ISSN 2176-5480

22nd International Congress of Mechanical Engineering (COBEM 2013)
November 3-7, 2013, Ribeirão Preto, SP, Brazil

COMPUTER SIMULATION VALIDATION TO OBTAIN THE AERODYNAMIC DRAG FOR A BUS SCALED MODEL

Vagner Abramchuk
Programa de Pós-Graduação em Engenharia Mecânica – PROMEC
Universidade Federal do Rio Grande do Sul – UFRGS
Rua Sarmento Leite, 425
CEP 90050-170 Porto Alegre, RS, Brasil
vagnerabram@gmail.com

Adriane Prisco Petry
Programa de Pós-Graduação em Engenharia Mecânica – PROMEC
Universidade Federal do Rio Grande do Sul – UFRGS
Rua Sarmento Leite, 425
CEP 90050-170 Porto Alegre, RS, Brasil
adrianep@mecanica.ufrgs.br

Nehemias Lima Lacerda
FEMTO Ciências Aplicadas Ltda
Parque Tecnológico de São José dos Campos
CEP 12.247-004 São José dos Campos, SP, Brasil
nehemias@femtociencias.com.br

Carlos Viero
Programa de Mestrado Profissional em Projeto e Processos de Fabricação– PPGPPF
Universidade Passo Fundo – UPF
Rodovia BR 285 - Km 292
CEP 99.052-900 Passo Fundo, RS, Brasil
viero@comilonibus.com. br

Abstract. *The technological improvement of the digital computes and the development of fluids simulation software providing increasingly sophisticated and effective results enabled a significant reduction in time and computational costs to develop better vehicles, regarding to operational and environmental concerns, due to its safer use and lower fuel consumption. However, even with quite widespread CFD use, there is still conflicting information on the expected level of agreement between the results obtained via simulation, when compared with the experimentally measured results. Within this context, it is often necessary to validate those mathematical models. The present study aims to compare results obtained by simulation with experimental tests to measure the aerodynamic drag performed in a 1:30 scale wind tunnel. The geometry of the model was similarly designed to the physical model, the Reynolds number considered is the order of* 10^5*, residue of convergence equal to* 10^{-5} *obtained with the K-Omega turbulence model and calculated in second-order discretization. The validation of these results will facilitate the use of the numerical method to develop coaches with a lower drag coefficient and greater aerodynamic stability with consequent reduction in fuel consumption.*
Keywords: *Mathematical model, Computational validation, Experimental tests, Bus aerodynamics.*

1. INTRODUCTION

Environmental concern is evidenced with emphasis in recent decades. Thus, technologies should be used in each new vehicle design so that they provide lower environmental impact. The study from aerodynamics vehicles is of utmost importance in Brazil, since much of passenger and load transport is carried by buses and trucks under the overland routes that travel at a favorable speed for this type of study, any possibility of reduction on fuel consumption provides less environmental pollution and, for the entrepreneur a significant reduction in costs which can be passed along to the consumer. Their geometry can be worked extensively for possessing frontal area and large lengths, the front tilt or adding rays at the model edges, for example, can reduce drag by 40% (Gilhaus, 1981).

Therefore, within this contest, the vehicular aerodynamic provides the possibility of reduction in fuel consumption that a vehicle consumes to perform a specific path very satisfactorily. Overall, aerodynamics has gained industrial significance with the arrival of airplanes that needed to get around with the lowest possible air friction, hence they would be faster spending less fuel. However, even with this variety of environmental, economic benefits and great technical possibility from drag reduction, it should be highlighted that the aerodynamic phenomena on vehicles are not easy to catch in real models, due to their high Reynolds number and even with experimental scaled essays, the results

ISSN 2176-5480

can be unsatisfying, because in most of cases, the wind tunnels do not make possible to achieve the same Reynolds number that a real model produces, compromising the study.

The computational simulation in CFD (Computational Fluid Dynamics) can provide dependable results by reproducing high Reynolds number on models in real scale.

It is in this context that the work is situated. An aerodynamic computational simulation in CFD will be conducted under a bus scaled model, which has undergone experimental essays in wind tunnel. Thus, the contour conditions and discretization of the mathematical model imposed for performing computational simulation analyzing similarity of aerodynamic forces acting on the vehicle will be validated.

2. AERODYNAMIC STUDY ON BUS

Currently high fuel consumption and consequent emission of pollutants in air are factors that propel several studies in automotive engineering. The study of aerodynamic forces could propitiate satisfactory outcomes, due to being directly linked to the best energetic performance of cars, trucks, buses, among others. Moving vehicles are subject to the action of air resistance known as the drag forces, lift forces, side forces and moments, are all linked directly to the geometry of the body, which with its displacement, cause vortexes in the posterior region, increasing their intensity along with velocity.

When a vehicle is in high speed, the resultant force contrary to the displacement on a bus body is derived, principally by air resistance. The intensity of this resistance depends on the type of vehicle and mostly of the speed of locomotion, which according to (Gotz, 1977), when it is higher than 80 km/h, aerodynamic force represents more than 50% of the imposed resistance on the vehicle, emphasizing resistance of friction that tires exert on road. Currently vehicles use 30% to 40% of the engine power only to win air resistance (Ahmed, 1983).

Therefore, the greatest interest to the reduction of drag is for body coaches, because they develop high speed, have a large frontal area and travel great distances. However, even with this amount of environmental, economic benefits and great technical possibility of drag reduction, it should be highlighted that aerodynamic phenomena on vehicles are not of easy uptake in real models, because of their high Reynolds number and even with essays on scale, results cannot be satisfactory, because in most of cases, wind tunnels do not make possible to reach the same Reynolds number that a real model produces, compromising the study. In order to unsatisfactory results do not occur, the validation of the methodology should be performed comparing an experimental case on reduced scale of the vehicle with the computing simulation, verifying the veracity of the results.

3. METHODOLOGY

The aim of this study is to validate computing tools as alternatives to a solution or an optimization of engineering problems, considering that their suitable use provides flexibility and agility in responses.

3.1 Dimensional and geometric analysis

Geometric similarity is defined when the ratio among any length in the model and the corresponding value in the prototype is constant. This ratio is known as the scaling factor. The kinematic similarity refers to matching motion, what necessarily implies equivalent length and time intervals. Two systems are dynamically similar when the absolute values of the forces at equivalent points of the prototype and model present a fixed ratio.

The forces that determine the behavior of the fluids have different origins. Typical dimensionless groups for application to the fluid flow study, play a fundamental role on the understanding of the operational system and its associated characterization. Table 1 shows such equations.

Table 1. Dimensionless groups in typical usage situations involving fluid flow.

ADMENSIONAL GROUP	NAME	ACTING FORCES	SYMBOL
$\frac{UL\rho}{\mu}$	Reynolds Number	$\frac{\text{inertial force}}{\text{viscous force}}$	Re
$\frac{U}{(Lg)^{\frac{1}{2}}}$	Froude Number	$\frac{\text{inertial force}}{\text{gravity force}}$	Fr
$U\left(\frac{L\rho}{\sigma}\right)^{\frac{1}{2}}$	Weber Number	$\frac{\text{inertial force}}{\text{surface tension force}}$	We
$\frac{U}{c}$	Mach Number	$\frac{\text{inertial force}}{\text{elastic force}}$	M

ISSN 2176-5480

A moving vehicle should react to forces and moments contrary to displacement. These are denominated bearing force and drag force, the former coming from friction of the mechanical components, and the latter from the resistant strength of the air over the vehicle that is parallel to the direction of free stream and depends on the dimensions of immersed body in the fluid, on relative speed displacement of air in relation to the solid body, on kinematic viscosity and on fluid specific mass (Fox, 2001). In this study of automotive vehicle, where the flow is considered viscous, both the pressure forces as the shear ones act on the model (Fox, 2001). Eq. (1):

$$\vec{F} = \int d\vec{F} = \int d\vec{F}_{\text{cisalhamento}} + \int d\vec{F}_{\text{pressão}} \quad (1)$$

Decomposing the resulting $\vec{F}$, the parallel component to the flow is the drag force. In the flow other forces also act, for example, lift force. There are few cases in which this force can be determined without computing or experimental results since the adverse pressure gradient in the flow usually leads to flow separation, which impedes the determination of the drag force analytically. For incompressible flow we have the mathematical definition of the drag force according to the equation (2):

$$F_D = \int dF_x = -\int p cos\theta dA + \int \tau_w sen\theta dA \quad (2)$$

To determine this force it is necessary to determine the format of the body, the distributions of shear stress on the wall and the pressure distribution along the surface of the body. In dimensionless form, this force is defined by the coefficient of drag. Eq. (3):

$$C_D = \frac{F_D}{\frac{1}{2}\rho U_\infty^2 A} \quad (3)$$

where:
C_D: Coefficient of drag;
ρ : Density of the fluid;
U : Relative speed of fluid in relation to the vehicle
A : Frontal area of the vehicle.

The drag coefficientC_D, a primary factor to measure the ease by which air resistance is overcome, is defined to be the ratio of the drag force and a characteristic force associated to the free stream dynamic pressure. This drag coefficient is the magnitude to be minimized and may result in economical , environmental and performance effects because the vehicle will have a lower force in opposition to its displacement, requiring lower horsepower destined to overcome the resistance of the air, with possible fuel economy, lower emission of gases in the atmosphere, besides better performance.

3.2 Experimental model proposal

(Orlando Moreira Junior, 1996) has obtained drag forces starting from experimental essays, lateral and of yaw moment on three bus models on reduced scale by differentiating tilt geometry of the frontal surface among them .The model stayed on a flat wooden plate , automotive table, which simulated the ground. To measure the acting forces, (Orlando, 1996) has designed and built a scale using extensometers, "strain-gage", allowing the capture of pressure values on the model surface and on the automotive table, besides getting drag forces with Reynolds number ranging from $8,23 \times 10^4$ to $13,02 \times 10^4$. The bus dimensions considered by the author are shown in figure 1, with the main transversal area equal to $A_t = 0,00726 m^2$ and characteristic length for calculation of the drag coefficient equal to $L_c = 0,0726 m$.

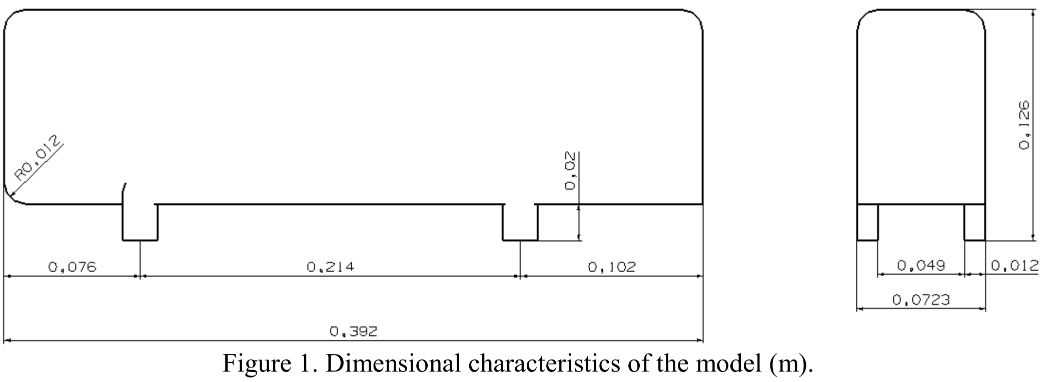

Figure 1. Dimensional characteristics of the model (m).

ISSN 2176-5480

Seven Reynolds numbers were used for the total drag essays, each essay was repeated three times and a simple average of these values was used, since there was not dispersion in the results, i.e differences were within the uncertainty index. (Orlando, 1996) has obtained the values according to table 2 and graph plotting of Figure 2 for the model without front tilt and yaw angle of β=0°.

Table 2. Experimental values of the drag coefficient and uncertainty intervals obtained by (Orlando, 1996).

Rex10⁴	**Cdt (total drag coefficient)**	**Wcdt (uncertainty values)**
8,232	0,51	0,02
9,2	0,51	0,02
10,08	0,51	0,02
10,89	0,51	0,02
11,65	0,50	0,02
12,35	0,50	0,02
13,02	0,50	0,02

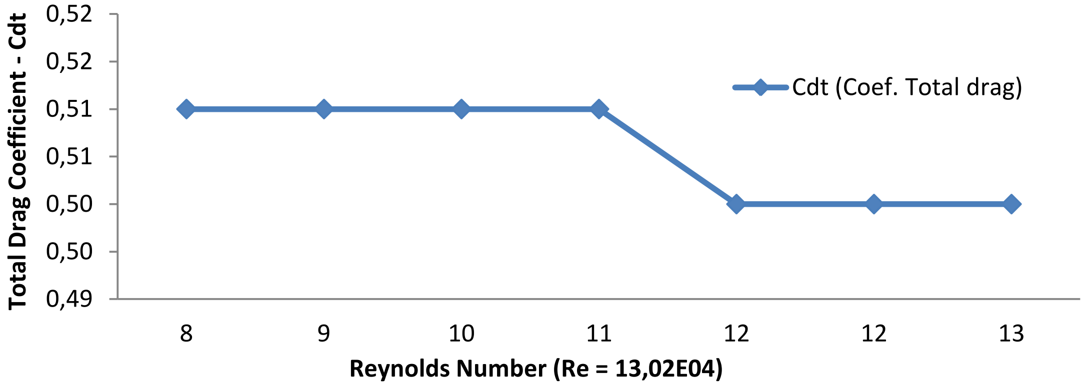

Figure 2. Variation of total drag coefficient with Reynolds number for yaw angle, β=0°, obtained by (Orlando, 1996).

It is observed on these results a slight difference in drag coefficient due to the variation of the Reynolds number, although within the interval of uncertainty.

3.3 Numerical simulation, model discretization

The numerical simulation for comparison purposes of experimental results obtained by (Orlando, 1996), were performed using a commercial software of Computing Fluid Dynamics with its own mesh generator, SIEMENS^{TM} NX^{TM}. The first stage was the creation of the CAD model, followed by the creation of the domain, according to figure 3, with 8 meters long, 2 meters high and 1 meter wide, discretized by volumetric mesh with tetrahedral and hexahedra elements in the boundary layer, thus allowing greater refinement on the wall, with the aim of describing the boundary layer formed in those regions adequately.

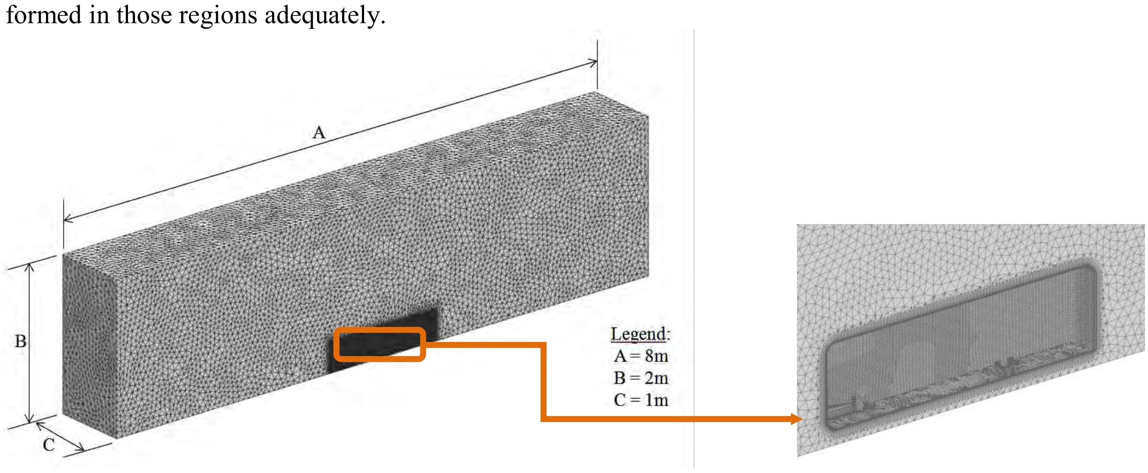

Figure 3. Volumetric mesh generated on the domain.

ISSN 2176-5480

Due to the high computational time was chosen symmetry of the model. The boundary conditions imposed on the domain model and using commercial software SIEMENS^{TM} NX FLOW ^{TM} are presented in Figure 4 and faithfully comply with the conditions considered by (Orlando, 1996) in their experimental testing. As the condition "use wall fuction" on the body of the bus establishing a friction between the surface and the flow (zero velocity on the wall). Side edges, top and bottom the condition of "slip-wall" is added or smooth wall, which possesses the dominant input speed to zero and shear so as to have minimal influence on the flow. At the lower end of the domain is imposed a condition wall moving velocity equal to the input.

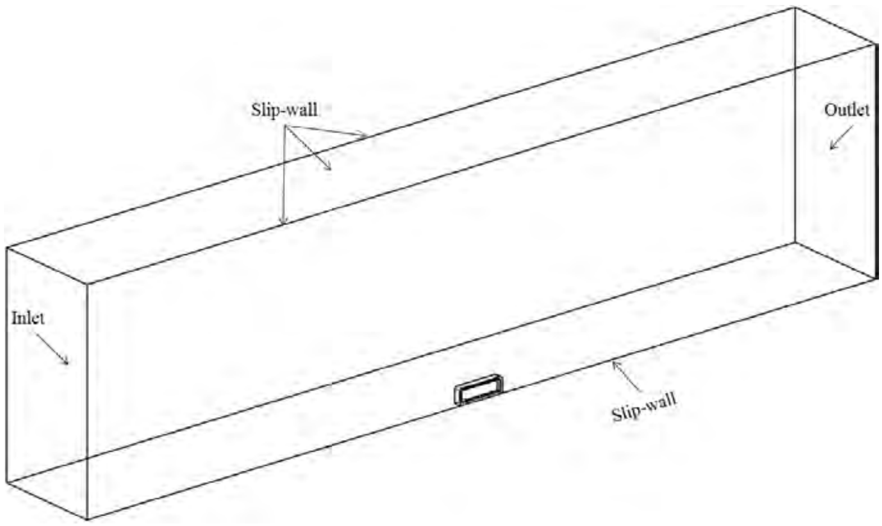

Figure 4. Boundary conditions established on the domain.

The output condition or open, which allows both input and output flow is imposed on the posterior side of the input field with a value of zero static pressure. The input will be the condition of "Inlet" where the entry is permitted only flow during the simulation solution. As initial condition for the flow to occur was imposed a speed of 27 m / s at the entrance of domain 1% turbulence intensity.

3.4 Mathematical equation

The numerical solution was performed through the method Reynolds averaged Navier-Stokes (RANS), coupled to a turbulence model k-Omega. Using differential equations to study the flow field in greater detail, what reduces according to (Anderson, 1995), the discontinuity of numerical oscillations of the equations. The equations of dissipation and the turbulent kinetic energy were discretized by the upwind method, which provides a strong numerical diffusion in the solution, however it avoids the appearance of dispersive numerical solutions since it softens the problem through gradient reductions, (Fortune, 2000). Transport equations are derived from the RANS equations. The continuity equation written in the form of Cartesian coordinates, follow as Eq. (4):

$$- \quad \nabla(\ \vec{V})$$

Because it is a problem of incompressible fluid flow, the Navier Stokes equations set changes at the moment and acceleration of a fluid particle thus, field strengths are despicable, which are written in Cartesian coordinates as Eq. 5, 6 and 7.

$$\rho\frac{Du}{Dt} = -\frac{\partial p}{\partial x}+\frac{\partial}{\partial x}\left(2\mu\frac{\partial u}{\partial x}-\frac{2}{3}\mu\nabla\vec{V}\right)+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial u}{\partial y}+\frac{\partial v}{\partial x}\right)\right]+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial w}{\partial x}+\frac{\partial u}{\partial z}\right)\right] \tag{5}$$

$$\rho\frac{Dv}{Dt} = -\frac{\partial p}{\partial y}+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial u}{\partial y}-\frac{\partial v}{\partial x}\right)\right]+\frac{\partial}{\partial y}\left(2\mu\frac{\partial v}{\partial y}-\frac{2}{3}\mu\nabla\vec{V}\right)+\frac{\partial}{\partial z}\left[\mu\left(\frac{\partial w}{\partial y}+\frac{\partial v}{\partial z}\right)\right] \tag{6}$$

$$\frac{Dw}{Dt} = -\frac{\partial p}{\partial z}+\frac{\partial}{\partial x}\left[\mu\left(\frac{\partial w}{\partial x}-\frac{\partial u}{\partial z}\right)\right]+\frac{\partial}{\partial y}\left[\mu\left(\frac{\partial v}{\partial z}+\frac{\partial w}{\partial y}\right)\right]+\frac{\partial}{\partial z}\left(2\mu\frac{\partial w}{\partial z}-\frac{2}{3}\mu\nabla\vec{V}\right) \tag{7}$$

ISSN 2176-5480

The transport equations are resolved for turbulent kinetic energy and for a term called specific rate of dissipation, which is the rate of dissipation per unit of turbulent kinetic energy. The representation of the modeled turbulence K-Omega (k, ω) is a model of two equations that presents an alternative to the K-Epsilon model. The wall treatment is a transition model based on the correlation with the gamma Re-theta variant. (Sorensen, 2009) makes a study with this model on thicker profiles to low Reynolds numbers obtaining an excellent concordance with the experimental results, which can make this model interesting in subsonic flow analysis.

4. RESULTS

With the analysis of the Cdt dimensionless parameter (total drag coefficient) derived from the numerical simulation of four distinct meshes, it is observed that numerical calculation has mesh independence, which makes it appropriate for comparison with experimental results. The graph in Figure 5 allows the visualization of Cdt for different mesh sizes ranging from 10% in its entirety for the first three and about 50% for the last mesh.

Number Elements	Coef. Total drag (simulation)
2,836,014	0,547
3,201,188	0,532
3,479,653	0,477
7,941,920	0,480

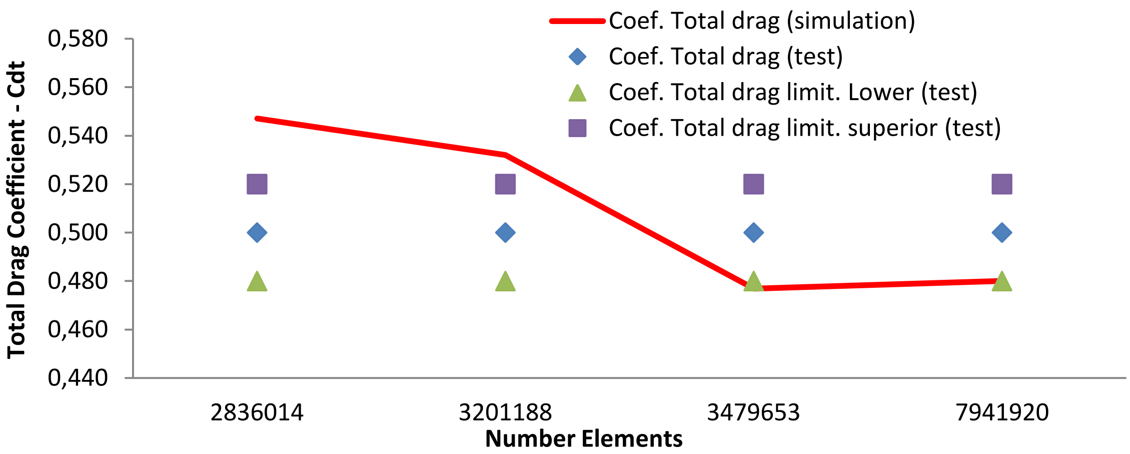

Figure 5. Cdt variation (coefficient of total drag) with the amount of elements.

Experimental Cdt obtained by (Orlando, 1996) has 4% variation for any Reynolds studied, allowing to conclude that the values reported by the numerical simulation are circling the limits set by the experimental testing. It is relevant to analyze that the mesh with 7,941,920 elements that has hit the exact value of the coefficient of total drag at the lower limit of the assay, has a total time of 45 computational hours and that the mesh 50% lower totaling 3,479,653 elements has obtained its computational time reduced to 9 hours, with the Cdt result almost unchanged.

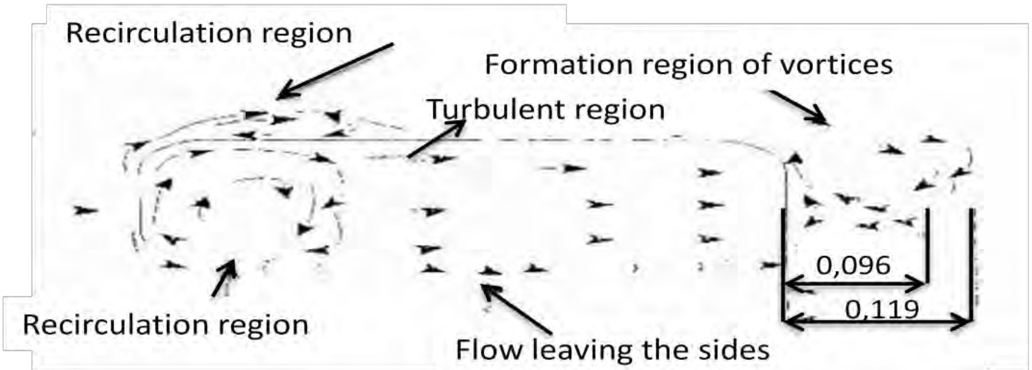

Figure 6. Left side view of the model, describing the wool yarn for testing visualization. Source: rewritten (Orlando, 1996).

The main regions of recirculation tested on the model by (Orlando, 1996) is shown in figure 6. Comparatively one has observed the great similarity of vortices in the simulation performed in this study, which is being presented in Figure 7.

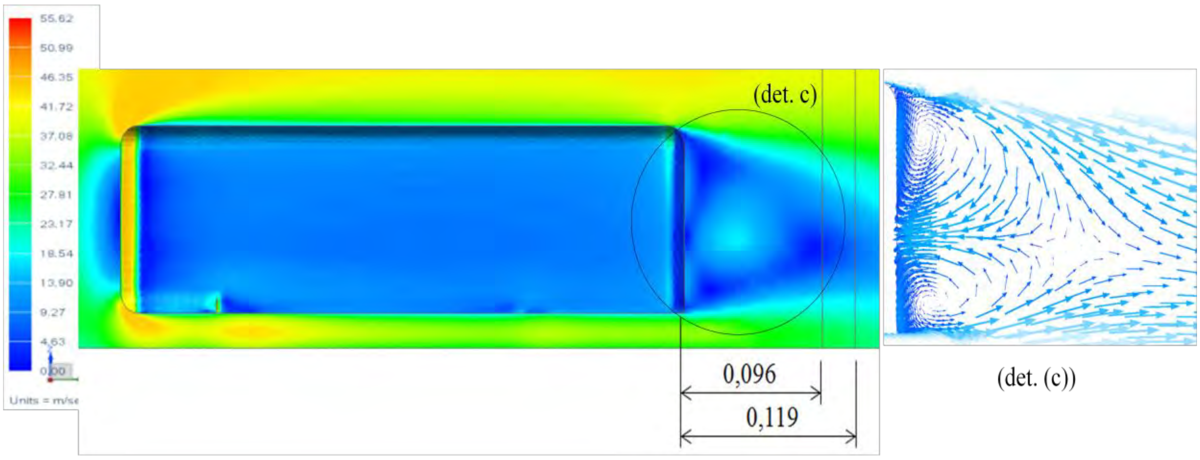

Figure 7. Left side view of the model, with the plotting speed coming from the numerical simulation, with detail on the rear recirculation region.

The recirculation vortices are created due to the winding of the free shear layers that come off the top and side surfaces. The longest vortex is formed as the boundary layer that gives off early in the model in the upper part and the other formed by the flow exiting the flow model and finds the separation of the lower field. The vortices have axes of rotation perpendicular to the flow direction, however the first vortex generated the largest, rotates clockwise and the second in a counterclockwise direction.

The dominant contribution to the aerodynamic drag of an intercity bus is the pressure differential between the forward- and rearward-facing surfaces of the body, with a minimal contribution from skin friction. According to (Cooper, 1985), about 60 to 70 percent of the total wind-averaged drag of a bus is attributed to pressure loads acting on the vehicle forebody, making it the principal area for drag reduction strategies.

5. CONCLUSIONS

The use of experimental and computational techniques (CFD - Computational Fluid Dynamics) enabled comparison between two different approaches of problem solving of fluid mechanics. The software SIEMENS^{TM} NX^{TM} has shown to be suitable for the simulation of fluid flow problem on transport vehicles, presenting consistent results with the available experimental data, including the posterior region of the model where it obtained airflow recirculation, besides offering great flexibility to analyze different configurations with low cost when compared to experimental methods. Nevertheless, it was evident the importance of experimental studies to validate the numerical simulations and that the experimental and numerical approach must coexist whenever possible, highlighting that this result correlation is mostly obtained only with the study of mesh independence of the problem, as evidenced with emphasis in this study.

The computational time is still an impacting factor on numerical simulation, but improvements in these types of studies are being created by the construction of increasingly powerful computers and elaboration of studies and numerical methods increasingly more efficient, resulting in more accurate data, which decreased the amount of experimental tests. The turbulence model K-Omega also has shown satisfactory results in obtaining the drag forces, but future studies should be performed with different turbulence models and comparing their accuracy.

6. ACKNOWLEDGEMENTS

V. Abramchuk thank the guidance of Prof. A.P. Petry and support of the company Applied Sciences FEMTO Ltda Comil and Bus SA.

7. REFERENCES

AHMED, S. R., 1983. "Influence of base Slant on the Wake Structure and Drag of Road Veycles". Journal of Fluids Engineerring, vol. 105, 1983.

ANDERSON, J.D., 1995. "Computational Fluid Dynamics - The Basic with Aplications." 1 ed., MecGraw-Hill.

COOPER, K.R., "*The effect of front-edge rounding and rear-edge shaping on the aerodynamic drag of bluff vehicles in ground proximity*", SAE paper No. 850288, 1985.

ISSN 2176-5480

FORTUNA, A. O., 2000. "Técnicas Computacionais para Dinâmica dos Fluidos - Conceitos basicos e Aplicações". EDUSP, Editora da Universidade de São Paulo.
FOX, R.W., McDONALD, A.T., 2001. "Introdução à Mecânica dos Fluídos". *L.T.C – Livros Técnicos e Científicos*. Rio de Janeiro, 2001.
GILHAUS, A., v. 2, 1981. "*The Main Parameters Determining the Aerodynamic Drag of Buses*". Colloque Construire a vec le vent.
GÖTZ, H., 1977. "Die Aerodynamik des Nutzfahrzeuges-Mabnahmen zur Kraftstoffeinsparung". *Fortschr, Berichte der VDI-Zeitschriften*. Series 12,1977.
ORLANDO, M. J, 1996. "*Estudo Experimental do Escoamento Sobre Modelos de Ônibus com Diferentes Inclinações da Superfície Frontal*". Dissertação (Mestrado), ITA - Instituto Tecnológico da Aeronáutica.
SORENSE, Niels N., 2009. CFD Modelling of Laminar-turbulent Transition for Airfoils and Rotors. Using the Gamma-Re-Theta Model. WIND ENERGY. 2009, Vol. 12, pp. 715-733.

8. RESPONSIBILITY NOTICE

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