Scientific Journal of Impact Factor (SJIF): 4.72
e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 4, Issue 6, June -2017

EFFECT OF VARIOUS AERODYNAMIC DRAG REDUCTION METHODS ON VEHICLE- A REVIEW

[1]DARSHAN M. DESAI , [2]IMRAN MOLVI

[1]*PG Scholar, Mechanical Department, PIET, Parul University*
[2]*Head of mechanical department, PIET, Parul University*

ABSTRACT - *Various technique to reduce the drag of bluff body through mechanism of back pressure recovery has been investigated including spoiler, tail plate, vortex generator, rear suction and guide vane. Pressure distribution over external surface has important significant for aerodynamic study of car. This review paper intends to provide information on recent approaches and their efficiency in minimizing aerodynamic drag of vehicle. Study mainly focuses on methods employed to delay flow separation at the rear end of vehicle body. Research carried out on different drag reduction technique and their effect on aerodynamic drag is highlighted.*

KEY WORDS:Aerodynamic drag, drag coefficient, boundary layer, flow separation

INTRODUCTION

Recent spike in fuel prices and uncertain future of fossil fuels as well as control over global warming give tremendous pressure on design engineering to improve current efficiency of vehicle. Efficiency of vehicle can be utilize by minimizing waste of power to propel auto. Perfect aerodynamic car not only consume less fuel but also overcome drag exerted by air while running at high speed and also provide good stability and handling behavior. "Aerodynamics" is the branch of fluid dynamics concerned with studying the motion of air, particularly when it interacts with moving objects. There are two major part of aerodynamics, External aerodynamics that deal with flow over the solid body with different shape and second one is internal aerodynamics that deal with flow pass through inner compartment of solid body. Flow over car, airplane and turbine blade are example of external aerodynamics and Air pass through automobile engine, jet engine, air conditioner are example of internal aerodynamics.

Aerodynamic drag is also one of factor which play a major role in vehicle stability, power consumption and overall efficiency. Aerodynamic drag consist mainly two component skin friction drag and pressure drag. Pressure drag account most of the total aerodynamic drag which is depend on external geometry due to boundary layer flow separation from rear window surface and wake region behind the vehicle [1]. Location of separation point determine the size of wake region and consequently it's determine the value of drag coefficient. Flow separation control of car is major interest in fundamental fluid dynamics as well as in various engineering applications. Many techniques have been determined to control the flow separation either by preventing it or reducing its effect.

Based upon whether the methods consume energy to control the flow or not, they are classified into active or passive control methods. Active control is performed by using actuators that require a power generally taken on the principal generator of energy of the vehicle.

LITERATURE REVIEW

Active flow control with suction

In Harinaldi et al.[2] study, a family van was modeled with a modified form of Ahmed's body by changing the orientation of the flow from its original form (modified/reversed Ahmed body). This model was equipped with suction on the rear side to comprehensively examine the pressure field modifications that occur. The investigation combines computational and experimental work. Computational approach used a commercial software with standard k-epsilon flow turbulence model, and the objectives was to determine the characteristics of the flow field and aerodynamic drag reduction that occurred in the test model. Experimental approach used load cell in order to validate the aerodynamic drag reduction obtained by computational approach (Fig. 1). The results show that the application of suction in the rear part of the van model gives the effect of reducing the wake and the vortex formation. They found that aerodynamic drag reduction is closed to 13.86% for the computational approach and 16.32% for the experimental.

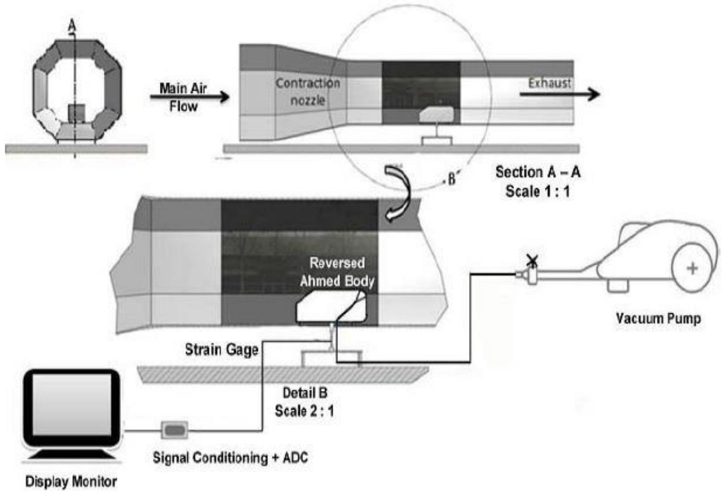

Figure 1 Scheme of aerodynamic drag measurement Passive control method with different spoiler

V.N.Kumar et al[3] worked on a sedan car with different types of spoilers for finding out drag and lift forces at different velocities. The design of sedan car was done on CATIA-2010 and analysis was performed in ANSYS-fluent solver.In this analysis two styles of spoilers were applied, primary spoiler was a "wing" type spoiler mounted 28cm above the surface of car's rear end , and the 2d spoiler was hooked up edge of the rear aspect of the car without leaving space between the spoiler and the surface of the car . Comparison of drag and lift forces without spoiler, with spoiler 1 and with spoiler 2 at different velocities were obtained .using wing shape spoiler drag coefficient is reduced from 0.333 to o.329.

The simulation of external aerodynamics is one of the most challenging and important automotive CFD applications. With the rapid developments of digital computers, CFD is used as a practical tool in modern fluid dynamics research. It integrates fluid mechanics disciplines, mathematics and computer science. With high-speed automobiles much more common nowadays, reducing the lift coefficient to enhance stability on the road is no longer just a concern for race cars. Spoilers are one of the well-known devices for producing down force on a moving vehicle.

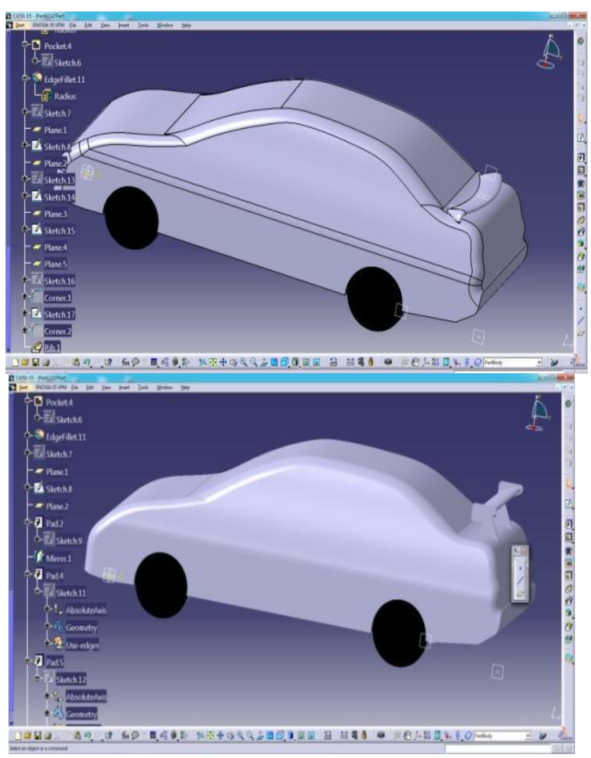

Figure 2 Assembly 3D CAD model of vehicle with different spoiler

Hu and Wong[4] studied the flow around a simplified high speed passenger car with a rear-spoiler and without a rear-spoiler. The standard k-ε model was selected to numerically simulate the external flow field of the simplified Camry model with or without a rear-spoiler. Through an analysis of the simulation results, a new rear spoiler was designed and it shows a mild reduction of the vehicle aerodynamics drag. This leads to less vehicle fuel consumption on the road. Usingspoiler at rear end 1.7 % drag reduction achieved at a high speed.

Tail plate

R. B. Sharma and Ram Bansal performed experiment using tail plate to reduce drag[5]. In this study model of generic passenger car has been developed in solid works-10 and generated the wind tunnel and applied the boundary conditions in ANSYS workbench 14.0 platform then after testing and simulation has been performed for the evaluation of drag coefficient for passenger car. In another case, the aerodynamics of the most suitable design of tail plate is introduced and analyzed for the evaluation of drag coefficient for passenger car. The base line model of generic passenger car is designed in Solid Works. figure3 show the generic passenger car used in the present CFD simulation. The maximum value of the coefficient of drag is 0.3512 and the Maximum value of the coefficient of lift is 0.2310 for the base model.The Tail plates are placed at back side of the roof and at the tail bumper of the passenger car at 12^{o} inclination angle. With Tail plates, the maximum value of the Coefficient of drag (CD) is 0.3376 and the Maximum value of the coefficient of lift (CL) is 0.1926. Thus, it is concluded that, the drag coefficient is reduced 3.87% and lift coefficient is reduced 16.62% with installation of tail plates. Hence, the tail plate is one of the powerful add on device to reduce the aerodynamic drag of a car.

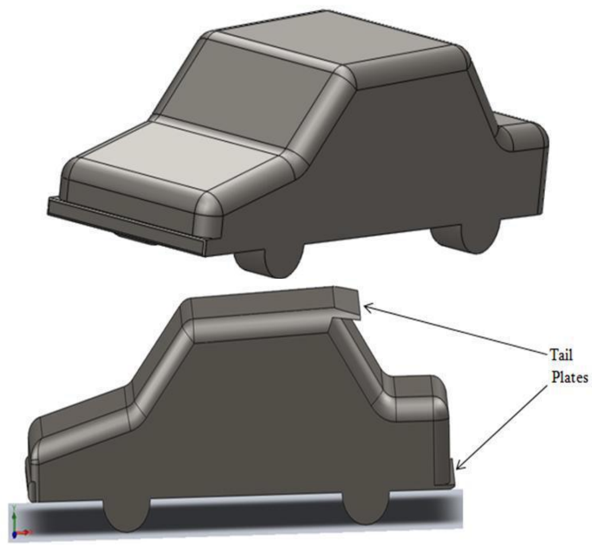

Figure 3Solid work model of Base Car and Passenger car with tail plates

Jeff howell modified a simple car by tapering of the rear upper body on both roof and sides. The effect of taper angle and taper length on drag and lift characteristics are investigated.

Figure 5 tapered windsor body for wind tunnel body

At a shorter taper length as initial sharp reduction in drag occurs, but this trend is reversed and a local drag peak is experienced for taper angle greater than 10°. At larger taper angles a significant increase in drag can occure for this taper length. A simular anomaly in the drag trend with taper length was identified for the body with roof taper only at one taper angle of 15°[6]. Pressure mesurement showed that strong suction is formed at the intersection of the tapered rear surface and the flat roof and bodyside, which affect both drag and lift.

Rear underbody slicing and under body diffuser

S.M.Rakibul Hassan[7] has worked on exclusive aspect evaluation of aerodynamic drag of racing cars and exclusive drag reduction strategies reminiscent of rear beneath physique modification and exhaust gas redirection towards the rear separation zones. Through a numerical process (Finite Volume Method) of solving the Favre-averaged Naiver-Stokes equations. On this work, numerical simulations are carried out to research the drag of a racing vehicle and a few tactics to scale down it with the aid of reducing the flow separation. As the speed of car increases, the stagnation stress raises and at the same time pressure at rear area decreases too because of develop in momentum of air.

To reduce the pulling-back effect, a unique idea is to slice the rear under-body at certain angles as shown in figure 6(a) which actually directs some flow from the under-body to the low pressure zone. Another popular way to reduce the rear end separation is to use under-body diffuser as shown in figure 6(b) which also adds elegance in aesthetics of the car, but it has less flow area as the rear under-body is not fully sliced out.

These must be considered while designing a car for higher speed and acceleration as well as for better fuel efficiency and control. Aerodynamic drag reduction by rear under body modification results in up to 22.13% and rear under-body diffuser results 9.5% reduction of drag coefficient.

Rear suction

Rohan Yadav and Justin Fischer[8] has introduced new technique suction slit to reduce drag generated at rear end of vehicle. The physical model is 1/10th scale generic SUV without side mirrors. A rectangular slit was added and merged at the rear of the vehicle. The slit length, width and centroid vertical coordinate were all normalized with reference to the vehicle height. Air is sucked uniformly through the slit and the boundary inlet velocity together with were considered as the design variables.

Orthogonal array optimization or Taguchi method is a statistical technique used to study the simultaneous effect of multiple variables on the performance of a process. In this technique they used numbers of arrays and iteration to get minimum value of drag by incorporating Solid works for modeling, Ansys for generation of proper meshes and fluent analysis. The model including the suction slit were imported to the Design modeler, and aligned with a control volume. A half model was used to allow quicker solution of the model with a more refined mesh. The control volume size was set according to Fluent’s best practice guide for vehicle analysis.

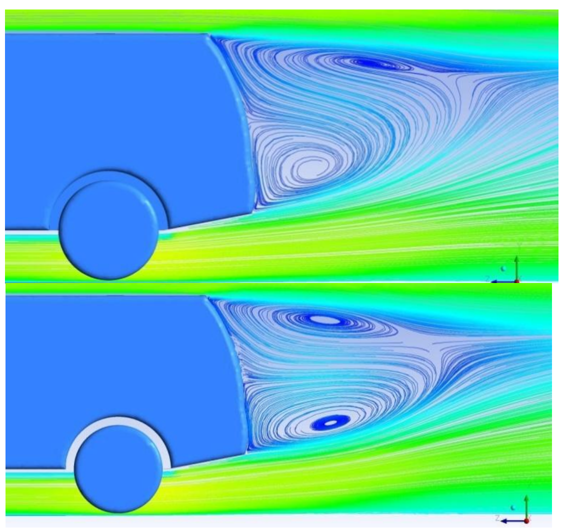

Figure 8 Streamlines colored by the pressure coefficient (no suction and with suction)

This result is also confirmed in figure.3.8 where the pressure coefficient (Cp) is plotted as a function of the normalized height. The pressure gain in the lower part of the suction slit outweighed the loss in pressure above the opening. Overall, the inclusion of suction reduced the pressure difference between the fore and after facing surfaces of the vehicle which thereby reduced drag.

E.M.Wahba et al[9] performed an experiment to investigate the drag reduction capacity of guide vanes considering two types of ground vehicles, a simplified bus model and a simplified sport utility vehicle (SUV) model. To obtain an optimal configuration, guide vane configuration, chord length and angle of attack are varied. Lateral guide vanes were installed near the rear end of the box model which direct the air flow into the low pressure wake region.

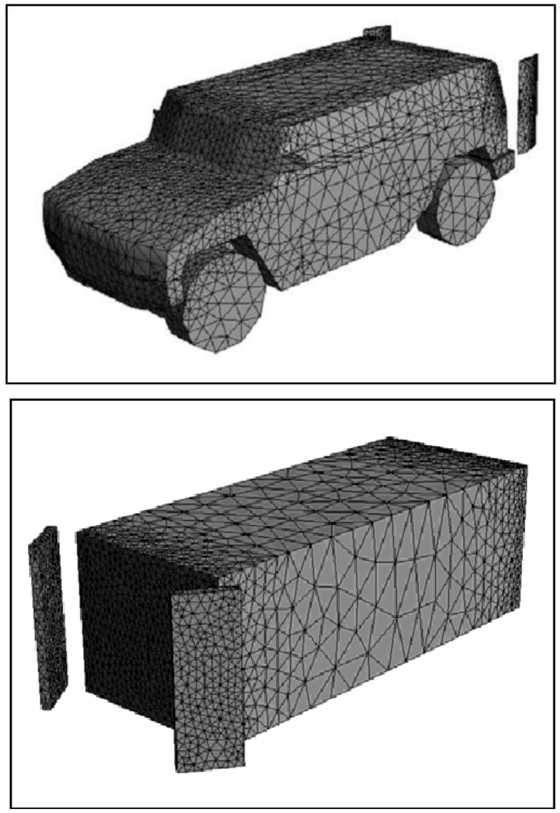

Figure 9 Schematic with lateral guide vanes installed for the Box and The SUV Model

An overall reduction in aerodynamic drag coefficient was up to 18% for bus and SUV model is observed with the proposed modifications. The guide vanes with symmetric cross section provided more drag reduction than asymmetric cross sections.

Vortex generator

Vortex generator is an aerodynamic surface which is basically a small vane that creates a vortex. VG widely used in the aerospace industry, mainly to control boundary layer transition and to delay flow separations. A different type of VG is used on race cars for manipulating the flow over and under the vehicle, mainly to generate down force.

K. Selvakumar et al[10] tested the effect of bump –shaped vortex generator attaching on the roof end of a sedan car using both computational tools and wind tunnel experiment.

Both experimental and computational method show that drag force increases with velocity for both base and VG model .After the application of VG, a comparative reduction in drag coefficient value is observed .Due to the application of VG, a downforce creates which pushes the car to the ground resulting more stability at high velocity.

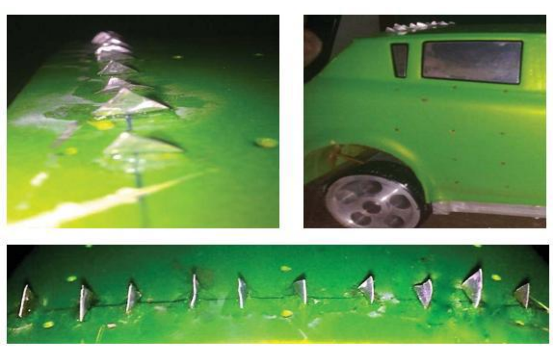

Figure 10 Installation of vortex generators in the base model [7]

Gopal P et al[11] carried out experimental investigation on coefficient of lift and drag with and without vortex generator on the roof of a utility automobile at various yaw angles (10°, 15°and 20°).A scale down model 1:15 of utility vehicle with velocities of 2.42, 3.7, 5.42 and 7.14m/s are tested in wind tunnel. An exceptional reduction in drag force can be obtained by the application of VG. Experimental results showed that drag coefficient reduced maximum for Vortex Generator with 15° yaw angle at lower velocity and the lift coefficient remains constant for VG with varying yaw angle at higher velocities.

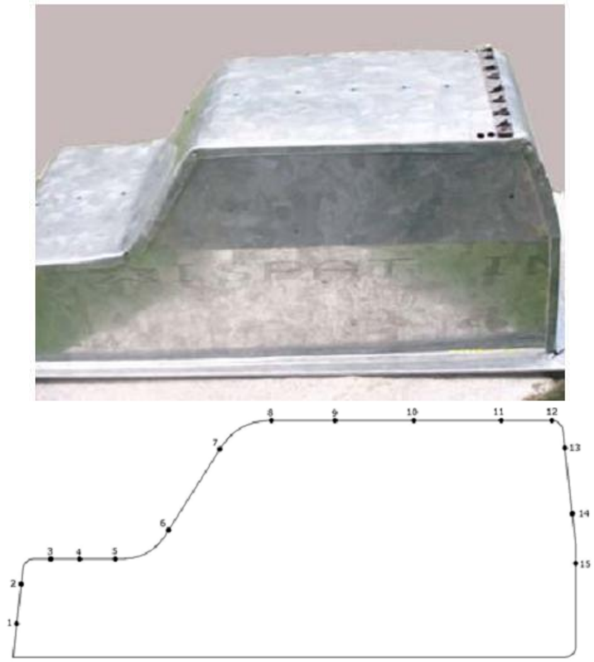

Figure 11 Scale Model and location of tapping points [5]

CONCLUSION

Aerodynamically designed vehicles provide significant improvement in fuel savings over its life cycle but all technically feasible designs are not incorporated owing to limitations such as space optimization, vehicular size optimization as well as aesthetics.

Various drag reduction devices as mention in literature chapter like bump shape and delta wing shape vortex generators, rear spoiler, guided vane, tail plate, rear under body slicing, underbody diffuser and rear suction can be used to reduce drag in SUV car. Along with alltechnics, vortex generator and rear suction method is more advance in automobile field and also future research in this field can be provide more absolute result in reducing drag.

Although wind tunnel experiments simulate the real world scenario, CFD studies too can be utilized for generating initial modeling and conceptualizing the designs before validating with experiments.

Selection of correct design and /or add-on devices depends upon factors such as cost, vehicle type and geography it is designed for.

REFERENCES

[1]Mayer, W., and Wickern, G. (2011) The new audi A6/A7 family-aerodynamic development of different body types on one platform, *SAE International Journal of Passenger Cars-Mechanical Systems*, Vol. 4(1), pp197-206.

[2] Harinaldi, B., Warjito, E. A. K., and RustanTarakka, S. P. S. (2012) Modification of flow structure over a van model by suction flow control to reduce aerodynamics drag. *Makara Seri Teknologi*, Vol. 16(1), pp15-21.

[3] V.N .Kumar,K.L .Narayan,L.N.V.Narasimha Rao and Y Sri Ram, "Investigation of Drag and Lift forces over the Profile of Car with Rearspoiler using CFD"in International Journal of Advances in Scientific Research,2015

[4] Hu, X. X., and Wong, T. T. (2011) A numerical study on rear-spoiler of passenger vehicle, *World Academy of Science, Engineering and technology*, Vol. 57, pp636-641.

[5] Sharma, R. B., and Bansal, R. (2013) CFD simulation for flow over passenger car using tail plates for aerodynamic drag reduction, *IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE)*, Vol. 7, No. 5, pp 28-35.

[6] Jeff Howell, Martin Passmore, Simon Tuplin "Aerodynamic drag reduction on a simple car – like shape with rear upper body taper", SAE International journal, Mech syst. 6(1):2013, doi: 10.4271/2013-01-0462.

[7] S.M. Rakibul Hassan, Toukir Islam, Mohammad Ali, Md. Quamrul Islam Numerical Study on Aerodynamic Drag Reduction of Racing Cars, Department of Mechanical Engineering, Bangladesh University of Engineering and Technology, Dhaka-1000, Bangladesh.

[8] Rohan Yadav and Justin Fischer"Aerodynamic Drag Reduction for A Generic Sport Utility Vehicle Using Rear Suction"- AbdellahAit Moussa.

[9]Wahba, E., Al-Marzooqi, H., Shaath, M., Shahin, M., and El-Dhmashawy, T. (2012) Aerodynamic drag reduction for ground vehicles using lateral guide vanes, *CFD Letters*, Vol.4(2), pp68-79.

[10] "Experimental Investigations on Optimization of Aerodynamic Characteristics in a Hatchback Model Car" Using vortex Generators ,December 2013- by K.Selvakumar et al.

[11] Gopal P and Senthilkumar T, "Aerodynamic Drag Reduction in a Passenger Vehicle Using Vortex Generator With Varying Yaw Angles"in Journal of Applied Science and Engineering, Vol-7, 2012.

.

