizontal axis represents drag coefficient, the bubble diameter refers to diffuser length and bubble color refers to diffuser height.

During the simulations we can notice that the FAST algorithm converged quick for this case, since after few simulations it was already possible to see the Pareto frontier formation. At the end of 10 generations and 50 simulations we obtained a well-defined Pareto frontier. We can see that both the Low drag and High downforce cases exhibit a limit of efficiency; if the rear diffuser height increased more than 370 mm the drag increased and downforce decreased. That could happen because the angle the flow exits is so hight that the flow detaches from the surface and creates a separation zone. Point 31 is the more efficient, we can see the details in Table 4. Compared with the original geometry results, the optimized case with rear diffuser proved satisfactory. Table 5 shows a percentual difference between C_D and C_L for both cases.

Table 4 Data for rear diffuser optimized geometry chosen.

Geometry Parameters (input)		Aerodynamic Parameters (output)		
Height	Lenght	C_D	C_L	Aerodynamic efficiency
0.20 m	2.2 m	0.155	-0.998	6.44

Table 5. Comparison between drag and lift coefficients for optimized and original geometry.

	Original Geometry	Optimized Geometry	% difference
C_D	0.146	0.155	6%
C_L	-0.378	-0.998	164%

We could not minimize both parameters simultaneously, because they are inversely proportional quantities. However, considering a race car, the aim is to increase the performance and decrease the lap time, and to do that the more effective mechanism is create downforce. The optimized geometry is shown in Figure 8.

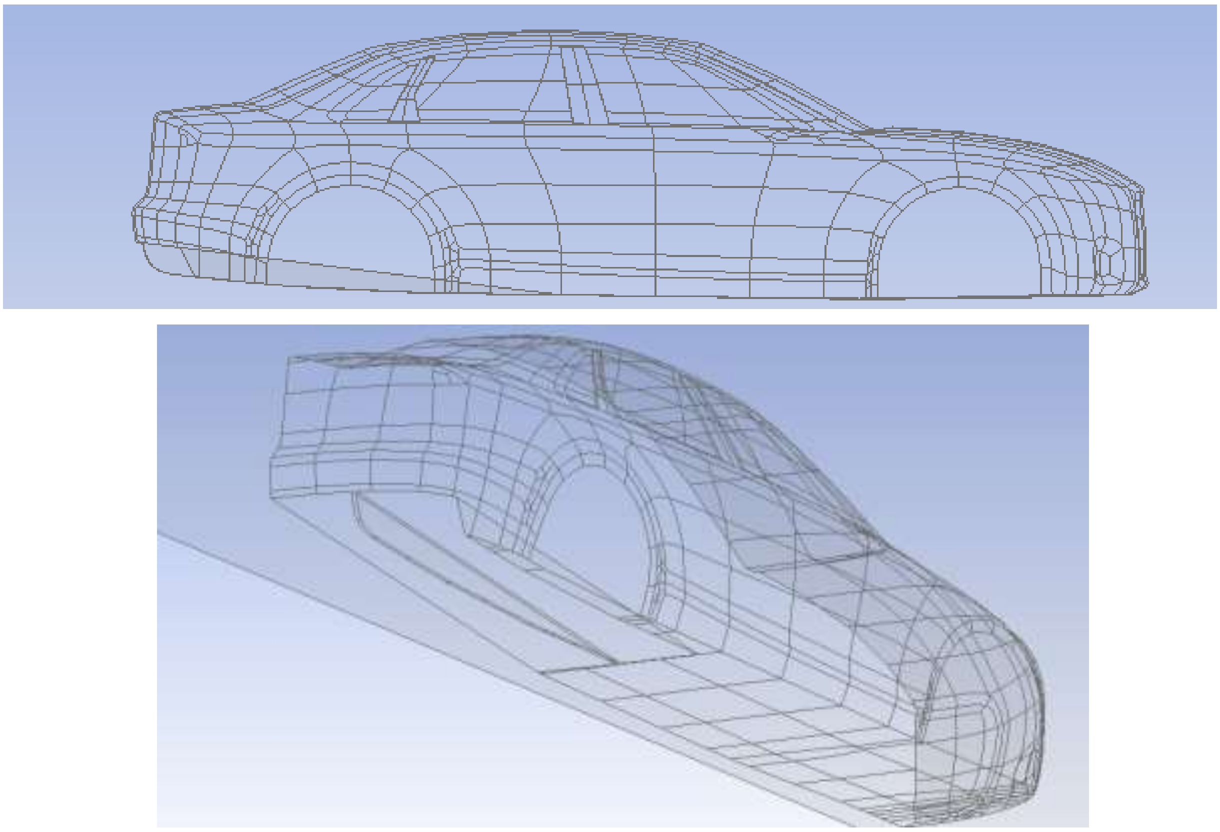

Figure 8. Car optimized geometry (rear diffuser).

Looking at the contours of velocity magnitude at the symmetry plane for the original and optimized geometry (Figure 9 and Figure 10), it is possible to see that the flow underneath the car accelerates more in the optimized than the original case, this is due the suction created by the rear diffuser. This suction is responsible for the creation of downforce. There is an important difference in the wake structure between the two cases: in the optimized geometry looks that the flow separated too early compared to the original case. This can be responsible for the increase in drag and reduces the efficiency of rear diffuser.

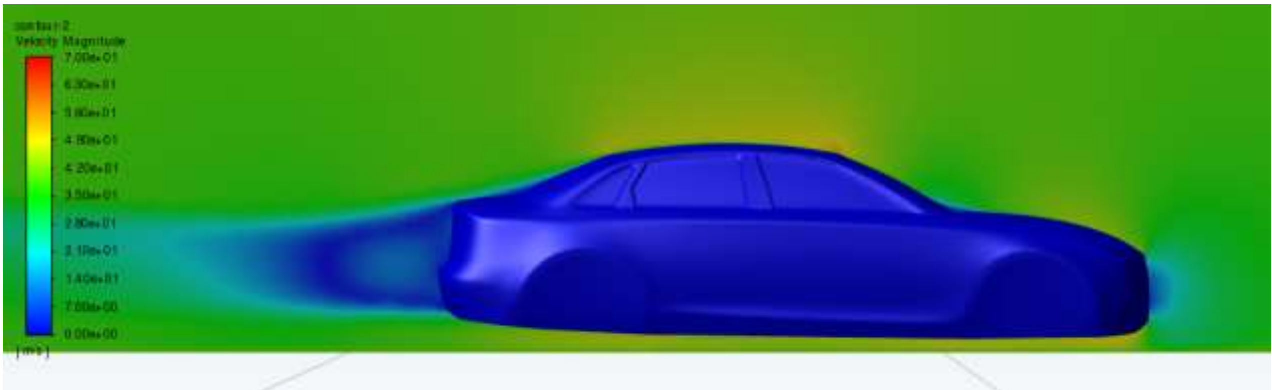

Figure 9. Contours of velocity at the symmetry plane for the car original geometry.

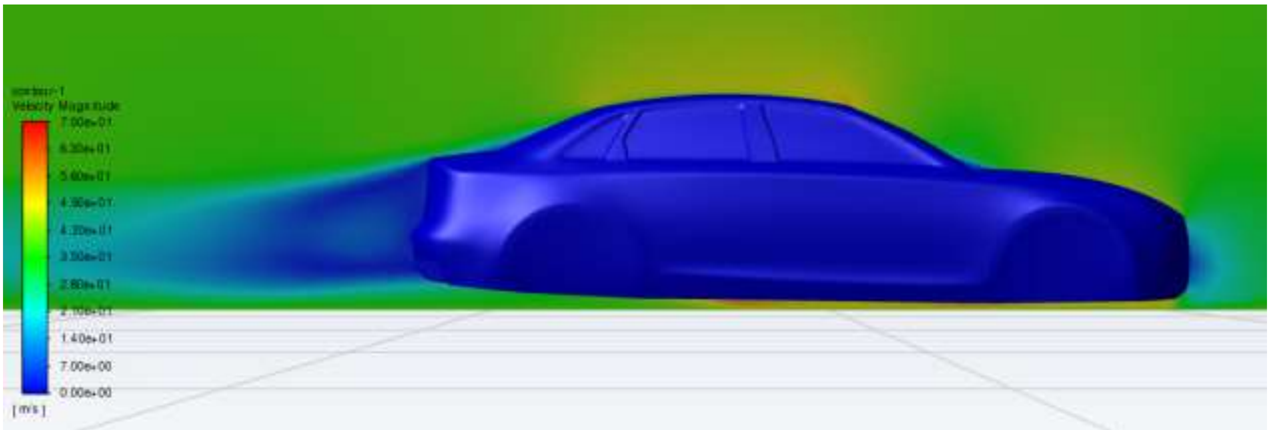

Figure 10. Contours of velocity at the symmetry plane for the car optimized geometry.

We now analyse the vertical and force distribution along the upper and bottom surfaces of the car, for the optimized and original geometries. Figure 11 shows that the distribution of vertical force along the upper surface is virtually the same for both cases (it starts close to zero at the front bumper, increases across the hood according with the hood curvature, and decreases until negative values when this curvature changes). At the roof the vertical force was predominant positive, at the rear windshield the vertical became negative. The negative values of vertical force happen in the separation zones. However, the vertical force in the upper surface is predominantly positive and that contributes to increase downforce. However, for the bottom surface, the differences are remarkable, as can be seen in Figure 12. The implementation of rear diffuser creates a suction in car underneath doing the vertical force distribution assumes high negative values compared with original geometry. The peak of negative force happens in the rear diffuser inlet this increase the downforce improving the car performance.

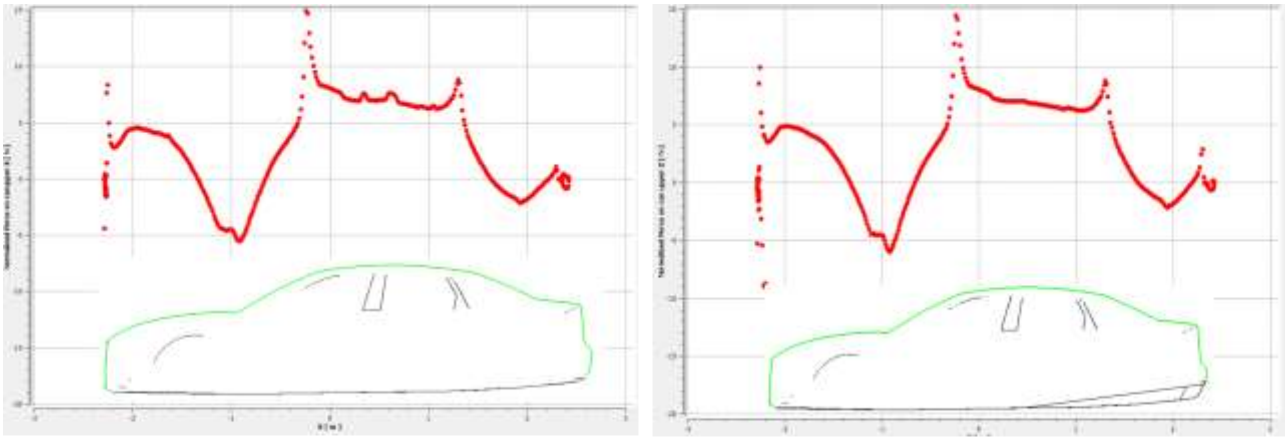

Figure 11. Vertical force distribution at the upper surface. Left: original geometry; Right: optimized geometry.

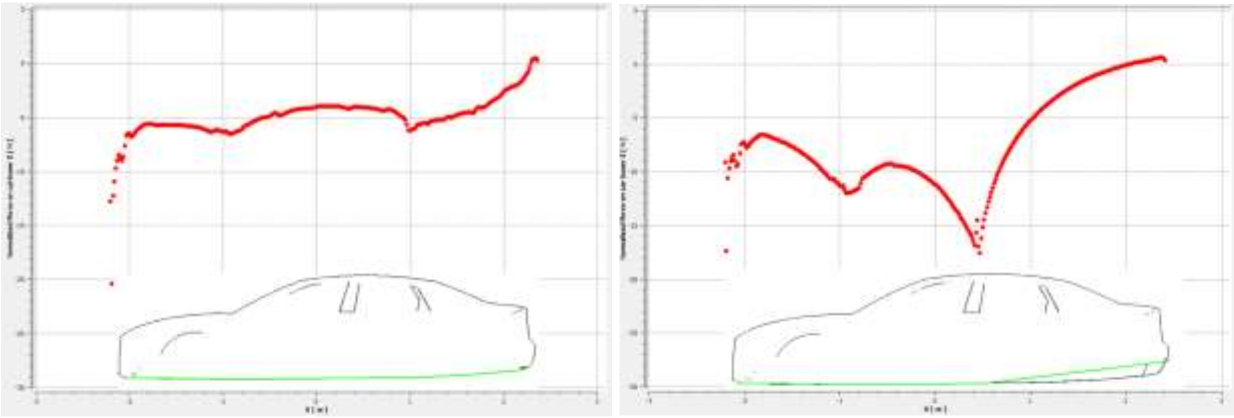

Figure 12. Vertical force distribution at the bottom surface. Left: original geometry; Right: optimized geometry.

Through the horizontal force distribution, it is possible to identify the regions that are more relevant to increase the drag. Looking at Figure 13, we see that there are no relevant differences between two geometries with respect to the force distribution at the upper surface. The horizontal force reaches its highest value at the front bumper, that is due the stagnation point. The next region with high horizontal force values is the junction of the hood with the wind shield, which is a region with high pressure. The junction between roof and rear windshield presents a slightly higher value of horizontal force due to the high wall shear stress at this point. Figure 14 shows that, at the bottom surface, the original and optimized geometries exhibit the same horizontal force distribution until the entrance of diffuser, the value is very close to zero. The force increases at the rear diffuser entrance reaching its maximum value and decreases slightly until zero. This behaviour contributes to increasing the drag.

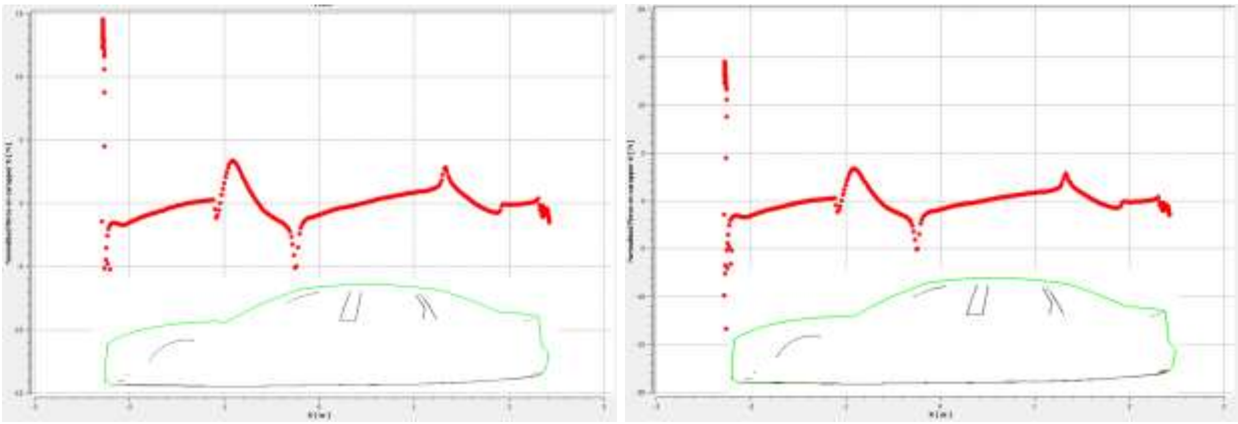

Figure 13. Horizontal force distribution at the upper surface. Left: original geometry; Right: optimized geometry.

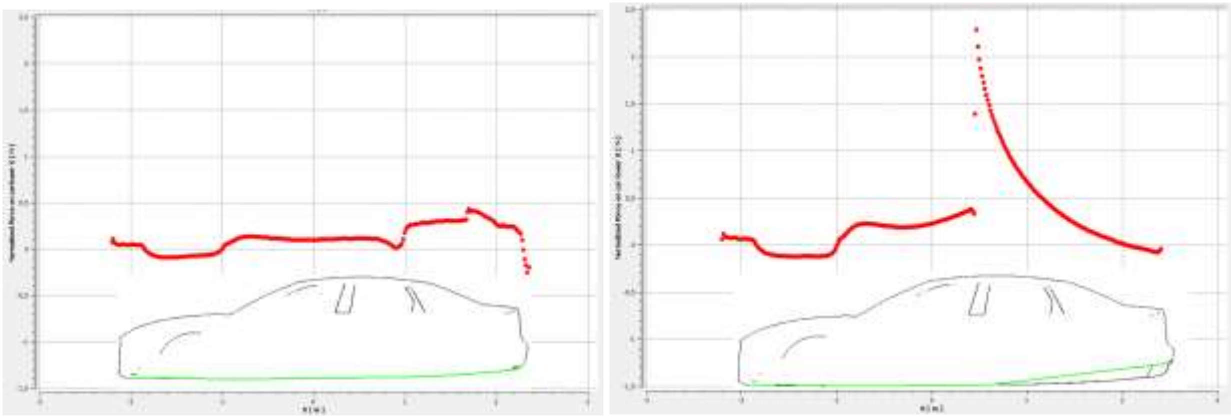

Figure 14. Horizontal force distribution at the bottom surface. Left: original geometry; Right: optimized geometry.

5. CONCLUSION

The purpose of this work was to develop knowledge in the field of high-performance automobile aerodynamic devices, in particular rear diffuser, which have crucial effects on vehicles performance. Such expertise was focused on the comparison of two different geometries, through a Multi Objective optimization integrated with a CFD-Simulation, developing an automated approach making the study fast and reliable.

From the studies performed in this thesis an of optimization considering MOO processes and CFD-Simulation for an optimal preliminary design was developed for a rear diffuser, but it can also be applied to any other automotive aerodynamic components. As in almost every high-performance vehicle manufacturer, the use of CAE tools proved to be an efficient simulation platform to assure the accuracy towards feasible products in a relatively short time frame. The MOO approach linked with optimization algorithms proved to be a fast and reliable alternative in search of the better solution, was notable the efficiency of RSM methods like FAST, since they use virtual optimization techniques, which decreases the number of simulations of low-quality individuals.

This work confirmed that ground effect elements are the most efficient in terms of lift/drag generated, providing expressive levels of downforce with minimal drag. Although the rear diffuser improves the overall downforce of the car, it also increases the drag. Nevertheless, the increase in drag coefficient was just 6% compared with original model, while the lift coefficient reduces 164%. The rear diffuser proved to be an efficient aerodynamic device in a way to improve the aerodynamic performance of a race car providing high levels of downforce adding a relatively low drag.

Regarding the complexity involved in a car CFD simulation, for future work we suggest performing some scale wind tunnel test to confront directly the numerical results with experimental results in order to validate the overall approach. Another point for future research is to study more deeply the geometry of the rear diffuser. The results presented in this paper showed some separations zones in the rear diffuser surface; one suggestion is to try to reduce these separation zones by implementing more vertical fins to keep the flow attached. It would be interesting to try more complex geometries for the rear diffuser like implementing a double angle, or a curved geometry. Thinking about the overall car geometry, the same optimization loop approach used for rear diffuser can be used to improve another car device. Looking the contours of velocity, pressure, wall shear stress and etc, we can see other regions that can be possible candidates for optimization, like front and rear wind shield, front bumper, among others.

6. ACKNOWLEDGEMENTS

B. S. Carmo acknowledges financial support from the Brazilian National Council for Scientific and Technological Development (CNPq) in the form of a productivity grant (grant number 312951/2018-3).

7. REFERENCES

Cooper, K. R., Bertenyi, T., Dutil, G., Syms, J., and Sovran, G., 1998. "The Aerodynamic Performance of Automotive Underbody Diffusers". *SAE Paper* 98-0030.

Cooper, K. R., Sovran, G., and Syms, J., 2000. "Selecting Automotive Diffusers to Maximise Underbody Downforce". *SAE Paper* 2000-01-0354.

Ehirim, O. H., Knowles, K., and Saddington, A. J., 2019. "A Review of Ground-Effect Diffuser Aerodynamics". *ASME Journal of Fluids Engineering*, v.141(2): 020801.

Ferziger, J. H., 1999. "Computational methods for fluid dynamics". Springer Verlag.

Gillieron, P., Samuel, S. and Chometon, F., 1999. "Potential of CFD in Analysis of Under-Bonnet Airflow Phenomena". *SAE paper* 1999-01-0802.

Jowsey, L., 2013. "An Experimental Study of Automotive Underbody Diffusers," Ph.D. thesis, Loughborough University, Loughborough, UK.

Lanfrit, M., 2005. "Best practice guidelines for handling Automotive External Aerodynamics with FLUENT". https://www.southampton.ac.uk/~nwb/lectures/GoodPracticeCFD/Articles/Ext_Aero_Best_Practice_Ver1_2.pdf

Nogueira, L. W., Carmo, B. S., 2018. "Numerical analysis and acoustic optimization of a detached splitter plate applied for passive cylinder wake control". *Journal of the Brazilian Society of Mechanical Sciences and Engineering*, v.40, 37.

Raghu, S., Heil, F. and Santamarina, A., 1999. "Windshield Spray Performance at High Vehicle Speeds". *SAE paper* 1999-01-0803.

Rigoni E., Turco A., Montrone T., 2014. "FAST Optimizers: General Description". *ESTECO Technical Report* 2014-001.

Roache, P. J., 1998. *Fundamentals of Computational Fluid Dynamics*, Hermosa Publishers, Albuquerque, New Mexico.

8. RESPONSIBILITY NOTICE

The authors are the only responsible for the printed material included in this paper.