XLI CILAMCE
XLI Ibero-Latin American Congress on Computational Methods in Engineering
16-19 November, 2020 FOZ DO IGUAÇU PARANÁ - BRAZIL

Aerodynamic analysis of a vehicle's body for the competition Shell Eco-marathon

Carlos E. Maia[1]

[1]*COEME, Federal University of Technology – Paraná*
Av. Prof. Laura Pacheco Bastos, 800, 85053-525, Paraná, Brazil
cadumaia96@gmail.com

Abstract. The Shell Eco-marathon is a competition that challenges students around the world to make a High Efficiency vehicle. One of the characteristics that influences the energy consumption is the body's aerodynamics. The main objective of this work is to choose, for the UTECO team of The Federal University of Technology – Paraná, Guarapuava Campus, the vehicle body for the 2020 edition of Shell Eco-marathon Brazil. First, there is a need to project three possible designs, using Autodesk Inventor software, and after approval of the design by the team, they will undergo a simulation using computational fluid dynamics, with the software ANSYS Fluent. With this, there is going to be an evaluation of the drag coefficient values obtained for the three different models. To simulate the conditions, the results were based on the Reynolds-Averaged Navier-Stokes equations and the turbulent model k-ε, after a statistical analysis, the model with the lowest drag coefficient was chosen. The variables used on ANSYS, such as Temperature and Pressure were based on the City of Rio de Janeiro – RJ and the meshing and other variables on ANSYS Fluent setup were limited by the computational resources available at the time.

Keywords: Simulation, ANSYS, Drag, Fluent, CFD.

1 Introduction

One of the biggest concerns, nowadays, is with the environment, and with higher prices in fuel, primarily in Brazil, people are after more economical vehicles, that uses less energy. And the usage of energy is one of the most researched subjects, one that a lot of universities and companies work on, as one of these companies is Shell. With the competition Shell Eco-marathon, students from all around the world are defied to project and build a vehicle that has the objective of being more efficient. [1]

The competition takes place all over the world, and for people in South America there is the challenger event Shell Eco-marathon Brazil, in which the team UTECO, from the Federal University of Technology – Guarapuava (UTFPR-GP), participates. There are two categories, and one of them is the Urban Concept, that consists in building a small car, similar to common ones, with attributes as two doors, lights, two seats, etc. The category that this team is in, is the Prototype, that has less limitations, and the vehicle's design is optimized for fuel consumption, that means that the only goal is to be the team that consumes less energy among the others, also following the adequate safety protocols.

A lot of characteristics can affect the fuel consumption, such as the tires, engine, transmission, bearings, brakes, and others, basically any force that acts against the movement of the car. The focus of this article, is the aerodynamic, and thus, comes the objective of this work. The team had to build the vehicle's body, around the chassis, that would cover and protect the pilot, and also, had to follow the procedures in the official rules. To carefully analyze the vehicle's aerodynamics, the tool that was used is the software ANSYS Fluent, to obtain the drag coefficient value through CFD. So, the objectives are as follow:

- Design three models, based on other teams' data;
- Run a CFD analysis on ANSYS Fluent;

- Obtain the results of drag coefficient for each model;
- Compare the results and choose the one with lowest drag value;

2 Concepts

When analyzing fluid dynamics, a lot of concepts are used, and amongst all of these concepts, some need to be explained and defined, such as what is fluid flow, the Reynolds number and Aerodynamic Drag:

2.1 Fluid flow

There are two main types of flows that were considered, the laminar and the turbulent. As Potter and Wiggert [2] establish, the first one is a more subtle flow, and the mixture between the particles is minimal, and more stable, and that results in less changes in the way the fluid flows across the surface. When the flow gets more turbulent, in external flows, the boundary layer gets detached, and results in an even more chaotic mixture of the particles in the same fluid. That can change accordingly with the flow velocity, for example, and one way to discover which type of flow is happening, is using the Reynolds number.

2.2 Reynolds Number

A lot of variables affect the transition between laminar and turbulent flow, such as geometry, roughness, velocity and pressure. The Reynolds number, according to Çengel and Cimbala [3], is a dimensionless number that indicates what type of fluid flow is happening, considering the data quoted previously. This number can be obtained by the equation:

$$Re = \frac{\rho V L}{\mu} \tag{1}$$

With ρ being the fluid density, L the characteristic length of the body, V the flow velocity and μ the dynamic viscosity.

2.3 Aerodynamic Drag

Aerodynamics is an area in Fluid Mechanics that focuses in studying the air flow. It has a lot of applications in engineering and separates into internal and external flow. As Anderson [4] stated, the external aerodynamic doesn't involve only geometry, but what happens in the surroundings of the body, such as the difference of pressure in airplane wing and the shockwaves created by supersonic jets breaking the sound barrier.

When the air flows through a body, changes occur in the boundary layer. In some areas, the pressure of the fluid is lowered, because of its geometry. Windlin [5] wrote that this phenomenon slows down the velocity of the flow, until it becomes zero and, theoretically, can even be negative, and that is what is called detachment of the boundary layer, as it can be seen in the Figure 1:

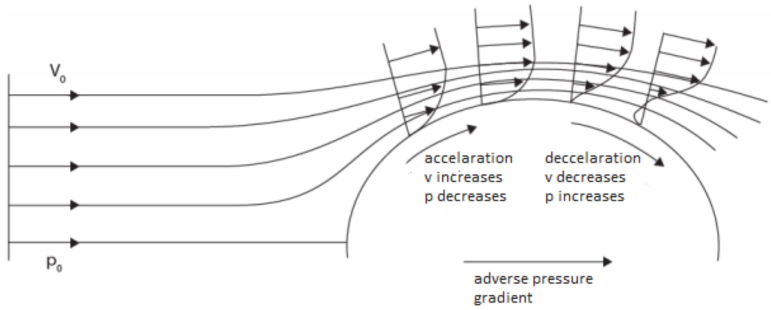

Figure 1. Velocity profile variation in a solid body. Modified from Windlin [5].

Aerodynamic drag is being studied for years, and its main focus was in aviation, according to Ciélinski [6], but when cars started to become faster, the subject was also started to be analyzed in the automotive industry, and the main goal, was to understand and get to the optimal shape that would result in the best movement of air flow around bodies.

As Abo-Serie et al. [7] wrote, is the force resulting from the air flowing past the vehicle, that generates a contrary force that goes against the movement of the body, as it can be seen in the Figure 2:

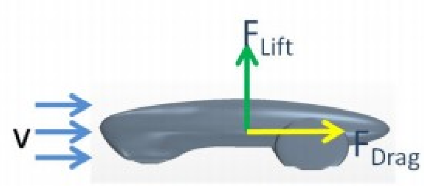

Figure 2. Drag and Lift force directions. [7]

The drag can be divided into components related to various parts of the vehicle, for example, wheel drag, underbody drag, skin friction, cooling drag and many others, and all of them are forces considered when analyzing aerodynamics in CFD, hence the difficulty of doing such analysis by hand, making the use of computers and wind tunnels more common.

The Drag Force, can be obtained, with the following equation:

$$F_D = \frac{1}{2} C_D \rho V^2 A_p \tag{2}$$

F_D being the Drag Force, C_D the Drag Coefficient and A_p the projected area, [6] which can be seen, in the Figure 3:

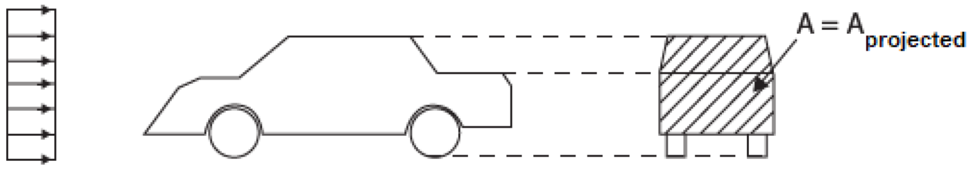

Figure 3. A_p the projected area of an example vehicle. [5]

2.4 Navier-Stokes

Given the complexity of the variables that influence in fluid movement, were created the Navier-Stokes equations. According to Potter and Wiggert [2] it is possible to obtain the results needed, using these equations and some differential ones. However, it still is really hard to get results in turbulent flows, and lots of approximations have to made, making the results less accurate. So, there are two way of analyzing aerodynamics now, using wind tunnels, that means that is a practical experiment, or using Computer Fluid Dynamics (CFD) simulations.

The Navier-Stokes equations are:

$$\begin{aligned} p\frac{du}{dt} &= pg_x + u\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \\ p\frac{dv}{dt} &= pg_y + u\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) \\ p\frac{dw}{dt} &= pg_z + u\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) \end{aligned} \tag{3}$$

With p being the fluid's density, g the gravity and u, v and w the velocity components in the x, y and z axis, respectively. Combined with conservation of mass equation and the system that follows in equations 4 and 5 [8]:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{4}$$

$$\begin{aligned} u &= \bar{u} + u' \\ v &= \bar{v} + v' \\ p &= \bar{p} + p' \end{aligned} \tag{5}$$

With p in Eq. 5 being pressure, we have the Reynolds-Averaged Navier Stokes equation:

$$pg_x - \frac{\partial \bar{p}}{\partial x} + \frac{\partial}{\partial x}\left(\mu \frac{\partial \bar{u}}{\partial x} - \overline{pu^{2'}}\right) + \frac{\partial}{\partial y}\left(\mu \frac{\partial \bar{u}}{\partial y} - \overline{pu^{\bar{\prime}}v'}\right) + \frac{\partial}{\partial z}\left(\mu \frac{\partial \bar{u}}{\partial z} - \overline{pu^{\bar{\prime}}w'}\right) = \rho \frac{d\bar{u}}{dt} \tag{5}$$

This equation is used as base for the turbulence models, as κ-ϵ for example, used in this work.

2.5 Computer Fluid Dynamics (CFD)

The CFD analysis uses designs made in computer, and various. In this work, it was used the concept of finite volumes and 3D drawings. The computer establish cells and meshes along the whole body, simulating the particles of the fluid studied. And then, it uses the Navier-Stokes equations, along with some other calculation methods and flow models, such as Laminar, Inviscid and κ-ϵ turbulent, as stated by Çengel and Cimbala [3].

3 Methods

To design it, the software was used to draw the vehicle's chassis, that was already built, by the 2017 team. With the chassis, using the box tool in Autodesk Inventor software, it was possible to make the bodies around the chassis. With this, the first model was made by simply covering the chassis. The other two followed some examples, based in designs made by other teams.

Figure 4. Models used as inspiration for the Models 2 and 3. (a) BYU Supermileage (CD = 0.107) by Such [8]; (b) Si Pitung G4 (CD = 0.13) by Sirojuddin et al. [9]; (c) Eco-Arrow (CD = 0.096) by Such [8]; Pac-Car II (CD = 0.075) by Such [8].

To make the analysis, the software used was ANSYS 2019 R3, in the setups of the UTFPR-GP, which has the settings: windows 10, intel core i5 processor and 4 GB of RAM. The tool used was Fluent.

The process of export the design form Inventor to ANSYS was simple, and a control volume was established. The body was cut in half, as the car is symmetric, and that would make the meshing more refined. The meshing was made by focusing in Quality and Inflation, that refined the most critical points in the car, and left the parts that the fluid would flow laminar bigger. The number of elements in each model was an average of 507,000,000, due to a limit of 510,000,000 allowed by the software license.

The next step is the setup, and there was chosen the fluid, the velocity, the surface analyzed, and the calculation method. The fluid is air, at 303.16 K and 1 atm of pressure, as it is based on the city of Rio de Janeiro, that hosted the last three editions in Brazil. The velocity of the air used was 30 km/h, an average in the competition, flow from the front of the car to the rear.

The model of flow used, was both Laminar and κ-ϵ turbulent. The laminar was used as a tool to make the value converge easier. To calculate, there were two types of pressure coupling methods, SIMPLE and Coupled. Pressure-Based Segregated algorithm uses a solution with the governing equations being solved sequentially, as they are non-linear and coupled, it is solved iteratively, obtaining a converging numerical solution. The SIMPLE method, is a segregated one, that means that the individual governing equations for the solution variables are solved one after another, hence its name, segregated, and these are memory-efficient, but the convergence is relatively slow. The Coupled algorithm, solves a coupled system of equations, comprising the momentum equations and the pressure-based continuity equation, thus, some steps in the segregated algorithm are replaced by a single one that have a system of equations solved. The convergence velocity is faster, but the memory usage is large [10]. The first one had more chances of converging, but the Coupled is more accurate. And, there was the

The First and Second Order Upwind calculations, basically work with the mesh elements properties, and the difference between First and Second is the amount of data used, with the Second Order using more elements than First. They also have the same attributes as the two items mentioned in this paragraph, the first converged easier and the second was more accurate. And so, a total of eight calculations were made.

As the author got by trial and error, and based on courses that used different methods of calculation, when making this research, the result more accurate was reached by using the combination of properties described in the Figure 5. The chart indicates the order of calculation, starting with a Laminar, using a SIMPLE pressure-based algorithm and First Order Upwind of data, change the Order of calculation, then the algorithm, then flow model. It was considered a Steady-State flow, when the pressure does not change over time.

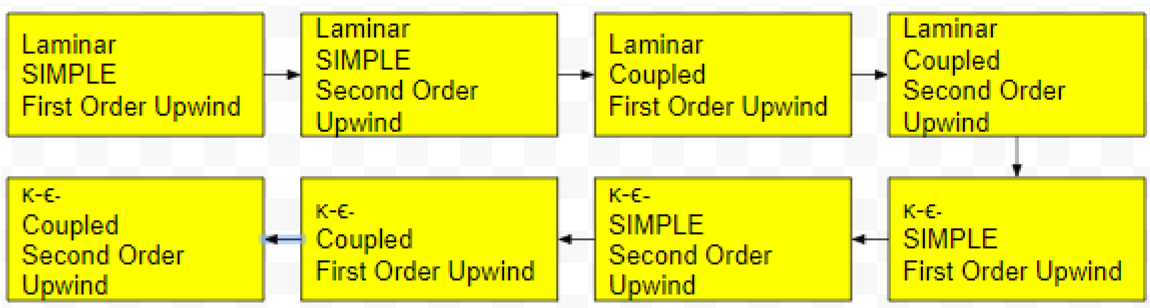

Figure 5. Chart indicating the order of the calculations.

The models' meshes are as follow in the next figure.

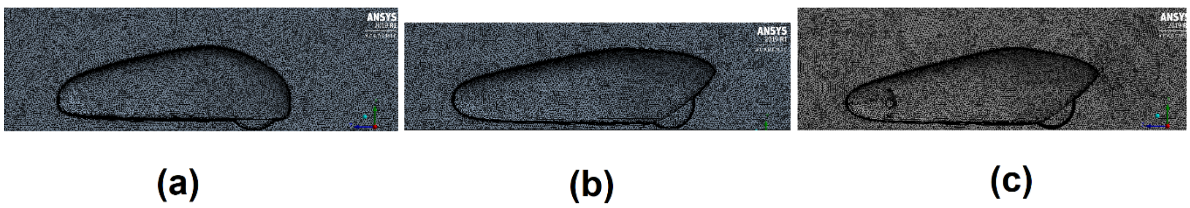

Figure 6. Models' meshes designed by the team. (a), (b) and (c) represents models 1, 2 and 3, respectively.

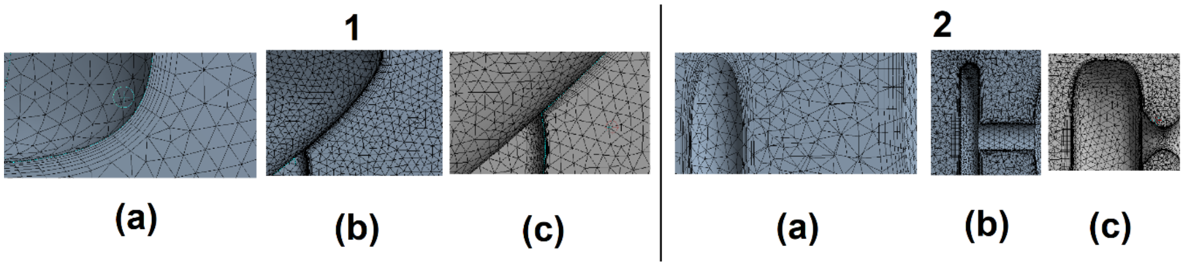

Figure 7. Meshes' details on the rear (1) and on the wheels (2).

4 Results

After running the simulations, and the CFD analysis, the values of the drag coefficient for each model were obtained, and are:

Model	1	2	3
C_D	0.10830	0.10093	0.09999

Table 1. Drag coefficient values obtained in each computer, and the average between the results of the same model.

With C_D, the drag coefficient. As stated in the introduction section, the objective was to find out which of the three models had the lowest drag coefficient value, and choose the one that would be more efficient. Thus, the model that is going to be built is the Model 3 (Fig. 5), which is 0.853% lower than the Model 2 and 7.68% than Model 1.

Out of the four objectives, all of them were completed, they being the 3D draw in CAD, the analysis in ANSYS Fluent, obtaining five results for each model, and choosing the model with an average drag coefficient lower than the other two. All that is left is the final design of the model chosen. And the final version of the Model 3, without the simplifications made for the simulation, with the actual bike wheels and details, are in the Figures 7, 8 and 9:

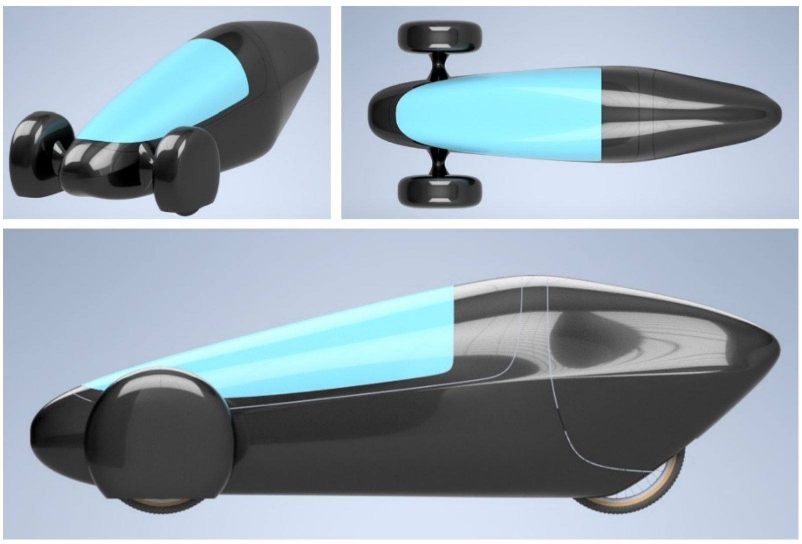

Figure 7. Final Version of Model 3.

5 Conclusion

It can be observed that de geometry of the Model 2 is more aerodynamic than the Model 1, but comparing it to the Model 3, is lower, because of the wheels' protection. This kind of protection is largely used by Shell Eco-marathon teams, as seen in section 3, in the examples. As the results show, the difference of drag value between the first and the second model (0.853 %) is considerably low, however, between the second and the third (7.68%) is significant, showing that slight changes in the design can make a lot of difference when studying aerodynamic drag. And even though there is difference between them, any of the designs could be used without compromising the movement. The method for building the real one is yet to be discussed, as there were delays because of the COVID-19 pandemic.

Acknowledgements. First and foremost, I'd like to thank my parents and siblings, that gave me the support to study and develop myself until this point, because without their support I wouldn't be where I am right now, neither be the engineer I dream to be. Then I wish to thank for the UTFPR, that gave all the infrastructure needed, for the existence of the UTECO team, and for my research, as I couldn't do it without them. Also, my teacher Raquel C. R. da Silva, who guided me through this research, ever since I got the idea, and told me of CILAMCE. And, the Shell Eco-marathon Brazil, that challenges us to work towards de development of new technologies, and allowed me to learn more about engineering and teamwork. Last, but not less important, the UTECO team, that gave me the opportunity to develop myself and helped me make lots of contacts, also teaching the importance of teamwork.

References

[1] Shell. "Make the Future: Shell Eco-Marathon". 2020. Available in: < https://www.makethefuture.shell/en-gb/shell-eco-marathon>, access in October 10 of 2020.
[2] M. C. Potter; D. C. Wiggert. "Mecânica dos Fluidos". Porto Alegre: Bookman, 2018
[3] Y. A. Çengel; J. M. Cimbala. "Mecânica dos Fluidos: fundamentos e aplicações". 3. ed. Porto Alegre: Bookman, 2015.
[4] J. D. Anderson. "Fundamentals of Aerodynamics". 5 ed. New York: McGraw-Hill, 2011.
[5] Fernando Windlin ; Fabio Tanaka ; Kamal Ismail ; Fernando Malvezzi. "Aerodinâmica Veicular". 2012. Available in : <https://www.researchgate.net/publication/277003631_Aerodinamica_Veicular>, access in august 04 of 2020.
[6] A. Cieslinski; W. Prym; M. Stajuda; D. Witkowski. "Investigation on Aerodynamics o Super-Effective Car for Drag Reduction". 2016. Lodz University of Technology Stefanowskiego 1/15, Lódz, Poland. Available in: <http://repozytorium.p.lodz.pl/bitstream/handle/11652/1651/Investigation_aerodynamics_super_Cieslinski_Prym_2016.pdf?sequence=1&isAllowed=y>, access in october 06 of 2020.
[7] E. Abo-Serie; E. Oran; O. Utcu. "Aerodynamics Assessment using CFD for a low drag Shell Eco-Marathon Car". 2017. Yildz Technical University Press, Istanbul, Turkey. Available in: <https://dergipark.org.tr/en/download/article-file/364700>, access in October 06 of 2020.
[8] Matheus Rosa Such. "Análise aerodinâmica de um veículo de eficiência energética". Joinville, 2018.
[9] Sirojuddin; R. Engineu; Wardoyo, (2019). "Aerodynamic Drag Reduction of Vehicle Si Pitung G4 UNJ for Shell Eco-Marathon Asia 2015". 3rd UNJ International Conference on Technical and Vocational Education and Training 2018, KnE Social Science, pages 304–311. DOI 10.18502/kss.v3i12.4096
[10] ANSYS Inc. "ANSYS Fluent 12.0 Theory Guide: 18.1.1 Pressure-Based Solver". 2009. Available in: <https://www.afs.enea.it/project/neptunius/docs/fluent/html/th/node361.htm#:~:text=The%20pressure%2Dbased%20solver%20uses,%2C%20segregated%20from%20one%20another).&text=The%20segregated%20algorithm%20is%20memory,memory%20one%20at%20a%20time.>, access in October 06 of 2020.

Authorship statement. The authors hereby confirm that they are the sole liable persons responsible for the authorship of this work, and that all material that has been herein included as part of the present paper is either the property (and authorship) of the authors, or has the permission of the owners to be included here.