24th COBEM - 2017

24th ABCM International Congress of Mechanical Engineering
December 3-8, 2017, Curitiba, PR, Brazil

COBEM-2017-2605
NUMERICAL AND EXPERIMENTAL ANALYSIS OF THE AERODYNAMIC BAHAVIOR OF A ROAD BUS IN A WIND TUNNEL

G. M. Rech
Marcopolo S/A, Caxias do Sul, Rio Grande do Sul, Brazil
giovanni.rech@hotmail.com

C. R. Altafini
C. A. Costa
Universidade de Caxias do Sul, Departament of Mechanical Engineering, Rio Grande do Sul, Brazil
craltafi@ucs.br
cacosta@ucs.br

Abstract. The aim of this work was to assess the aerodynamic parameters of a bus model, comparing the results obtained from CFD (Computational Fluid Dynamics) analysis, using ANSYS FLUENT® software, with those obtained from wind tunnel experimentation. The vehicle was modeled on a scale of 1:24. An *Ahmed* body with dimensions close to the bus was used in this work as reference to compare with literature values. For *Ahmed* body analysis, six models of turbulence were used, whereas for the bus only the standard κ - ε model was used. Tests of pressure distribution and aerodynamic drag (C_d) were carried out in the experimental analysis, varying the height of the free gap between the automobile table and the lower surface of the models, and air velocity. Comparing the results of the experimental tests with those obtained in the numerical analysis, there were variations in the C_d for the models. In the pressure test there was no significant variation in the pressure coefficient (C_p) between the experimental and CFD analysis for both models. It was also performed visualization tests using tufts distributed on the outer surface of the bus model, confirming the visual analyses of the CFD tool. For *Ahmed* body the value of the C_d found in the CFD was 0,449 while for the bus the value was 0,406. In the experimental trials the value was 0,650 and 0,530.

Keywords: Aerodynamics, drag coefficient, CFD, road bus, wind tunnel.

1. INTRODUCTION

Aerodynamics has become an important issue on vehicle design, either passage and freight vehicles. Based on these kinds of studies, is possible to improve the performance of a vehicle, avoiding significant losses in fuel consumption, emissions of harmful gases in the atmosphere, among others. The reduction of aerodynamic drag without affecting the comfort of the occupants of the vehicle is achieved by innovative design and has become a differential in the customer's choice.

According to Tunay, Yaniktepe and Sahin (2016), the flow around vehicles is extremely complex, mainly because of the turbulence, which making difficult the experimental and numerical analyses.

Katz (1995) states that the drag force has its direction parallel to the flow and in the opposite direction, points to the rear of the vehicle. Since C_d is dimensionless, it does not depend on the speed at which the vehicle is, so it is related only to the shape of the object. The drag coefficient can be defined by Eq. (1),

$$C_d = \frac{F_d}{\frac{1}{2}\rho U_\infty^2 A_f} \tag{1}$$

where C_d is the drag coefficient (dimensionless), F_d is the drag force (N), ρ is the specific mass of the air (kg/m^3), U_∞ is the free flow velocity (m/s) and A_f is the frontal area of the body immersed in the fluid (m^2).

In accordance to Hanfeng et al. (2016), *Ahmed* body is a very simple geometry of a vehicle and due to its flow in the rear region has become the object of many studies. In analyses performed by Ahmed et al. (1984), results were obtained for the drag coefficient (C_d) as a function of the angle of inclination of the back surface (named slant angle) of the model as shown in Fig. 1. It can be verified that an increase occurs in the C_d, as the angle of the chamfer increases from approximately 15° to 30°.

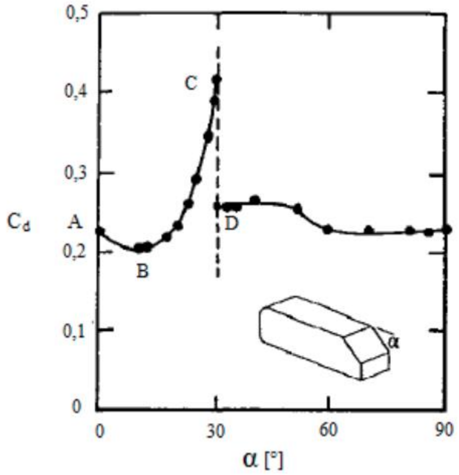

Figure 1. Experimental results on *Ahmed* body

2. EXPERIMENTAL AND COMPUTACIONAL PROCEDURE

An open-circuit wind tunnel was used for the experimental tests. Fig. 2 illustrates the wind tunnel used, with the test section on octagonal form, having the circumferential internal diameter of 400 mm.

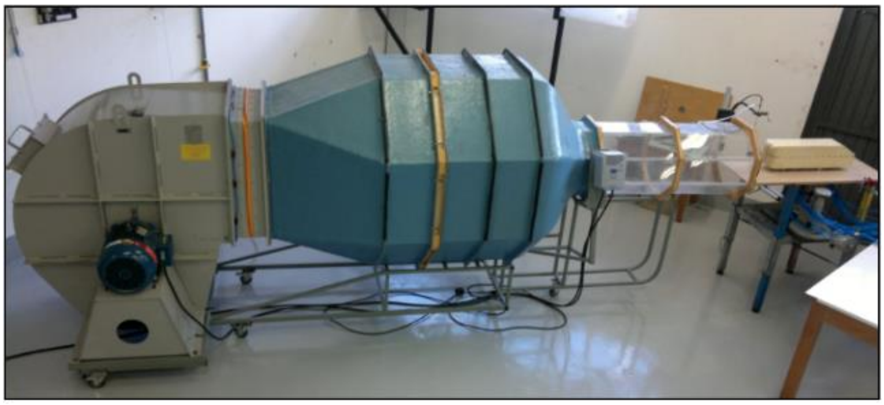

Figure 2. Wind tunnel used in the study

To simulate the effect of the soil an automobile table was built in MDF, being bevelled on the front face. This face was supported on the angled walls of the octagonal test section, staying around 200 mm into that section. The other end was supported on an external metal table, being properly levelled.

The model of the bus has a front area of 0.01653 m², depicted in Figure 3.

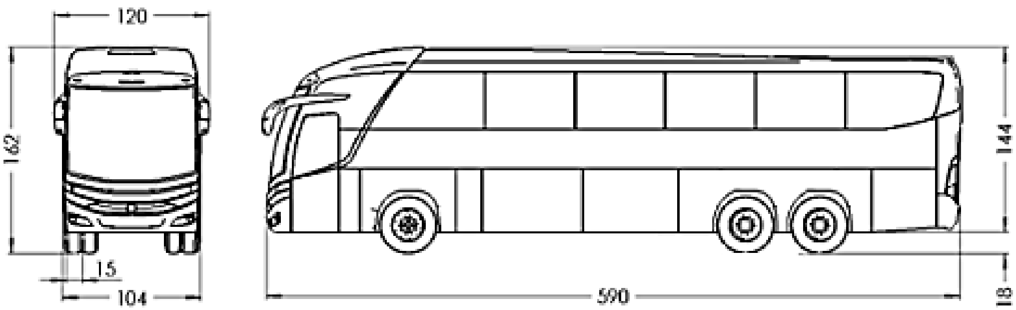

Figure 3. Dimensions (mm) of bus models

The front area of the *Ahmed* body is 0.01809 m² (Fig. 4).

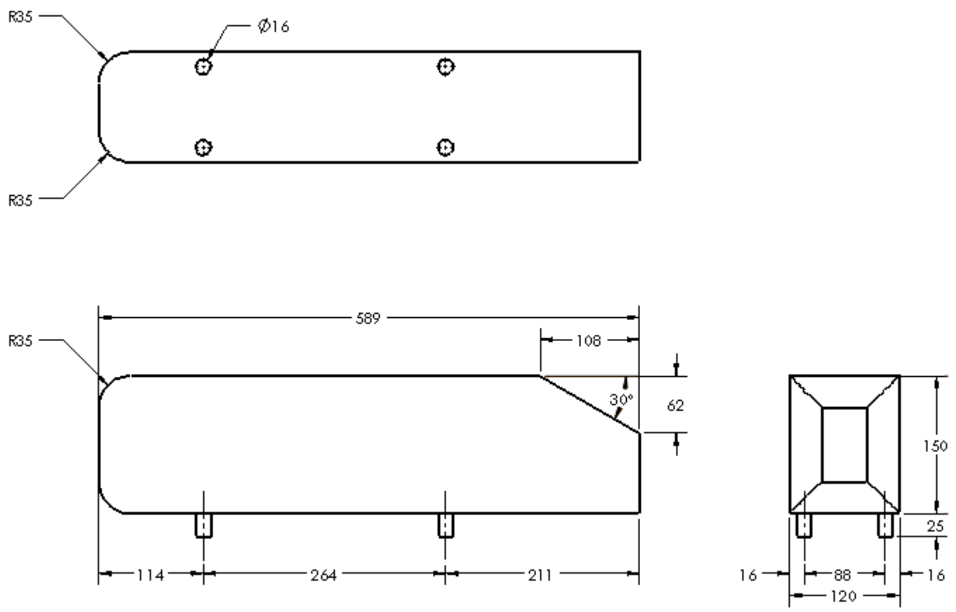

Figure 4. Dimensions (mm) for *Ahmed* body

The equipment used to measure air flow velocity and pressure was a digital pressure differential micromanometer of the brand KIMO model MP120. The velocity measurement was made indirectly by a Pitot tube of external diameter of 3 mm, through its total and static pressure sockets whose difference measures the dynamic pressure.

According to KIMO (2016) the micromanometer has a reading accuracy of ± 0,5% and resolution of 1 Pa and its accuracy for speed reading is ± 0,7 m/s in the speed range of 2 to 5 m/s and ± 0,5% of the measured velocity ± 0,3 m/s in range of 5 to 40 m/s and a resolution of 0,1 m/s.

To measure the drag force, a load cell of the brand AEPH, Single-Point Low Mini family (SPLMI) was used, with 5000 partitions and made of anodized aluminum, insensitive to moments of bending and twisting and with capacity of 2 kg (19,62 N).

To carry out the measurement of the drag force the load cell was mounted on vertical position coupled to a rod. The rod in turn extends upwards, through the center hole of the automobile table and at its end was fixed the model. The load cell assembly is illustrated in Fig. 5.

Figure 5. Mounting the load cell

For these tests, the defined speed was 32 m/s, and the System 5000, Model 5100 Scanner from Vishay Measurements Group was used for data acquisition.

Pressure tests with the *Ahmed's* body were performed by 45 pressure plugs distributed along the outer surface and at free flow velocities of 20, 24, 28 and 32 m/s. Figs. 6a (front), 6b (rear), 6c (lateral) and 6d (upper surface) represent the pressure ports on *Ahmed* model.

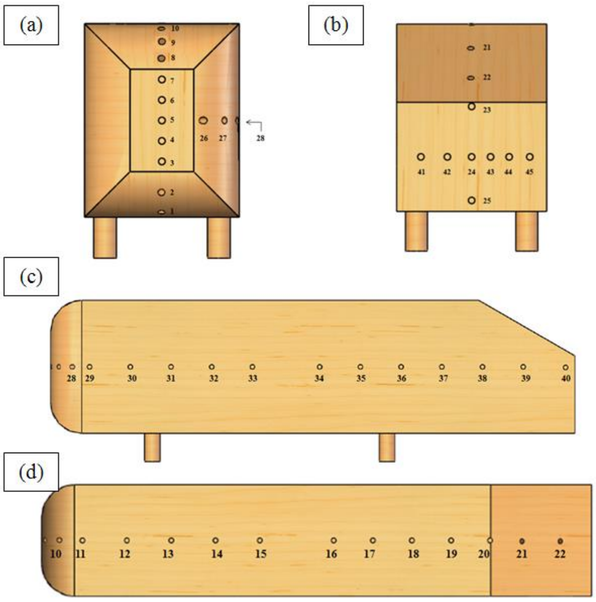

Figure 6. Pressure plugs on *Ahmed* body

Using equation 2 it was possible to determine the pressure coefficients.

$$C_p = \frac{p_i - p_\infty}{\frac{1}{2}\rho U_\infty^2} \tag{2}$$

Where p_i is the pressure of the model, p_∞ is the static pressure.

The software used was ANSYS FLUENT®. For the numerical analysis, it was considered as a computational domain a half cylinder, with dimensions equivalent to those of the wind tunnel, i.e., diameter of 400 mm, length up to the front surface of the model of 1000 mm and length of the back surface of the model to the end of the domain of 1500 mm.

The boundary conditions used in the computational domain for all models are: free flow velocity of 32 m/s; right side face defined as symmetry face; outlet face defined with a manometric pressure of zero Pa; floor and side defined as no slip wall.

Six models of turbulence were used in the computational tests for *Ahmed* body: Spalart - Allmaras, Standard κ - ε, κ - ε RNG, Standard κ - ω, κ - ω SST and SST. For the bus model only the Standard κ - ε turbulence model was used.

In the visualization test, wool yarns were used around the bus model, and the test speed was at 32 m/s.

3. RESULTS AND DISCUSSION

Table 1 shows the computational values of C_d in *Ahmed* body for the six turbulence models mentioned.

Table 1. C_d for *Ahmed* body with CFD

Turbulence Model	C_d for *Ahmed* body
Spalart - Allmaras	0,481
Standard κ - ε	0,460
κ - ε RNG	0,451
Standard $\kappa - \omega$	0,449
κ - ω SST	0,472
SST	0,474

The computational C_d value for the bus using the Standard κ - ε model was 0,406. For the experimental tests the C_d was 0,650 for *Ahmed* body and 0,530 for the bus model.

By adopting the C_d obtained for the bus model (0,530) for the 1:1 scale model and using Eq. (1) and U of 22,2 m/s (80 km/h), to overcome this drag, a power of 32,5 kW will be required from the vehicular traction engine.

Pressure tests performed for *Ahmed* body are represented in Fig. 7 by the C_p curve for each velocity, along the plane of symmetry of the model.

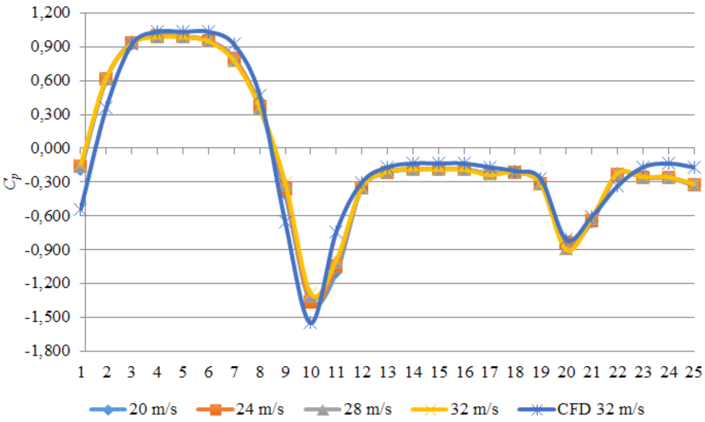

Figure 7. C_p curves for *Ahmed* body in the plane of symmetry

Figure 8 shows the C_p curves in the lateral region of the model.

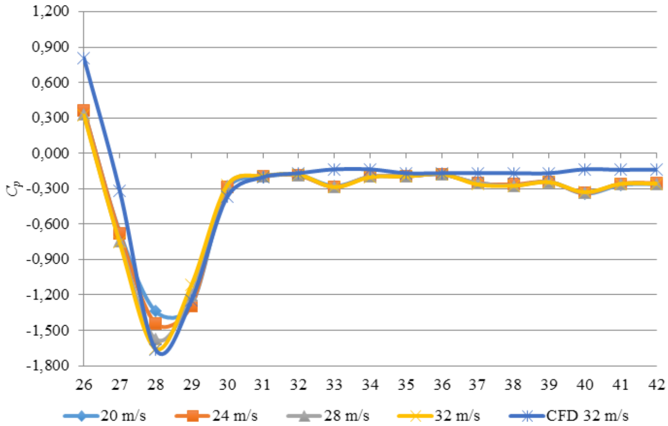

Figure 8. C_p curve for *Ahmed* body on lateral surface

Figures 9a (front face), 9b (lateral surface) and Fig.10 (rear face) show the visualization test using wool yarns for the bus model.

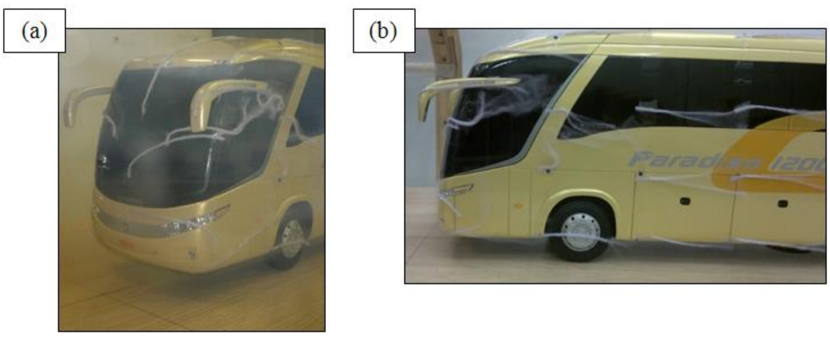

Figure 9. Test using wool yarn at the front and side of the bus

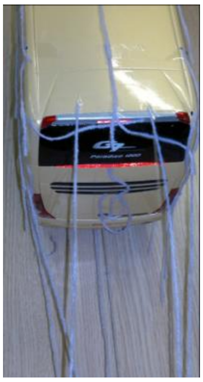

Figure 10. Test using wool yarn on the back side of the bus

Figure 11a and 11b represent the stremlines for *Ahmed* body.

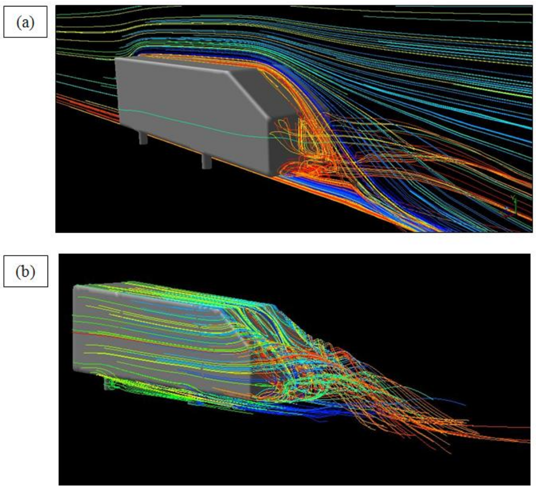

Figure 11. *Ahmed* body streamlines

Figures 12a, 12b and 12c represent the stremlines on the back surface, top and side for the bus model.

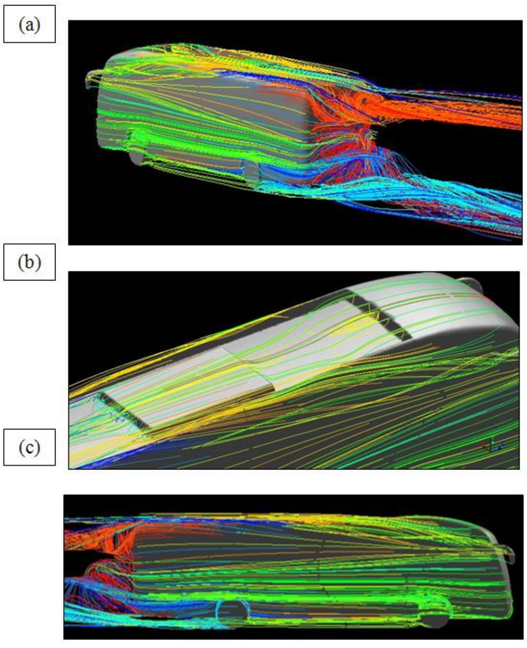

Figure 12. Bus model stremlines

3.1 Conclusions

From this work, it was verified that bus on 1:1 scale studied, about 11% of the engine power is used to overcome the drag force at a speed of 80 km/h.

Given the results obtained in the experimental and numerical analyses it is possible to conclude that there is a divergence between the values for the C_d, which can be minimized using a 3D model of the bus with less imperfections in the surface, better discretizing the geometric mesh used for the numerical analysis, etc.

In the pressure distribution experiments, the software proved to be effective when compared to the experimental analysis, showing good agreement in the pressure coefficients with those measured in the wind tunnel experiments.

Figure 10 illustrates the wool yarns at the rear of the bus, the two vortices being generated, one at the top of the model and one at the bottom. It is also observed that an air recirculation zone, between the lower and upper part of the bus.

In view of the factors, it was concluded that it was possible to reach the main objective of the work, since the models were made in CAD and compared the results obtained in the wind tunnel with those acquired in the CFD analysis.

4. REFERENCES

J. Katz, 1995. *Race Car Aerodynamics-Designing for Speed*. Robert Bentley Publishers, Cambridge, MA.

KIMO® INSTRUMENTS. “Technical Data Sheet”. Jun.2016 <http://www.kimo.fr/produits/1384/mp_120.html?lang=en>

S. R. Ahmed, G. Ramm, G. Faltin, 1984. “Some salient features of the time-averaged ground vehicle wake”. In *SAE Technical Paper*.

T. Tunay, B. Yaniktepe, B. Sahin, 2016. “Computational and experimental investigations of the vortical flow structures in the near wake region downstream of the Ahmed vehicle model”. In *Journal of Wind Engineering and Industrial Aerodynamics.*

W. Hanfeng, Z. Yu, Z. Chao, H. Xuhui, 2016. “Aerodynamic drag reduction of an Ahmed body based on deflectors”. In *Journal of Wind Engineering and Industrial Aerodynamics*.

5. RESPONSIBILITY NOTICE

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