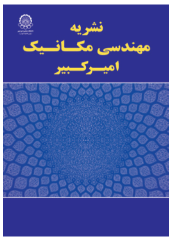

Amirkabir Journal of Mechanical Engineering

Amirkabir Journal of Mechanical Engineering, 49(3) (2017) 157-160
DOI: 10.22060/mej.2016.749

Experimental Study of Drag Reduction on a Model Car Using an Active Flow Control

A. Bak Khoshnevis*, V. Barzanooni

Mechanical Engineering Department, Hakim Sabzevari University, Sabzevar, Iran

ABSTRACT: Aerodynamic drag is one of the most important determinants of vehicle's fuel consumption. Pressure drag which is the main component of total drag is a result of boundary layer separation from vehicle surface. In this paper, we investigate experimentally, the possibility of separation from the upper surface of model and the effect of base bleeding as an active flow control method on aerodynamic drag reduction of Ahmed body with 35 degree rear slant angle. Then optimum position of blowing on the slope and vertical surfaces at the end of model was found. Blowing increased static pressure in the wake and reduces the drag. In addition, pressure profile across the surface model, velocity and turbulence intensity in this paper is studied. The results showed that the jet flow in the best position causes 21 % reduction in the drag coefficient. If the jet flow is applied on the upper position of inclined surface, it increases the drag coefficient. All tests are done in an open circuit blowing wind tunnel with maximum turbulent intensity of 1 % and maximum speed of 30 m/s. Velocity measuring components are carried out by hot-wire flow meters.

Review History:

Received: 6 July 2015
Revised: 23 December 2015
Accepted: 24 January 2016
Available Online: 8 November 2016

Keywords:

Aerodynamic drag reduction
Active flow control
Ahmed model
Flow separation
Wind tunnel

1- Introduction

Heavy vehicles' average covered distance, annually is between 130000 and 160000 km; and 52 % of fuel consumption is consumed on overcoming aerodynamic forces, therefore, even slight adjustments leading to increase in aerodynamic efficiency in these type of vehicles, will reduce fuel consumption amount greatly and helps to avoid air pollution [1].

The model used in this study is the model suggested by Ahmad et al. [2] which is a simplified model of ground motor vehicles. Ahmad examined a simplified model of vehicles with rear window in different angles (0 to 45) and found out in critical angle of 30, there is a plunge in the drag. Following him, many researchers assessed the model and carried out research on it. Watkins and Vino [3] carried out studies on changes in the drag and lift coefficients. Matthew and Patrick [4] analyzed the flow in a square-back geometry model and effects of formed vortexes in the rear of the model. Ghaligne and Thomas [5] investigated the active controlled wake's flow in Ahmed's two-dimensional model .Spohn and Gillieron [6] revealed flow and assessed separation's place. Khaleghi et al. [7] investigated mechanism of drag reduction for a square-back vehicle's modified model due to the changes in final geometry of model and changes in the wake, a reduction of 10 % in turbulence density and 20 % in drag's coefficient was reported. Krajnovic and Davidson [8] Studied two-dimensional geometric of vortexes formed in the end of Ahmad model, in average Reynolds number of 3000, and showed in lower Reynolds, the output flow will be independent of Reynolds Number. However, this number was smaller than the actual Reynolds, but had a good conformity with the results of work done by Rouméas et al. [9] at Reynolds 280000. Krajnovic and Davidson [10] examined vortex's flow in low Reynolds numbers confirmed that the output flow from vehicles behind for high Reynolds's number will be independent of this parameter. Charles and Bruneau [11] investigated active controlled validity and fluctuation in the wake of a square-back model in low Reynolds numbers, and eventually reached a reduction of 30 percent in drags coefficient. In the present study, the probability of separation in the upper surface of the model and the effects of an active control approach of wakes flow on reduction of aerodynamic drag in Ahmad's model with the final angle of 35 degrees as a method of controlling the flow of drag reducing experimentally were assessed. Also, effects of position of application of the flow on drag's coefficient and optimal position of flowing on a slope and final vertical surface were defined.

2- Experimental Setup and Measurement Technique

All the studies were carried out utilizing open circuit blowing wind tunnel at University of Hakim Sabzevari. This tunnel is in 40 cm by 40 cm dimensions and length of 180 cm. The maximum turbulence intensity in the test case is 0.1 % and the maximum speed is 30 m/s. Airflow speed control is by controlling fan's revolution in the range of (0 to 30 m/s). Flow meter of hot wire's probe is from one-dimensional kind and its sensors are from Tungsten with five microns, and measured data by hot wire's flow meter were conveyed through A/D card to a computer and also analyzed by software.

In order to measure pressure, 30 channel electronic manometers were used and the output was transferred to a computer and with software, it was analyzed. The utilized monometer was a differential manometer and its efficiency

Corresponding author, E-mail: khosh1966@yahoo.com

range is around 1250 Pa and its accuracy is around 0.25 %

For moving the probe, in order to measure the airflow profile, transmission mechanism was used, and its accuracy is 0.1 mm in coordinate axis direction. The studied model in this research is suggested model by Ahmad et al. [2] with final angle of 35 degrees. Considering blockage coefficient is the first determinant in making a model. Suggested volume for laboratorial models is less than 0.1, so that, the effect of fluid flow on the side walls of the object can be overlooked. In this study, the chosen coefficient considering laboratorial conditions and wind tunnel is 0.09 and scale factor is 0.14. In order to measure the pressure, on the surface, some holes were made into model's surface, and pressure sensors are embedded in the holes. In order to provide air for jet flow wake actions, a 2.5 hp compressor with a 25 liter tank and the maximum pressure of 12 bars is used. The produced speed of jet flow in the model's wake is 15 meters per second.

3- Governing Equations

Equations that are used to measure the force of drag can be easily extracted by applying the laws of conservation of mass, momentum on a control volume. Van Dam [12] found an equation to calculate drag coefficient in which Reynolds tensions and turbulence intensity of flow existed. However, fluid intensity changes and sliminess terms were over looked. Lu and Bragg [13] had numerous investigations with regard to the effects of distribution in calculating drag coefficient. According van dam's equations:

$$c_d = 2\int\left(\frac{\bar{u}}{U_\infty}\right)\left(1-\frac{\bar{u}}{u_\infty}\right)d\left(\frac{y}{l}\right)+\frac{1}{3}\int\frac{\overline{u'^2}}{U_\infty^2}d\left(\frac{y}{l}\right) \quad (1)$$

4- Conclusions

In this study, the effect of blowing as an active flow controlling method, reduces the aerodynamic drag in Ahmad's model with 35 degrees of final slope angle was examined.

Results show feasibility of studying at low Reynolds number in model's wake flow (Reynolds number chosen in this study) showed best applying position of jet flow for active control of flow in the wake is on a vertical surface and near the slope and in this position, applying jet flow causes the reduction of drag coefficient approximately by 21 %. Around 3 percent of calculated drags coefficient is attributed to wake's turbulence and the other 97 % is owing to reduction of momentum pressure loss in the wake. Applying jet flow in positions p1 and p2 (Figs. 1 and 2) causes increase in fluctuant term than other positions of jet flow application. Applying jet flow on upper surfaces of the slope is not justified and leads to increase in drag coefficient.

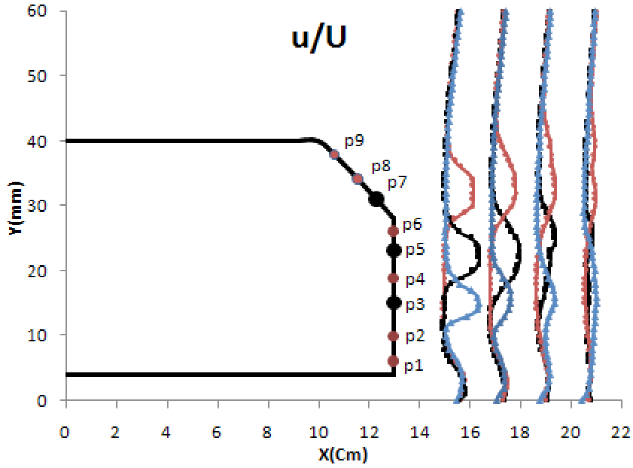

Figure 1. Average velocity of the jet flow in the wake of applied positions

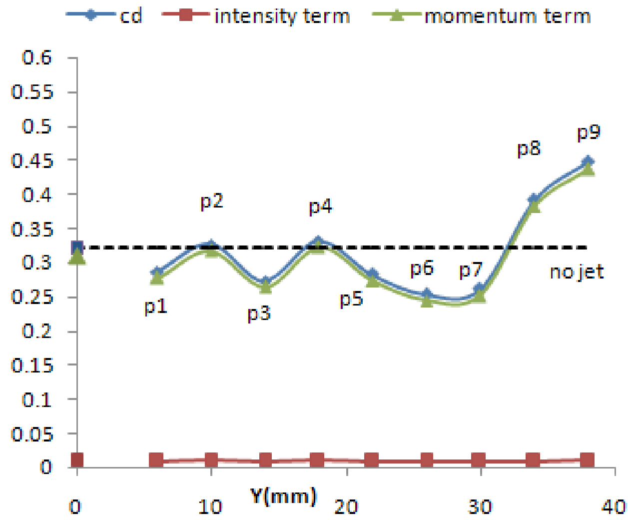

Figure 2. Drag coefficient in different applied positions of the jet

References

[1] M. Hazim, A. Abdulkadir, K. Iftekhar, A. Firoz, W. Simon, A study on aerodynamic drag of a semi-trailer truck Harun Chowdhury, *Procedia Engineering* 56 (2013) 201– 205.

[2] S.R. Ahmed, R. Ramm, G. Faltin, Some Salient Features of the Time Averaged Ground Vehicle Wake, *SAE Technical Paper Series* 840300 3(20) (1998) 104-115.

[3] S. Watkins, G. Vino, The Effect of Vehicle Spacing on the Aerodynamics of Representative Car Shape, *J. of Wind Engineering and Industrial Aerodynamics* 96(3) (2011) 1232-1239.

[4] R. Mathieu, G. Patrick, Analysis and control of the neare-wake flow over a square-back geometry, *computr& fluids* 21(38) (2009) 60-70.

[5] S. Ghaligne, C. Thomas, Active control of the flow behind a two-dimentional bluff body in ground proximity, *comptes rendus mecanique* 2(341) (2013) 289-297.

[6] A. Spohn, P. Gillieron, Flow separeated generated by a simplified geometry of an automotive vehicle, *Society of Automotive Engineers Technical Paper* 23(41) (2011) 89-97.

[7] B. Khalighi, S. Zang, C. Koromilas, S. Balkanyi, L.P. Bernal, G. laccarino, P. Moin, Experimental and Computational Study of Unsteady Wake Flow Behind a Body with a Drag Reduction Device, *SAE Technical Paper* 14(17) (2006) 12-21.

[8] S. Krajnovic, L. Davidson, Flow around a simplified car, part 2, *Journal of Fluids Engineering* 12(7) (2008) 919-928.

[9] P. Roms, A. Spohn, Flow Separations Generated by a

Simplified Geometry of an Automotive Vehicle, *SAE Technical Paper* 21(12) (2007) 31-44.

[10] S. Krajnović, L. Davidson, Flow Around a Simplified Car, Part 1: Large Eddy Simulation, *ASME J. Fluids Eng 127* (2005) 907-918.

[11] H. Charles, A. Bruneau, B. Emmanuel, A. Delphine, c. Gilliéron, Coupling active and passive techniques to control the flow past the square back Ahmed body, *Computers & Fluids* 39 (2010) 1875-1892.

[12] C.P. Van Dam, Recent Experience with Different Methods of Drag Prediction, *Progress in aerospace* 35(8) (1999) 751.

[13] LU, B. and Bragg, M. B., 2002. “Experimental investigation of the wake-survey method for a bluff body with highly turbulent wake”. *AIAA*, 23(11), Dec, pp. 130-1430.

Please cite this article using:

A. Bak Khoshnevis and V. Barzanooni, Experimental Study of Drag Reduction on a Model Car Using an Active Flow Control, *Amirkabir J. Mech. Eng.*, 49(3) (2017) 435-444.

DOI: 10.22060/mej.2016.749