MACHINE DESIGN, VOL.10(2018) NO.4, ISSN 1821-1259 PP. 153-156

DOI: 10.24867/MD.10.2018.4.153-156
Research paper

3D MODELLING AND AERODYNAMIC SIMULATION OF A PASSENGER CAR

Nadica STOJANOVIĆ[1] - Jasna GLIŠOVIĆ[1] - Ivan GRUJIĆ[1,*] - Aleksandar DAVINIĆ[1] - Sunny NARAYAN[2] - Muhammad Usman KAISAN[3] - Shitu ABUBAKAR[3]

[1] University of Kragujevac, Faculty of Engineering, Sestre Janjić 6, 34000 Kragujevac, Serbia
[2] Qassim University, Faculty of Engineering, Saudi Arabia
[3] Ahmadu Bello University, Faculty of Engineering, Zaria, Nigeria

Received (01.05.2018); ***Revised*** (04.06.2018); ***Accepted*** (06.06.2018)

Abstract: *The shape of a passenger car is very important, both in terms of vehicle dynamics and for winning the leading position on the market. The vehicle's aerodynamics greatly affects fuel consumption, vehicle stability and safety during braking process. Today's software tools make it much easier to develop a new product - a car. This further contributes to the manufacturer to be competitive on the market, as well as reducing the necessary investment in developing a new model of car. The influence of air resistance on the fuel consumption and the occurrence of noise and vibrations, using the ANSYS software package, is analysed in the paper. Then the coefficient of aerodynamics of the vehicle was determined. Also, the loads that occur during the movement of the vehicle and what are the most exposed areas are shown.*

Key words: *dynamics, fuel consumption, ANSYS, loads, the most exposed areas*

1. INTRODUCTION

The aerodynamics of the vehicle have a significant place in the development phase of the car's body due to [1]:

- Better vehicle dynamics;
- Less fuel consumption;
- Increased lateral stability and
- Safest stopping.

During the movement of the vehicle, the air mass of the fluid flows along the exterior of the vehicle. This phenomenon occurs for two reasons. The first reason is that it is not possible to make an ideal smooth surface, and another because of the fluid's viscosity. The currents do not pass from one layer to the other, and there is no lateral movement relative to the direction of air flow. Turbulence occurs in the case of currents mixing, and in these cases, there is a change in speed. The vortexes are undesirable because they increase the resistance to vehicle motion. When the air currents encounter the vehicle's body, the currents are separated to reconnect when they go around the vehicle. Elements on the vehicle, such as bumpers, lights, mirrors, and other elements create additional air resistance due to the breakdown of the currents. When the currents arrive behind the vehicle, due to the difference in pressure on the front and rear end of the vehicle (the pressure is higher on the front than on the rear), a vortex occurs or the turbulence.

The values of the drag coefficient, for new-vehicle models on the market, are in range from 0.24 to 0.32, Table 1. It is interesting that the frontal area, even if it has higher values in some vehicles, has no negative impact on the drag coefficient [2]. The most influential factor is the drag area. In addition to the requirements it has to satisfy: to have been as low as possible drag coefficient and to have good stability at higher speeds, it is necessary that new-vehicle model conquers the largest market. As there is major resistance around tyres, the front of the vehicle must be such that it prevents the formation of such resistance. Heat exchangers are designed not to require airflow. Spoilers and fences guide air away from the front wheelhouse, and exiting air is routed under the car instead of out the wheel openings. The rear diffuser corrects the air currents, and in this way, it minimizes the lifting of the vehicle.

Table 1. C/D wind tunnel test results [2]

	Drag coefficient, [-]	**Frontal area, [m^2]**	**Drag area, [m^2]**
CHEVROLET VOLT	0.28	2.2	0.62
MERCEDES – BENZ CLA 250	0.30	**2.16**	0.65
NISSAN LEAF SL	0.32	2.28	0.72
TESLA MODELS P85	**0.24**	2.34	**0.58**
TOYOTA PRIUS	0.26	2.22	**0.58**

When it comes to the aerodynamics of a vehicle, accent is placed on the exterior design of the vehicle, while the interior is ignored and therefore, only the outer contour of the vehicle is modelled. A large number of world car manufacturers do not move from scratch, but improve the

*Correspondence Author's Address: University of Kragujevac, Faculty of Engineering, Sestre Janjić 6, 34000 Kragujevac, Serbia, ivan.grujic@kg.ac.rs

already existing model. So a large number of researchers take the contours of an already existing car and modify it so that they get the lowest possible drag coefficient. In the first phase, it starts from simplified models, i.e. the 2D model is analysed first [3] and in the case of fulfilling the expected results, then the 3D model is considered [4]. The aerodynamics of the vehicle is very complex, and it is required from the newly manufactured vehicles to be tested in air tunnels, which allow the measurement of drag, side force, lift, yaw, pitch and roll and yaw angles in the range of +/- 180deg, as well as smoke visualization [5]. The application of the software has enabled the faster development of the vehicle. It was found that the numerical analysis can be applied, but first the identification of parameters that the traction values approaches to the experimental must be performed [6]. However, a large number of studies are concerned only with the study of the influence of the vehicle's body shape on aerodynamics. So the geometry of wheels and suspension systems is ignored. As their influence is only 0.4% [7], and will be ignored in this paper, too.
Furthermore, some researches dealt with the influence of vehicle moving behind a larger vehicle (such as a bus). The critical distance is between 4 and 5 m, where the drag coefficient is higher, which also directly affects the fuel consumption [8]. On the other hand, some studies investigated optimal distances between a passenger vehicle and a truck's trailer, where this distance was 20 m [9]. Unlike these studies, there are those who deal with the influence of altitude on fuel consumption [10]. At higher altitudes, the air has a lower density, and therefore, the drag coefficient is also lower.

2. 3D MODEL AND BOUNDARY CONDITIONS

The model of the city vehicle (hereinafter referred to as the vehicle) contour was created in the Autodesk Inventor software package, figure 1. In the paper [3], an analysis of the 2D model of the vehicle has been already been carried out, and the optimal angle of the windscreen was determined for it. The optimal angle is the angle for which the lowest value of the drag coefficient will be obtained, and this is, in fact, the starting point for research in this paper. That is, the angle of the windscreen is 30 degrees. The software package that was used for the analysis is Ansys, and its module Fluid Flow CFX.

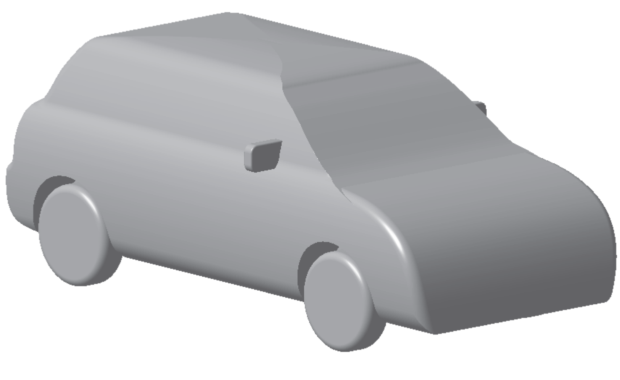

Fig.1. 3D model of the city car

Before starting the modules for determining the stresses acting on the vehicle while moving, as well as determining the drag coefficient, it is necessary to set up the mesh. The display of the generated mesh is shown in figure 2, where it can also be noticed that in the zone close to the surface of the vehicle, the mesh size is little smaller. The mesh size is smaller in the vehicle zone (size of mesh is 30 mm, type is tetrahedrons) to better present the loads, as well as the air currents that occur during the vehicle movement.

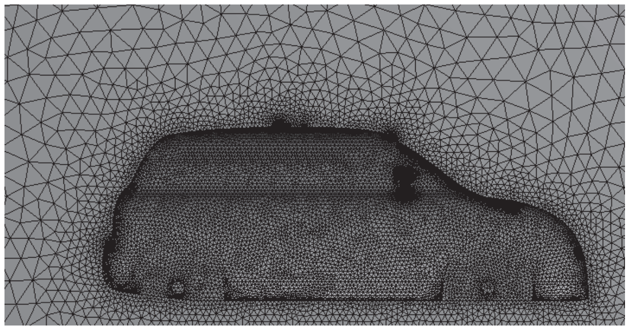

Fig.2. Generated mesh

Boundary conditions that are typical for such an analysis are:

- Air temperature, 25°C;
- Environment pressure is 101325 Pa, the pressure difference in considered area and outside it is 0 Pa;
- Air density 1.225 kg/m^3;
- Vehicle speed is 100 km/h, or more accurate of the air.

It is considered only the case when the air acting normally on the front side of the vehicle. The applied model is k-ω and it is very similar to k-ε. For vehicles, turbulent flows are characteristic and for this reason the k-ε model is applied [11]. The advantage of this method is that it can, near the walls to determine a very small Reynolds number and is more accurate than any other applied method, but the disadvantage is that the calculations are more robust.

2.1. k-epsilon Model

k is the turbulence kinetic energy and is defined as the variance of the fluctuations in velocity. ε is the turbulence eddy dissipation the rate at which the velocity fluctuations dissipate. The values of k and ε come directly from the differential transport equations for the turbulence kinetic energy and turbulence dissipation rate show in equations (1) and (2) [12].

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho U_j k\right)= \\ =\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+P_k+\rho\varepsilon+P_{kb} \tag{1}$$

$$\frac{\partial(\rho k)}{\partial t}+\frac{\partial}{\partial x_j}\left(\rho U_j k\right)=\frac{\partial}{\partial x_j}\left[\left(\mu+\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+ \\ +\frac{\varepsilon}{k}\left(C_{\varepsilon 1}P_k-C_{\varepsilon 2}\rho\varepsilon+C_{\varepsilon 1}P_{\varepsilon b}\right) \tag{2}$$

where $C_{\varepsilon 1}$, $C_{\varepsilon 2}$ and σ_ε are constants. P_{kb} and $P_{\varepsilon b}$,- represent the influence of the buoyancy forces, which are described below. P_k is the turbulence production due to viscous forces.

In addition to the independent variables, the density, ρ, the velocity vector, U_j, the molecular (dynamic) viscosity, μ and the turbulence viscosity μ_t.

The model constants are given:

- $C_{\varepsilon 1} = 1.44$
- $C_{\varepsilon 2} = 1.92$
- $\sigma_\varepsilon = 1.3$

2.2. k-omega Mode

One of the advantages of the $k-\omega$ formulation is the near wall treatment for low-Reynolds number computations. The model does not involve the complex nonlinear damping functions required for the $k-\omega$ model and is therefore more accurate and more robust. It solves two transport equations, one for the turbulent kinetic energy, k, and one for the turbulent frequency, ω, show in equations (3) and (4) [12].

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j k\right) = \\ = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + P_k + \beta' \rho k \omega + P_{kb} \tag{3}$$

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial}{\partial x_j}\left(\rho U_j k\right) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + \\ + \alpha \frac{\omega}{k} P_k - \beta \rho \omega^2 + P_{\omega b} \tag{4}$$

P_{kb} and $P_{\omega b}$ represent the influence of buoyancy forces. P_k is the turbulence production due to viscous forces. α, β, β', σ_k and σ_ω are constants.

The model constants are given:

- $\alpha = \frac{5}{9}$
- $\beta = 0.075$
- $\beta' = 0.09$
- $\sigma_k = 2$
- $\sigma_\omega = 2$

3. RESULTS AND DISCUSSION

Figure 3 presents a velocity distribution along the vehicle’s longitudinal plane of symmetry in the space. The pressure distribution that occurs during the vehicle's movement is shown in figure 4. The maximum pressure on the vehicle at a speed of 100 km/h is 574.5 Pa. Based on the numerical analysis, it can be concluded that the greatest pressures occur in the part of the air intake, at the transition between the hood and the windshield, as well as on the wheels. In the analysis, there are no air lines from the air intake that lead the air further, for example, to the engine or brakes, so the air is blocked here and for this reason, pressures occur. Of course, if there were such lines, such a vehicle would have a very good cooling of the mentioned components.

Fig.3. Velocity contour

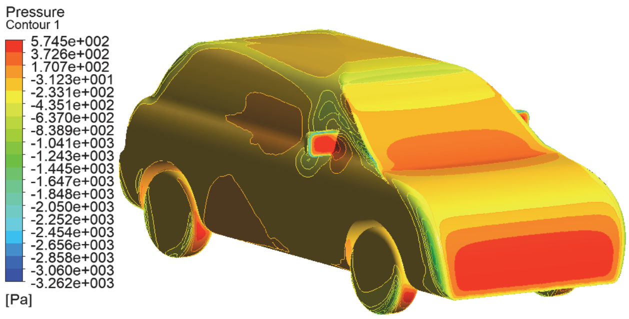

Fig.4. Pressure contour on the vehicle

Turbulent kinetic energy represents a measure of turbulent flow in the immediate vicinity of the vehicle. The consequence of a turbulent flow is the movement of the vehicle and the air. The maximum value of turbulent kinetic energy is 83.33 J/kg, figure 5.

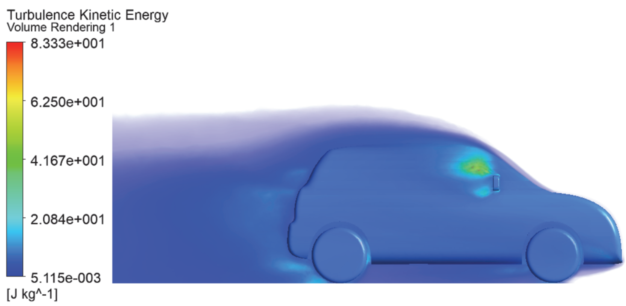

Fig.5. Turbulent kinetic energy

The drag coefficient is a very important indicator in terms of vehicle dynamics and fuel consumption. Drag coefficient of the analysed vehicle model is 0.370633 at vehicle speed of 100 km/h. In addition to the obtained drag coefficient, the analysis of what happens when the vehicle is moving at a lower or higher speed of 100 km/h is performed, figure 6. The vehicle speeds ranged from 50 to 150 km/h for the same border conditions.

By observing the diagram shown in figure 6, it can be noticed that the drag coefficient is growing up to 100 km/h, while with a further increase of speed, it decreases. This is almost the same case as with the bullet [12]. The

value of the drag coefficient, shown on the diagram, ranges from 0.362185 up to 0.370276. Furthermore, it can be concluded at which speeds the fuel consumption will be greater, which directly affects the amount of emissions of exhaust gases.

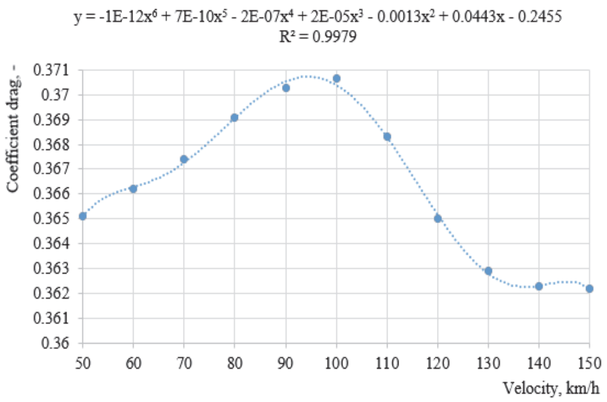

Fig.6. Drag coefficient versus velocity

4. CONCLUSION

Such an approach is significantly gaining in time, and this enables manufacturers to be competitive on the market. If the considered body shape does not provide the expected values of the drag coefficient, the model can be easily altered and simulated again without any major investment. When the numerical method obtains the lower value of the drag coefficient in comparison with competitor's vehicle, it can be accessed by the production of the same. Using the numerical analysis, the realistic values of the drag coefficient, for the considered body shape, are obtained in the paper. By this approach, the manufacturer reduces the costs that would arise when finishing operations would be done on the physical prototype of the vehicle's model, rather than in the program. Of course, after making the vehicle's body, it is necessary to confirm the results from the numeral analysis by experimental research in the wind tunnel. Furthermore, the paper presents a change in the drag coefficient with the vehicle's speed. As fuel consumption is in relation to the drag coefficient, it is possible to determine at what speed such a vehicle would be more economical.

ACKNOWLEDGMENTS

This paper was realized within the researching project "The research of vehicle safety as part of a cybernetic system: Driver-Vehicle-Environment" ref. no. 35041, funded by Ministry of Education, Science and Technological Development of the Republic of Serbia.

REFERENCES

[1] Janković, A. (2008). *Vehicle Dynamics,* Faculty of Mechanical Engineering University of Kragujevac, ISBN 978-86-86663-22-1, Kragujevac

[2] Sherman, D. (2014). Drag Queens Five Slippery Cars Enter a Wind Tunnel, *Available from:* https://www.tesla.com/sites/default/files/blog_attachments/the-slipperiest-car-on-the-road.pdf *Accessed:* 2018-02-10

[3] Grujić, I.; Stojanović, N.; Glišović, J.; Davinić, A. & Milojević, S. (2017). Modeling and aerodynamic simulation of the passenger vehicle, *Proceedings of International scientific conference on advances in mechanical engineering*, Oct. 12-14, Hungary, ISBN 978-963-473-304-1, Bodzás, B. & Mankovits, T., pp. 172-177, Department of Mechanical Engineering, Faculty of Engineering, University of Debrecen, Debrecen

[4] Damjanović, D.; Kozak, D.; Ivandić, Ž. & Baškarić, T. (2011). CFD analysis of concept car in order to improve aerodynamics. *Járműipari innováció,* Vol. 1/2, pp. 108-115, ISSN 1788-2699

[5] A Horiba Company MIRA (2014). Full Scale Aerodynamic Wind Tunnel *Available from:* https://www.horiba-mira.com/our-services/full-scale-wind-tunnel-(fswt) *Accessed:* 2018-02-10

[6] Prajwal, B.; Unune, D.; Aherwar, A. & Kishan, J. (2015). Modelling, simulation and validation of results with different car models using wind tunnel and Star-CCM+. *Journal of the Serbian Society for Computational Mechanics,* Vol. 9, No. 1, pp. 46-566, ISSN 1820-6530

[7] Angelo, E.; Angelo, G.; Di Giovanni Santos, P. H. & de Ansrade, D. A. (2015). Numerical Study of the Influence of Elements inside the Wheelhouse on the Passenger Vehicle Aerodynamic. *Open Journal of Fluid Dynamics,* Vol. 5, pp. 199-207, ISSN 2165-3860

[8] Amirnordin, S. H.; Wan Salim, W. S.; Ariffin, M. F. & Md Seri, S. (2010). Numerical Analysis on the Effects of Air Flow in the Wake of a Large Vehicle on Trailing a Passenger Car, *Proceedings of Malaysian Technical Universities Conference on Engineering and Technology*, June 28-29, Malaysia, pp. 11-19, Publisher, Melaka

[9] Adarsh, P.; Andrew, G. C.; Arun, K. S. & Kumar, D. R. (2014). Numerical investigation of drag on a trailing aerodynamic sedan vehicle. *International Journal of Mechanical and Production Engineering,* Vol. 2, No. 4, pp. 81-86, ISSN 2321-2071

[10] Zervas, E. (2011). Impact of altitude on the fuel consumption of a gasoline passenger car. *Fuel,* Vol. 90, No. 6, pp. 2340-2342, ISSN 0016-2361

[11] SHARCNet. The k-omega and SST Models, *Available from:* https://www.sharcnet.ca/Software/Ansys/16.2.3/en-us/help/cfx_mod/i1345900.html *Accessed:* 2018-02-12

[12] IANSYS. ANSYS CFX-Solver Theory Guide 2011, *Available from:* https://users.encs.concordia.ca/home/m/m_mamu/ANSYS%20CFX%20documentation/cfx_thry.pdf*:* 2018-02-12