IFERP connecting engineers... developing research

ISSN (Online) 2394-2320

International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)

Vol 4, Issue 4, April 2017

Research on Vehicle External Aerodynamics

[1]Ankur Sinha,

[1]Department of Mechanical Engineering, Galgotias University, Yamuna Expressway Greater Noida, Uttar Pradesh

[1]ankur.sinha@galgotiasuniversity.edu.in

Abstract: As the manufacturers of the motor vehicles generally use their own wind tunnels to test the real vehicle's performances, especially in the final stage of vehicle design, though the development of prototype starts from defining external aerodynamics of the vehicle model using experimental research and numerical simulation. Today, there are a number of different commercial software designed for the numerical simulation of the external aerodynamics, that show the different air flow images around and especially behind the vehicle depending on the different turbulence models. Therefore, the improvement of prototype implies the testing of the vehicle model in the wind tunnel in order to reach the optimal numerical model for the air flow simulation around the vehicle. This paper presents the results of the numerical simulations of the air flow around the vehicle model compared to the experimental results. The results obtained by numerical simulations and experimental research, enable an analysis appointing the existing problems and appropriate numerical methods considering improvement of the vehicle aerodynamics based on the development of the vehicle model.

Keywords: CFD, vehicle aerodynamics, wind tunnel.

INTRODUCTION

During the vehicle motion the road resistances can appear as follows: rolling resistance, aerodynamic strain, accelerating resistance, road grade and eventually trailer resistance. The consequence followed by increase of these resistances is the fuel consumption and the pollutants emission raise. As the accelerating resistance dominates in the overall road resistances during the urban driving due to the frequent changes of the vehicle speed meanwhile air strain dominates during the drive along the highways, where the vehicle speed achieves 80 km/h or more. Therefore, considering the motor vehicle development, a maximum attention is dedicated in the field of the air strain reducing. The parameters that directly affect to the air resistance are: A - frontal area of the vehicle, v – speed of the vehicle and C_x – aerodynamic strain coefficient[1].

Considering the wishes of the users of the transportation means related to achieve higher speed, options like reducing the frontal area of the vehicle and the strain coefficient cx to reduce aerodynamic strain must be achieved. As reducing the frontal area of the vehicle have an impact to the comfort reduction, the most influential parameter for the aerodynamic strain optimization becomes the aerodynamic strain coefficient C_x.

Determination of the strain coefficient requires very complex experiments in the wind tunnels. The experimental method requires existing of very expensive equipment, like the wind tunnel at the first place. Only very successful car manufacturers and some test centers are able to have expensive wind tunnels enabling the real motor vehicles research. External aerodynamics research also can be performed by the vehicle model, scaled in specified ratio as it is presented in. In this way, using the less complex laboratory equipment, the good results can be obtained which could be useful in the verification of the many numerical software simulation methods related to the air stream around the vehicle body[2]–[4].

In this paper, the possibilities of the external aerodynamics research around the vehicle body, experimental results gained and their comparison with the numerical methods are presented. Based on the experimental results and using the numerical methods, appropriate recommendations are given for the further research work.

EXPERIMENTAL RESULTS OF THE AERODYNAMICS OF THE VEHICLE MODEL

The first steps in the field of the vehicle model external aerodynamics research by these authors had started at Mechanical Engineering Faculty, University of Sarajevo in year 2006. In these days, the tests had been based on determination of the aerodynamic strain force when the balance (equality) between the weights of the known mass and the air strain force have been established. Continued research that followed during 2008 has been based on the implementation of the modern measurement methods where the use of the strain gauges could accurately determine the aerodynamic strain force and determine the aerodynamic strain coefficient as well. Obtained results of this experiment are presented.

However, basic drawback of the previously performed measurements laying on the fact that the vehicle model is 3–5 times of its length away from the outlet nozzle of the wind tunnel and didn't give the possibility to determine the pressure field around the model of the vehicle what is very important in order to define the external aerodynamics. Considering those drawbacks, the idea consists of forming the extension nozzle came up and so, in fact, placing the vehicle model in the nozzle superstructure, as shown in Fig. 1. Thus, the uniform velocity of the air flow in front of the vehicle model have been achieved, and in addition to that, the use of the nozzle superstructure made the installation of the devices measuring the pressure around the vehicle's model possible to perform[5].

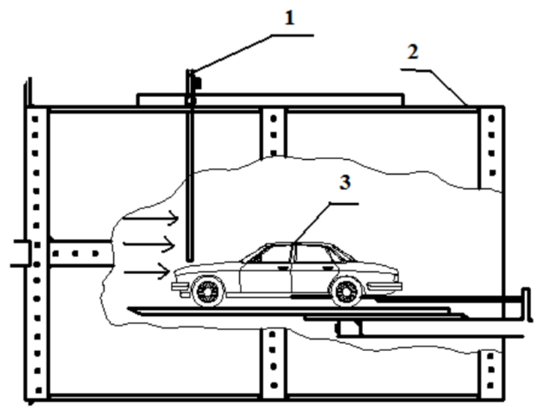


Fig 1: Superstructure of the nozzle extension of the wind tunnel

Description:

1 – Measuring probe carrier for determining the pressure around the vehicle model

2 – Nozzle superstructure (extension) of the wind tunnel

3 – Vehicle model

As the vehicle testing model of the external aerodynamics, the model of Peugeot 407 Coupe in the scale of 1:18 have been used. At that time, the values of the air velocity, pressure around the vehicle model and the air resistance were measured. For this purpose, the measuring equipment have been used was next: pressure gauge FLUKE FL 922 with the Pitot – Prandtl probe and with the characteristics as follows: ± 4000 Pa / 1 Pa / ± 1% + 1 Pa, and dynamometer LUTRON FG 5005 with the characteristics as follows: ± 5000 g / 1 g / ± 0, 4 %. Using the pressure gauge FLUKE FL 922 data containing the values of the air stream velocity in front and behind of the vehicle model have been obtained, as shown in Fig. 2, as well as the values of the static pressure, for a different flow velocities, around the vehicle model, as shown in Fig. 2[6].

IFERP
connecting engineers... developing research

ISSN (Online) 2394-2320

International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)

Vol 4, Issue 4, April 2017

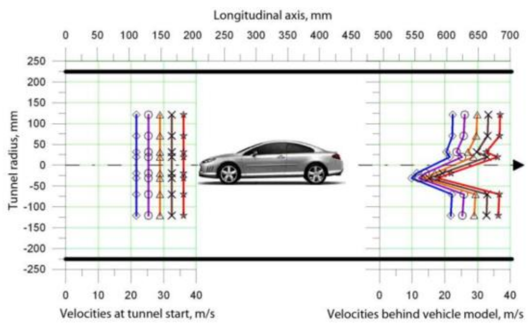


Fig 2: Velocity profile of the air stream in front and behind of the vehicle model

Based on the results shown in Fig. 2, it is obvious that the superstructure attached to the wind tunnel nozzle achieves that the air stream in front of the vehicle model be kept uniform what justifies the initial assumptions. However, the velocity of the air flow have been measured behind the vehicle model as well, where the disruption of the air flow and possible turbulence was noticed behind the vehicle model and the platform keeping the vehicle model in place in order to measurement makes possible. Since this paper only analyzes the flow around the vehicle model in terms of the speed and the pressure of the air stream, on this occasion, the value of the aerodynamic strain coefficient of the vehicle model is not going to be determined[7].

The values of the static pressure around the vehicle model, for the velocities of air flow given in Fig. 2, are given in Fig.3.

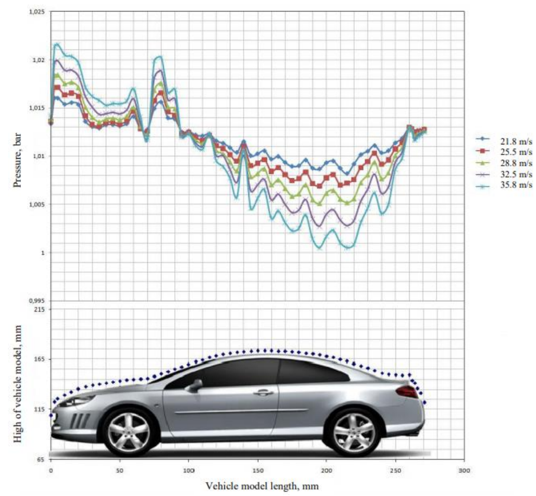


Fig. 3. Distribution of the static pressure values along the vehicle model for different velocities of the air stream

Considering the results shown in Fig. 3 it can be concluded that the increased air flow rate leads to the increased pressure in the front of the surface of the vehicle model below the hood and on the windshield as well. Following the shape of the roof towards the rear glass and the hatchback, the pressure decreases. Significant fluctuations in the pressure caused by the formation of the boundary layer and turbulence along the end side of the vehicle model can be seen as well. Especially this formation of the turbulence after the rear glass and hatchback of the vehicle model has a dominant influence in defining the turbulence, where many numerical models are trying to give the answer what is going to be presented in the next chapter[8].

NUMERICAL SIMULATIONS OF THE VEHICLE MODEL EXTERNAL AERODYNAMICS

The fact is in order to do the research of the external aerodynamics, the substantial funds for the experimental setup are necessary even for testing vehicle model analyzed in this example, but applying the numerical methods, the spending can be significantly reduced. Although, the use of the

IFERP
connecting engineers... developing research

ISSN (Online) 2394-2320

International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)

Vol 4, Issue 4, April 2017

numerical methods makes sense in order to obtain the first results, which are very helpful in the phase of the prototype development. Therefore, almost all manufacturers of motor vehicles practicing to do numerical simulation at first, then design a vehicle model, perform the experimental tests, complete the verification of the experimental results with the results of the numerical simulations and continue with further development in order to achieve the optimal solution. In this way the significant savings in time and money can be accomplished.

Today, various software implementing the numerical simulation in the field of fluid mechanics can be found on the market. The software chosen for the purposes of this work was Ansys Fluent, because it has the different methods of the turbulence calculation has been built[9].

In order to reduce the hardware resources, the longitudinal cross-section of the adopted vehicle model have been chosen. The numerical mesh is generated automatically with some additionally refined characteristic area. The problem of the external aerodynamics of the selected vehicle model has been analyzed as a quasisteady problem where k-ε model of the turbulence calculation was used. It is important to note that in the case studies of the external aerodynamics as an unsteady problem i.e. transient problem, the use of DES and LES model of the turbulence calculation in lieu of k-ε is recommended. The same parameters of the air flow around the vehicle model obtained by the experiment, have been used in the numerical simulation as well, thus the initial conditions applied for this case are the next:

Air flow velocities: v0 = 21.8; 25.5; 28.8 and 32.5 m/s,

Static pressure: p0 = 101325 Pa,

Density of air: ρ = 1.21 kg/m3.

Due to the limited space provided for the text of this paper, the comment and the analyze of the results given below refer only to the calculations of the pressure and the velocity of the air stream flowing along at the speed of 32.5 m/s (Fig. 4 and Fig. 5). The results of the numerical simulations at the other velocities above (21.8; 25.5 and 28.8) will be shown in order only to give a picture of that how the air streamlines around the vehicle model are changing by changing the initial speed, as it is shown in Fig. 6.

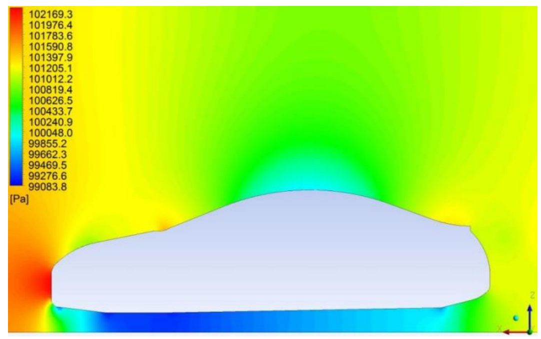


Fig. 4. Air pressure around the vehicle model

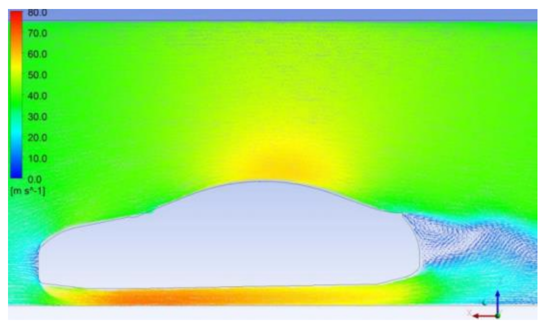
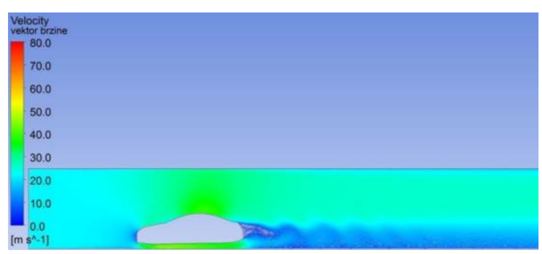


Fig. 5. Velocity of the air stream around the vehicle model

In Fig. 6, the results of the air velocity for a given speeds of the vehicle (the air stream velocities) of 21, 8; 25, 5 and 28, 8 m/s are presented.



IFERP
connecting engineers... developing research

ISSN (Online) 2394-2320

International Journal of Engineering Research in Computer Science and Engineering (IJERCSE)

Vol 4, Issue 4, April 2017

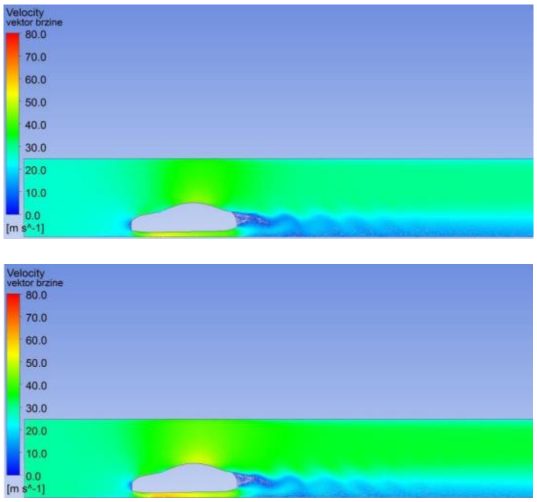


Fig. 6. Representation of the streamlines around the vehicle model for different velocities (21.8; 25.5 and 28.8 m/s) of the air

Regarding the results of the numerical calculation, it might be concluded that its similarity with the experimental results is more than satisfactorily considering the values of the static pressure (Fig. 4). This is obvious in terms of the static pressure values profile in the frontal area of the vehicle below the hood and on the windshield. Emergence of the turbulence behind the hatchback for different cases of the initial velocities of the air stream, presented in Fig. 5 and Fig. 6, is corresponding with the experimental results shown in Fig. 2. Also, it is obvious that increased initial velocity leads to an increase of the turbulence behind the vehicle model, what is clearly visible in Fig. 6[10].

CONCLUSION

This paper shows that the numerical simulation of the external aerodynamics of the vehicle model as well as of the real vehicles now may be implemented by the average hardware resources. The development and improvement in the external aerodynamics of the vehicles using a computer with a various CFD software installed, not only makes possible the solving of the engineering problems quick and efficient but it has become the standard in the companies engaged in the production of the vehicles, planes and vessels. These are all reasons why CFD is becoming the standard for any development of the vehicles aerodynamics in the future especially considering the time and costs savings avoiding the modeling of the external aerodynamics of the vehicles by the experiment. Although, in order to make a serious approach to the above problems, in the case when precise results are required, the experiment, that most often consists of the wind tunnels tests must be performed. However, there are some aspects of the calculation of the specific components of the air stream that still require the professional skills and experience in the research work owing to the complex analysis in the form of 3D simulations for what the high-capacity computer systems are required.

REFERENCES

[1] S. Hetawal, M. Gophane, B. K. Ajay, and Y. Mukkamala, “Aerodynamic study of formula SAE car,” in *Procedia Engineering*, 2014, doi: 10.1016/j.proeng.2014.12.398.

[2] T. Fujimoto and T. Suzuki, “Aerodynamic design for SR11 (formula SAE racing car),” in *SAE Technical Papers*, 2013, vol. 2013, doi: 10.4271/2013-32-9100.

[3] D. Kelly *et al.*, “Computational-Based Aerodynamic Design for a Formula SAE Vehicle,” *SAE Int. J. Passeng. Cars - Mech. Syst.*, vol. 11, no. 1, 2018, doi: 10.4271/06-11-01-0003.

[4] S. Chepkasov, G. Markin, and A. Akulova, “Suspension kinematics study of the ‘Formula SAE’ sports car,” in *Procedia Engineering*, 2016, vol. 150, pp. 1280–1286, doi: 10.1016/j.proeng.2016.07.288.

[5] T. Nakata, H. Liu, Y. Tanaka, N. Nishihashi, X. Wang, and A. Sato, “Aerodynamics of a bio-inspired flexible flapping-wing micro air vehicle,” *Bioinspiration and Biomimetics*, 2011, doi: 10.1088/1748-3182/6/4/045002.

[6] P. Leśniewicz, M. Kulak, and M. Karczewski, “Aerodynamic analysis of an isolated vehicle wheel,” in *Journal of Physics: Conference Series*, 2014, doi: 10.1088/1742-

6596/530/1/012064.

[7] F. Mariani, C. Poggiani, F. Risi, and L. Scappaticci, "Formula-SAE racing car: Experimental and numerical analysis of the external aerodynamics.," in *Energy Procedia*, 2015, doi: 10.1016/j.egypro.2015.12.111.

[8] A. Kotikalpudi, H. Pfifer, and G. J. Balas, "Unsteady Aerodynamics Modeling for a Flexible Unmanned Air Vehicle," 2015, doi: 10.2514/6.2015-2854.

[9] N. Winkler, L. Drugge, A. S. Trigell, and G. Efraimsson, "Coupling aerodynamics to vehicle dynamics in transient crosswinds including a driver model," *Comput. Fluids*, 2016, doi: 10.1016/j.compfluid.2016.08.006.

[10] T. Favre, *Aerodynamics simulations of ground vehicles in unsteady crosswind*. 2011.

[11] S. Balamurugan, K. Amarnath, J.Saravanan and S. Sangeeth Kumar, "Scheduling IoT on to the Cloud : A New Algorithm", European Journal of Applied Sciences 9 (5): 249-257, 2017.

[12] S.Balamurugan et.al., "Smart Healthcare: A New Paradigm", European Journal of Applied Sciences 9 (4), 212-218, 2017

[13] Vishal Jain and Dr. S. V. A. V. Prasad, "Analysis of RDBMS and Semantic Web Search in University System", International Journal of Engineering Sciences & Emerging Technologies (IJESET), Volume 7, Issue 2, October 2014, page no. 604-621 having ISSN No. 2231-6604.

[14] Vishal Jain and Dr. S. V. A. V. Prasad, "Evaluation and Validation of Ontology Using Protégé Tool", International Journal of Research in Engineering & Technology, Vol. 4, No. 5, May, 2016, page no. 1-12 having ISSN No. 2321-8843.

