DESIGN ANALYSIS IMPROVEMENTS APPLIED TO A FORMULA SAE RACING CAR AERODYNAMIC DEVELOPMENT

Luiz Antonio Negro Martin Lopez
FEI University Center, Mechanical Engineering Department
Av. Humberto A. C. Branco, 3972 – São Bernardo do Campo – SP – Brazil – 09850-901
Tel# 55-11-43532900 r. 2172 – *luizlope@fei.edu.br*

Frederico Augusto Alem Barbieri
FEI University Center, Mechanical Engineering Department
Av. Humberto A. C. Branco, 3972 – São Bernardo do Campo – SP – Brazil – 09850-901
Tel# 55-11- 43532900 r. 2143 – *fabarbieri@fei.edu.br*

Jean Tavares Horcaio
FEI University Center, Mechanical Engineering Department
Av. Humberto A. C. Branco, 3972 – São Bernardo do Campo – SP – Brazil – 09850-901
Tel# 55-11- 43532900 r. 2143 – *jyzf@uol.com.br*

Abstract. *The aerodynamic features which occur in a Formula race car are complex and difficult to evaluate. Analytical methods use fluid dynamics and thermodynamics relations which are most of the times imprecise. Wind tunnel experimentation is made to overcome this limitation, but this method is expensive and time demanding. Nowadays Computational Fluid Dynamics (CFD) has been extensively used as a quick, cheap and flexible tool for aerodynamic behaviour prediction. Nonetheless, CFD results shall be validated. In this work, ANSYS CFX software is used to evaluate the aerodynamic behaviour of a Formula SAE racing car. Aerodynamic lift and drag forces as well as aerodynamic flow paths are evaluated in order to reduce drag and to increase down forces for better vehicle performance. The simulations are carried out in 3-D models of the vehicle body. The software results are compared with experimental results obtained in a reduced scale wind tunnel.*

***Keywords**: Design, Formula SAE, Aerodynamics, CFD*

1. INTRODUCTION

The Formula SAE is a competition among graduate and undergraduate university teams held in five countries all over the world (Australia, Brazil, Italy, United Kingdom and USA). Nowadays, FEI University Center is the Brazilian leader team. The objective of this competition is to award the best race car design, judging the fabrication and performance results for a small monoplace formula style racing car. There are several rules that must be followed in the competition, but there are not so many restrictions on the aerodynamic issue, so that creativity and imagination in body styling can be stimulated.

In recent Formula SAE aerodynamic design, a lot of "amazing" solutions are presented. Frequently, the reason of such solutions is the misunderstanding of aerodynamic phenomenon, making the car a mutant moving laboratory.

In despite of the field tests seem to be a good approach because they are made under real boundary conditions, they are not appropriate for design development because environment conditions change continuously. In this way, the boundary conditions are disordered and cause-effect relations are difficult to analyse. Furthermore, speaking economically, field tests are time demanding and quite expensive.

The traditional techniques for aerodynamic drag and lift forces evaluation used in a race car design combine analytical and empirical methods, which are most of times imprecise for performance analysis.

To compensate this limitation, reduced scale models are used but they also present many limitations due to similarity problems. As a result of this, the most reliable manner to verify the design is the full-scale laboratory experimentation by using climatic wind tunnels which simulate, under controlled laboratory conditions, the actual environmental conditions that a race car is subjected to. However full-scale tests are very expensive due to wind tunnel dimensions (Hucho, 1998).

To overcome the above mentioned problems, Computational Fluid Dynamics (CFD) has been extensively used as a quick, cheap and flexible tool to predict the thermodynamic and fluid dynamic behavior of streams around a race car enhancing the aerodynamic design (Anderson, 1995). However, the experimental verification is still the only way to validate a numerical simulation model. Therefore, the complete domain of aerodynamic design techniques requires the theoretical knowledge of transport phenomenon, experimental stands and computer aided simulation.

In this work, CFD aerodynamic development is made through ANSYS CFX software allowing the aerodynamic behavior evaluation of a Formula SAE racing car. Furthermore, for the sake of economy, a 1:5 reduced scale model is used to experimentally simulate the car performance in a laboratory reduced scale wind tunnel.

Experimental and computational simulation results are obtained allowing a future comparative work between CFD results and a full-scale model experimental results.

2. AERODYNAMIC ANALYTICAL AND EMPIRICAL EVALUATION

The analytical and empirical evaluation of the aerodynamic behavior of a Formula SAE racing car is based on the drag (D) and lift (L) forces determination. Those forces are the result of air shearing stresses acting over car areas, and their behaviors must be obtained.

The local frictional shearing stress due to the air flow along the car surface is given by the well known Newton's law of friction:

$$\tau_{sx} = \mu \frac{\partial u}{\partial y} \tag{1}$$

where the subscript x denotes the fluid flow direction parallel to the surface and y is the direction normal to it. The absolute air viscosity μ is the proportionality fluid property between the shearing stress and the velocity gradient that is normal to the surface and is given by $\partial u / \partial y$ (Schlichting, 1968). The term u is the air velocity component parallel to the surface. For the sake of exemplification, the velocity gradient in the laminar boundary layer along a flat plate (Kreith, 2003) is:

$$\frac{\partial u}{\partial y} = 0.332 \frac{u_\infty}{x} \sqrt{Re_x} \tag{2}$$

where u_∞ is the air velocity in the free airstream out of surface influence and parallel to it, x is the distance from the surface leading edge and Re_x is the Reynolds number at x position that is given by:

$$Re_x = \frac{\rho\, u_\infty\, x}{\mu} \tag{3}$$

In the Reynolds number equation, ρ is the air density. Substituting Eq. (2) into Eq. (1) result:

$$\tau_{sx} = 0.332 \frac{\mu\, u_\infty}{x} \sqrt{Re_x} \tag{4}$$

The local dimensionless coefficient for drag is defined as:

$$C_{Dx} = \frac{\tau_{sx}}{P_d} \tag{5}$$

where P_d is the dynamic head (Ieno and Negro, 2004) that is given by:

$$P_d = \frac{1}{2} \rho\, u_\infty^{\,2} \tag{6}$$

Therefore, substituting Eq. (4) and Eq. (6) into Eq. (5), becomes after simplification:

$$C_{Dx} = \frac{0.664}{\sqrt{Re_x}} \tag{7}$$

Equation (7) presents the local drag coefficient at position x. The drag coefficient along a distance L is:

$$C_D = \frac{1}{L} \int_0^L C_{Dx}\, dx \tag{8}$$

Substituting Eq. (7) into Eq. (8) and integrating we obtain:

$$C_D = \frac{1.328}{\sqrt{Re_L}} \tag{9}$$

where Re_L is the Reynolds number calculated for a length L.

The total drag force acting on the surface is given by:

$$D = C_D \, P_d \, A \tag{10}$$

where A is the surface area. Finally, by substituting Eq. (6) into Eq. (10), becomes:

$$D = C_D \, \frac{1}{2} \rho \, u_\infty^{\,2} A \tag{11}$$

A similar equation is used for the total lift force:

$$L = C_L \, \frac{1}{2} \rho \, u_\infty^{\,2} A \tag{12}$$

where C_L is the lift coefficient.

The above presented analytical aerodynamic approach is quite limited to be applied to the several shapes and positions of car surfaces. To overcome this limitation, a shortcut is used through the lift and drag coefficients experimental determination.

The drag coefficient C_D is scheduled for standard shapes as sphere, cylinder, plate, etc. (Brandt, 1999). The lift coefficient C_L is frequently presented together with the drag coefficient C_D for airfoils as a function of attach angle. By considering that those coefficients do not exist for a particular aerodynamic Formula SAE racing car design, lift forces and drag forces are obtained experimentally and through CFD as presented below.

3. REDUCED SCALE EXPERIMENTATION

The Formula SAE racing car aerodynamic design requires the experimental determination of the above presented lift and drag forces (Badih *et al*., 2001). To achieve this objective, a geometrically similar reduced scale car model was made in the 1:5 scale as shown in Fig. 1 and mounted inside the reduced scale wind tunnel as shown in Fig. 2. The dimensions of the reduced scale model are 0.566 m length, 0.295 m width and 0.235 m height. The full scale Formula SAE racing car is 5 times bigger.

The reduced scale wind tunnel allows air velocities from 0 to 20 m/s. The drag and lift aerodynamic forces were obtained through strain devices as presented in Fig. 3.

Reduced scale wind tunnels present as advantages low acquisition and operational costs as well as small area stand. Furthermore, reduced scale car models are cheap and easy to be constructed and modified. However, reduced scale models have the disadvantage of geometric and cinematic similarity losses. Geometric similarity loss is due to construction difficulties of the reduced scale model details.

Cinematic similarity loss occurs when viscous forces and inertia forces do not present the same relation in the reduced scale model and in the full scale car. The non-dimensional Reynolds number relates those forces and it is used for the air velocity scale model determination. Empirical aerodynamic data were obtained by performing similarity analysis equating the non-dimensional numbers associated to the flow processes.

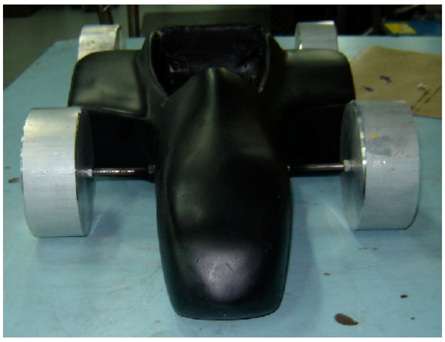

Figure 1. Reduced scale car

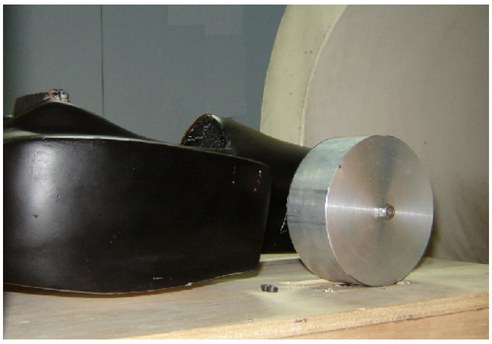

Figure 2. Reduced scale wind tunnel

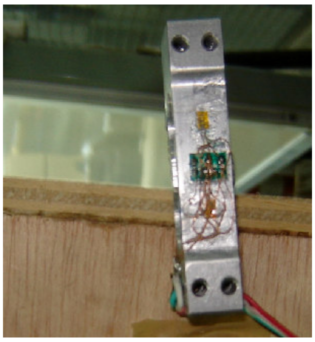

Figure 3. Force measuring strain device

Taking into account that the reduced scale model and the full scale racing car are geometrically similar,

$$Re_r = Re_f \tag{13}$$

where Re_r and Re_f are, respectively the Reynolds numbers of the reduced scale model and of the full scale car.

Equating Reynolds numbers of the reduced scale model and of the full scale car, result:

$$\frac{\rho_r \; u_{r\infty} \, L_r}{\mu_r} = \frac{\rho_f \; u_{f\infty} \, L_f}{\mu_f} \tag{14}$$

where L is the car length and,

$$L_r = \frac{L_f}{s} \tag{15}$$

and s assumes the value 5 at the 1:5 scale.

Therefore, by substituting Eq. (15) in Eq. (14) and by considering that the air viscosity and the air density are the same in the reduced scale model and in the full scale car, it follows that:

$$u_{r\infty} = s \, u_{f\infty} \tag{16}$$

According to Eq. (16), the air velocity in the reduced scale wind tunnel must be 5 times the air velocity in the full scale car. Hence, the use of reduced scale models usually requires high reduced wind tunnel air velocities. Due to wind tunnel air velocity limitation, cinematic similarity loss can occur.

To overcome this limitation, a turbulent air flow was imposed in this work, with Reynolds number bigger than 10^5. Consequently, the viscous effects can be ignored prevailing only the inertia forces and allowing the reduced scale wind tunnel use. In this work, the similarity analysis was made by using the non-dimensional Froude number (Schlichting, 1968), presented in Eq. (17) instead of the Reynolds number.

$$Fr = \frac{u_\infty}{\sqrt{g \, L}} \tag{17}$$

where u_∞ is the air velocity in the free air stream, g value is 9,81 m/s^2 and L is the length of the car.

4. COMPUTATIONAL FLUID DYNAMICS

The reduced scale wind tunnel and the reduced scale Formula SAE racing car geometries were used to generate a virtual reduced model by using the ANSYS CFX software.

The computational simulation tool (Maliska, 1995) uses CVFEM (Control Volume Finite Element Method) to solve the well known differential three-dimensional fluid mechanics Navier-Stokes equations of motion for flows with friction and the equation of continuity for the prevision of the air stream behaviour around the racing car.

In this work it was considered the following boundary conditions: incompressible air flow, steady state conditions, constant air viscosity, constant air temperature and despicable air weigh. Therefore, the three equations of motion in the x, y and z directions (conservation of momentum) are simplified to:

$$\rho\left(u\frac{\partial u}{\partial x} + v\frac{\partial u}{\partial y} + w\frac{\partial u}{\partial z}\right) = \mu\left(\frac{\partial^2 u}{\partial x^2} + \frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) - \frac{\partial p}{\partial x} \tag{18}$$

$$\rho\left(u\frac{\partial v}{\partial x} + v\frac{\partial v}{\partial y} + w\frac{\partial v}{\partial z}\right) = \mu\left(\frac{\partial^2 v}{\partial x^2} + \frac{\partial^2 v}{\partial y^2} + \frac{\partial^2 v}{\partial z^2}\right) - \frac{\partial p}{\partial y} \tag{19}$$

$$\rho\left(u\frac{\partial w}{\partial x} + v\frac{\partial w}{\partial y} + w\frac{\partial w}{\partial z}\right) = \mu\left(\frac{\partial^2 w}{\partial x^2} + \frac{\partial^2 w}{\partial y^2} + \frac{\partial^2 w}{\partial z^2}\right) - \frac{\partial p}{\partial z} \tag{20}$$

The equation of continuity (conservation of mass) is simplified to:

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{21}$$

In the above equations, u, v and w are respectively the air velocity components in the longitudinal (x), transversal (y) and vertical (z) directions as presented in Fig. 4. The air density is ρ, the air absolute viscosity is μ and the air pressure is p.

A half portion of the computational car and wind tunnel domains are presented in Fig. 4 and Fig. 5 .

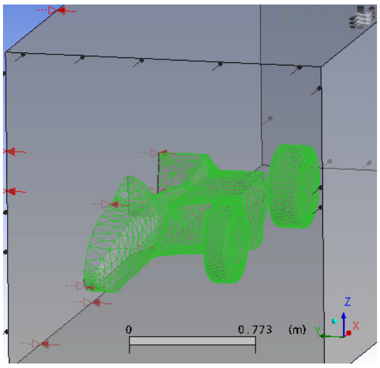

Figure 4. Virtual reduced scale racing car

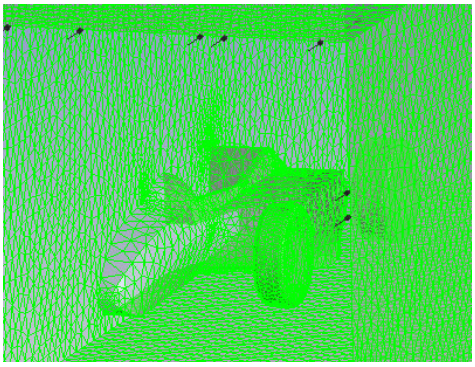

Figure 5. Virtual reduced scale wind tunnel

The CVFEM was applied for a tri-dimensional tetrahedral elements mesh (Anderson, 1995). Near the surface boundaries, smaller mesh length scales were used providing better knowledge of the velocity and pressure gradients normal to the surface. The consequent better resolution was particularly important for lift and drag forces determination. The mesh contains around 8 x 10^5 elements as a result of convergence model analysis. The mathematical treatment of turbulence was the traditional k-epsilon model.

5. RESULTS AND DISCUSSION

Process parameters such as pressure and air speed of the laboratory stand were used as input data for software using. The air velocity was selected taking into account the scaling similarity conditions previously described in Section 3.

Figure 6 presents a typical air pressure distribution CFD result of the air flow surrounding the Formula SAE racing car. In this simulation, the air pressure stream varies from approximately 540 Pa (positive) in the nose and frontal car areas to 1,370 Pa (negative) near to the tires centerline horizontal plan trailing edge. For the sake of computational time reducing, the tires air pressure stream effect was not considered in this example.

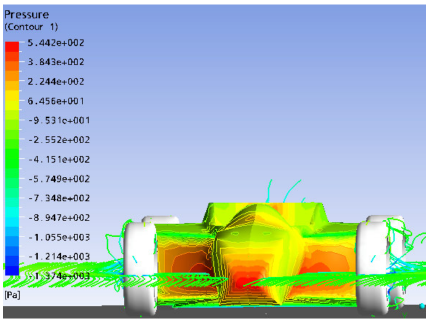

Figure 6. Air pressure distribution

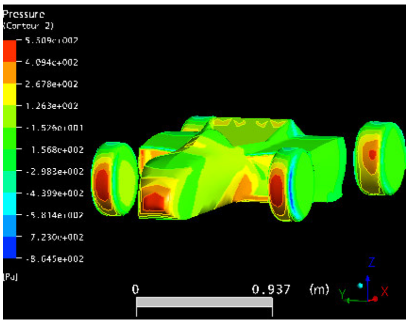

Figure 7. Surface pressure distribution

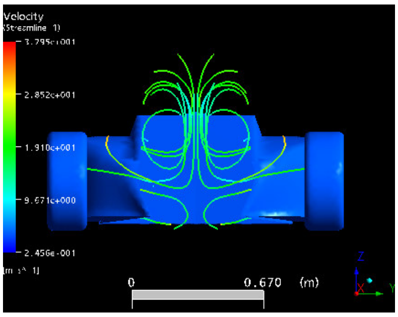

Figure 8. Air speed stream lines

The surface pressure acting over all car areas, including the tires, is presented in Fig. 7. This result allows the lift and drag forces determination as shown in Fig. 9 and Fig. 10. In this work the tires were considered as non-rotating cylinders as well as the floor was considered as being stopped. This simplification is a consequence of experimental difficulties which causes some similarity losses between reduced car simulations and full scale car real behavior.

Figure 8 shows air speed streamline transverse paths. The transverse flow generates longitudinal vortices which creates lift forces. These results allow the correct location of aerodynamic surfaces which reduces lift and drag forces. In this typical simulation result, the air velocity reaches approximately 30 m/s in the upper rear surface while the air velocity far away from the reduced scale car is 20 m/s.

The experimental lift and drag forces were obtained by using strain gage measurements devices as shown in Fig. 3. Those devices were properly installed in the reduced scale car and in the reduced scale wind tunnel. The drag and lift forces CFD results were compared against the experimental results.

The graph in Fig. 9 shows the drag force results obtained for the reduced scale 1:5 model, against the CFD results of the corresponding virtual reduced scale car.

Drag forces were obtained for air velocities varying from 2 m/s to 20 m/s. Although it seems to be a low speed for a Formula race car, it is important to note that the average speed for the race track is around 17 m/s. In the same way, Fig. 10 presents the lift force results against wind tunnel air velocities. The negative lift force values reveal that, in fact, there are down forces. Initial results revealed large instrumentation errors up to ± 0.5 N while the maximum measured forces were around 5N. Further instrumentation accuracy upgrade allowed the reduction of errors to acceptable values around ± 0.1 N.

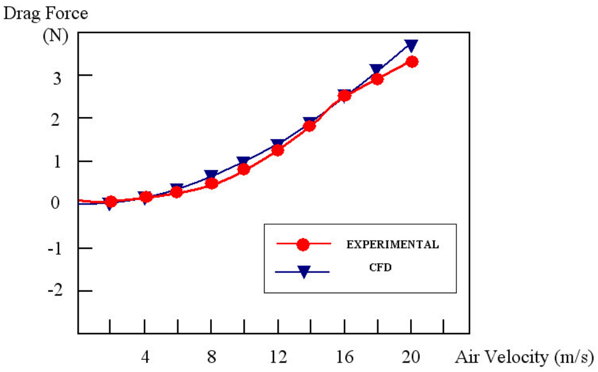

Figure 9. Experimental versus CFD drag force

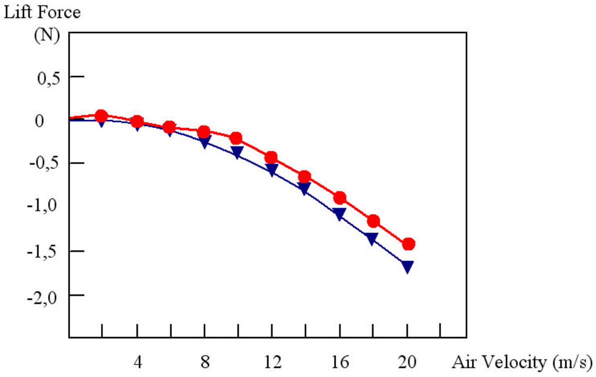

Figure 10. Experimental versus CFD lift force

6. CONCLUSIONS

The validation of aerodynamic studies made by CFD of a Formula SAE racing car is an expensive and time demanding challenge if full scale car and full scale wind tunnel are used. Field tests present many difficulties for aerodynamic development due to environmental conditions continuous changing. Reduced scale computational simulation with the corresponding reduced scale experimentation is a promising development field because it is relatively cheap and fast.

In this work a comparative study between reduced scale experimental and computational lift and drag forces is presented. The results show that the mathematical models are appropriated for a proper simulation.

The reduced dimensions of the wind tunnel imposed larger differences between experimental and CFD results at high experimental air velocities due to similarity losses in the experimental stand. The higher the relation between the full scale and the reduced scale car, the greater the similarity loss. To overcome this limitation, a future work using a larger wind tunnel (1:4 maximum scale) and a more accurate instrumentation acquisition is suggested .

In this work, the tires were considered as non-rotating cylinders as well as the floor was considered as being static due to experimental stand limitations. These assumptions shall be revised in a future work in order to increase the similarity between the full scale Formula SAE car and the virtual/experimental reduced scale model.

Even considering the above mentioned restrictions, lift and drag forces were evaluated. The know how presented in the present work allows a future comparative study between CFD modeling and a larger experimental set-up.

7. REFERENCES

Anderson, J.D., 1995, “Computational Fluid Dynamics”, 1.Ed., Mac-Graw Hill.

Badih, A. J., Longnecker M. M., Nagy K., Timmer J. S., 2001, “Aerodynamic Drag and Fuel Economy in a Formula SAE Racing Style Vehicle, SAE Technical Paper Series.

Brandt, S., 1999, “Introduction to Aeronautics: A Design Perspective”, AIAA, USA.

Hucho, 1998, “Aerodynamics of Road Vehicles”, SAE international, 4th edition.

Ieno, G., Negro, L., 2004, “Termodinâmica”, Ed. Prentice Hall, 1. Ed., 229p.

Kreith, F., Bohn, M. S., 2003, “Heat Transfer Principles”, Pioneira Thomson Learning Ed., São Paulo.

Maliska, C. R., 1995, “Transferência de calor e mecânica dos fluidos computacional”, 1.Ed., LTC.

Schlichting, H., 1968, “Boundary-Layer Theory”, Ed. McGraw-Hill, USA, 748 p.

8. RESPONSIBILITY NOTICE

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