RESEARCH ARTICLE OPEN ACCESS

Validation of Experimental and Numerical Techniques for Flow Analysis over an Ahmed Body

Rodrigo Mendes Alves*, Odenir de Almeida**
*(Department of Mechanical Engineering, Universidade Federal de Uberlândia, Brazil
** (Department of Mechanical Engineering, Universidade Federal de Uberlândia, Brazil

ABSTRACT
The impact of improvement in vehicle aerodynamics mainly reflects in lower fuel consumption and lower carbon dioxide emissions into the atmosphere. The governments of many countries support continuous aerodynamics' improvement programs as a way of mitigating the energy crisis and atmospheric pollution. This work has the main goal to validate experimental and numerical techniques for application in road vehicles. The experimental results were obtained through the analysis of the flow around a standard body with simple geometry called Ahmed Body, using hot wire anemometry from experiments in wind tunnel. It was also proposed a computational validation using a commercial software (Star CCM +) to further analyze the flow and to corroborate the experimental results. Both results were compared and allowed characterizing the flow around the vehicle. The results obtained analyzing the Ahmed Body aimed further application on aerodynamics of heavy-duty vehicles, which is an ongoing research being developed at the Experimental Aerodynamics Research Center – CPAERO, in Brazil.

Keywords- Aerodynamics, Ahmed Body, Experimental Techniques, Flow analysis, Road Vehicles

I. INTRODUCTION

The main reason to study the aerodynamics of road vehicles is reducing the drag with the objective to reduce the power necessary to motion, and, by consequence, decreasingthe emission of pollutants. Fig. 1 shows the relation between power necessary to overcome aerodynamic drag and rolling resistance for a 40 tons truck with a Cd (Drag Coefficient) equal to 0.50. From this figure, it is visible that over 80 km/h it is critical the increasing of aerodynamic drag. That is the reason of the efforts applied to mitigate the drag force in heavy-duty vehicles industry and via academic researches around the world.

According to the DOE (Department of Energy of the United States) annual review of 2012, the class 8 trucks (vehicles with gross weight rating exceeding 33000 lb) are responsible for about 13% of the fuel consumption in the United States. The reduction of 12% on fuel consumption would lead to 13.2 billion dollars savings per year and reduction of 28 million tons of carbon dioxide emission [1].

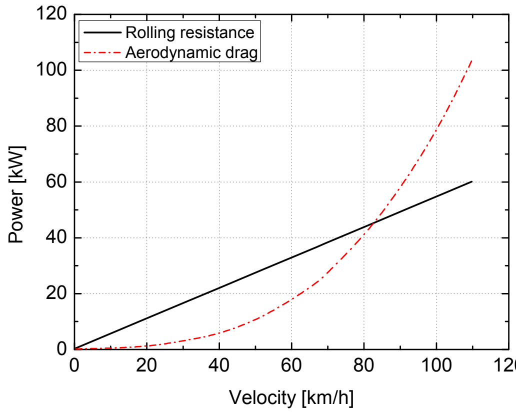


Fig.1. Relation between the power necessary to overcome aerodynamic drag and rolling resistance for a 40 ton Truck with 0.5 Cd. Source: ICCT (International Council on Clean Transportation).

Fig. 2 shows the energy losses in an 80000 lbtruck travelling with a velocity of 105 km/h. The energy loss caused by aerodynamic is about 53% of the total losses, while the rolling resistance is responsible by 32% of total. Therefore, the study of the interaction of the flow with truck's structure is very importance to reducing fuel consumption and increasing energy efficiency.

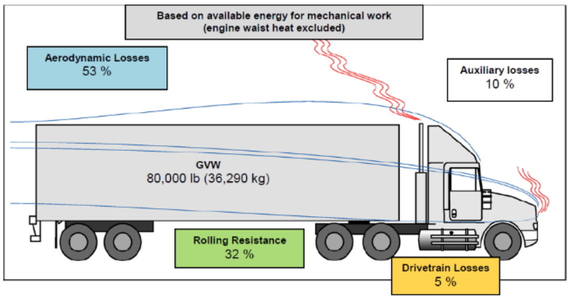


Fig. 2. Energy losses in a moving truck, after [2].

In order to get a better understanding of the flow and its interaction with the truck, it is necessary to work in two different approaches: experimental and numerical techniques applied to characterize the flow dynamics.

One of the most usual and efficient equipment to develop an experimental aerodynamic analyses is the wind tunnel. Similar conditions to real can be simulated in the wind tunnel in different scales ratio. Since trucks are big structures, the real condition simulation would require a very big wind tunnel, high investment and technology. These are the reasons for working with scale models, such approach could lead to reasonable results when compared to real cases flow analyses. This is true only if a lot of expertise and care is taken during the experiments.

Historically, the improvement of trucks' aerodynamics had an important effort in developing and implementing the air deflector by1950s as seen in Ref. [3]. Then trucks' shape has been changed over the years (small radiator, rounded corners, etc.) as seen on Ref. [4]. Nowadays, new structures have been added to trucks for aerodynamics improvements like skirts, wedges and base flaps; and good results already obtained from experimental and numerical analysis [4].

For the numerical approach, it is usual to work with Computational Fluid Dynamics (CFD) analysis. This tool offers a chance to observe the flow in limitless conditions in all scales with lower costs and shorter time compared to the experimental analyses. There are several commercial software developed for aerodynamic investigation, which has demonstrated good results compared to experiments. For this specific work, STAR CCM+® software was used based on the reliability of its results already demonstrated in other researches like the work of Ref. [5].

Hakansson and Lenngren [5] has developed interesting analyses for the aerodynamics of trucks using CFD, testing some trailer devices in order to understand the influence of those on the flow and drag force analyses. The results indicate that those devices lead to lower drag for different yaw angle and shows that the optimization of the carriage would be a great deal for the industry of trucks.

The objective of this work is to study the flowaerodynamic over an Ahmed Body with a Reynolds number ranging from 0.7×10^5up to $1.4\ 7 \times 10^5$,through experimental and numerical techniques aiming to apply the results on heavy-duty vehicles' aerodynamics. In order to achieve such goal wind tunnel experiments were carried out by using a hot-wire anemometry system to characterize flow profiles in the Ahmed body's wake. The limitations of using hot-wire anemometry for performing velocity profiles measurements in the near-wake of the body were investigated in this work.

Additionally a numerical approach, by using the STAR CCM+ code, was validated against ERCOFTAC(European Research Community on Flow, Turbulence and Combustion) results [6]. The numerical technique has been applied to corroborate the experimental results with the purpose to validate these tools for later use in the analyses of the flow around heavy-duty vehicles such as trucks and buses. With the purpose of validating such techniques some remarks were registered during the development of this work.

II. METHODOLOGY

The experimental procedure used in this work aimed to analyze the flow around the model in the wind tunnel using hot wire anemometry to measure velocity profiles in the wake of the body. The objectives here were to validate the newly constructed small wind tunnel in the Experimental Aerodynamics Research Center - CPAERO of the Federal University of Uberlândia(UFU) and also toverify the applicability of hot-wire sensors to generate experimental data, for further comparison purposes with the numerical model.

The numerical procedure was firstly validated against experimental results obtained by ERCOFTAC [6], aimed to offer a better understanding about the interaction of the flow with the Ahmed body. For this, the commercial software STAR CCM + has been used for numerical simulation of the flow with the same physical characteristics of the body and conditions of the experiment.

1.1 Experimental Approach

1.1.1 Wind Tunnel

The wind tunnel of the LEEAR was used for the experimental part of this work. Some of the specifications of the tunnel are described in sequence:

- Maximum flow velocity of 33 m/s in free test-section;
- Test-section with a cross section of 0.6 × 0.6 m and 1 meter in length;

- 6 meter in length for the whole tunnel;
- Maximum width of 1.90 meter;
- 1.90 meter in height;

The Fan has 1.25 m in diameter and about 43,000 m³/h of flow rate. The engine has 24 HP, 220/380V, IP55, 60 Hz, with rotation of 1150 RPM. The engine's velocity is controlled by a frequency inverter of 220/380V in three phase. The wind tunnel also was equipped with a Pitot tube for velocity measurement inside the test-section. Fig. 3 shows the test section of the wind tunnel with the Ahmed body used for the experiment. A complete set of flow measurements inside the test section and also flow visualization allowed to well characterize the flow pattern, as part of the verification of the flow uniformity in this wind tunnel.

Fig.3 shows the velocity profiles for the wind tunnel boundary layer development at different test conditions.According to the results, the boundary layer thickness is around an average of 4.0 mm in the middle of the test section.The turbulent intensity of the tunnelis around 0.81% for the range of velocities investigated in previously work, being around 0.6% for 7 m/s and reaching a peak level of 1% at 25.5 m/s.

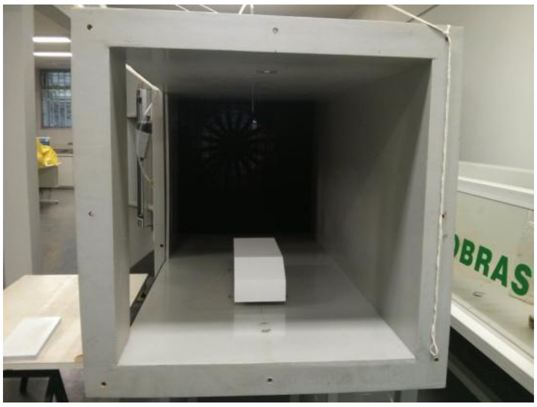


Fig.3. Images from the test section of the wind tunnel with the Ahmed body.

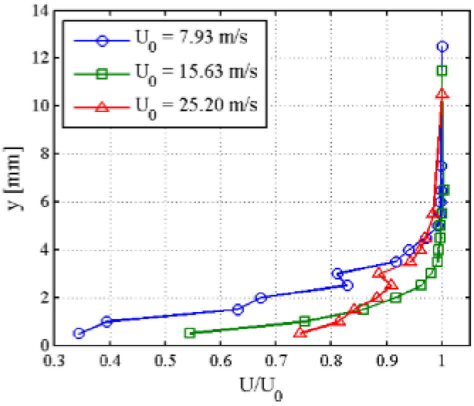


Fig. 4. Normalized profiles for boundary layer development at different flow velocities.

1.1.2 Hot Wire Anemometry

The technique for flow velocity measurement used in this work was the hot-wire Anemometry. This technique has been widely used in aerodynamic researches and constantly optimized due to the technological development and the need for a detailed description of turbulent flows. Fig. 5 shows the measurement system used for the experiment of this work [7].

The probe is located in one of the four sides of a Wheatstone bridge and an electronic circuit powers it. According to the operation principle of the Wheatstone bridge, all sides have the same electrical tension, since the resistance is the same on all four sides. Therefore, the electric current passing through the bridge has the same intensity at the four points.

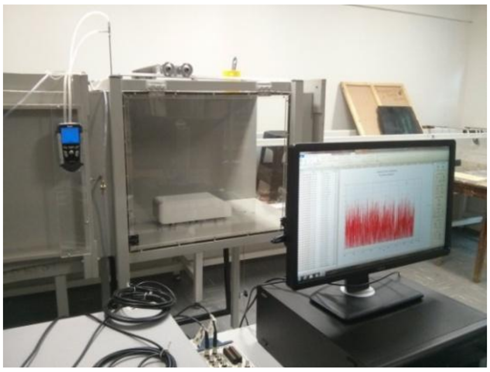


Fig. 5. Measurement system used on the experiment.

When the sensor is inserted into the flow, the heat exchange of the wire heated by the electric current causes an imbalance in the bridge for the change in temperature implies changing the resistance of the sensor side. This electric tension change can be measured and related to the flow velocity through the 4th order polynomial seen in Fig. 6. The correlation is given by the polynomial $U = -0.000002E^4 + 0.0001E^3 + - 0.0041E^2 + + 0,0764E + 1.5827$.

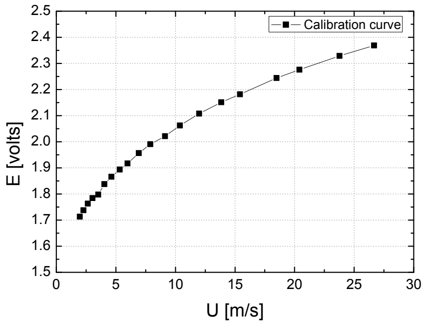


Fig. 6. Calibration curve of the hot-wire anemometer.

The typical dimensions of the heated wire are 5 μm in diameter and 1 to 3 mm in length [8]. The hot-wire used in the experiment is from the manufacturer DANTEC DYNAMICS, model 55P11 1D[7]. Fig. 7 shows the Hot-wire anemometer on the back of the Ahmed Body inside the test-section. A mechanical support was used to place the anemometer inside the test-section with reasonable accuracy in location and also avoiding vibration.

Fig. 7. Hot-wire anemometer on the back of Ahmed Body.

1.1.3 The Ahmed Body

The Ahmed Body is a simplified model of a vehicle, invented by Ahmed in 1984 [9]. It is widely used on experiments over the world, which is the reason it is possible to find many data from different sources and laboratories. The geometry of the Ahmed Body is similar to a general car with a flat front and a slanted back. Detailed dimensions and geometry used in this work is shown in Fig. 8 and Fig. 9.

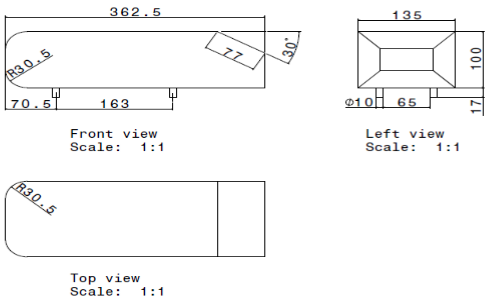


Fig. 8. Dimensions of the physical model used on the main experiment of this work.

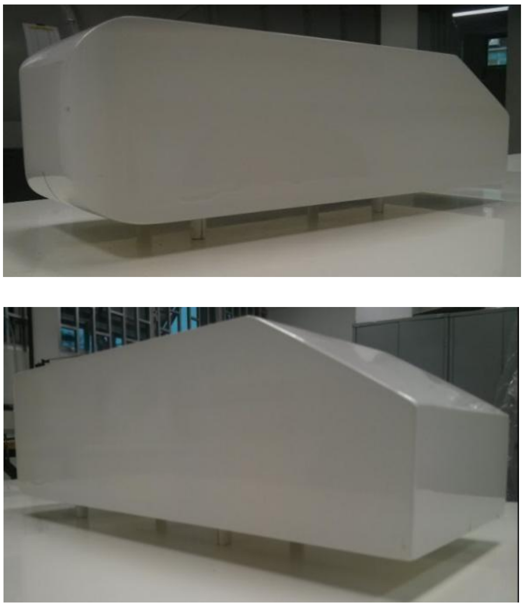


Fig. 9. Physical model for wind tunnel.

Beside the analyses of the velocity field, the Ahmed Body is also used for drag force and lift force analyses and rolling, pitching and yawing moments. The slant angle on the back allows studying the separation region and its influence on the velocity profile and forces. In this case we have selected 30° of slanted angle in the back.

1.2 Numerical Approach

The main reason to use computational simulation was to compare the results from this simulation with the experimental results and data from literature. According to the results of this comparison, it is possible to check more reliability for the results from the numerical technique. The main parameters analyzed are the velocity field, lift coefficient and drag coefficient of the body. The STAR CCM+ software was used for simulation.
The dimensions of the computational domain are 31L x 10L x 7.5L where L is the length of the body, as seen in Fig. 10.

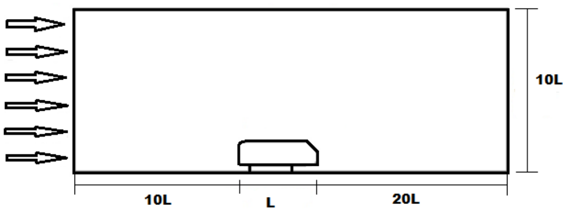


Fig. 10. Computational domain used in computational simulation.

The simulation was performed in steady state regime with 5000 iterations reaching a residual level of order 1×10^{-4}. The turbulence model was RANS k-ω SST (Shear Stress Transport). This turbulence model was used since the flow is considered to be completely detached on the back of the Ahmed body followed by a turbulent flow with regions of recirculation behind the body.

In order to get a better simulation, the domain was divided into ground, inlet, outlet, roof, side wall and plane of symmetry. The boundary conditions applied to the simulations were:

- Inlet: uniform velocity
- Outlet: Atmospheric pressure
- Roof: stationary wall
- Ground: stationary wall
- Side wall: stationary wall
- Plane of symmetry: symmetry
- Ahmed Body: stationary wall

It was used tetrahedral mesh and prismatic layer for the boundary layer region. The total amount of cells was about 2.3 millions. The mesh was refined on critical regions like regions close to the body and behind the body (wake flow region). The mesh conditions were chosen based on satisfactory results of Souza [10]. The computational mesh is seen in Fig. 11.

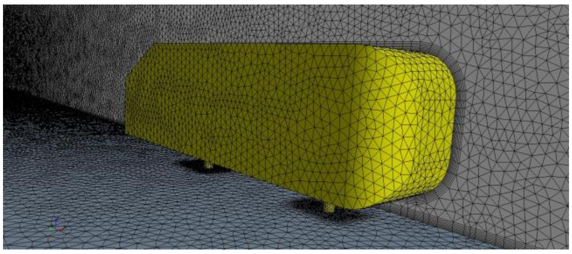


Fig. 11. Computational domain. Detail of the mesh in the body and plane of symmetry.

III. VALIDATION OF THE NUMERICAL APPROACH

In order to validate the numerical approach using STAR CCM+ software, it was compared the experimental data from ERCOFTAC available online with the results of the computational simulation. For this work, the "Case 9.4 – Flow around a simplified car (Ahmed Body)" [6] was used for comparison.

This case has been simulated using the same conditions as the Case 9.4 from ERCOFTAC and the same dimensions of the Ahmed Body (Fig. 12). The parameters used are slant angle of 25°, kinematic viscosity of $1.5\,\mathrm{x}10^{-5}\mathrm{m}^2/s$ and flow velocity of 40 m/s. The computational domain was described on section 2. The results from STAR CCM+ showing velocity contours and streamlines can be seen on Fig.13.

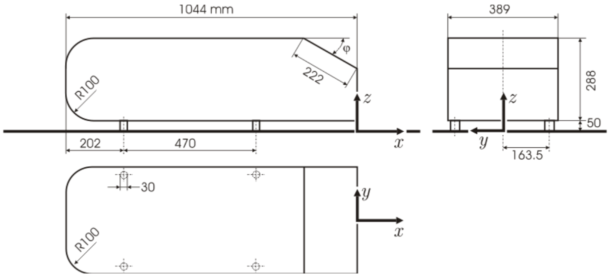


Fig. 12. Dimension of the Ahmed Body [6]

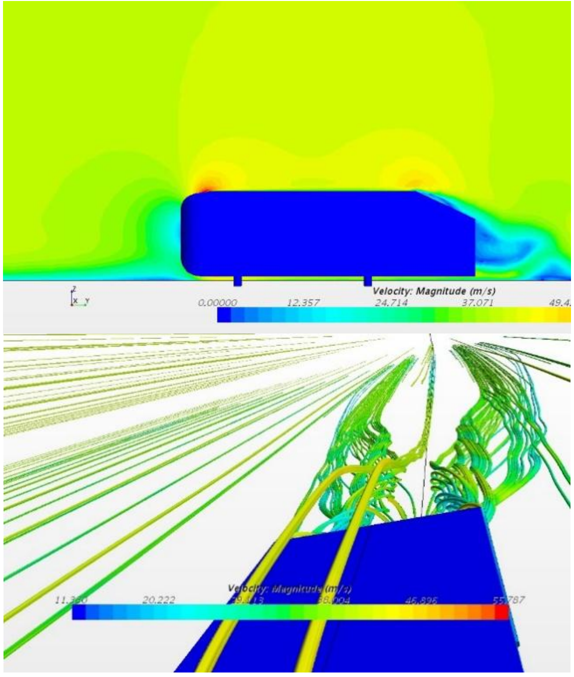


Fig. 13. Velocity contours and streamlines.

The data from the streamwise velocity was collected in four different positions: 38 mm, 88 mm, 138 mm and 188 mm from the back of the Ahmed Body (the origin reference is the rear end of the Ahmed body). Fig.14and Fig. 15 show the pattern of the u-velocity component on the symmetry plane.

Both profiles in Fig. 14 and Fig. 15 presented the lowest velocitiesalong 50 mm and 250 mm in height in all positions, which probably were occasioned by the recirculation region (low pressure region) behind the body. As the radial position is moved downstream in the wake, it is possible to observe a reduction in this velocity profile inflexion. Outside this region of lower velocities it is possible to see some small discrepancies between the ERCOFTAC data and our current simulations. However, it can be seen a satisfactory similarity of the profiles with reasonable identification of the flow actual flow structures, which gives reliability for using the computational approach selected in this work.

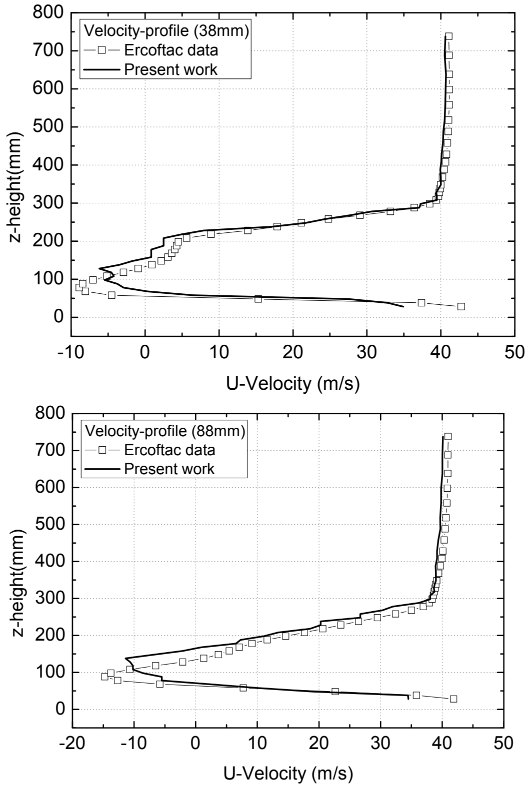
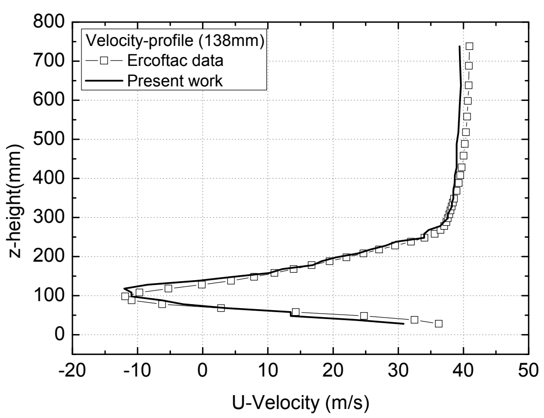


Fig. 14. Velocity profiles for validation of the numerical technique – 38 and 88mm.

It is important to comment that the numerical modeling was performed by using a RANS approach with turbulence modeling to characterize a flow pattern that is naturally transient in actual condition with large regions of flow detachments and recirculating flows in the wake of the body. As a mean flow-prediction's method this numerical model is showing reasonable results allowing the use of this approach for doing design selections, at least, for industrial applications with a low cost when compared to other numerical techniques such as LES (Large Eddy Simulation) and DNS (Direct Numerical Simulation).



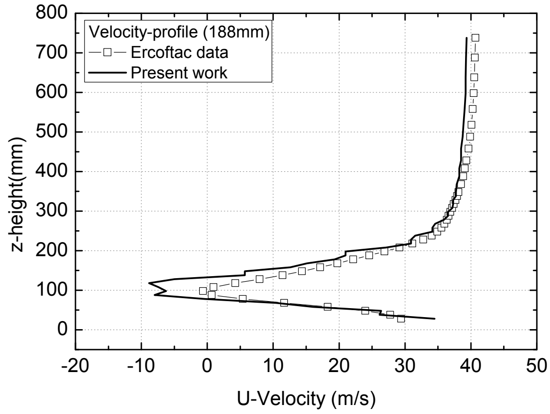


Fig. 15. Velocity profiles for validation of the numerical technique – 138 and 188mm.

IV. RESULTS AND DISCUSSION

After the validation of the numerical approach used in this work, the next step was to generate new computational simulations for the experiments with the Ahmed body,carried out in thewind tunnel at CPAERO as explained in the Methodology.

The main differences between the Ahmed bodies used for validation and the physical model in our experiment were the slant angle (inourexperiment the angle was 30° degrees), the dimensions of the body (Fig. 8) and the velocity range. For the physical model in our experiments we planned to work with most critical scenario for further turbulence analyses on Ahmed Body where separation flow occurs, that is for a 30º slant angle based on literature [9]. Based on the dimension of the test section of the wind tunnel, we calculated the best dimension for the Ahmed Body for a reasonable blocking ratio (3.84% in this case).

After obtained the resultsfrom the validation for computational data, the next step was comparing experimental data with computational data, aiming a better comprehension of the flow around the body and also identifying potential limitations from each technique employed. Then it was manufactured a physical Ahmed body (Fig. 9) with dimension shown in Fig. 8 for the experiment in wind tunnel.

The experiment in the wind tunnel was set for three different velocities: 10m/s (Re = 0.7×10^5); 15m/s (Re = 1.05×10^5) e 20m/s (Re = 1.4×10^5). Computational simulations were performed in STAR CCM + looking for the same geometric and physical conditions of the experimental approach and same conditions as showed on section 2.2.

Three positions were set for the velocity measurement and profiles comparison in all three cases. The first position (P1) was located 160 mm from the front of the body, the second position (P2) was located 48 mm from the end of the body, and the third position (P3) was located 97 mm from the end of the Ahmed body (49 mm from the second position), as seen in Fig. 16. The maximum height measured in the experiment was 160 mm. As mentioned, a mechanical support was used to place the hot-wire probe inside the flow in the indicated positions.

In the next subsections the u-velocity profiles will be shown, including the comparison with the CFD simulations performed for each specific position P1, P2 and P3.

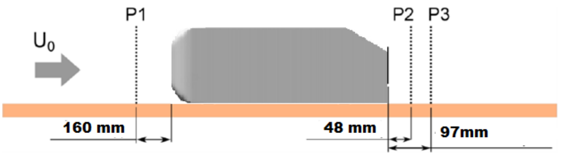
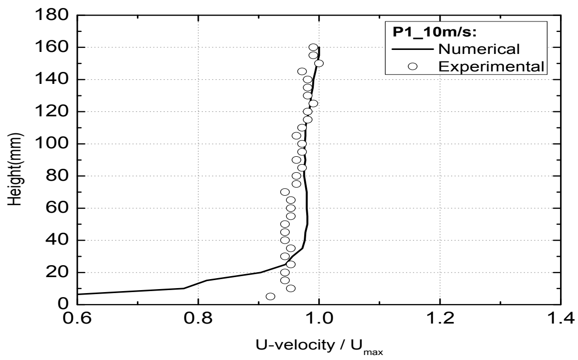
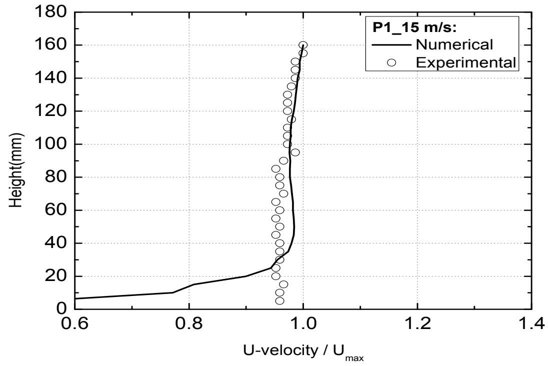


Fig. 16. Positions for the velocities profiles: wind tunnel experiment and computational simulation.

1.3 Velocity Profile Comparison in P1

The comparison between experimental and simulation data for the position P1 is shown in Fig. 17. The position P1 was in front of the body, where the flow is completed well established. The first boundary layer measured point in the experiment was about 1.5 mm, increasing up to 160 mm. The velocities profiles were normalized independently by dividing the measured velocity (u-velocity) by the maximum velocity (U_{max}) measured along the vertical experimental line. This allowed verifying the similarity in the velocity profiles in both experimental and numerical data.





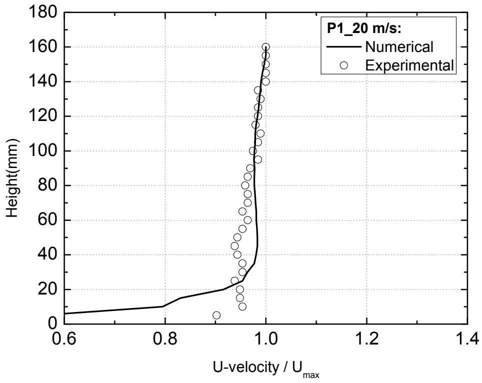


Fig. 17. Velocity Profile for position P1.

It is notable the similarity on both data at P1 for different velocities. The position in front of the body have almost none influence of the body, so it is expected a constant velocity as seen in Fig. 17. The region where there is no match between data was on the boundary layer region.

One of the main reasons for the non-coincidence of data would be the in fact that, in the computational model, there was an imposition of a uniform velocity profile as a boundary condition. This boundary condition imposition associated with the wall-function modeling led to a development of the boundary layer which is closely different in the first measured positions close to the wall. On coming flow simulations have been performed to evaluate the influence of the boundary conditions in the final simulations, taking into account the actual boundary layer profile.

1.4 Velocity Profile Comparison in P2

The comparison between experimental and simulation data for the position P2 is shown in Fig. 18 for the three velocities investigated.

In all cases, the effect of the recirculation region (also called recirculation bubble) behind the body is seen, creating an inflection in the velocity profiles at intermediate positions measured.Consequently, it is seen a velocity drop in this region (with negative velocities calculated, indicating the recirculating flow). However, as can be seen, the result from the computational simulation has this inflection much more pronounced than the experimental results with values much smaller than those collected by the sensor.

Thecause of this discrepancy was due to the fact that the sensor (hot-wire) was not able to measure negative velocities. In fact, this result can be associated to the limitations of the measuring technique, once values outside the calibration region (including negative velocities) are not correctly acquired. Also, according to the numerical results, seen further, the points P2 and P3are well inside where the recirculation bubble was located.

The hot-wire technique is not a proper approach for measuring inside recirculating flows and should be used for well-established flow regions.

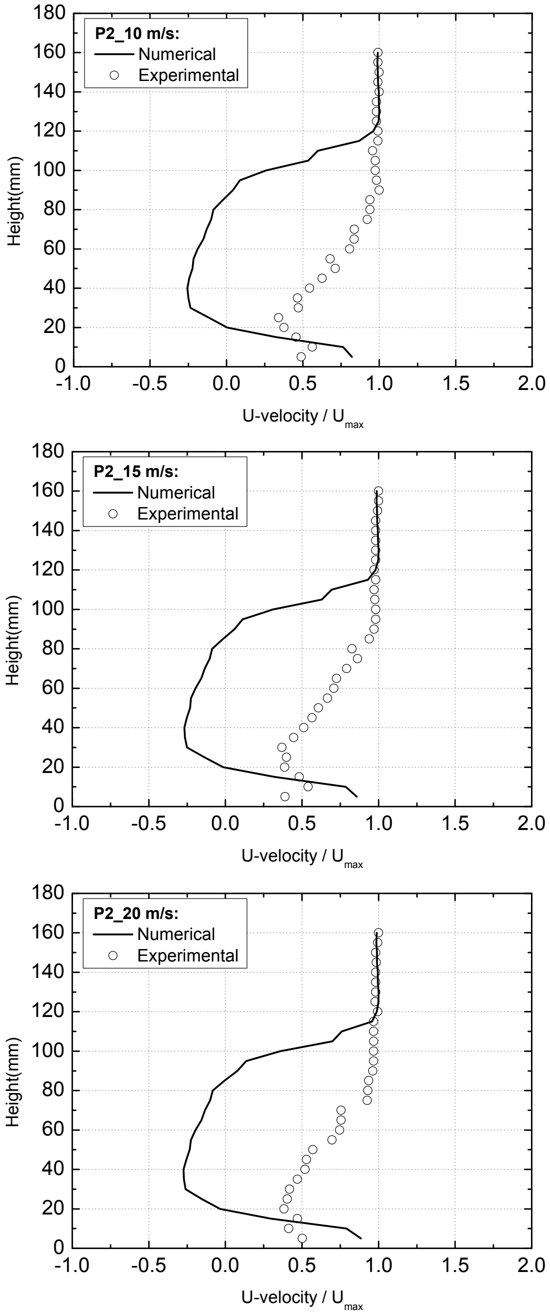


Fig. 18. Velocity Profile for position P2.

1.5 Velocity Profile Comparison in P3

In the position P3, one can observe the complete qualitative and quantitative disparity in the recirculation region in both cases. Although the values in the free flow region are very close in both cases, the profiles have opposite trend in the other regions, as seen in Fig. 19, for the reasons already explained.

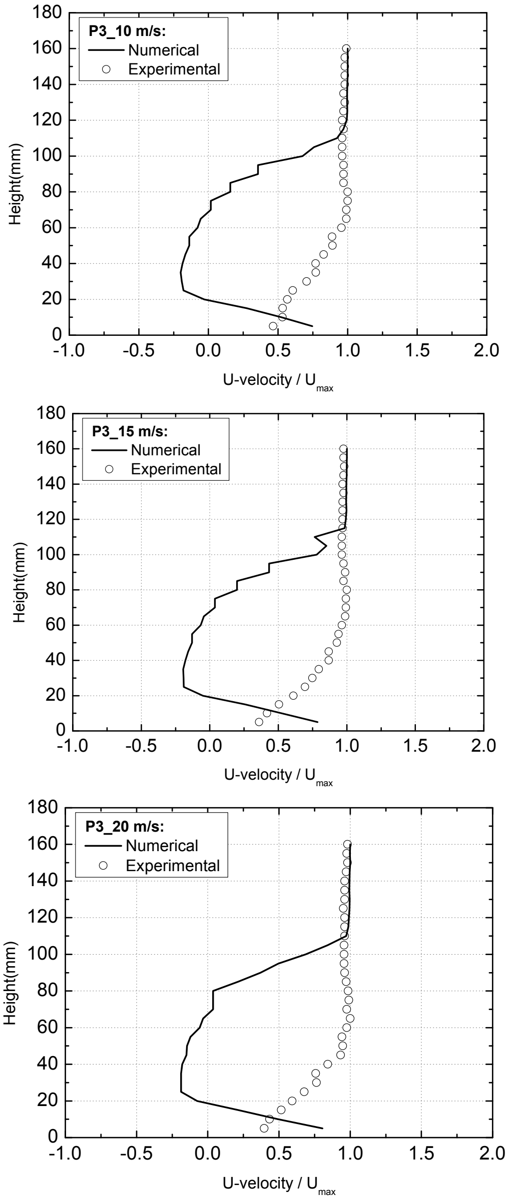


Fig. 19. Velocity Profile for position P3.

While the velocity profiles from the CFD simulation indicated negative velocity values in the recirculation region, the velocity values of the experiment shows a profile trend as it was located outside the recirculation with smother curves. This analysis leads to the possibility of the recirculation bubble in the simulation being larger than the bubble generated in the experiment.According to the flow analysis performed through visualization tools by using the software STAR CCM+, one could estimate the size of the recirculation bubble for the three cases investigated. Table 1 illustrates the recirculation region sizing behind the Ahmed body obtained numerically. Fig.20 presents the flow field in the back of the Ahmed body, visualized by vector plots, for 15 m/s.

Table 1. Recirculation region – Numerical results.

Recirculation bubble		
10 m/s	15 m/s	20 m/s
0.150	0.140	0.135

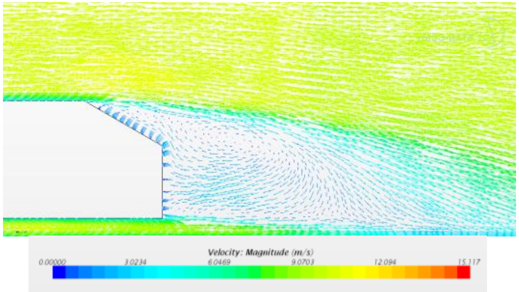


Fig. 20. Vector plot of the flow behind the Ahmed body.

Since there was no information about the size of the recirculation bubble before the experiment, the velocity acquisition points were chosen in the tunnel 48 mm and 97 mm from the rear of the body. As commented, for a massive recirculation of the flow, using the anemometer as a measurement technique is not indicated. Therefore, the experimental profiles should be taken in positions fartheraway from the back of the body. As an ongoing research, new velocity profiles will be undertaken and a deeper analysis will be performed.

Finally, by analyzing the computational result for drag coefficients evaluated by Equation 1, it can be seen from the Table 2 that the drag coefficient drops as the velocity increases. The values presented in Table 2are very close to those ones listed in literature.

Table 1. Drag coefficient – Numerical results.

Drag Coefficient (Cd)		
10 m/s	15 m/s	20 m/s
0.3042	0.2943	0.2886

Table 1

$$C_D = \frac{D}{\frac{1}{2}\rho V^2 A} \qquad (1)$$

V. CONCLUSION

The results obtained from experiments using the Ahmed body allowed some understanding about how the flow behaves around such geometry. The techniques employed in this work could also be used further to study aerodynamics of heavy-trucks. Mainly through the work of Hakansson and Lenngren [5], it can be seen current and future applications in the improvement of the aerodynamics of trucks.

The Ahmed body’s computational simulation results in this work were validated by the experimental results from ERCOFTAC [6]. Despite this, the comparison of the computational results with the experimental results obtained in the wind tunnel in this work contains some inconsistencies, especially for the recirculation region behind the body.

One source of errors identified was the location of the body's recirculation bubble. By analyzing velocity profiles behind the body, the velocity values captured in the simulation are much smaller than those captured in the experiment confirming the presence of a recirculation bubble just behind the body. Next experiments efforts could confirm this, including a sizing for this recirculating region behind the body.

As these results are part of an ongoing study, for future work it is proposed a study of velocity profiles in the slanted region of Ahmed body, and study of velocity profiles in the post-recirculation region of Ahmed body. It is also proposed visualization with "tufts" and "China Clay" (aiming a better understanding of the flow around the body in the wind tunnel) and study of mesh refinement, aiming a better accuracy in numerical results.

REFERENCES

[1] Cooper K. R.,Commercial Vehicle Aerodynamic Drag Reduction: Historical Perspective as a Guide, *National Research Council of Canada*, 2004.

[2] J. Woodroofee, Reducing Truck Fuel use and Emissions: Tires, Aerodynamics, Engine Efficiency, and Size and Weight Regulations,*Report No. UMTRI-2014-27, University of Michigan*, 2014.

[3] K. R.Cooper.,Commercial Vehicle Aerodynamic Drag Reduction: Historical Perspective as a Guide,*National Research Council of Canada*, 2004.

[4] F. Browand, Reducing Aerodynamic Drag and Fuel Consumption,*Global Climate and Energy Project, Workshop on Advance Transportation*, 2005.

[5] C. Hakansson, M. J. Lenngren, CDF Analysis of Aerodynamic Trailer Devices for Drag Reduction of Heavy Duty Trucks,*Chalmers University of Technology*, 2010.

[6] B. Basara, CASE 9.4 - Flow around a simplified car body (Ahmed body), 2001,*Available in http://www.ercoftac.org/fileadmin/user_upload/bigfiles/sig15/database/9.4*. Access on August 2016.

[7] MiniCTA Anemometer Package, How to get started - a quick Guide. *Dantec Dynamics A/S*, 2004.

[8] Hot-wire measurements, Lecture notes. Experimental Aerodynamics Class, *Middle East Technical University*.

[9] S. R. Ahmed, Some salient features of the Time Average Ground Vehicle, *SAE-Paper 840300*, 1984.

[10] P. R. C. Souza, *Numerical and Experimental Analysis of subsonic jets*, Undergraduation thesis, Federal University of Uberlândia, Uberlândia, 2012.