Vandana Publications
IJRASB

Volume-5, Issue-1, January 2018
International Journal for Research in Applied Sciences and Biotechnology
Page Number: 1-5

Assessment of Flow Control using Passive Devices around Bluff Bodies

Ankush Raina[1*], Akhil Khajuria[2]
[1]Assistant Professor, Department of Mechanical Engineering, SMVD University, Katra, 182320, J&K, INDIA
[2]Professor, Department of Mechanical Engineering, NIT Jalandhar, Punjab, INDIA

[1]**Corresponding Author**: ankush.smvd@gmail.com

ABSTRACT

This article presents a review study on the experimental investigation of aerodynamic force on a car like bluff bodies along with the simulations using different turbulence models used in CFD (Computational Fluid Dynamics). The aim of the study is to find a useful method for the better design of a car body. It was observed that combination of wind tunnel experiments and CFD computation can lead to better aerodynamic design. Significant reduction in coefficient of the lift and drag for a car model were found when a more streamlined body design was adopted. Appropriate change in the slant angle for the car body can significantly reduce the fuel consumption. Also, the use of simulations in combination with the experimental observations helps in predicting the flow behavior more accurately.

Keywords— Bluff Body, Passive devices, Drag coefficient.

I. INTRODUCTION

The issues pertaining to the reduction of vehicle power consumption presented a challenge to the researchers, since the inception of invention of vehicle itself. The power consumption is related to speed of vehicle, drag presented by the air, road conditions etc. The design of vehicles is dictated by the host of factors which includes drag force even under smooth road conditions. The combined effect of the rolling resistance of wheels and aerodynamic drag is responsible for opposing the forward motion of the vehicle plying on a road. Aerodynamic drag is a force produced as a result of the distribution of pressure around the vehicle. The major part of it comes from the pressure difference between the rearward facing parts and the forward facing parts. The pressure distribution around a vehicle is produced by a number of interacting influences, one of the most important being the boundary layer and the drag produced by this effect depends largely on where flow separation occurs, e.g., a circular plate held normal to the flow will produce separation around the periphery of the leading face resulting in a wide wake with high drag coefficient, while a teardrop shape with a long tail can retain attached flow right to the end, with consequently low coefficient of drag. So by delaying the flow separation a considerable amount of reduction in drag coefficient can be obtained [1].

II. TYPES OF FLOW CONTROL DEVICES

Vehicular geometry presents difficulties regarding the drag reduction particularly with high speeds and the minimization of drag is consequently an iterative process in design due to host of varying conditions under which a vehicle has to operate. One of the ways to reduce the drag is the use of passive devices which have the capability to reduce the drag and lift forces. Various approaches as regards passive devices to reduce drag are shown in Figure 1, and can be put in following categories.

1. Vortex Air Flow Generators (VGs).
2. Splitter Plates.
3. Deflectors.
4. Flaps.
5. Rear Fairings
6. Front Panels

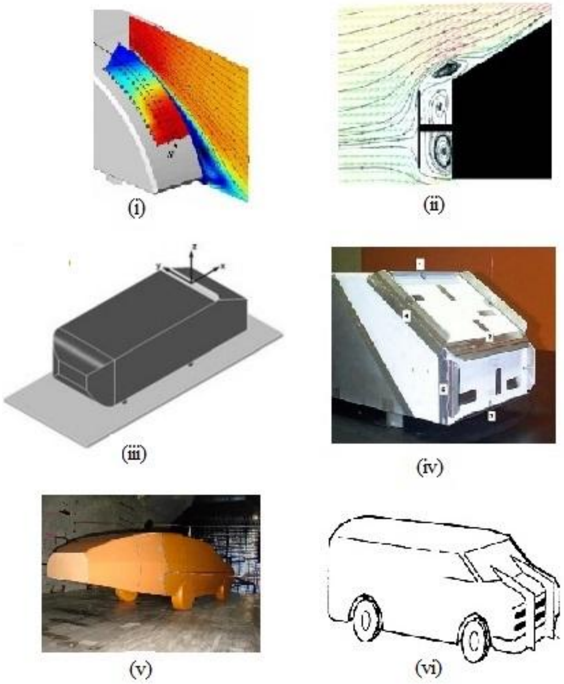

Figure 1 Various passive devices on the vehicles (i) Vortex generators (ii) Splitter plates (iii) Deflectors (iv) Flaps (v) Rear fairings (vi) Front panels [2,3,7,8,9,16].

III. VORTEX AIR FLOW GENERATORS

An investigation on a modified Ahmed body with a curved rear part, using VGs (Vortex Generators) was performed by Aider et al. [2]. An extensive parametric study of the influence of the vortex generators on the aerodynamic forces was performed by considering the four cases. In the first case the influence of the longitudinal position (d) as shown in Figure 2 and angle of the vortex generators (α) with respect to the rear edge of the model were considered (60^o and 120^o with the vehicle top surface). The first observation has strong influence on the drag coefficient: for both angles, the drag reduction of about 12% for d=0.2 m. Similar observation as regarding the lift coefficient for α = 60^o has been reported with a reduction of 54% w.r.t the case without VGs on Ahmed body. The optimal position of the VG line is also different in both the cases: the maximum lift reduction for α =120^o is observed at d = 0.2 m which is 0.02m downstream of α = 60^o configuration.

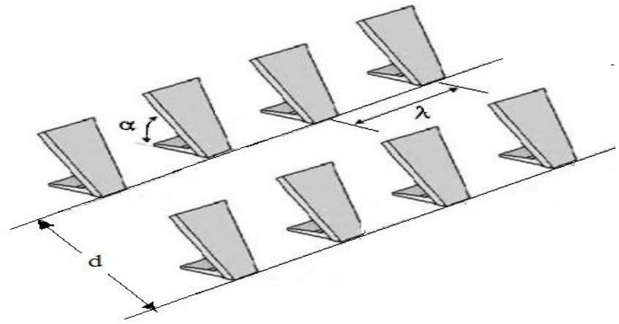

Figure 2 Arrangement of vortex generators on the rear face of the Ahmed Body in the longitudinal direction.

In the second case, influence of the spacing (λ) along the transverse direction) between the VGs was considered in [2]. The spacing between the VGs is varied from λ= 0.02 m to λ = 0.015 m. As the VGs are distributed along the all width of the rear slant in the transverse direction, the number of VGs is also changed from 17 to 22. It was observed that the maximum drag or lift reductions of 18% was obtained with λ = 0.015 m spacing for nearly all the positions and for the two different angle represented in [2].

In the third case, the influence of velocity was introduced to check the efficiency of VGs for higher Reynolds number. A series of velocities of 20 m/s, 30m/s and 40 m/s were used to obtain the effect of Reynolds number on the drag and lift coefficient, and corresponding reduction of 12.2%, 7.1 % and 3.7 % was observed for respective cases.

In the fourth case the influence of the distribution of the vortex generators along the width of the model for a row of VGs at S = 0.20 m was considered where drag is minimum w.r.t the other reported cases in [2].Some of the VGs homogeneously distributed along the width were removed and two new configurations were then studied with same spacing of λ = 0.015 m. In the first case 4 VGs located on each side edge of the rear part of model were removed and in the second case 14 VGs located in the center of the line of VGs were removed. It was observed that maximum drag reduction of 3 % with respect to the complete line of VGs and 13.6% with respect to the reference flow (without vortex generators) was obtained for the first case reported above, whereas the second case led to the increase in drag coefficient. Thus, a better understanding of the interactions between the VGs and the overall flow structure, the velocity field in the near-wake of the bluff-body for three different configurations of VGs was reported in [2].

A series of vortex generators placed on the back edge of the roof was investigated by Rohatgi [3] as shown in Figure 2.3. A decrease in the drag coefficient of 1.24% was observed w.r.t the case without vortex generators.

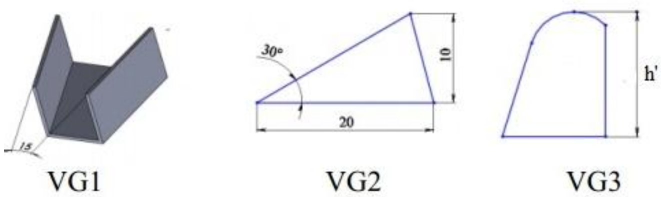

Figure 3 The types of vortex generators [3].

Koike et al [4] investigated the flow using VGs around a sedan car at a flow velocity of 50 m/s. Particle Image Velocimetery (PIV) method was used to measure the velocity distribution. It was observed that the optimum height of the VG is almost equivalent to the thickness of the boundary layer i.e. 15 to 25 mm. Optimum method of placement was found in the form of arrangement in a row in the lateral direction 100 mm upstream of the roof end at intervals of 100 mm [5]. The delta-wing shaped VG as shown in Figure 3 demonstrated high effectiveness with a drag reduction of about 0.006 w.r.t the case without vortex generators. The results were also validated by analyzing the flow field using computational fluid dynamics.

Kumar et al [6] studied the drag force calculations over the Audi R8 car model. The major criteria for the drag study were the position of VGs on the test model. Measurement of the static pressure distribution was performed by drilling the needles of 0.6 mm diameter on the hood and on the shield (topmost portion) of the vehicle. Drag force and static pressure were measured by using a digital system as an output of the wind tunnel model. Due to the delay in the flow at the increased flow velocity at the trailing edge, the results obtained showed a drag reduction of 0.2 when the VGs were placed at 45 degrees w.r.t rear edge of the model for a flow velocity of 10 m/s.

IV. PLATES AND DEFLECTORS

Gillieron and Kourta [7] studied the reduction of drag using splitter plates placed at the front or the rear of Ahmed Body as shown in Figure 1(ii). The height and width of splitter plates were 0.6, 0.7, 0.8 and 0.9 times the height and width of the Ahmed body. The experiments were performed with splitter plates positioned downstream and upstream of the Ahmed body as used in investigation [7] by varying the distance between the base of the Ahmed body and the splitter plate. Results for three types of downstream vertical splitter plates were obtained for an upstream velocity of 30 m/s. It was observed that the maximum drag reduction of 12% is obtained with a splitter plate measuring 0.9 H_A * 0.9 w_A, placed at $x/H_A = 0.5$, where x is the distance between the splitter plate and the rear face of the Ahmed body.

Effects of splitter plates on a small scale model of General Motor SUV (Sports Utility Vehicle) having length of 1710 mm were studied by Rohatgi [3]. The model was tested in wind tunnel for expected wind conditions and road clearance. Also the splitter plates were installed at various distances from the model's rear surface and the corresponding values of drag coefficient were obtained. It is observed that at splitter plate at a distance of 220 mm has the minimum drag coefficient.

Fourrie et al [8] experimentally studied the passive flow control on Ahmed body by placing the deflector on the upper edge of the Ahmed model (as shown in Figure 1 (iii)) at a flow velocity of 16 m/s and 40m/s corresponding Reynolds numbers of 3.1 * $10^5$and 7.7 * 10^5.respectively.It was observed that between $\theta = -25^o$ (w.r.t top surface of Ahmed body) and $\theta = 0^o$, the drag coefficient increases with the deflector angle and for a 0^o deflection angle there is a sudden decrease in drag and lift coefficients. This is due to the fact that deflector increases the separated region over the model rear window. This phenomenon leads to a drag reduction of up to 9% w.r.t. to the case without deflector.

V. FLAPS

In another type of study on passive flow control devices using Ahmed body by Beaudoin and Aider [9],the influence of different flaps (as shown in Figure 1(iv)) located on every edge of the two rear panels of a classic body was investigated for three free stream velocities (V=20, 30 and 40 m/s). The two most efficient configurations of [9] were the two flaps on the side edges of the rear slant (17.6% of drag reduction) and the flap on the top of therear slant (15% of drag reduction). In view of the above results it is observed that the flaps around thebase of the Ahmed body were less efficient as compared to the flaps on the side edges of the rear portion andthat allowed a reasonable drag and lift reduction of 7% and 14% respectively.

Wahba et al. [10] investigated the use of flaps for reduction of drag using computational fluid dynamics. Two types of ground vehicles were considered, a simplified bus model and a simplified sport utility vehicle (SUV) model. The flaps were used to direct air into the low-pressure wake region in order to enhance pressure recovery, which in turn reduced the drag. Flap cross-section, chord length and angle of attack were varied in order to obtain the optimal configuration for improved aerodynamic performance. An overall reduction in the drag coefficient of up to 18% for the bus and SUV models with the use of flaps was obtained. Grid-independence tests and comparison with available data in the literature was carried out to validate the numerical procedure, as laid down in [10].

VI. OTHER DEVICES

The effect of rear fairings (as shown in Figure 1(v)) was studied in [3]. Rear fairings imply a structure in vehicles back part, included in the separation area in order to provide no separation to the flow. It was observed that installation of full size fairing led to the reduction of drag factor by 26 %.

Half-length truncated rear fairing also has high aerodynamic efficiency and led to the reduction in the drag factor by 22.6%.

Pamadi, Taylor and Leary [5] proposed another method of drag reduction of the bluff bodies by installing the panels on the forward surface of the vehicle facing the air stream (as shown in Figure 1(vi)). The tests were conducted on the medium capacity van and a medium size passenger car for different types of panel configurations. Based on the road tests and flow visualizations one specific configuration was optimized and reduction of 27% in aerodynamic drag and fuel reduction of 18% was obtained at flow velocity of 17.77 m/s for a medium capacity van. For a medium size passenger car at speeds below 13.33 m/s there is substantial reduction in drag to the extent of 60 % but falls to 6.5 % at 24.44 m/s.

Raju et al [11] studied the effect of collapsible wind friction reduction attachment at the rear portion of the car. Depending up on the requirement the attachment can be opened and closed. The results were reported for power consumption and millage of the car with and without the attachment at different flow velocities and it was observed that on the installation of the attachment, drag coefficient was reduced by a factor of 0.2. This led to the increase in the efficiency of the model especially at higher velocities.

VII. SIMULATIONS

The use of simulations also gained much importance owing to their computational capacity in the recent past. Khalighi Zhang and Koromilas [12] conducted an experiment and computational investigation of a drag reduction device for bluff bodies. The main goal of the research was to gain the better understanding of the drag reduction mechanism in the bluff body square back geometries. The models were referred to as square back (SB1) for baseline configuration and modified square back (MSB) for the modified configuration with extension plates called flaps. Two types of CFD simulation approaches were used. First, steady state RANS (Reynolds Average Navier Stokes) equation was solved to investigate the general features of the wake flow fields for both models. Second, unsteady RANS was used to the problem and flow unsteadiness was obtained. The mean pressure results show a significant increase in the base pressure when the drag reduction device is in place. Also there is a reduction of turbulence intensity as well as rapid upward deflection of the underbody flow due to the add-on device (flaps on MSB) as compared to the baseline configuration.

Serre et al. [13] presented a comparative analysis of simulations, conducted in the framework of a French German collaboration of Complex flows, for the Ahmed body at Reynolds number 768000 and slant angle 25^o as shown in Figure 2.13. The approaches used include two Large Eddy Simulations (LES), a stabilized spectral method and a Detached Eddy Simulation (DES) using unstructured grid. It was observed that DES model showed a variation of 15% in drag coefficient when compared to the experimental values obtained through literature; whereas LES models showed a variation of 40% when same type of comparison was made.

Liu and Moser [14] investigated the air flow over Ahmed body by means of transient RANS turbulence model. They evaluated the performance of several RANS turbulence model and compared it by using the two differencing schemes, second order differencing and upwind scheme. Turbulence models used were Durbin's K-ε-V^2, K-ε, SST and RSM. Experiments in wind tunnel with flow velocity of 40 m/s and the comparison for mean velocity profile were made with the numerical results at the separation zone were reported. It was observed that K-ε-V^2 model gave better results as compared to the standard K-ε model

Barbut and Negrus [15] performed a case study on the influence of the lower part of road vehicles on the drag characteristics at inlet flow velocity of 25 m/s. The CFD code used was DxUNSp, based on unstructured domains with additional option of grid refinement close to the solid surfaces. The objective of that analysis was to demonstrate the importance of CFD analysis in car optimization taking in to account the flow under the car. The drag coefficient for sedan car was reduced from reference value of 0.2411 to 0.2105 (12.5%) by modifying its lower part.

VIII. CONCLUSION

The aim of the study was to review the performance of different passive flow control devices around the bluff bodies. Different flow control devices were discussed and their effect on the flow behavior was analyzed. It was observed that the use of these flow modification devices leads to the overall improvement the drag and lift forces. Also, the effect of different turbulence models on the drag and lift forces was evaluated. It was observed that the use of the appropriate turbulence model in conjunction with the experimental observations helps in predicting the flow behavior more accurately.

REFERENCES

[1] Barnad R.H., "Road Vehicle Aerodynamic Design", MechAero Publishing, ISBN 978-0-9540734-0-4, 2001.

[2] Aider J. L., Beaudoin J. F. and Wesfreid J.E, "Drag And Lift Reduction Of A 3D Bluff-Body Using Active Vortex Generators", Exp Fluids 48:771–789, DOI 10.1007/s00348-009-0770-y,Springer-Verlag, 2009.

[3] Rohatgi U.S, " Methods Of Reducing Vehicle Aerodynamic Drag", Brookhaven National Laboratory, U. S. Department of Energy, BNL-98039-2012-CP, 2012.

[4] Koike M, Nagayoshi T and Hamamoto N " Research On Aerodynamic Drag Reduction By Vortex Generators" Mitsubishi Motors A Technical Review no. 16,2004

[5] Pamadi B.N., Taylor L.W. and Leary T.O.,"A Method For The Reduction Of Aerodynamic Drag Of Road Vehicles", NASA Technical Memorandum 102589, 1990.
[6] Kumar C. R, Chowdary J and Reddy A. K, "Study Of Aerodynamic Drag Reduction Using Vortex Generators" International Journal of Advanced Engineering Sciences and Technologies, ISSN: 2230 - 7818, Vol. No. 10, Issue No. 1, 181-183, 2011
[7] Gillieron P. and Kourta A, "Aerodynamic Drag Reduction By Vertical Splitter Plates", Exp Fluids 48:1–16, DOI 10.1007/s00348-009-0705-7, Springer-Verlag, 2009.
[8] Fourrie G., Keirbulck L., Larbi L. and Gillieron P,"Bluff-Body Drag Reduction Using A Deflector", Exp Fluids 50:385–395, DOI 10.1007/s00348-010-0937-6, Springer-Verlag, 2010.
[9] Beaudoin J. F. and Aider J. L, "Drag And Lift Reduction Of A BluffBody Using Flaps", Exp Fluids 44:491–501, DOI 10.1007/s00348-007-0392-1, Springer-Verlag, 2008.
[10] Wahba E. M, Marzooki H, Shaath H, Shahin H, and Dhmashawy T, "Aerodynamic Drag Reduction For Ground Vehicles Using Lateral Guide Vanes ", CFD Letters, Vol. 4(2), 2012.
[11] Mohan Raju D. K. and Reddy G. J," A Conceptual Design Of Wind Friction Reduction Attachments To The Rear Portion Of A Car For Better Fuel Economy At High Speeds", International Journal of Engineering Science and Technology, ISSN: 0975 - 5462, Vol. No. 4, pp-2366-2372, 2012.
[12] Khalighi B., Zhang S. and Koromilas C.,"Experimental And Computational Study Of Unsteady Wake Flow Behind A Bluff Body With A Drag Reduction Device", Society of Automotive Engineers,2001.
[13] Serre E, Minguez M,Pasquetti R,Guilmineau E, Deng G. B, Kornhaas M and SchäferM, "On Simulating The Turbulent Flow Around The Ahmed Body: A French-German Collaborative Value Of LES And DES", Computers and Fluids, 2011.
[14] Liu Y. and Moser A., "Numerical Modelling Of Airflow Over Ahmed Body", Air and Climate Group, Swiss Fedral Institute of Technology. CH-8092 Zurich.
[15] Barbut D. and Negrus E.M.,"CFD Analysis For Road Vehicles", INCAS Bulletin, Volume 3, Issue 3, pp. 15 – 22, 2011.
[16] Pamadi B.N., Taylor L.W. and Leary T.O.,"A Method For The Reduction Of Aerodynamic Drag Of Road Vehicles", NASA Technical Memorandum 102589, 1990.