

Computational simulation of an aerodynamic profile of a vehicle SAE formula type using OpenFOAM

Simulação computacional de perfil aerodinâmico de um veículo do tipo fórmula SAE utilizando OpenFOAM

Lúcio Vargas de Albuquerque Nunes¹; Evelise Roman Corbalan Góis Freire²;
Jonas Laerte Ansoni³

Abstract

Improving vehicle design is essential for esthetic reasons and ensuring better efficiency and lower fuel consumption. The present study intends to provide a computational approach to an actual physical engineering problem: the aerodynamics of automobiles. The focus of this study was to use the open-source software OpenFOAM to study the aerodynamic effects on the external fairing of a Formula SAE vehicle. The vehicle used was the Z03 model of the ZEUS team of the Federal University of Lavras (UFLA). The team participates in university competitions of Formula SAE and, therefore, an aerodynamic improvement of the vehicle for the following versions can be fundamental for the team, increasing efficiency of the vehicle, resulting in a model of greater competitiveness. For analysis purposes, the drag and lift aerodynamic coefficients are analyzed. A procedure for performing aerodynamic simulations of automotive vehicles was systematized, and, in addition, satisfactory results were found for the presented simulation in comparison with results found in literature.

Keywords: OpenFOAM; aerodynamics; computational fluid dynamics; SAE formula.

Resumo

Melhorar o design de veículos é importante não somente por questões estéticas, mas também para garantir melhor eficiência e menor consumo de combustível. O presente estudo tem a intenção de prover uma abordagem computacional para um problema físico real de engenharia: a aerodinâmica de automóveis. Este estudo foi focado na aplicação do software livre OpenFOAM para estudar os efeitos aerodinâmicos na geometria externa de um veículo do tipo Fórmula SAE. O veículo utilizado foi o modelo Z03 do time ZEUS da Universidade Federal de Lavras (UFLA). Este time participa ativamente de competições universitárias na categoria Fórmula SAE, e a melhoria aerodinâmica do equipamento aumenta a eficiência e a competitividade do carro. Os coeficientes de arrasto e sustentação foram estudados como forma de melhoria de eficiência e estabilidade do veículo. O procedimento de simulação foi sistematizado e os resultados encontrados foram satisfatoriamente validados com dados da literatura.

Palavras-chave: OpenFOAM; aerodinâmica; fluidodinâmica computacional; fórmula SAE.

¹ Undergraduate Student, UFLA, Lavras, Minas Gerais, Brazil; E-mail: lucio.nunes@estudante.ufla.br

² Dr., Math and Applied Math Department, ICET/UFLA, Lavras, MG, Brazil; E-mail: evelise.freire@ufla.br

³ Dr., Math and Applied Math Department, ICET/UFLA, Lavras, MG, Brazil; E-mail: jonas.anson@alumni.usp.br

Introduction

Aerodynamics is the study of the airflow forces exerted on a vehicle. The study of this area can improve the performance, consumption and stability of a vehicle since reducing the resistance of its movement increases the speed for the same amount of energy expenditure (fuel) or reduces the expense to the same speed (JATHAR; BORSE, 2014). By association, it is easy to note that this area of knowledge is closely linked with the area of fluid mechanics and, therefore, several equations used in these studies, as is the case of the Navier-Stokes equations, are partial differential equations of high complexity and therefore cannot be solved analytically.

Computational Fluid Dynamics (CFD) tools can be used effectively to solve this problem. CFD is an essential tool for engineering since the analysis obtained through this method can reduce the costs of an engineering project by effectively analysing a physical phenomenon without creating experimental prototypes in the initial phase of the project (HOLZMANN, 2017).

The CFD tool uses numerical methods for the solution of the Navier-Stokes equations. For that, processors are used in their resolution, allowing them to find the fields of velocities, pressures, temperature, and other parameters in the flow region. Thus, the engineer can optimize the project from the data obtained, reducing operating costs and improving performance (FREIRE; SELEGHIM JUNIOR, 2020). In addition, CFD-based simulations can reduce the number of experiments and explore phenomena that could not be studied experimentally in laboratories. In the industry, CFD tools are also used to understand aerodynamic phenomena, especially aircraft development and ground vehicles (JATHAR; BORSE, 2014).

The focus of this study is to use the CFD open-source software OpenFOAM to understand the aerodynamic effects on the external fairing of a Formula SAE vehicle. The vehicle considered is the Z03 model of the ZEUS team, which belongs to the Federal University of Lavras (UFLA). The project in question participates in university competitions of Formula SAE and, therefore, an aerodynamic improvement of the vehicle for the following versions can be fundamental for the team, for increasing efficiency of the vehicle and, thus, resulting in a model of greater competitiveness.

Drag coefficient and lift coefficient

By definition, aerodynamics involves studying air movement (or other gaseous fluids) concerning a solid body that moves through it (NASA, 2011).

In this study, the interest in aerodynamics is linked with the search to reduce the vehicle's resistance to movement and, consequently, improve its performance. The goal is to reduce the drag coefficient. An essential factor for this objective is the change in the car's shape. The drag force is a function of the vehicle format (HOLZMANN, 2017). This force can be expressed by equation (1):

$$F_D = \frac{1}{2}\rho V^2 C_D A, \quad (1)$$

where F_D (N) is the drag force, C_D is the drag coefficient, A (m²) is the front area of the vehicle, V (m/s) is the flow speed and ρ (kg/m³) is the density of the air.

The lift force consists of the force that causes the vehicle to be sustained. If applied in the wrong direction, it can cause an excess of force in the vehicle's tires. It should then be kept below a set limit to avoid excess in the tires or a vehicle's lifting that could cause a lack of stability. The force formulation is explained by equation (2):

$$F_L = \frac{1}{2}\rho V^2 C_L A, \quad (2)$$

where F_L is the lift force and C_L is the lift coefficient.

It is known that an increase in the attack angle can cancel negative lift values (GAGNON; RICHARD, 2010), which makes it extremely important to evaluate the geometry/design possibilities that can interfere with this parameter.

Computational fluid dynamics and vehicles design

Several approaches to the aerodynamics of motor vehicles have been carried out in the scientific field. Most of them were performed in commercial software, such as Fluent and CFX (PRASANTH et al., 2016). Few of these studies, however, were conducted using free programs, such as OpenFOAM, which is the software used in this study.

Nebenfuhr's work (NEBENFUHR, 2010) conducted a study comparing Fluent (commercial software) with OpenFOAM (free software) in predicting airflow behaviour on a vehicle. The data obtained through computational

simulation between both software were compared with experimental data performed with a model in a wind tunnel. For this purpose, three OpenFOAM solvers were applied.

The PotentialFoam was used to converge the initial field. The SimpleFoam was used to solve the external flow, both the laminar and the turbulent parts, and PimpleFoam was used to resolve the transient part. For the "steady-state incompressible Reynolds Averaged Navier-Stokes" (RANS), a speed of 27.8 m/s was used; for this case, the main focus was stability and accuracy. For the transient regime, called "incompressible Large Eddy Simulation" (LES), performed in a Sport Utility Vehicle (SUV), the goal was to reduce the level of noise generated by airflow through the side mirrors and the windshield (A-pillar). For this, a speed of 39.0 m/s was used.

For comparison purposes, two mesh generators were used: Harpoon and snappyHexMesh. The mesh was checked through checkMesh. The cell numbers used for the meshes built-in harpoon were in the order of 17.7M and 19.7M. The model car consisted of an approach of the Volvo XC90. The simulation was performed in SimpleFoam through a convergence process after 1.000 iterations. The authors concluded that OpenFOAM was able to deliver results as good as commercial softwares. In other words, very satisfactory results were obtained from both softwares.

Gagnon and Richard (2010) studied the drag force of an aerodynamic profile. The meshes have been modified to the same geometry. In the profile simulation, snappyHexMesh was used to generate the mesh, and the k-omega-SST model was used to model the turbulent effect. The drag and lift coefficients were calculated and analyzed for several mesh patterns used as reference (GAGNON; RICHARD, 2010).

The work of Jahtar and Borse (2014) brought an aerodynamic study using OpenFOAM, intending to analyze airflow on two types of car design (a sedan and a hatchback). BlockMesh was used for geometry creation, and snappyHexMesh was applied for mesh generation. The solver used was PotentialFoam. For turbulence analysis, the k-omega model was used, and post-processing was performed by ParaFoam. In the processing, simulations were performed to calculate drag coefficient, lift coefficient, drag and lift force. After the study, a more efficient model was reached from the point of view of aerodynamics for the vehicle. In other words, the modifications were based on smoothness vehicle shape,

rounded corners, high angle for the windscreen, tapered rear end and minimized body seams. In this computational model, the values of the drag coefficient were reduced. The most efficient model has a smoother contour, rounded corners, greater angle of the windshield and more tapered rear. In addition, it has been proven that the use of a more prominent rear glass angle allows for a reduction of drag. Therefore, it was concluded that the reduction of drag depends both on the configurations and dimensions, as well as on the arrangement of the glasses and the configuration of the rear model.

As can be seen, despite requiring a longer learning time, OpenFOAM proved to be a tool with great potential for conducting numerical studies in vehicle aerodynamics since it delivers results of the same quality as commercial software.

Methodology

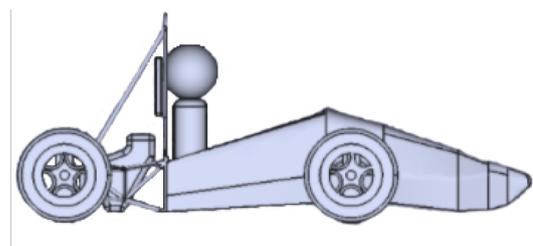
The OpenFOAM 7 software was used to perform the computational simulation of the airflow on the Formula SAE vehicle.

Because it is an external flow, it was necessary to create a control volume, composed by an *inlet* and an *outlet* and the domain for all airflow. The motor vehicle is then inserted within the control volume in the form of an obstacle. In this section, all the steps performed in this study are described: Pre-processing, Processing, and post-processing.

Geometry of formula SAE vehicle

The initial geometry was developed using the SolidWorks computational environment® Premium 2018 x64 Edition. It was developed and provided by the ZEUS project of the Federal University of Lavras (UFLA), as can be observed in Figures 1 and 2.

Figure 1 – Computational geometry car prototype.



Source: The authors.

Figure 2 – Original car model from Formula ZEUS UFLA.



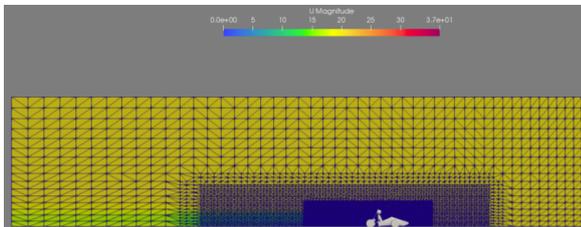
Source: The authors.

The car is 1377 mm wide, 2735 mm long and 1157 mm high at its maximum point. The chassis is made of SAE 1020 steel tubes and the nozzle and side fairings are laminated with fiberglass and polyester resin. The same is shown in Figure 2.

Meshing details and convergence analysis

The mesh generation process was divided into two stages: generation of the main computational domain and inclusion of the geometry of the motor vehicle. The computational mesh of the main domain was made using the OpenFOAM generator. For this, blockMesh was used to generate the computational domain, which is shown in Figure 3.

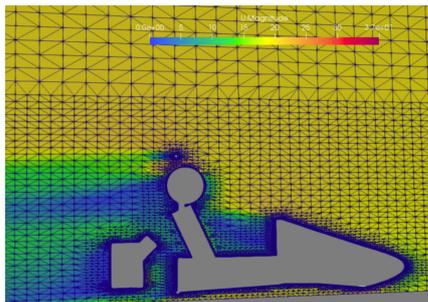
Figure 3 – Mesh computational domain.



Source: The authors.

From the first computational domain, snappyHexMesh was used to introduce the geometry of the vehicle into the domain as an obstacle and generate the mesh, as can be observed in Figure 4.

Figure 4 – Details of the refinement box around the car.



Source: The authors.

For all simulations, the convergence criteria were set 10^{-3} for pressure, velocity and turbulent components, as developed by Guerrero and Castilla (2020). The percentage of relative error was evaluated using equations (3) and (4), as follow:

$$\%ER_j = \frac{|X_j^i - X_j^{ref}|}{|X_j^{ref}|} \times 100, \quad (3)$$

$$MER_j = \sum_{j=1}^N \frac{ER_j}{N}, \quad (4)$$

where X is the drag coefficient at point j , N is the number of cells in the mesh, and the indexes i and ref refer to coarse and fine mesh, respectively. MER_j is the factor of safety. As indicated in Roache (1998), the MER_j was set equal to equation (4).

The mesh convergence analysis was performed comparing three different degrees of refinement, as described in Table 1.

Table 1 – Details of grid convergence analysis

Mesh	Number of elements	Average relative error (%)	Number of iterations
Mesh 1	602,873	3.90%	154
Mesh 2	1,469,558	3.82%	237
Mesh 3	2,199,418	Reference mesh	962

Source: The authors.

Observing Table 1, it is possible to notice that the decrease in the average relative error values is directly proportional to the mesh refinement.

Table 2 shows the results obtained for thickness and y^+ . Higher thickness values guarantee a higher percentage of the mesh with refinement compatible with the boundary layer.

Table 2 – Details of grid thickness and y^+ average

Mesh	Thickness (%)	y^+ Average
Mesh 1	87.2	22.7129
Mesh 2	88.5	16.0630
Mesh 3	89.7	13.0336

Source: The authors.

Where y^+ is a non-dimensional distance applied to describe how fine or coarse a mesh is. The good value of y^+ is able to guarantee the ability of the mesh to capture flow details in domain regions close to walls or obstacles. More complex geometries require increasing the refinement level only in regions with small length scales.

The identification of these regions could be interesting to reduce the computational effort. To handle a thin region, the region’s characteristic thickness, or length scale, must be identified.

Thus, considering the values of relative error, thickness and y^+ , the mesh chosen for the execution of the simulations was the most refined mesh, with 2,199,418 elements.

Boundary conditions

The external flow was considered incompressible, steady on a permanent basis. The vehicle speed of 29 m/s (104.4 km/h) was considered a boundary condition. The k-omega Shear Stress Transport model was applied in order to perform the turbulence effects. This model was used for its ω proven reliability in separation zones and its ability to blend a good freestream model to a good boundary layer model (GUERRERO; CASTILLA, 2020; JATHAR; NIKAN; BORSE 2014; PORCAL; TOETI; GAMEZ-MONTERO , 2021; RAVELLI; SAVINI, 2018).

Processing

The simulation was performed using OpenFOAM. Two solvers were applied: PotentialFoam (potential flow solver which solves for the velocity potential, to calculate the flux-field, from which the velocity field is obtained by reconstructing the flux) and SimpleFoam (Semi-Implicit Method for Pressure-Linked equations).

The PotentialFoam has the function of generating a potential initial flow, in order to provide an approximate field of reality and reduce the work of the next solver (PORCAR; TOETI; GAMEZ-MONTERO, 2021).

The SimpleFoam, it is an appropriate solver for incompressible, steady, turbulent flows, and able to provide the turbulence and gear field results (GUERRERO; CASTILLA, 2020). In the simulation in question, an initial time of 0 was used, and a final time of 100. The time interval was 0.5 seconds. As described by Guerrero and Castilla (2020), the goal analysis was not to study transient phenomena, but study the vortex average on a medium flow, in order to evaluate the effect on average stability of the car.

The simulations were carried out in parallel (24 threads) on a cluster running under CentOS Linux release 7.4.1708 and OpenFOAM v7 (OpenFOAM, 2021). Table 3 shows the cluster specifications applied to perform the simulations.

Table 3 – Workstation specification

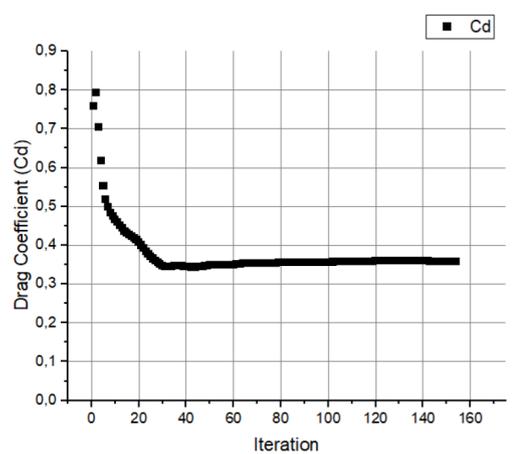
Cluster	Workstation specifications
Head Node	Intel(R) Xeon(R) CPU X5670 2.93GHz; RAM: 16GB; HD 1TB
9 compute nodes	Intel(R) Xeon(R) CPU X5660 2.80GHz; RAM: 12 GB; HD 500GB

Source: The authors.

Post-processing and results

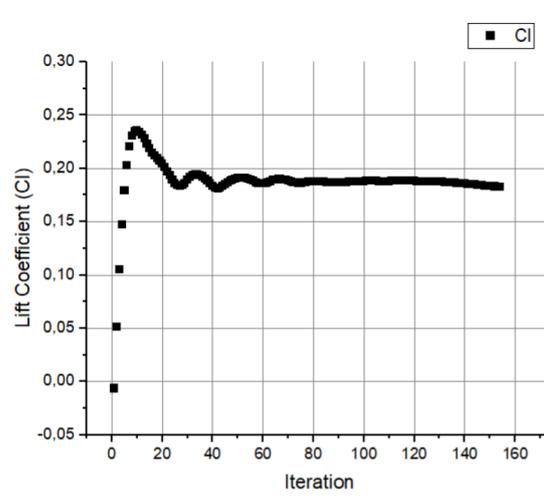
By simulating the flow around the vehicle, it was possible to obtain the drag coefficient and lift coefficient values. The results can be seen in Figures 5 and 6.

Figure 5 – Drag coefficient as a function of number of iterations.



Source: The authors.

Figure 6 – Lift coefficient as a function of number of iterations.

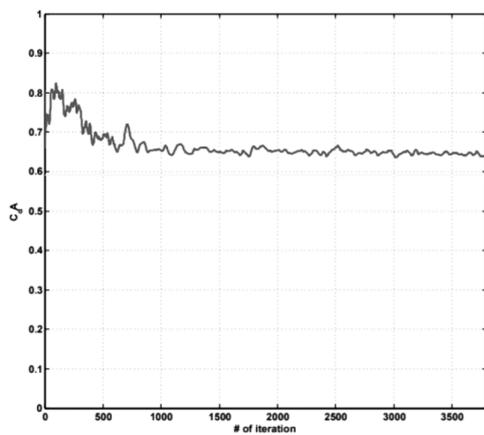


Source: The authors.

Figure 5 shows that the drag coefficient is higher during the beginning of the vehicle movement. Newton's break-in inertia can explain this. After this break of inertia, the drag coefficient stabilizes and remains about constant within a range. Similar behaviour can be observed for the lift coefficient in Figure 6. The mean drag coefficient is 0.316, and the average sustain coefficient is 0.119.

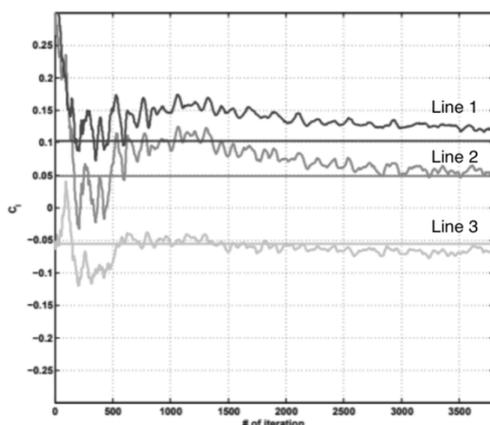
The simulation results were compared to those obtained in the study performed by Nebenfuhr (2010), which have similar boundary conditions. The coefficients found in Nebenfuhr (2010) study are represented by Figures 7 and 8:

Figure 7 – S80 CdA, obtained by OpenFOAM.



Source: Adapted from Nebenfuhr (2010).

Figure 8 – Cl of an S80, obtained through OpenFOAM. Line 1 represents the fluent reference value. Line 2: total support; Line 3: front support (NEBENFUHR, 2010).



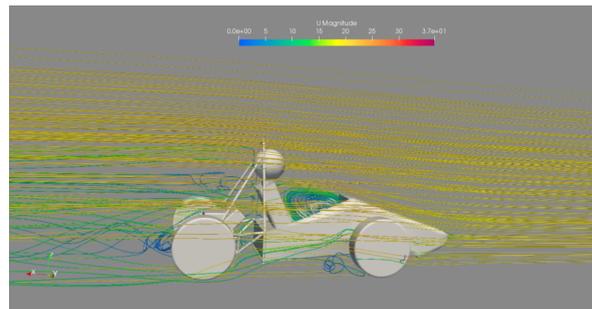
Source: Adapted from Nebenfuhr (2010).

Despite using a more extended sampling of time in the simulations, the behaviours observed in Nebenfuhr (2010) are qualitatively quite similar to the results shown in Figures 5 and 6. In the case of Nebenfuhr simulations, the drag coefficient values are substantially higher, showing lower aerodynamics than those of the Formula car simulated here. It is possible to observe that the coefficients reach a peak at the beginning of the simulation, with the curve stabilizing with time.

This difference in the car's aerodynamics is expected because, in Nebenfuhr (2010) study, the flow around an SUV vehicle was simulated. In addition, it is possible to observe some lift coefficient negative values. It has already been shown to be responsible for generating unwanted pressure on the vehicle's wheels.

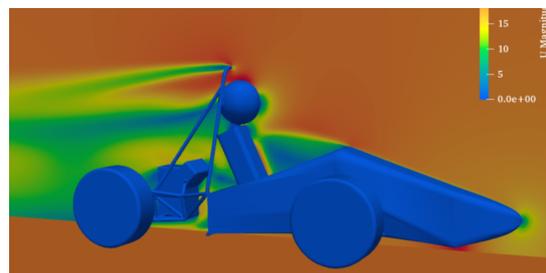
The behaviour of streamlines, velocity and pressure contours can be observed in Figures 9, 10 and 11.

Figure 9 – Streamlines colored by axial vorticity.



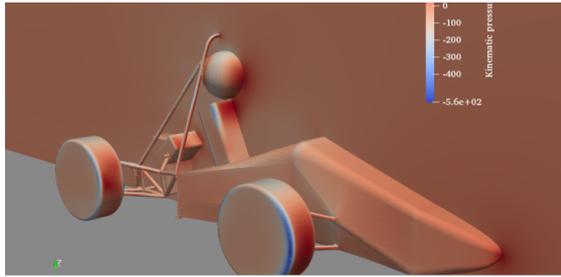
Source: The authors.

Figure 10 – Velocity contour.



Source: The authors.

In Figure 11, at the vehicle's front end, it is possible to notice a maximum flow pressure. There is also a high-pressure region where the top of the driver's seat is located. At the initial moment of the flow, it is possible to observe the regions that suffer from higher flow speed. They are the top of the vehicle and the end of the vehicle's front.

Figure 11 – Pressure coefficient contours.

Source: The authors.

There is, outside these areas, a uniformity in the flow of fluid, being this, at approximately 10 m/s.

The lift and drag coefficient evaluation, together with the regions of high pressure, suggest possible changes in the shape of the vehicle's front end. Improvements in the design would guarantee less suffering for the structure since it would be possible to reduce the quality of the material in the region and cheapen the cost of the vehicle without reducing the integrity of the project.

Conclusions

This study aimed to analyse the drag and support coefficients from the airflow around a prototype SAE type race vehicle developed by the ZEUS Team of the Federal University of Lavras. The influence of vehicle design on their efficiency was considered.

Performing Computational Fluid Simulation from OpenFOAM, it was possible to achieve accurate drag and lift coefficients for the formula vehicle. In addition, a computational procedure was systematized using the tool in question. The results obtained were validated with results found in the literature.

Future studies seek to change the shape of the front end of the vehicle to find a shape under pressure. Thus, it would be possible to reduce the quality of the material in the identified regions, reducing project costs and increasing the car's efficiency in competitions.

Acknowledgments

The authors would like to thank the CNPq (Conselho Nacional de Desenvolvimento Científico e Tecnológico), grant 428792/2018-9, for the financial support and the ZEUS Project of the Federal University of Lavras for the granting of prototype geometry.

This research was carried out using the computational resources of the Center for Mathematical Sciences Applied to Industry (CeMEAI), granted by FAPESP (2013/07375-0).

References

- FREIRE, E. R. C. G.; SELEGHIM JUNIOR, P. optimization of fermenters for ethanol production: residence time analysis applying computational fluid dynamics. *Semina: Exact and Technological Sciences*, v. 41, n. 1, p. 51-58, 2020. DOI: <http://dx.doi.org/10.5433/1679-0375.2020v41n1p51>.
- GAGNON, L.; RICHARD, M. Parallel CFD of a prototype car with OpenFOAM. *In: WORKSHOP OPEN-FOAM, 5., 2010, Gothenburg. Proceedings [...]*. Gothenburg: OpenFOAM, 2010.
- GUERRERO, A.; CASTILLA, R. Aerodynamic study of the Wake Effects on a Formula 1 Car. *Energies*, Basel, v. 13, n. 19, 2020. DOI 10.3390/en13195183.
- HOLZMANN, T. *Mathematics, numericals, derivations and OpenFOAM*. 4th ed. Germany: [s. n.], 2017.
- JATHAR, L.; BORSE, S. Study of Flow over Car by Changing Different Parameters using OpenFOAM. *International Journal of Fluids Engineering*. Delhi, v. 6, n. 1, p. 29-39, 2014.
- JATHAR, L.; NIKAN, K.C.; BORSE, S. Determination of Coefficient of Lift And Lift Force on Sedan Car Body by Changing Different Parameters Using OpenFOAM. *International Journal on Recent Technologies in Mechanical and Electrical Engineering*, Rajasthan, v. 1, n. 5, p. 10-18, 2014.
- NASA - NATIONAL AERONAUTICS AND SPACE ADMINISTRATION. *What is Aerodynamics*. Washington: Nasa, 2011. Available from <https://www.nasa.gov/audience/forstudents/k-4/stories/nasa-knows/what-is-aerodynamics-k4.html>. Access: Mar. 20, 2022.
- NEBENFUHR, B. *OpenFOAM: a tool for predicting automotive relevant flow fields*. Gothenburg: Chalmers University of Technology, 2010.

PORCAR, L.; TOETI, W.; GAMEZ-MONTERO, P. J. Study of the Effect of Vertical Airfoil Endplates on Diffusers in Vehicle Aerodynamics. *Designs, Basel*, v. 5, n.3, 1-23, 2021. DOI: <https://doi.org/10.3390/designs5030045>.

PRASANTH, A.; BISWAL, S.; GUPTA, A.; BARODAWALA, A. *Complete Design and Optimization of the Aerodynamics of a FSAE Car using Solid works ANSYS & XFLR5*. London: Proceedings of the World Congress on Engineering 2016. v. 2.

RAVELLI, U.; SAVINI, M. Aerodynamic Simulation of a 2017 F1 Car with Open-Source CFD Code. *Journal of Traffic and Transportation Engineering*, [s. l.], v. 6, p.155-163, 2018. DOI 10.17265/2328-2142/2018.04.0013.

ROACHE, P. J. Verification of codes and calculations. *AIAA Journal*. Reston, v. 36, n. 5, p. 696-702, 1999.

Received: Jan. 25, 2022
Accepted: Mar. 24, 2022
Published: May 15, 2022