The Aerodynamic Study of a Body Truck

Bodea Sanda-Mariana(✉), Prodan Calin-Vasile,
and Scurtu Iacob-Liviu

Technical University of Cluj-Napoca, Cluj-Napoca, Romania
{sanda.bodea, vasile.prodan,
liviu.scurtu}@auto.utcluj.ro

Abstract. Nowadays, the required standards of automotive manufacturing involves complex studies with purpose of the fuel and noise reduction produced by the autovehicles in functioning. Shape optimization of the car body involves lowering the aerodynamic force on the car; the airflow must be have a more laminar distribution. The aims of this study are to determine the airflow distribution in functioning of the truck with trailer and the method to decreasing the drag coefficient. In the first part of the paper are presented the studies and graphical methods used to determine the airflow simulation. Using the CAD design and the modeling techniques, the body shape of the truck with trailer is modeled in SolidWorks software. The geometry is inserted in a virtual wind tunnel created in the Flow Simulation module. The virtual environment created imitates the real working conditions. The aerodynamic model is tested on the 25 m/s velocity of the truck. After running the simulation, the obtained results are analyzed and the shape of the truck is improved with air deflectors in the space between the cabin and trailer. Rear side of the trailer is slanted to reduce the swirl flow and the vacuum volume. All obtained result are compared, establishing the optimal geometry of the bodywork. The end of this study presents the conclusions and the future research.

Keywords: Aerodynamic study · Virtual wind tunnel · CFD
Bodywork shape

1 Introduction

The aerodynamic shape of the autovehicles has been improved lately filling the weaknesses and required standards. The main object of the autovehicles designer is to develop an optimal car body shape to reduce the drag force and lift force who act to external shape of the car body shape. Rapid development of numerical solver and the advanced computing system present great advantages to find the optimal body shape. Using dedicated CFD software advances studies can be virtually performed, highlighting the advantages of each component of the body structure. A significant role in autovehicles dynamics on the road is determinate by the aerodynamic forces, in this case the front air of the autovehicles is compressed, the air pressure increase and the pressure decreases in the rear of the car body. Decreasing the air resistance can be achieved by developing a better aerodynamic shape of the car body. The water droplet shows the smallest resistance to airflow, this shape is the best example for develop a car

N. Burnete and B. O. Varga (Eds.): AMMA 2018, PAE, pp. 73–79, 2019.
https://doi.org/10.1007/978-3-319-94409-8_9

body shape. The first aerodynamic car in the world was designed and built by the romanian engineer Aurel Persu (1890–1977). In 1924, Persu has obtained the patenting of the structure in Germany and in 1969; the car prototype was donated to the National Technical Museum Dimitrie Leonida from Bucharest [3]. Another CFD study is applied on the Ahmed body, a simplified car body, created by Ahmed in 1984, is a car rounded in the front side and in rear side has a slant angles between 0° and 40°. This geometry is modeled and simulated at 35/s of the air velocity using Fluent software (Ansys 15). The study of this geometry describes methods for reducing the drag coefficient [2]. The structure of the Ahmed body it used in automotive design serving as a CFD turbulence model benchmark for wind tunnel calibration [5]. A truck CFD analysis is simulated in Abaqus importing geometry from Catia. It is presented the pressure contour and the velocity vectors for various speeds [4].

1.1 Force on Autovehicles Body

During the autovehicles running, forces that acts on the body structure influence the autovehicles stability. Principal force that acts on the autovehicles body are: lift force, drag force and downforce from over body flow. Lift force act in the perpendicular direction of the relative fluid flow, causing the automobile lifting in the air when it is applied in positive directions, and when it is applied in the negative directions the effect is reversed, resulting an excessive wheel down force [5]. The relation 1 gives the expressions of this force.

$$F_y = \frac{1}{2} \cdot \rho \cdot V^2 \cdot C_L \cdot A \tag{1}$$

Where:
F_y = Lift force,
ρ = Air density = 1.225 kg/m^3,
V = Stream velocity,
C_L = Lift coefficient

Aerodynamic drag force is the force that act in opposite forwarding direction of the autovehicles. The value of this force is very important to establish the final shape of the car body. Relation 2 present the formula for aerodynamic force calculation.

$$F_x = \frac{1}{2} \cdot \rho \cdot V^2 \cdot C_D \cdot A \tag{2}$$

Where:
C_D = Drag coefficient

The lower drag coefficient facilitates a better ability of the car structure to achieve a higher speed relative to a similar vehicle with same power of engine but with a higher aerodynamic coefficient and lower noise resulted from air friction and fuel consumption. In the case of trailer trucks, the strength of air resistance is much higher due to the shape of the cabin surface. In order to improve the aerodynamic coefficient of the vehicles, designer mount deflectors, small shield, elerons to reduce the turbulent flow behind the car bodywork. The expressions of the drag and lift coefficients are presented below in relation 3 and 4 [5].

$$C_D = \frac{2 \cdot F_x}{\rho \cdot V^2 \cdot A} \tag{3}$$

$$C_L = \frac{2 \cdot F_y}{\rho \cdot V^2 \cdot A} \tag{4}$$

To understand how the aerodynamics forces act on the truck body while running on the road, the flow streamline distribution, in Fig. 1, are presented:

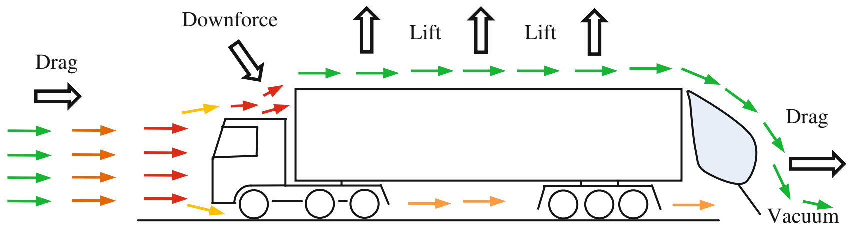

Fig. 1. Force on truck.

Due to design of the trailer truck, it can be observed as behind of the body a vacuum volume is formed, having a negative effect to forward on the road.

2 Virtual Computational Domain

Distribution of the airflow streamlines in the virtual environment can be determined following the steps presented in Fig. 2, where presented the logical flowchart of the graphical simulation process (Fig. 3).

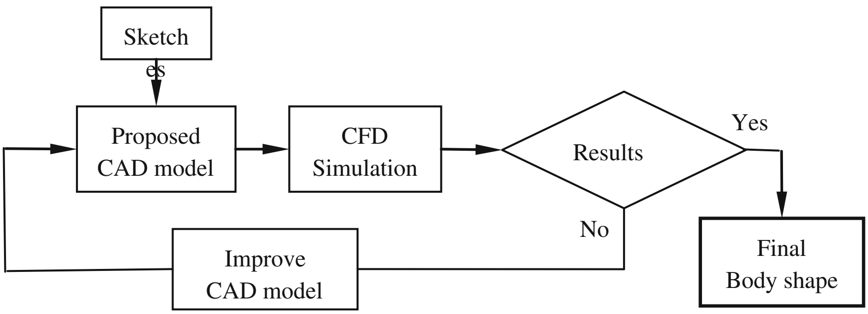

Fig. 2. Flowchart of the iterative CFD simulation process.

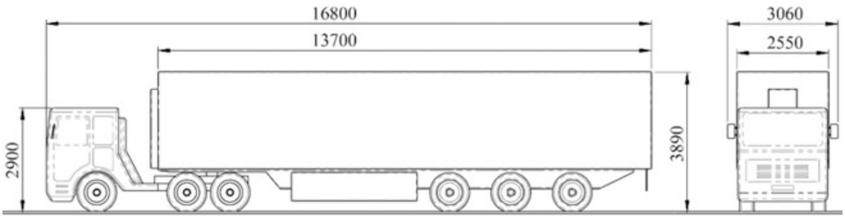

Fig. 3. Overall dimensions of the truck geometry.

The start point of the process is determined by the basic sketch design, where is establish the basic shape of the truck cabin and the trailer. Using advanced modelling techniques, the base sketch is modeled resulting the proposed body of the truck. The working conditions are applied for the proposed CAD model and this is introduced into a virtual wind tunnel. The obtained results are analyzed and if the proposed geometry satisfy the requirements, will be the final body geometry. In case if the requirements are not fulfilled the geometry of the proposed model are improved until the geometry satisfy the requirements.

The volume of the computational domain created is approximate 1440 m^3, including all geometry of the truck. The analysis length of the road is 40 m, a width of a 6 m and height 6 m. Dimensions of the computational domain are chosen in order to observe the distribution of the airflow while the truck running on the road.

3 Simulation Results

Computational resources for the modelling and CFD simulation are a HP z400 graphic workstation, with frequency 3000 Hz Intel Xeon processor, 16 GB RAM memory and NVidia Graphical card 1 GB.

In the first simulated version, is presented the basic geometry of the track without any element destined for reduce the swirl flow of air. Figure 4 shows the distribution of the airflow lines on all three-projection planes and the resulted value of the airflow velocity. After running the simulation, it can be seen in the space between the cabin and the trailer occur the turbulences that slowing down the advance on the road and in the rear of the trailer occur a swirl effect that has a vacuum effect.

Simulation results of the improved model are presented in Fig. 5, where can be view the lateral, top and front distribution of the velocity air streamlines of the improved geometry of the body truck, with deflectors located in the upper part of the cabin, in the space between the cab and the trailer sealing rear side.

After simulation of the optimized truck shape, can be observed a better laminar distribution of the air streamlines and reducing the vacuum volume behind the truck. The forces and coefficients listed in Tables 1 and 2 are calculated for two trucks body variants.

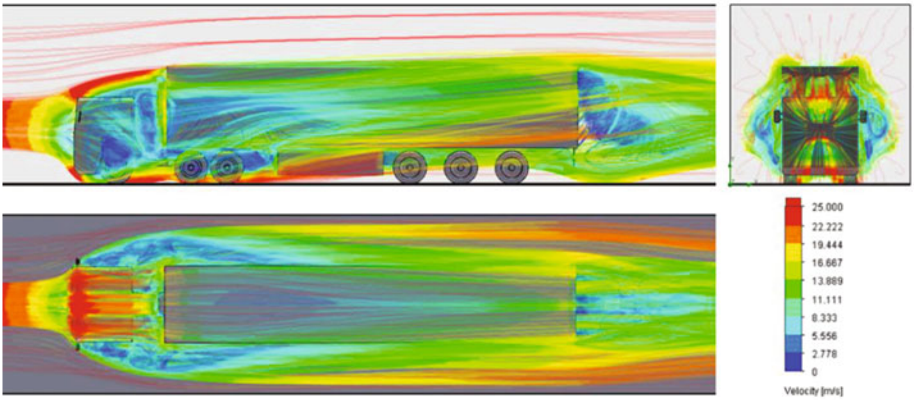

Fig. 4. Orthogonal view of the streamlines distribution of the base model (lateral view, top view and front view).

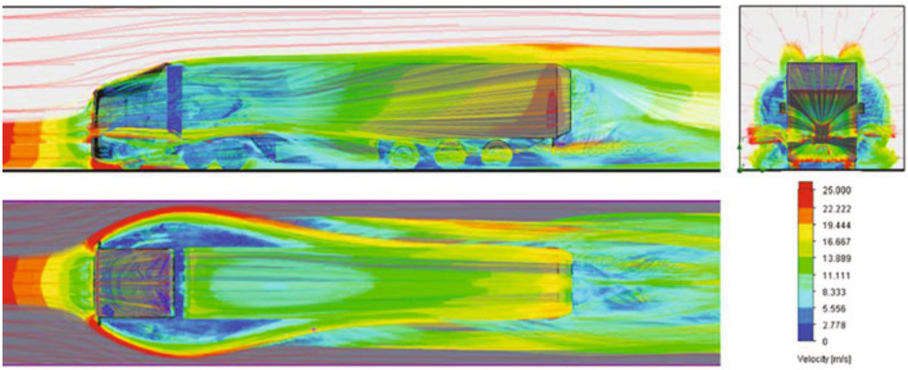

Fig. 5. Streamlines distribution of the improved model.

Table 1. Base model results

Basic geometry truck model	
Velocity	25 m/s
Lift force, F_y	1269.47 N
Drag force, F_x	2859.45 N
Drag coefficient, C_D	0.789
Lift coefficient, C_L	0.350
CPU time	879 s
Iterations	151

Table 2. Improved model results

Improved geometry truck model	
Velocity	25 m/s
Lift force, F_y	548.99 N
Drag force, F_x	2383.95 N
Drag coefficient, C_D	0.658
Lift coefficient, C_L	0.151
CPU time	658 s
Iterations	168

The lower drag force resulted on the improved model obtain a better aerodynamic coefficient compared to the base truck model. The resulted air pressure value on the improved truck geometry and the streamline distribution is presented in Fig. 5. The distribution of the airflow in computational domain and the resulted air pressure are presented in Fig. 6.

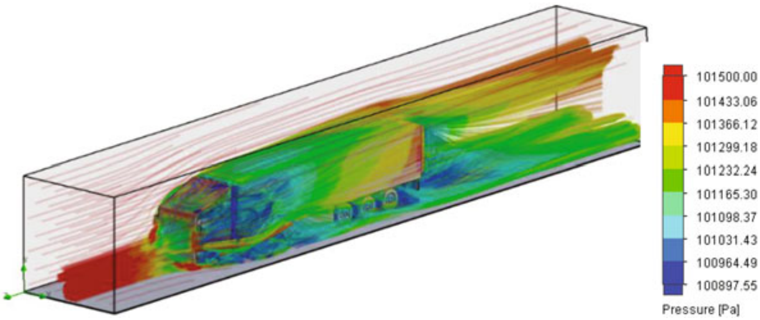

Fig. 6. Pressure air streamline distribution

It can be seen the maxim pressure is distributed on the cabin elements and the swirl effect created by the base model are removed by using the air deflectors and slant of the rear side of the trailer.

4 Conclusions

This study it is designed to show the advantages of using CFD method to determine the distribution of the air streamline to obtain a lower aerodynamic coefficient by installing the air defectors and adjusting the rear shape of the trailer. The advantages of using the CAD and CFD software are highlighted by the possibility of realizing several proposed car body geometry without the involvement of the material cost and the geometric optimization of the proposed models.

References

1. Franck, G., Nigro, N., Storti, M., D'elía, J.: Numerical simulation of the flow around the ahmed vehicle model. Latin Am. Appl. Res. **39**, 295–306 (2009)
2. Mitali, M.J., Suhani, S.P., Dhruvin, V.S., Arpit, N.P.: Computational study of air flow interactions and drag reduction techniques in simplified passenger vehicle. Int. J. Tech. Res. Appl. (43), 37–42 (2017). e-ISSN: 2320-8163. www.ijtra.com
3. Persu, A.: Streamline Power Vehicle. In: United States Patent Office, Serial no: 671,727 (1923)
4. Priyanka, D., Jayachandraiah, B.: Aerodynamic improvement of a truck body by using CFD. Int. J. Emerg. Technol. Comput. Sci. Electron. **13**(2) (2015). ISSN: 0976-1353
5. Shashank, A., Pankaj, G., Vishal, S., Shubham, M., Sourabh, T., Vishal, S.: Simulation of passenger car by attaching add-on device to reduce drag force by CFD approach. Int. Res. J. Eng. Technol. **4**(4) (2017). p-ISSN: 2395-0072, e-ISSN: 2395-0056