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

COB-2021-0851
NUMERICAL SIMULATION APPLIED TO THE AERODYNAMIC STUDY OF A FORMULA SAE VEHICLE

Carlos Henrique de Sousa
Farney Coutinho Moreira
Faculdade de Engenharia de Resende
carlos.sousa@aedb.br
farney.coutinho@aedb.br

Abstract. *Computer simulation has, over the past decades, continuously contributed to companies to develop their products and production processes. The present work aims to make an aerodynamic study of a Formula SAE vehicle using the Ansys Fluent extension, where the Navier Stokes equations are solved by the Finite Volume Method, starting from the boundary conditions coherent to the event to be simulated. The simulation analysis methodology adopted was through comparison between the model used to simulate vehicle body proposal and a validated model to analyze another vehicle with similar aerodynamic characteristics, using the drag coefficient as a parameter. The results obtained prove the reliability of the numerical tool in correctly capturing the aerodynamic coefficients of the vehicle.*

Keywords: *modern engineering, CAE, numerical simulation, aerodynamic*

1. INTRODUCTION

Scientific knowledge is characterized by seeking explanations about physical, chemical, biological events and objects, according to certain acceptance criteria about what can be an explanation, a good explanation or a better explanation. In this process, observations, conjectures, experiments, verifications, refutations, concepts, models, theories, are at the core of the construction of scientific knowledge. In other words, this knowledge builted, depends on the questions asked, the definitions, the metaphors and the models used (Moreira, 2013).

The engineer is the professional qualified to develop goods and services in the most varied areas using technical and scientific knowledge inherent to his field of training and performance. His work tools, which until the middle of the 20th century were used manually, gained more agility and precision with the intervention ofs that help him in the analysis and decision making in projects and control (Silva, 2017).

The use of simulation for modeling physical events dominated the field of product development. Predicting the behavior of components under numerous working conditions is an essential factor for the engineer to have a panoramic view of the effects that these events will provide to the product. This allows the professional to have total control of the key variables and achieve acceptable solutions. In addition to identifying the physical phenomena involved in a given event, it is necessary to know how to insert the appropriate boundary conditions in each case, what are the possibilities for simplifying the model, understand how the software used performs the mathematical equations involved, perform a coherence analysis results obtained and identify which ones are relevant for decision making.

Today, engineering companies face several challenges during their projects. The entire process associated with the development of an equipment, product or structure must present a good balance of crucial factors, such as quality, cost and time, and the scenario observed today brings several difficulties to achieve this balance. For example, the complexity of products is increasing, resources (both human and raw material) are scarcer and the deadlines for completing projects and launching products shorter (Silva, 2017).

Engineering relies on specialized software for the development of new products. This technology is known as CAE (Computer Aided Engineering) and encompasses a whole series of systems that assist the professional from the analysis of basic physics to more complex systems. Despite simplifying the project development process, their correct operation requires an engineer who has knowledge in the physical sciences and the ability to abstraction to create a computational model from a real product (Software, 2016).

2. LITERATURE REVIEW

The main physical concepts that integrate a flow analysis over a submerged solid will be mentioned in this chapter, the theoretical foundations of the numerical method that is generally used by software to obtain the solutions of the desired quantities for analysis.

2.1. DIFFERENTIAL APPROACH X INTEGRAL APPROACH TO FLOWS

In the study of fluid mechanics, there are two distinct approaches aimed at particular cases of analysis: Approach in integral form and differential form. In the first case, a region of space is studied as the fluid flows through it. Equation (1) is the generic way of expressing that the rate of variation of a given extensive property of the N system is equivalent to the sum of the rate of variation of the amount of the N property within the control volume and the rate at which the N property is exiting the surface of the control volume. (Fox, 2014)

$$\left.\frac{dN}{dt}\right|_{system} = \frac{\partial \int_{CV} \eta\rho d\forall}{\partial t} + \int_{CS} \eta\rho \vec{V} d\vec{A} \tag{1}$$

$\int_{CS} \eta\rho \vec{V} d\vec{A}$	Rate of change of the extensive property of the N system.
$\left.\frac{dN}{dt}\right\|_{system}$	Rate of change in the quantity of property N within the control volume. The term ∫CV ρηd∀ calculates the instantaneous value of N within the control volume (∫CV ρηd∀ is the instantaneous mass within the control volume).
$\frac{\partial \int_{CV} \eta\rho d\forall}{\partial t}$	The rate at which property N is leaving the surface of the control volume. The term calculates the rate of heat transfer going out through the area element of the control surface; multiplying by η the flow rate of the property is calculated N through the element; and consequently the integration calculates the net flow of N out of the control volume.

On other side, according to the same author, the differential approach is used when the interest is to analyze the flow in detail. This type of analysis is based on infinitesimal control systems and volumes. Forces acting on a fluid element can be classified as field force and surface forces. Looking at figure 1, taking only component x, the mass dm and volume dV=dxdydz, only the stresses that act in the x direction will give rise to surface forces in the x direction.

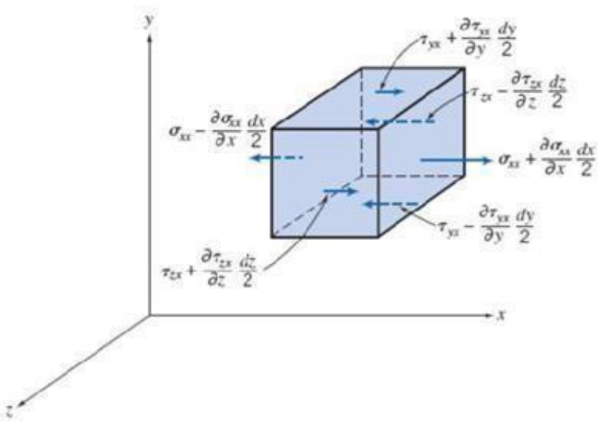

Figure. 1: Stresses acting on the x-axis on any differential element.
Source: Fox, 2014.

Introducing the equation of active efforts described in Equation (2), multiplying the stresses by the area differential, in second law of Newton described in Equation (3) we have the equation of motion with respect to the x axis. Placing the stresses as a function of speed gradients and fluid properties, we achieve the most coherent solution of Equation (4), globally known as the Navier-Stokes Equation.

$$dF|_x = \left(\rho g_x + \frac{\partial \sigma_{xx}}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z}\right) dxdydz \quad (2)$$

$$dF|_x = dm\left(\frac{Du}{Dt}\right) = dm\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t}\right) \quad (3)$$

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z} + \frac{\partial u}{\partial t}\right) = \rho g_x - \frac{\partial P}{\partial x} + \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \quad (4)$$

The same idea is applied for the y and z directions, described in Equations (5) and (6):

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z} + \frac{\partial v}{\partial t}\right) = \rho g_y - \frac{\partial P}{\partial y} + \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial^2 z}\right) \quad (5)$$

$$\rho\left(u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z} + \frac{\partial w}{\partial t}\right) = \rho g_z - \frac{\partial P}{\partial z} + \mu\left(\frac{\partial^2 w}{\partial^2 x} + \frac{\partial^2 w}{\partial^2 y} + \frac{\partial^2 w}{\partial^2 z}\right) \quad (6)$$

ρ	Especific mass
μ	Fluid viscosity
u, v, w	Velocity components
$\frac{\partial \vec{V}}{\partial t}$	Local acceleration
$\left(u\frac{\partial \vec{V}}{\partial x} + v\frac{\partial \vec{V}}{\partial y} + w\frac{\partial \vec{V}}{\partial z}\right)$	Connective acceleration
$\frac{\partial P}{\partial x}$; $\frac{\partial P}{\partial y}$; $\frac{\partial P}{\partial z}$	Rate of chance of thermodynamic pressure in relation to the position coordinates
g_x ; g_y ; g_z	Gravity acceleration components
$\left(\frac{\partial^2 \vec{V}}{\partial^2 x} + \frac{\partial^2 \vec{V}}{\partial^2 y} + \frac{\partial^2 \vec{V}}{\partial^2 z}\right)$	Viscous forces

2.2. DRAG AND LIFT FORCES

One of the main output variables of an aerodynamic analysis is strength. This, arising from the pressure distribution generated by the contact of air with the surfaces of the object, can be favorable or unfavorable for the project. This is the main reason why this variable should be analyzed, aiming at reducing or increasing its intensity and the ideal direction and direction of the vector.

According to Fox (2014), drag force is the component of the force on the body that acts parallel to the direction of the relative movement. This force may be the result of pure friction between the flow and the surface of the plate, so that the air particles tend to slide on the surface and the friction between both causes it to force the body in the opposite direction to the movement; of pure pressure when the flow is perpendicular to the flow surface, so that the air particles tend to collide head-on with the body surface also generating the effort contrary to the movement; and pressure and friction acting simultaneously.

Following Fox (2014), lift is defined as the component of the fluid's force perpendicular to the fluid's movement. They can be directed downwards, with the objective of better stability and upward (lift force), as the objective of lifting the object providing flight condition. Generally, the down force is more directed towards high performance ground vehicles and the lift force is totally related to the wings of airplanes and air vehicles.

3. COMPUTATIONAL FLUID MECHANICS

The following items will briefly conceptualize the entire computational extension in a dynamic analysis within fluid mechanics, also known worldwide as CFD. This technique joins computational resources to solve numerical methods applied to specific solutions for systems of partial differential equations that describe flows in a given medium.

3.1 MESH CONCEPT AND DISCRETIZATION

A computational mesh can be defined as a set of points and elements that describe an analysis region or geometry. These elements are polygons connected by the points forming the nodes and by their faces, which, when joined, should approach the object of study. In each element there is a mathematical concept, usually formed by systems of differential equations that are solved by specific methods so that there is continuity between neighboring elements. This whole execution is called discretization.

In essence, discretization is the process by which a mathematical expression, be it a function or equations in the differential or integral form involving functions, all of which are seen as having infinite continuous values of values in some domain, are approximated by analogous expressions that they prescribe values only at a finite number of points or discrete volumes in the domain. (Anderson,1995)

The figure 2 illustrates an example of a mesh created for aerodynamic analysis of an wing.

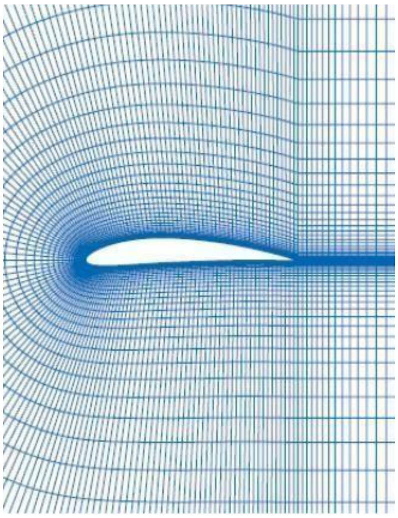

Figure 2: Example of a mesh used to resolve the flow around an airfoil.
Source: Fox, 2014.

3.2 FINITE VOLUME METHOD

Among the numerical resolution methods for integral equation systems, the method covered in this study is the Finite Volume Method. The need to solve the Navier-Stokes equations implies the use of specific resources, while the same is also directed to solve cases with this profile. Analytical solutions of partial differential equations involve specific cases with expressions that continuously vary the dependent variables over the domain. In contrast, numerical solutions can provide answers specifically at discrete points in the domain (Anderson, 1995).

According Osses (2016), an important property of this method is that the conservation principles (mass, momentum and energy), which are the basis of mathematical modeling for continuum mechanics, by definition, are respected by the equations deduced by the volume method finite. The method is not limited to fluid mechanics problems, and, in general, involves the following steps:

I. Decompose the domain into control volume;

II. Formulate the integral conservation equations for each control volume;

III. Numerically approximate integrals;

IV. Approximate the values of the variables on the faces and those derived with the information of the nodal variables;

V. Assemble and solve the algebraic system obtained;

4. METODOLOGY

The simulation execution process is mapped as shown in figure 3. The evaluation criteria of the results obtained were based on an analysis of coherence in the speed field across the domain and on the comparison between the drag coefficient calculated for the vehicle and this same coefficient of a vehicle with similar technical characteristics, adopting the same boundary conditions.

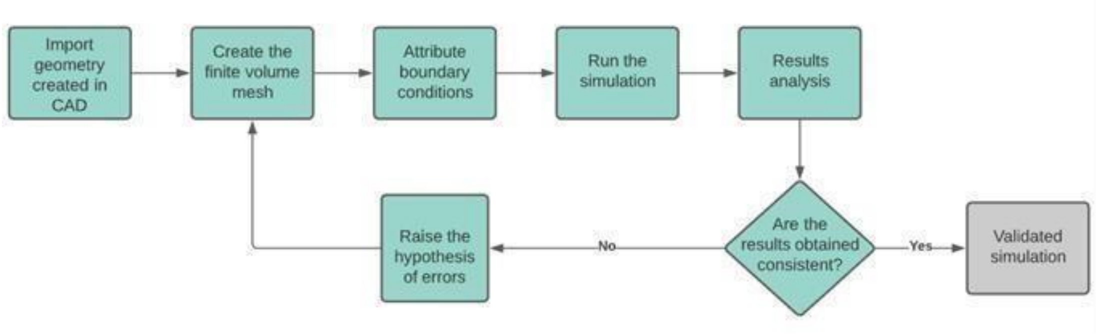

Figure 3: Flow chart for validating the simulation.

5. RESULTS

The initial concept of the vehicle was created in Catia V5 software. Design Modeler and Spaceclaim are Ansys extensions available to prepare geometry for simulation if there is incompatibility in any region in the geometry that needs to be corrected. First, the project was developed with front and rear in the expectation of a good distribution of down force, ensuring vehicle stability on asphalt, as shown in figure 4. As a parameter to validate the model, a simulation made by Fernandez (2019) was used, where the author was interested in measuring the drag coefficient of a vehicle with similar aerodynamic characteristics, used in the same competition category.

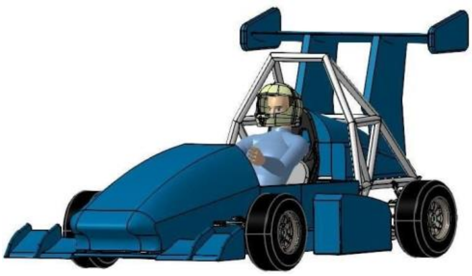

Figure 4: Three-dimensional model of the proposed Formula SAE projected in CAD.

In cases of external flow, the domain is the entire volume between the surfaces of the vehicle and the boundary faces of the enclosure, as shown in figure 5. The domains discretization was done in a mesh with tetrahedral volumes, not structured, while this one is directed to domains with irregular surfaces, as is the case of vehicles with aerodynamic devices. The Finite Volume Method acts in the transformation of the Navier-Stokes system of equations in each domain control volume into discretized equations for defining the velocity and pressure field contained in the mesh point, respecting the boundary conditions. The boundary conditions adopted for the model are wall effects and distant velocity field, which also considers the velocity field in themost distant zones of the boundary layer of the model.

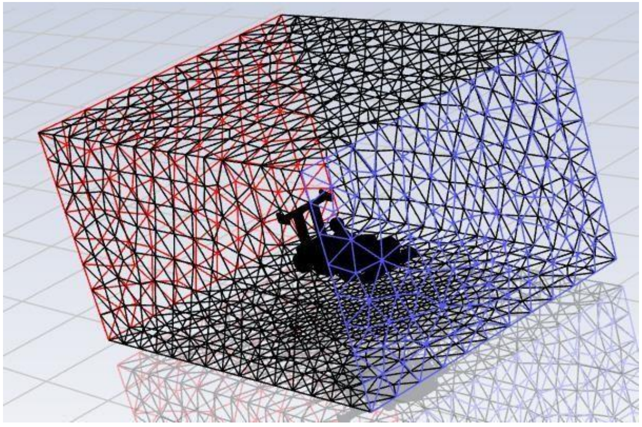

Figure 5: Mesh generated.

The regions adjacent to the vehicle surfaces, those that make up the boundary layers estimated along the domain, in addition to the surfaces that have a high degree of curvature and complexity, should be treated with smaller finite volumes, as they need greater precision in the variables of interest located therein, as shown in figure 6.

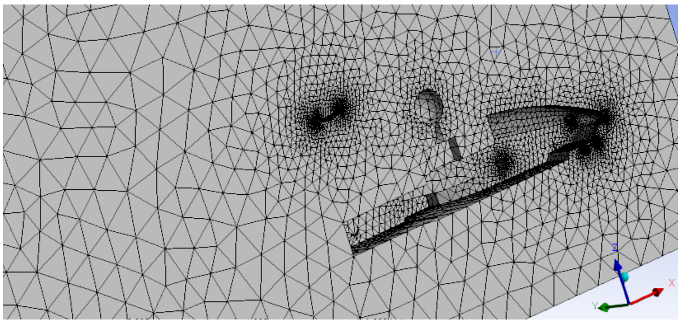

Figure 6: View of the symmetry plane to verify the mesh development.

Usually, the Reynolds number is the parameter for predicting the nature of the flow: laminar, transitional or turbulent. This parameter relates inertial forces and viscous forces. Simulations for the study of aerodynamics commonly resort to RANS (Reynolds Average Navier Stokes) and, according to Ansys (2009), the Reynolds Average Navier-Stokes equations govern the transport of average flow quantities, with the entire range of scales of turbulence being shaped. The RANS-based modeling approach therefore greatly reduces the effort and computational resources required and is widely adopted for practical engineering applications.

The most important step is to define which turbulence model will be used to meet the particularities of the model. As the Ansys theoretical guide covers, the k-ε and k-ω models are composed of two transport equations related to turbulent kinetic energy and eddy dissipation rate terms. The k-ω model has fewer terms than the k-ε, as it does not consider functions related to damping. From these two models there are other derivations such as the Shear-Stress Transport k-ω, which considers the modified turbulent viscosity equation to take into account the transport effects related to turbulent shear stress. The model chosen for the simulation was the standard k-ε (Figure 7), as it is a well accepted model in cases of turbulent viscous flow. A The "Realizable" shunt was chosen because it offers superior treatment with respect to average fluxes in complex structures.

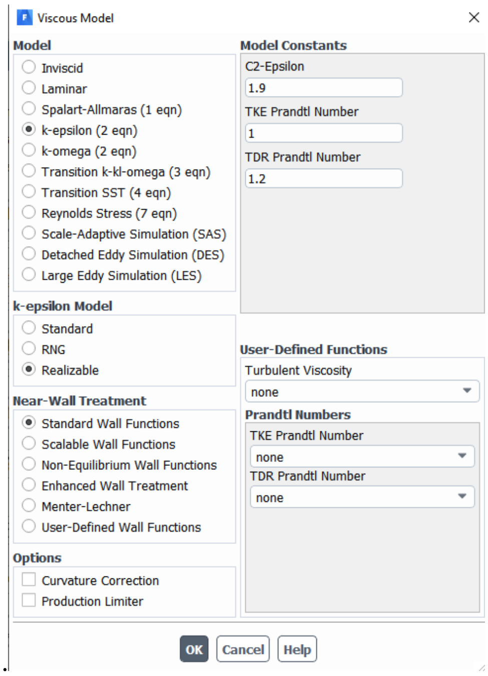

Figure 7: Definition framework of the model adopted for the simulated flow.

The path lines of the air particles during the flow are in accordance with figure 8. As can be seen, the particles that hit the nose of the vehicle (stagnation point) are redirected to the vicinity of the upper or lower surfaces of the hood, passing by by the surface of the helmet, being also distributed in the inferior and superior zones. Particles that flow from the bottom to the rear of the vehicle and those that flow from the top to the rear wing. In the lower rear region of the vehicle, the particles that leave the vehicle tend to rub against the air particles that are located on the extreme wall of the vehicle, causing fluctuations that characterize the part of the flow that undergoes turbulence.

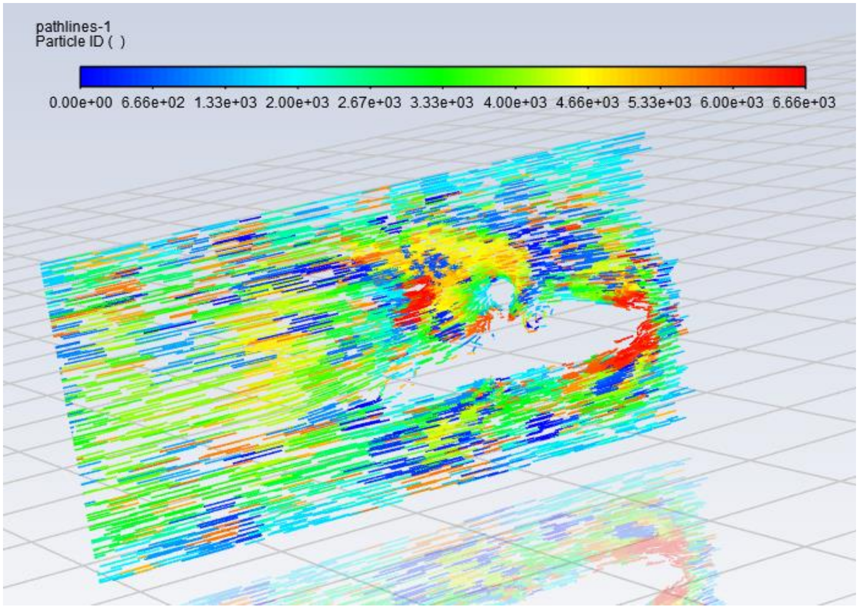

Figure 8: Path lines of flow representation in a symmetry plane of vehicle.

Analyzing the result for the velocity field in figure 9, the vectors furthest from the surfaces have values very close to 20m/s (green color), while the particles near the surface tend to have a velocity vector reduced (blue color) as they are in direct contact with the solid during flow, generating frictional drag and pressure.

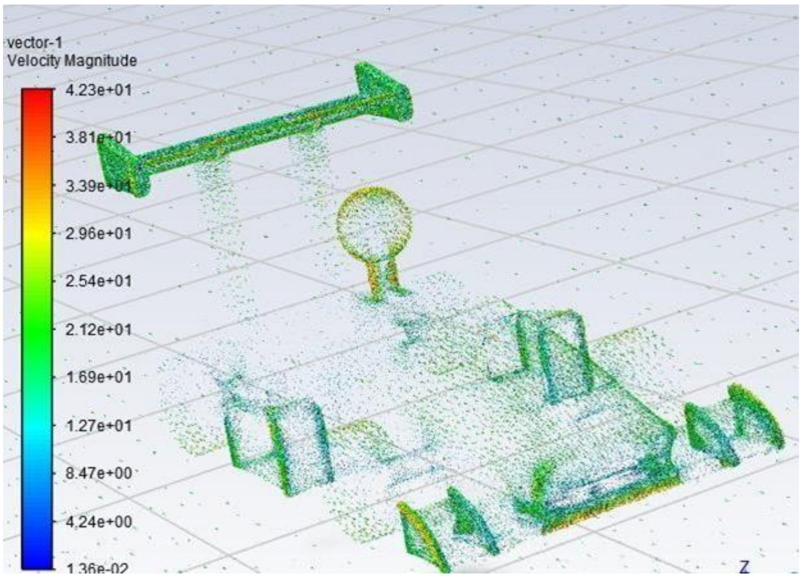

Figure 9: Vector diagram referring to the velocity field.

From the measured drag and the effective area shown in figures 10 and 11, respectively, added to the air density (1,225kg/m³), the flow velocity, it is possible to calculate the drag coefficient reached by the vehicle, through equation 9. Performing the calculation, with the respective values of each variable defined, a drag coefficient of 1.18 is obtained.

Drag	(n)
wall-solid	-197.72188

Figure 10. Drag force calculated.

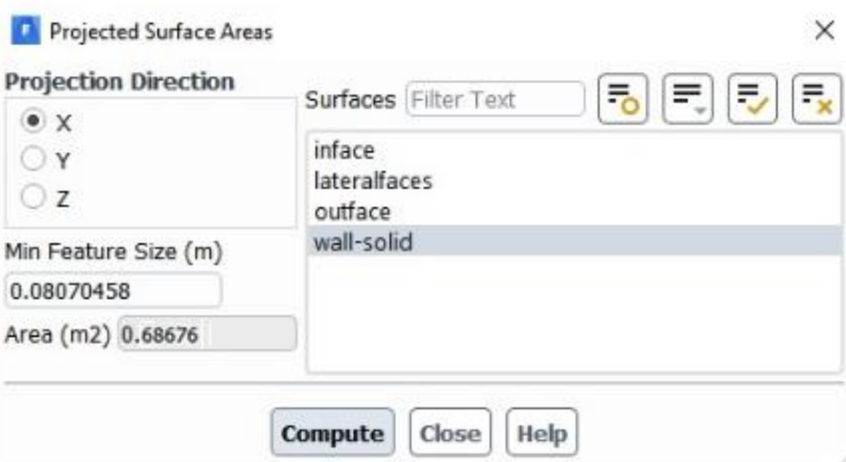

Figure 11: Effective area of vehicle.

$$Cdg = \frac{2.Fdg}{\rho.Adg.V^2} = \frac{2.[197,72]}{[1,225].[0,68].[20]^2} \quad (7)$$

Fdg – Drag force – [N]

Cdg – Drag coefficient – [-]

Adg – Effective Area – [m²]

V – Velocity – [m/s]

ρ – Especific Mass – [kg/m³]

In an attempt to verify the reliability of the model, a comparison was made with Fernandez (2019), where the author performed a CFD simulation of a vehicle from a German Formula SAE team, on the Simscale platform. Adopting the same speed of 20 m/s, the author traced the curved curve of the coefficient of aerodynamic drag accumulated as a function of the vehicle's length, which is added from the front wing to the end of the rear wing, as shown in figure 12.

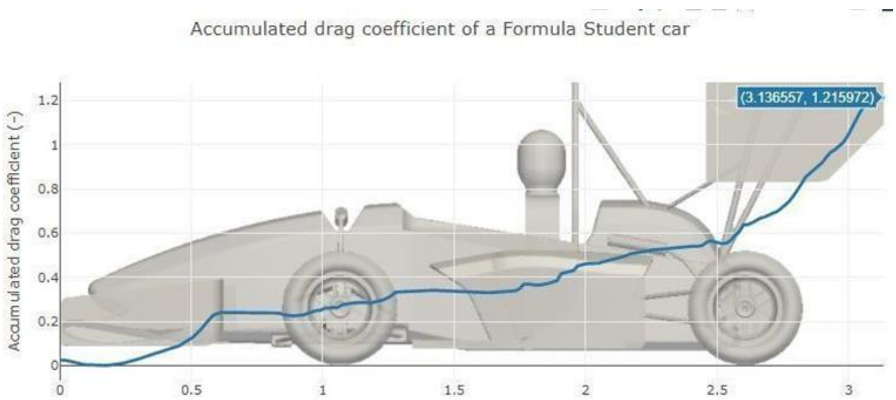

Figure 12: Drag coefficient as a function of vehicle speed -20m/s
Source: Fenandez, 2019.

With this information, it can be said that the drag coefficient values are very close. Some evident factors can justify the coefficient of the Brazilian prototype being smaller, as the wings of the German car are larger in the front and rear, which tends to increase due to friction and pressure drag.

Analyzing downforce and liftforce, in figures 13 and 14, respectively, it is verified that the values are acceptable, concluding that the force pushing the vehicle against the ground is much greater than the ignition force resulting from the air flow through the faces in the lower region the same. Therefore, when the vehicle reaches its maximum speed in a straight line, there is no risk of instability related to the aforementioned forces.

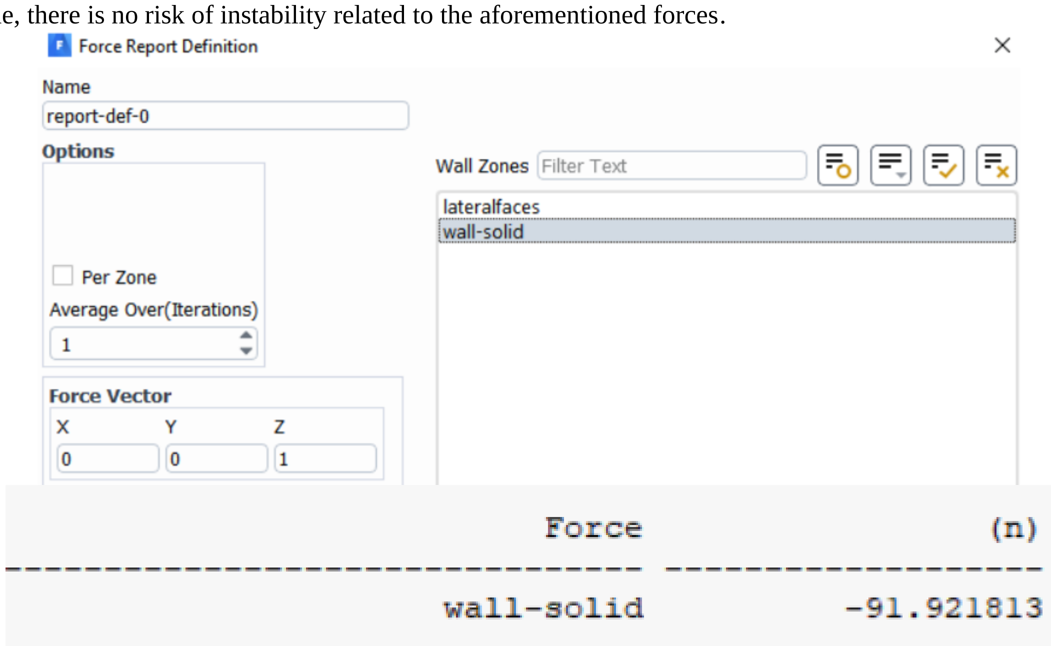

Figure 13: Total downforce resultant.

Lift	(n)
wall-solid	1.7176172

Figure 14: Total liftforce resultant.

6. CONCLUSION

From what has been presented, it is evident that fluid dynamics simulation is an extremely exploitable tool in decision making in aerodynamic projects. From the drag parameter in the validated model, there is reliability in the verification of other parameters such as down and lift force distribution. In addition, it is necessary to think of other possibilities in the geometry of the bodywork and the front and rear wings, making a comparative study so that the solution is converged to an ideal aerodynamic concept for the project.

On the other hand, the study covered was an opportunity to apply knowledge of fluid mechanics in CFD, to make the entire simulation from the input data, boundary condition and creation and editing of the mesh for the continuous medium of interest, definition of the appropriate calculation methodology for the model, up to the analysis of the results at the output, making coherence analyzes and comparing with similar models validated using other methodologies.

However, it is unquestionable that the software used by the engineer must be studied, mainly in relation to the calculation fundamentals explored by the same in the extensions used, so that there is the possibility of selecting the appropriate methodology for each demand, increasing the chances of obtain credible results in the development stages by making the necessary forecasts, verifications and variations so that the direction to an acceptable technical conceptualization occurs according to scope definitions.

7. REFERENCES

Anderson, J. Computation Fluid Dynamics: The Basic with Aplications , McGraw-Hill, 1ª edition, 1995.

Ansys Fluent 12.0 – Theory Guide. ANSYS, Inc, 2009.

Fernandez, P. Formula Student aerodynamics. 2019. Link: https://theansweris27.com. Acess in 28/03/2021 Fox, W., R.; Mc Donald, T.; A.; Introduction to Fluid Mechanics. 8th edition. LTC, 2014

Moreira, A., M. Scientific models, mental models, computational modeling and mathematical modeling: epistemological aspects and implications for teaching. 2013. Tese (Doutorado em Ensino de Ciência e Tecnologia) - Universidade Tecnológica Federal do Paraná, Ponta Grossa, 2013.

Osses, J. The Finite Volume Method. Engineering Simulation and Scientific Software, 2016.

Silva, R. Numerical, analytical and experimental method: Concurrent or complementary in Engineering? Engineering Simulation and Scientific Software, 2017.

Software, S., S., E.; What is the importance of the engineer in computer simulation? Engineering Simulation and Scientific Software, 2016.

White, M., F. Fluid Mechanics. 8th edition. McGraw-Hill, 2018.

8. RESPONSIBILITY NOTICE

The author(s) is (are) the only responsible for the printed material included in this paper.