RESEARCH ARTICLE OPEN ACCESS

Aerodynamic Study about an Automotive Vehicle with Capacity for Only One Occupant

Almeida R.A[1], Ferreira Tiago Simão[2], Pedro Américo Almeida Magalhães Júnior[3]

[1] (Rua Das Paineiras, N.12, CEP 3471204-Sabara-MG-Brasil)

[2] (Instituto Federal De Minas Gerais (IFMG), Departamento De Engenharia Mecânica, Av. Av. Michael Pereira De Souza, 3007 - CEP: 36.415-000- Campinho - Congonhas - MG)

[3] Pontifícia Universidade Católica De Minas Gerais, Av. Dom José Gaspar, 500 - Coração Eucarístico - Belo Horizonte - MG - CEP 30535-901 – Brasil)

ABSTRACT

The presented study describes the aerodynamic behavior of a compact, single occupant, automotive vehicle. To optimize the aerodynamic characteristics of this vehicle, a flow dynamics study was conducted using a virtual model. The outer surfaces of the vehicle body were designed using Computer Aided Design (CAD) tools and its aerodynamic performance simulated virtually using Computational Fluid Dynamics (CFD) software. Parameters such as pressure coefficient (Cp), coefficient of friction (Cf) and graphical analysis of the streamlines were used to understand the flow dynamics and propose recommendations aimed at improving the coefficient of drag (Cd). The identification of interaction points between the fluid and the flow structure was the primary focus of study to develop these propositions. The study of phenomena linked to the characteristics of the model presented here, allowed the identification of design features that should be avoided to generate improved aerodynamic performance.

Keywords **-** Automotive Engineering, Aerodynamics, energy efficiency, virtual simulation, Fluid Dynamics, Project.

I. INTRODUCTION

The objective of this paper is to illustrate the virtual analysis performed on a single passenger automobile concept design using CFD and the methods developed to improve the car’s aerodynamic performance.

The focus of the analysis was restricted to friction coefficients related to the down forces. By using CFD and interpreting the dynamic fluid flow simulation results, one can propose the surface changes required on the design to improve its aerodynamic performance.

The performance, driveability, safety and energy efficiency of a vehicle can be significantly affected by the forces acting on a body in a fluid flow regime, in this case air [4,5]. Therefore, it is important to investigate which regions are most subject to such fluid dynamic forces along the longitudinal length of the vehicle and thereby generate design recommendations to favor the desired drag and downforce characteristics.

The proposed compact vehicle, as described, has the capacity for only one occupant, but also must comply with design assumptions such as ergonomics, vehicle driver safety and basic packaging dimensions for chassis, powertrain assembly and other electromechanical systems of a conventional vehicle [2].

To determine the basic dimensions of the project vehicle following requirements were considered:

- Vehicle design aesthetic;
- Internal space of the car;
- Maximum dimensions of the engine compartment to incorporate the powertrain assembly and other basic vehicle systems.

Important to mention, the three-dimensional model of the internal region of the vehicle and electromechanical systems of the vehicle are outside the scope of this study, as they do not directly influence the phenomena to be studied. Those dimensions are therefore omitted. Presented in 2D layouts are the external dimensions of the body in white.

II. BUILDING OF VIRTUAL MODELS

A sketch was created, based on a technical design concept, with the basic dimensions of the vehicle described below (Figure 01). From this, we created a virtual model that was designed in CAD software, NX version 5.0 from Siemens (Figure 2).

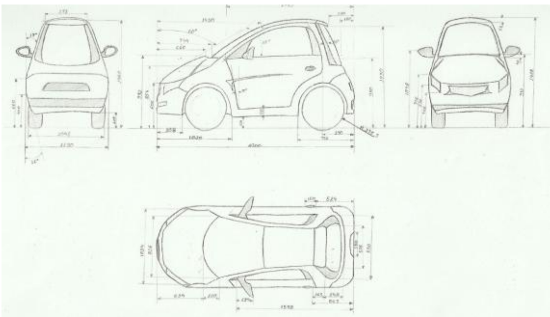

Figure 01 - Sketches containing the basic dimensions

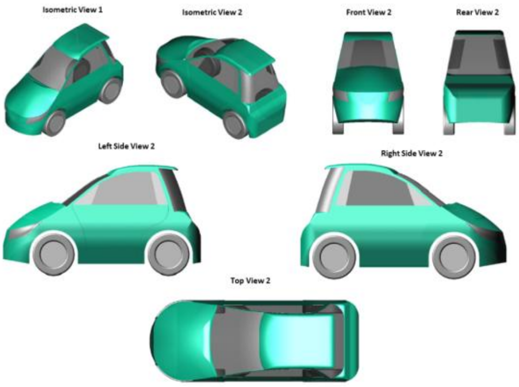

Figure 02 - 3D CAD Model

Once developed, the CAD model was segmented for the construction of a simplified structural mesh, based on the finite element model. The software, Gridgen, was used. Due to the transversal symmetry of the vehicle, we were able to simplify the study and only analyze one half of the CAD model, splitting it along the central median longitudinal plane (Figure 03). The generated mesh had the following characteristics: 63 Domains, 169 Connectors, 102 Nodes, 405 DBs.

For the CFD analysis, VSAERO version 7.1 was used. This software is based on the solution of the simplification of the Navier Stokes equations [9] using the method of panels [1]. This is a widely used method for analysis of three-dimensional aerodynamic bodies with or without lifting.

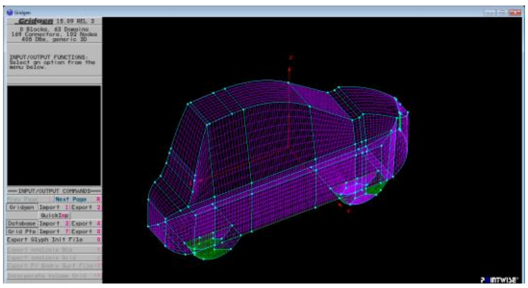

Figure 03 - Structured mesh generated in Gridgen

III. SIMULATIONS AND RESULTS

In this work, we obtained graphs and charts generated by VSAERO which were studied to develop correlations between the fluid surface and aero performance. The software considers air as incompressible fluid, where the Reynolds number used was 3,6x10^3 for an assumed speed of 80 km/h. The characteristic length (L) was set at 2500mm.

In slow flows, the inertia effect is negligible and is inversely proportional to the Reynolds number. Under these conditions, the body shape also has little influence in determining the drag coefficient. Over 45 Km/h aerodynamic drag becomes significant [3].

The following analysis neglects the effects of surface roughness, knowing that surface roughness generally causes an increase in drag coefficient in turbulent flows [3]. One must take care that for some blunt bodies such as a ball, the drag can be reduced with added roughness (if flow is within a specific interval on the Reynolds scale) which create turbulence in the back of the body and decrease the pressure drag. A classic example of this effect is a golf ball with its high roughness. In our context, the automobile body responds to lift or downforce (negative lift), where the contribution of viscous effects are negligible. The lift forces can be considered entirely due to the pressure difference acting on the body, and therefore the shape of the body has great importance [3].

For Turbulent flow (higher Reynolds numbers) the average drag is generated primarily by the pressure drag component, since the Reynolds number is inversely proportional to the viscosity of the fluid, as shown in equation 1. Therefore, with a high Reynolds number, viscosity becomes negligible and shear forces are less significant. For laminar flow (low Re) viscosity exerts enough influence, so the friction drag is most significant.

$$Re = \frac{\rho V D}{\mu} \qquad (1)$$

The graph shown in Figure 4 is the pressure drag behavior along a centerline section of the vehicle, which is the basis of our evaluation and analysis of the longitudinal flow. The white graph curve represents the vehicle profile and the magenta curve represents the corresponding pressure coefficient along the section of the vehicle.

Pressure drag is proportional to the frontal area and the difference in pressures acting on the front and rear region of a body. Pressure drag is the predominant force acting on blunt bodies, as can be seen in equation 2. The pressure coefficient (Cp) is a dimensionless number that describes the relative pressure through a flow field in fluid dynamics. The drag pressure becomes more significant when the fluid velocity is very high, since the flow has difficulty following the curvature of the body and consequently separates, similarly generating a low pressure region below the streamlines.

$$P = \frac{1}{2}\, C_P\, \rho\, V\, A^2 \tag{2}$$

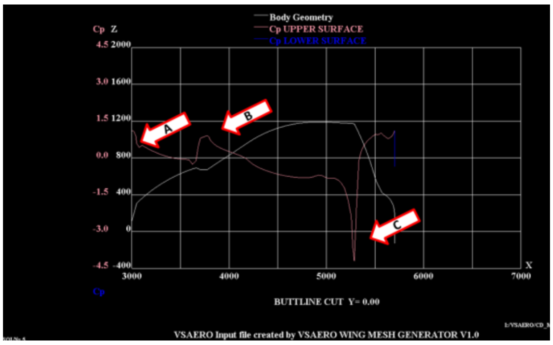

Figure 4 - Graph of the pressure coefficient (Cp) as a function of the average profile model

It evaluated in principle the frontal division between the hood and the bumper. Referencing the graph in Figure 4, there was a sharp drop in the pressure coefficient (Cp) in the area indicated by ARROW A. This behavior causes an undesirable impact on the coefficient of drag. Additionally, flow separation causes the formation of portions of disengaged fluid in rotation, called vortices. The vibrations generated by a vortex near the vehicle may cause resonance if the frequency of the vortex is close to the natural frequency of the vehicle. But at a minimum, it may create an undesirable noise input for the occupant.

A high coefficient of pressure is observed at the frontal region of the vehicle (the bumper). This occurs because a significant area is subjected to fluid resistance force to the flow thereby creating a typical high pressure that region. As can be seen in Equation 2, the pressure (P) is directly proportional to the pressure coefficient (Cp), and that in this region, the flow force is primarily generated by the pressure component.

Figure 5 shows a diagram which represents the variation of the friction coefficient or attrition (C_f), over the the streamlines. It was observed that this region has not occurred the lack of interaction between the fluid and the structure of the boundary layer, which is positive. The flow separation region may also be called a mat and is typically downstream, characterized by low pressures and reduced velocity of the fluid. In this region the viscous and rotational effects (vortex) are more significant.

Also in figure 5, it is shown that variation is occurring in the laminar flow regime as it changes to turbulent flow, identified by changing color of the streamlines. The magenta color represents a low friction coefficient and indicates a laminar flow regime. The green color represents higher C_f and the beginning of a transition process flow. The flow then begins to gradually stabilize into a turbulent profile represented by the color red. This turbulent flow extends along the longitudinal surfaces, and is the longitudinal flow equilibrium condition of the vehicle.

It was observed that the flow regime of variation could be minimized by increasing the fillet radius (or a parabolic shape) between the surfaces defining the hood and the bumper. Typically, rounded forms have better drag coefficient than the acute forms [3]. But in applications in commercial automotive projects, it is always necessary to evaluate all of the variables involved, such as style requirements, hood stamping feasibility or the bumper injection mold, before opting in favor of an aerodynamic improvement.

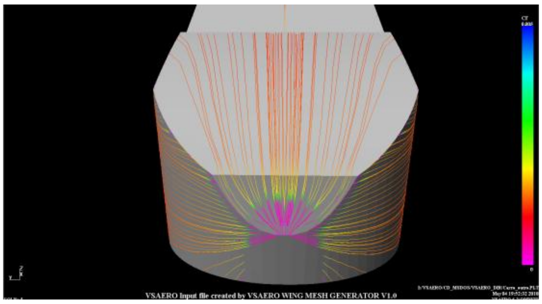
Figure 5 - Diagram of the variation coefficient of friction as a function of the streamlines

The fairing is a good alternative to reduce the pressure drag which occurs due to layer separation flow limit. The fairing help create a continuity of the curvature of the profile body reducing the pressure difference between the front and rear region of the vehicle, but it adds frictional drag because of the increased surface. Important to evaluate the predominant effect in order to try to reduce to the maximum the sum of the two effects.

The fairing also provides a reduction of vibration and noise (decreased turbulent profile). They are useful for blunt bodies subjected to flow at high speeds (higher Reynolds numbers) for which flow separation is more significant. At low speeds the drag is almost entirely frictional [3,4]. This explains why the application of a fairing is more common on a high performance sports car. In low-speed vehicles, cowling may possibly only increase the contact surface and frictional drag. An unnecessary or poorly designed fairing may also actually increase drag rather than decrease it.

Our next area of focus was the region between the hood and the windshield. Typically, in most vehicles this region filled with a plastic panel. Taking the graph of figure 4 again, in the region indicated by ARROW B, a slight pressure drop is observed at the hood's trailing edge, accompanied by a dramatic increase of pressure in the region aft. This can be explained by the sudden change of the profile geometry. Thus taken up a new diagram shown in Figure 6 has been reported that the behavior of the streamlines from the pressure coefficient ranging from a range in which the color magenta is a negative -2.69 Cp a blue color with Cp 1.02.

The existence of a breakdown or detachment between the fluid and the body was observed. The pressure dropped abruptly and remained constant until a new interaction between the fluid and the structure was established. In the case of our model, the inclined windshield provided this next interaction surface. Between these two interaction points was a recirculation 'bubble'. This behavior can be more clearly seen in the diagram in Figure 6.

It is important to recognize that this interaction between the fluid and the flow structure is a factor that impairs the drag coefficient, resulting from the pressure variation that causes drag force. While it is something undesirable for our project vehicle, it is also something typically minimized in commercial designs.

To improve this design condition, it would be suggested to minimize the variation of Cp by obtaining a more uniform geometry in the region and avoid a sudden change of profile. This can be done by modeling a plastic close-out panel at the base of the windshield to bridge between the two surfaces, providing better tangency with the hood and a transition to the inclined angle of the windshield. An example vehicle application can be seen in the FIAT Idea, as shown in Figure 7.

Also assessed, was the fillet radius between the side surfaces and the windshield, technically known in the industry as the A-Pillar. It was observed in Figure 6, there was a sharp pressure drop in this area (low Cp), identified by a red color region. For the streamlines to follow the body profile an increased acceleration of the flow is necessary. The peak pressure can generate a lack of interaction between the fluid and the structure of the flow, and presents a problem resulting from the suction force which can occur in this place.

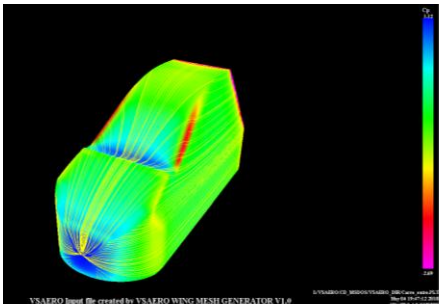

Figure 6. Diagram of the streamlines after the interaction between the fluid and the flow structure on the windshield

Figure 7 - Frame windshield (FIAT Idea)

Another project suggestion would be to increase the fillet radius (or a parabolic shape) between these surfaces to soften the pressure gradient. Figure 8 shows a commercial example, seen on the FIAT Doblo, where the door profile has a rather sharp radius.

Finally, the rear region of the vehicle presented what may be the most critical conditions in terms of pressure change, as shown by arrow C in the graph of Figure 4. The entire blue region of Figure 9 indicates a lack of interaction between the fluid and the flow structure, creating a region of very low pressure. This is also clearly evidenced by the graph of Figure 4. This negative pressure causes an undesirable suction force, acting in the opposite direction to the displacement of the vehicle, causing a loss of dynamic performance and other consequences such as increased fuel consumption. This is a common phenomenon, particularly in hatchback or SUV vehicle segment, for example.

Figure 8 - Fiat Doblo's side door

The separation position on the surface depends on several factors beyond the flow speed, such as Reynolds number, surface roughness and the level of free flow fluctuations. It is difficult to determine the exact position unless there are sharp corners or abrupt changes in the geometric surface, as is the present case.

One possible suggestion to minimize the effect of this phenomenon is to add a roof top spoiler (deflector air) in the rear region to increase the area of interaction between the fluid and the structure, and to promote turbulence at the end thereof. This would create a more desirable situation by creating interaction between the fluid and the turbulent flow structures on the posterior surface. The presence of a small radius at the edge of the spoiler would favor the effect described above. It is advisable also to avoid sharp edges along its surface, as these would be detrimental to the interaction between the fluid and the structure with level variations which might have important consequences in the spoiler's flow. A vehicle example of a spoiler with the suggested characteristics is applied to the Range Rover Sport SE (Figure 10).

For the proposed vehicle design, it is concluded that it would not be a good alternative to use of a spoiler. In the case of the spoiler, it is interesting to create downforce effect which is responsible for generating a resultant downward force and acting contrary, for example, at the time of pitching. Put random parts without assessing the overall result may not have the desired effect.

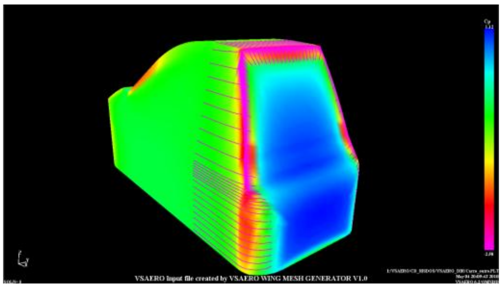

Figure 9- Diagram of the pressure coefficient (Cp) of the rear part of the vehicle.

Figure 10- Range Rover Sport SE Spoiler's

Considering the car profile as an airfoil, if the stagnation point occurs before the trailing edge (rear corner), the underflow lines try to divert to occupy the freed space creating a turbulent counterclockwise vortex tending to add to the underflow and subtract from the upperflow. This cannot be achieved, however, when the flow speed is relatively high, and consequently the flow detaches. As a result, the stagnation point moves to the trailing edge, as can be seen in Figure 11. At the Bernoulli's beginning again, there is a increase in the flow speed below and decreased pressure. The reverse occurs at the bottom, with the resultant force of pressure is vertically pointing down (Downforce). On the other hand, the airfoil greatly increases aerodynamic drag. When the angle is perpendicular to the horizontal, speed in a straight line is privileged [4]. When there is a change of angle, the greater the angle, the more drag, and less speed.

The Bernoulli equation is used to mathematically express the downforce and says that an increase in flow speed causes a decrease in pressure [3], as seen in Equation 3. The pressure is substantially independent of surface roughness, because the surface roughness affects mainly shear forces, not pressure.

$$P_1 + \rho g H_1 + \frac{1}{2}\rho V_1^{\,2} = P_2 + \rho g H_2 + \frac{1}{2}\rho V_2^{\,2} \qquad (3)$$

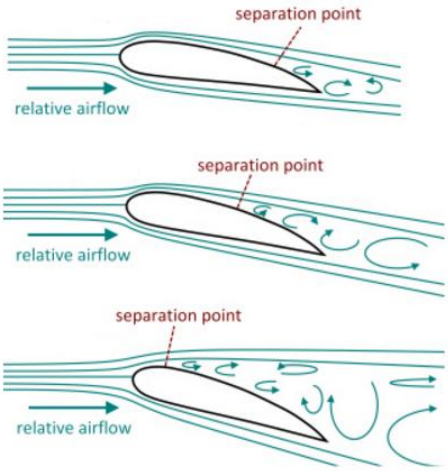

Figure 11- Irrotational flow around spoilers 2D

IV. CONCLUSIONS

This study does not, nor was it intended to, provide quantitative results on the aerodynamic coefficient of drag and lift. It is also known that, whether in the automotive or aerospace industry, the numerical models of CFD are not yet sufficient to fully validate a project. The complexity of the boundary conditions and the subsequent development of mathematical models to cover all numeric variables is not possible and consequently does not allow us to achieve sufficiently accurate results with only virtual simulations.

From a vehicle dynamics perspective of the vehicle concept described in the study, we realize the dimensions of the body design have a strong influence on its dynamic performance. To quantitatively determine the drag and lift coefficients, experimental testing is needed in a wind tunnel with physical scale models. This is suggested as a proposal for future work.

It is concluded however, that the study presented here provides models with significant approximations. The power coefficient graphs and friction coefficient obtained by VSAERO assist in understanding the dynamic behavior of the forces in flow along the vehicle profile. The results were extremely valid for the identification of potential design improvement regions. Thus, the proposed changes presented here should help develop a more robust design that allows the optimization of aerodynamic behavior such as handling and stability. The optimization of the energy efficiency of the product is another important factor, since the solutions shown here certainly favor the reduction in aerodynamic pressure, thus reducing power demand (or engine power) to move the car, and therefore reducing vehicle fuel consumption.

In addition to this work and, also to support the proposals made over the item "Simulations and Results", would be interesting a new virtual analysis looping. But the main objective of this work, which is to study the flow characteristics is to propose solutions that drive for good dynamic performance, was achieved.

As discussed also, in order to be essential a physical experimental validation for commercial application of the proposed model and also, due to qualitative interpretation of power and friction coefficients, it was observed that the simplifications made to meshing shown under "Construction of models virtual "did not have significant influence within the main purpose of this work.

This study also demonstrates in general, why CFD methods have become increasingly indispensable in modern engineering to design better projects, making this a practice with a lot of application within the manufacturers of automotive vehicles during product development.

LITERATURE REFERENCES

[1]. ANDERSON, J. D., 1991, Fundamentals of Aerodynamics, McGraw-Hill, 2ª Ed.
[2]. BOSCH, 2005, Manual de Tecnologia Automotiva; Edgard Blücher, 25ª Ed.
[3]. ÇENGEL,Y.A., Mecânica dos Fluidos, Fundamentos e aplicações; 2007. Mc Graw Hill, 1st Ed.
[4]. HUCHO, W.-H. 1998, Aerodynamics of Road Vehicles; SAE International, USA, 4ª Ed.
[5]. KAMAL A.R I. 2007, Aerodinâmica Veicular; Editora. KAMAL,
[6]. KATZ, J. 1995, Race Car Aerodynamics; Designing for Speed. Massachusetts. Bentley Publishers, 2.ed.
[7]. PRANDTL,L. 1952 , Essentials of Fluid Dynamics; Hafner publications, New York.
[8]. PRANDTL,L. 1930, The Physics of Solids and Fluids: With Recent Developments; Blackie and Son.
[9]. STOKES, C.G. 1845, On The Theory of the internal friction of fluids in motion and of the equilibrium an motion of elastic solids; vol.8-287.Cambridge Phil.Society, England.
[10]. STREETER, V.L. 1966 , Fluid Mechanics; McGraw–Hill Inc., New York.

Sites:
Figure 11 on http://www.woseba.de/08_stall.html (accessed on March 03, 2016)