World Academy of Science, Engineering and Technology
International Journal of Mechanical and Mechatronics Engineering
Vol:9, No:1, 2015

Aerodynamic Analysis of a Frontal Deflector for Vehicles

C. Malça, N. Alves, A. Mateus

***Abstract*—**This work was one of the tasks of the Manufacturing2Client project, whose objective was to develop a frontal deflector to be commercialized in the automotive industry, using new project and manufacturing methods. In this task, in particular, it was proposed to develop the ability to predict computationally the aerodynamic influence of flow in vehicles, in an effort to reduce fuel consumption in vehicles from class 3 to 8. With this aim, two deflector models were developed and their aerodynamic performance analyzed. The aerodynamic study was done using the Computational Fluid Dynamics (CFD) software Ansys CFX and allowed the calculation of the drag coefficient caused by the vehicle motion for the different configurations considered. Moreover, the reduction of diesel consumption and carbon dioxide (CO_2) emissions associated with the optimized deflector geometry could be assessed.

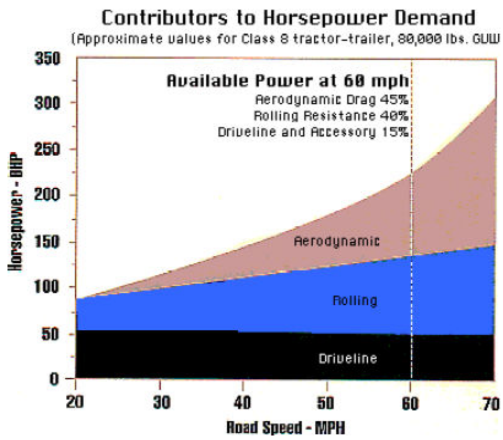
***Keywords*—**Aerodynamic analysis, CFD, CO_2 emissions, Drag coefficient, Frontal deflector, Fuel consumption.

I. Introduction

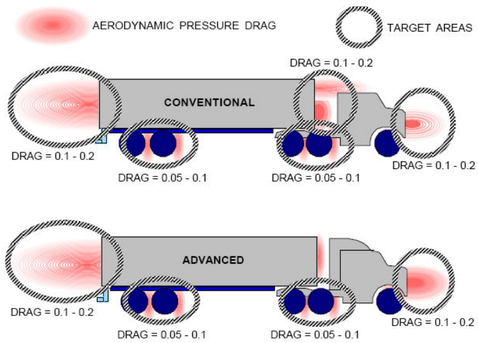
DRAG force reduction remains one of the main challenges in vehicle aerodynamics research, since fuel consumption and, consequently, CO_2 emissions can be significantly reduced if aerodynamic drag decreases [1]-[3]. Due to their large dimensions and weight, trucks are submitted to high drag forces, which are higher the greater the vehicle speed forces, in particular those derived from aerodynamic drag as shown in Fig. 1. The drag force is an aerodynamic force that opposes to the vehicle motion, due to the combined effects of air friction in the vehicle walls and pressure forces of the air motion. The drag coefficient is calculated analytically through (1) where C_d is the drag coefficient, F_d the drag force, ρ the air density, v the vehicle velocity and A the frontal area.

$$C_d = \frac{F_d}{0.5 \times \rho \times v^2 \times A}. \quad (1)$$

It is reported in the literature that a typical truck with an average drag coefficient of 0.6 and driving at 110 km/h spends 65% of its fuel overcoming aerodynamic drag [4], [5]. In addition, it is pointed out that 70% of the break power of a vehicle engine is consumed to overcome the aerodynamic drag generation of the vehicle at 100 km/h [2]. A reduction in drag force results in a fuel consumption reduction, which would both save money and conserve natural resources [1]-[6]. This can be attained by several strategies. It is recognized that one of the most effective methods to reduce drag is to change the body vehicle geometry through e.g. the use of deflectors. This is because the aerodynamic force mainly depends on the body shape of the vehicle [5], [6]. It is, however, of paramount importance to have efficient means to understand the flow-field around the vehicle and in its wake in order to design, develop, and test under operational conditions aerodynamic devices that would improve vehicles fuel economy [6]. In this work, Computational Fluid Dynamics (CFD) flow simulations were performed using Ansys CFX, to assess the flow structure around the vehicle and the influence of frontal deflector geometries on vehicle aerodynamic coefficients, with the aim of increasing fuel efficiency in vehicles from class 3 to 8.



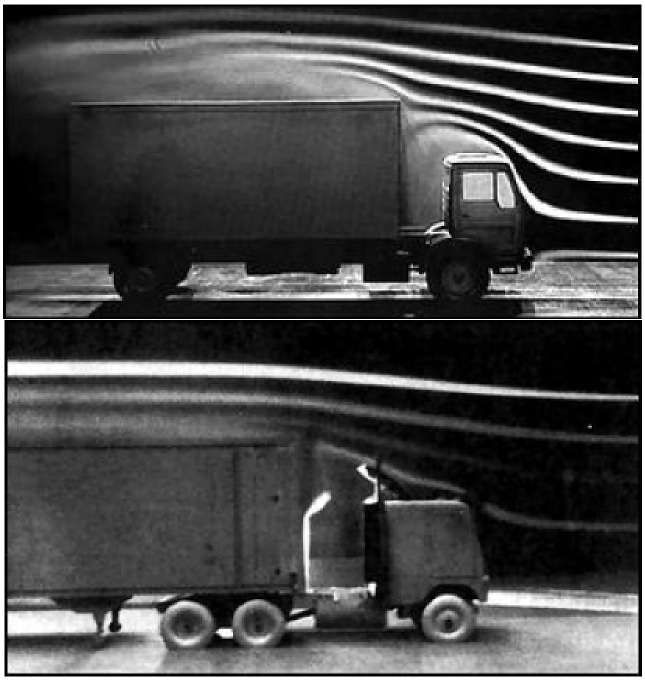
(a)



(b)

C. Malça is with the Mechanical Engineering Department, Polytechnic Institute of Coimbra, R. Pedro Nunes, 3030-199 Coimbra, Portugal and with the Centre for Rapid and Sustainable Product Development, Polytechnic Institute of Leiria, Rua de Portugal, 2430-028 Marinha Grande, Portugal (e-mail:candida@isec.pt).

N. Alves and A. Mateus are with the Centre for Rapid and Sustainable Product Development, Polytechnic Institute of Leiria, Rua de Portugal, 2430-028 Marinha Grande, Portugal (e-mail: nuno.alves@ipleiria.pt, artur.mateus@ipleiria.pt).



(c)

Fig. 1 (a) Horsepower requirements versus vehicle speed; (b) Target areas and (c) streamlines of aerodynamic drag distribution for a truck with and without deflector [3]

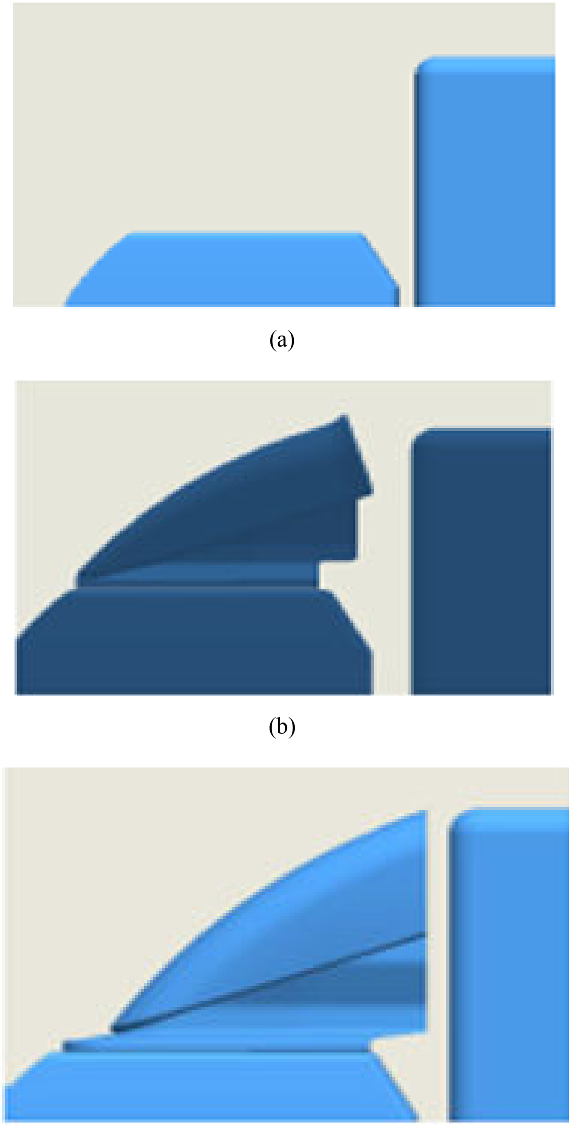
II. Computer Fluid Dynamics Simulation

The Ansys CFX computational tool was selected to perform the aerodynamic analysis of the three alternative configurations considered as illustrated in Fig. 2: (a) without deflector; (b) 1st deflector model and (c) 2nd deflector model.

For each configuration, two simulations at two different speeds, 60 km/h and 120 km/h, were conducted.

In terms of simulation parameters, the boundary conditions of flow were set up and the turbulence k-ε method was chosen. Although being the simplest, using only two equations, this method is recognized as robust and suitable for early iterations [2], [7]. Taking into account the experimental conditions and requirements for obtaining a higher computational accuracy, the following simplifications were considered for simulation purposes: i) the influence of small devices such as wing mirrors, is not considered; ii) the temperature, pressure and viscosity of the reference air are constant; iii) the air behaves as an incompressible fluid; and iv) the flow around the truck is stationary.

Fig. 3 shows, for a velocity of 120 km/h, the pressure distributions on the front of the truck for the three alternative configurations. It can be concluded that without deflector there are two main zones with a high pressure in the vehicle front. With the deflector, the high pressure zone above the truck cab is dissipated, which means that the air that is incident on the vehicle front has a more uniform distribution. This dissipation is greater for the 2nd deflector model, which proves its better aerodynamic behavior. Figs. 4 and 5 show, respectively without deflector and with the 2nd deflector model configurations, the pressure and velocity contours for the lateral panel of the truck at a velocity of 120 km/h. It can be concluded that with the configuration corresponding to the 2nd deflector model both pressures and velocities are significantly reduced. Once again it is demonstrated the superior aerodynamic performance associated with the 2nd deflector model.



(a)

(b)

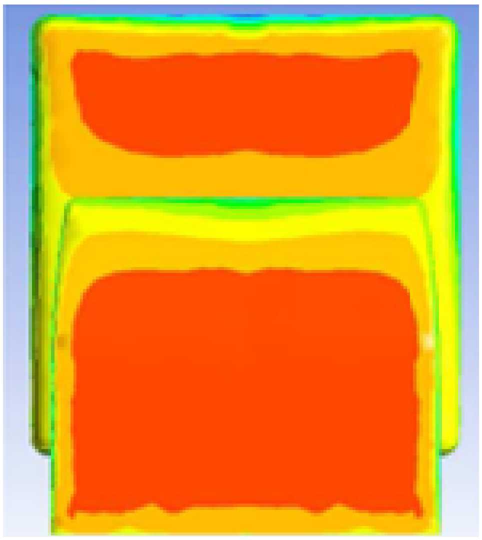
(c)

Fig. 2 The three alternative configurations: (a) without deflector; (b) 1st deflector model and (c) 2nd deflector model

Knowing the pressures involved and the forces applied on the different configurations, drag coefficients were calculated for speeds of 60 km/h and 120 km/h using (1).

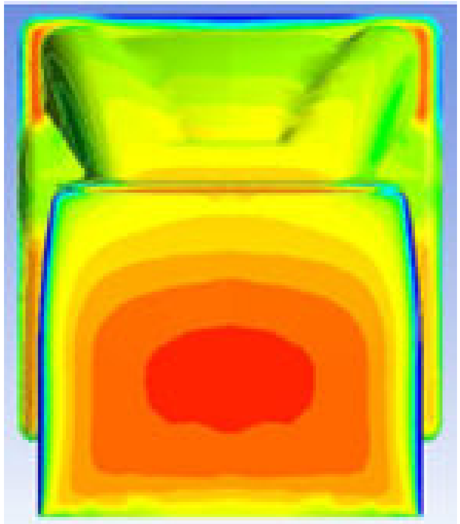
Table I summarizes the drag coefficient values for the configurations and velocities considered. From the results presented in Table I, it can be concluded that with the 2nd deflector model, the drag coefficient significantly decreases; resulting in improved vehicle performance and thereby reducing fuel consumption.

A calculation of fuel consumption was carried out for the different configurations and velocities. For a velocity of 120 km/h, the use of the 2nd deflector model leads to a fuel consumption corresponding to a less 5.62 L/100km/h when compared with the no deflector configuration as shown in Fig. 6. In short, a reduction of fuel consumption of about 15% and 7% was achieved with the 2nd deflector geometry when compared with the configurations of no deflector and 1st deflector model, respectively.



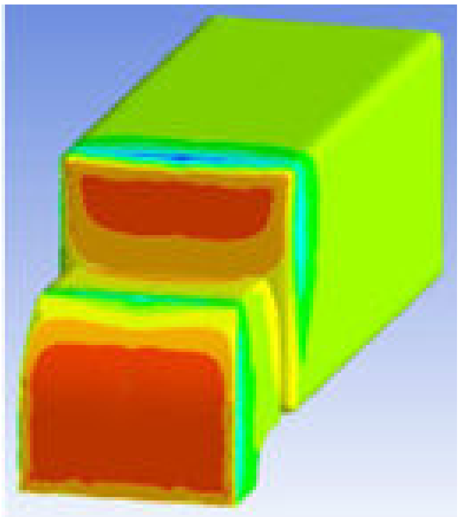
(a)

(b)

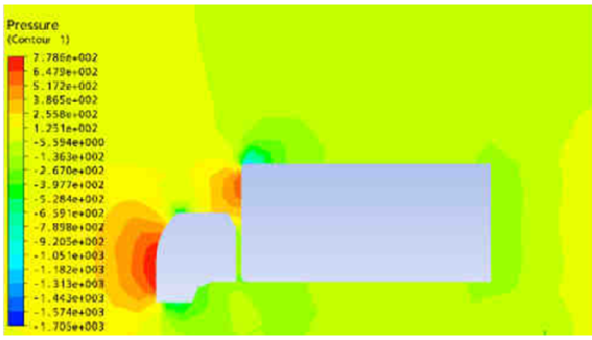


(c)

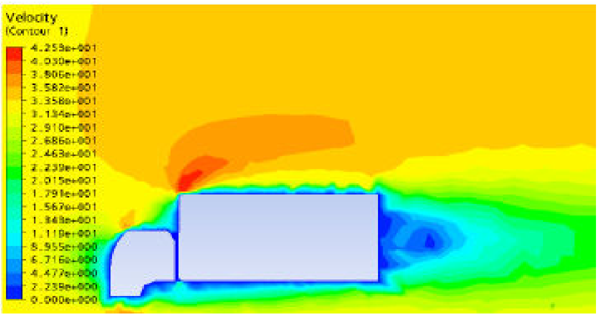
Fig. 3 Pressure fields for the configurations: (a) without deflector, (b) 1st model and (c) 2nd model deflector



(a)



(b)



(c)

Fig. 4 Simulation results without deflector configuration: (a) pressure contour on the front of the truck; (b) pressure contour on the lateral panel of the truck and (b) velocity contour on the lateral panel of the truck

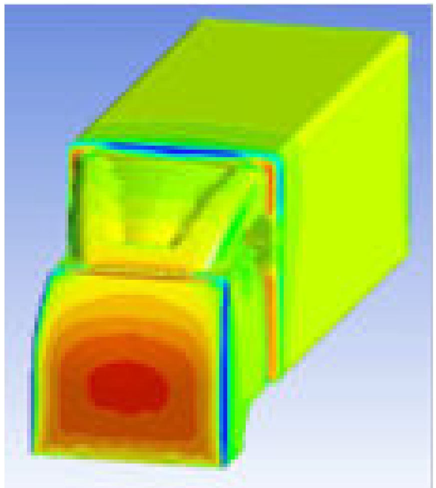
TABLE I
DRAG COEFFICIENT FOR THE CONSIDERED CONFIGURATIONS

Configurations	C_d
Without deflector – 60 km/h	1.080
Without deflector – 120 km/h	1.084
With 1st Deflector – 60 km/h	0.990
With 1st Deflector – 120 km/h	0.993
With 2nd Deflector – 60 km/h	0.9251
With 2nd Deflector – 120 km/h	0.9254

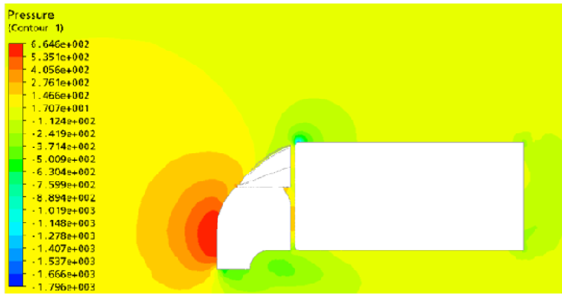
III. CONCLUSION

In order to reduce the drag force to which vehicles from class 3 to 8 are submitted, two alternative models of frontal deflectors were developed. The aerodynamic influence of the application or not of a frontal deflector in vehicles was analyzed through CFD simulations. A 2nd deflector model resulted from an optimization procedure of the 1st deflector model developed.

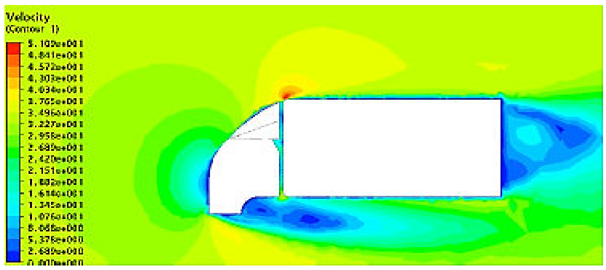
From CFD simulation results, it was concluded that applying the 2nd deflector model, at 120 km/h, the aerodynamic performance increased approximately 7% and 15% in comparison with the 1st and no deflector models, respectively. The differences found between the two alternative deflector models developed were due to the fact that the 1st model did not have the height well-adjusted to the back cargo. Using the 2nd model of the frontal deflector annual savings of 6825 € can be realized, assuming an annual average distance traveled of 100000 km and a diesel price of 1.024 €/L. Distributors can, in this way, contribute for the reduction of CO_2 emissions. Using the 2nd deflector model, the emissions of CO_2 are reduced by 17860 kg CO_2 for the same 100000 km/year, assuming that the combustion of 1 liter of diesel fuel releases 2.68 kg of CO_2. Finally, it is worth to report that a prototype of the 2nd frontal deflector model has been produced. The production process will be addressed in a forthcoming publication.



(a)



(b)



(c)

Fig. 5 Simulation results with the 2nd deflector model configuration: (a) pressure contour on the front of the truck; (b) pressure contour on the lateral panel of the truck and (c) velocity contour on the lateral panel of the truck

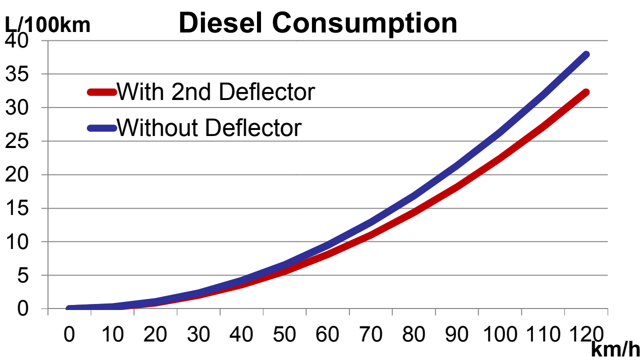


Fig. 6 Diesel consumption at 120 km/h for the configurations with and without the 2nd deflector model

ACKNOWLEDGMENT

Research supported by the Manufacturing2Client Project. Project N.º 5722 funded by Portuguese Innovation Agency.

REFERENCES

[1] R. J. Englar, "Advanced aerodynamic devices to improve the performance, economics, handling and safety of heavy vehicles," SAE Paper 2001- 01-2072, 2001.
[2] C. Kim, "A streamlined design of a high-speed coach for fuel savings and reduction of carbon dioxide," International Journal of Automotive Engineering, vol. 2, 2011, pp. 101-107.
[3] R.M. Wood, S. X. S. Bauer, "Simple and Low-Cost Aerodynamic Drag Reduction Devices for Tractor-Trailer Trucks," SOLUS – Solutions and Technologies, SAE International 2003-01-3377, 2003.
[4] P. Dasa, M. Tsubokuraa, T. Matsuukib, N. Oshimaa, K. Kitohc, "Large eddy simulation of the flow-field around a full-scale heavy-duty truck," Procedia Engineering, vol. 56, 2013, pp. 521–530.
[5] K. Ihlein, K. Toro, R. B. White, "On Drag Coefficients of Tractor Trailer Trucks," Truckin', ME 241Professor Roger Gans, 2007.
[6] H. Feng, "Low mass vehicle and its aerodynamic study," MSc Thesis, California state university, 2011.
[7] W. Yuana, H. Zhoua,J. Yaob, J. Xiec, "Numerical simulation research on the aerodynamic performance of concave building section", Procedia Engineering, vol. 31, 2012, pp. 206 – 212.