

DRAG REDUCTION USING VORTEX GENERATOR ON A PASSENGER VEHICLE

M.Kanagaraj¹, Dr.S.Periyasamy²

¹Assistant Professor, Department of Mechanical Engineering, Ranganathan Engineering College, Coimbatore,
Tamilnadu, India¹

Associate Professor, Department of Mechanical Engineering, Government College of Technology, Coimbatore
Tamilnadu, India²

ABSTRACT :Now a day's passenger cars like light utility vehicle are facing aerodynamic drag. One of the main causes of aerodynamic drag in vehicle is flow separation near the vehicle's rear end. To control the flow separation vortex generators are used. A vortex generator is an aerodynamic surface, consisting of a small vane that create vortex. In this project design and analysis of different shapes of vortex generator was done using CFD software. Vortex generator themselves create drag, but they also reduce drag by preventing flow separation at downstream. The overall effect of vortex generators was calculated by totaling the positive and negative effects .From the CFD analysis drag, velocity streamline and pressure gradient were obtained. Based on the result best vortex generator was identified for a passenger vehicle.

KEYWORDS: Aero dynamics, body, Computational Fluid Dynamics (CFD), Flow simulation.

I.INTRODUCTION

A vortex generator (VG) is an aerodynamic device, consisting of a small vane usually attached to a lifting surface such as an aircraft wing or a rotor blade of a wind turbine. VGs may also be attached to some part of an aerodynamic vehicle such as an aircraft fuselage or a car. When the airfoil or the body is in motion relative to the air, the VG creates a vortex, which, by removing some part of the slow-moving boundary layer in contact with the airfoil surface, delays local flow separation and aerodynamic stalling, thereby improving the effectiveness of wings and control surfaces, such as flaps, elevators, ailerons, and rudders.

II. WORKING OF VORTEX GENERATOR

As stated above, the vortex generators modify the boundary layer. It is the layer of fluid in the immediate vicinity of a boundary surface where the effects of viscosity are significant. In the boundary layer significant are viscosity and friction forces. The viscosity of the fluid and the friction of the object surfaces generates high transverse velocity gradients. Therefore, the boundary layer is an area of particular importance. Outside the boundary layer, where viscous forces play a minor role, usually assume that the flow is in viscid. The flow in the boundary layer can be either laminar or turbulent. During the flight, the air flows around the wing. The viscosity of the air and the surface friction of the wings causes the air molecules in contact with the surface of the wing. Molecules slightly farther away from the wing surface moves in the direction of flow, but they are slowed. The farther from the wing surface, air speed is greater, up to the point where the air has a constant speed, as the speed of the incoming air. The layer of air from the surface of the wing up to this point we take as the boundary layer. It can be laminar or turbulent. The flow in the laminar boundary layer is orderly and gentle. In this layer do not move perpendicular to the direction of flow, or any turbulence. Moving air molecules to form successive layers, sliding against each other, which, despite the difference in speed, they do not mix with each other. This is because the mass and momentum transfer between the layers occurs only at the microscopic level, in a macroscopic scale, this transfer is not affected. Randomly generated disturbances are

immediately damped; because the viscous forces dominate over inertial forces here Laminar flow is called also stable flow .In the turbulent flow particles of fluid in addition to the main movement also oscillate. Layers also move perpendicular to the direction of flow.

III.METHODOLOGY

Vortex generators were studied to install immediately upstream of the flow separation point in order to control separation of airflow above the vehicle's rear window and improve the aerodynamic characteristics. To delay flow separation, bump shaped vortex generators are tested for application to the roof end of a sedan [5]. Commonly used on aircraft to prevent flow separation, vortex generators themselves create drag, but they also reduce drag by preventing flow separation at downstream. The overall effect of vortex generators can be measured by totaling the positive and negative effects. So the effectiveness of the Delta-wing-shaped VG is more than bump shaped vortex generator. The reason for why delta-wing-shaped VGs are more effective than bump-shaped VGs can be explained as follows: Delta-wing-shaped VGs have a smaller frontal projection area, which means that they themselves create smaller drag [3]. Moreover, the vortex generated at the edge of a delta-wing shaped VG keeps its strength in the flow downstream of the edge since it barely interferes with the VG itself because of the VG's platy form [4]. With bump-shaped VGs, on the other hand, the vortex is generated at a point close to the downstream edge of the bump, which causes the vortex to interfere with the bump and lose its strength Many aircraft carry vane vortex generators from time of manufacture, but there are also after-market suppliers who sell VG kits to improve the STOL performance of some light aircraft [2]. Installation of vortex generators can usually bring about a slight reduction in stalling speed of an airplane and therefore reduce the required one-engine-inoperative climb performance. The reduced requirement for climb performance allows an increase in maximum takeoff weight, at least up to the maximum weight allowed by structural requirements [7]. Flow Visualization is a powerful tool in experimental fluid mechanics. The unique advantage of this technique is that certain properties of the flow field become directly accessible to visual perception and the insight into a physical process becomes clearer. Most fluids are transparent media and their motion remains invisible to the human eye during direct observations. However, the motion of such fluids can be recognized by making use of techniques by which the flow is made visible and such techniques are called flow visualization techniques. It is also possible to derive quantitative data from the flow pictures obtained by such techniques. Information about the complete flow field can be arrived without physically interfering with fluid flows. Flow measuring methods involve the use of a sensing device or probe causing disturbance to the flow field. Also probe measurements provide data at only one point at any given instant. It is therefore necessary to introduce a number of probes to collect data from many points in a section at any given instant. Introduction of a number of probes may completely change the flow conditions. Such difficulties can be overcome by use of flow visualization techniques. The methods of flow visualization can be normally classified into three groups (1) Injection of foreign material to the flowing fluid, (2) Optical method for compressible flows, and (3) Introduction of contaminants in the form of heat or electrical energy. First methods involve foreign material which involves hydrogen bubble if water is the flowing medium since the basic process involved is electrolysis of water. This process needs a electrolysis process in which Very thin wires produce very small hydrogen bubbles in size and the buoyancy forces become negligible compared to hydrodynamic drag forces causing little disturbance of actual flow conditions. That is why this process has some bindings in flow visualization. For the purpose of visualization in air in wind tunnel experiments smoke lines are used in the same way as dyes and other injecting materials in liquid flows. Smoke is usually generated by vaporization of a mineral oil or by the burning or smoldering of wood, paper or tobacco. Velocity profiles can also be visualized in liquids and air by electrolytic and photochemical dye production. Controlled rate of dye production is done electrically to determine the time and magnitude of voltage applied between electrodes. The dye production can be carried out at all points or along a defined curve, but exact temporal control should be maintained for better and accurate results. Recent developments indicate that smoke visualization in wind tunnels, one of the oldest flow visualization techniques, will continue as an important experimental tool in the study of complex flow dynamic phenomena. Improvements in generation and injection of smoke as well as in lighting (laser as a light source), in techniques of acquisition and computation have continued to increase the scientific value of this method. Similar results are obtained by flow visualizations with fog and vapor. The smoke can be very useful in a wind tunnel with low turbulence. There exists no upper limit of speed for smoke line visualization (it was possible to extend the range of smoke line visualization even to supersonic flow velocities). Smoke line can be generated in a wind tunnel

(smoke tunnel) by introducing smoke (produced by smoke generated devices) through small pipes placed in front of a test model, or through holes on the model surface. The choice of using smoke in a wind tunnel depends on several aspects. The smoke must be dense and white for visibility, non toxic and non corrosive. The quality of the observed or photographed smoke line depends also on the choice of the illumination system.

There are three basic types of smoke suitable for wind tunnel experiments: smoke generated by the vaporization of a mineral oil (paraffin, kerosene) mist resulting from the vaporization of certain substances containing bromide or chloride and smoke from burning or smoldering wood, paper or tobacco. The burning or vaporization is done in a smoke generator.



Fig 1. VG DIFFERENT SHAPES

IV.DESIGN CALCULATION

1) BOUNDARY LAYER THICKNESS (δ)

$$\delta = \frac{5.0 \times X}{(Re)^{1/2}}$$

$$= \frac{5.0 \times 0.616}{(3500)^{1/2}}$$

$$\delta = 0.052 \text{ m}$$

2) REYNOLDS NUMBER (Re)

$$Re = \frac{\rho \times V \times X}{\mu}$$

$$= \frac{1.205 \times 110 \times 5.74 \times 10^{-4}}{1.983 \times 10^{-5}}$$

$$Re = 3500$$

3) DRAG FORCE

$$F_d = \frac{1}{2} \times c_d \times A \times \rho \times v^2$$

$$= \frac{1}{2} \times 1.924 \times 0.024 \times 1.205 \times (110)^2$$

$$= 508.540 \text{ N}$$

where

F – drag force N
A – the surface area m²
c_d – coefficient of discharge
 ρ – density of air Kg/m³
V – velocity of air m/sec

4) DIMENSION OF VORTEX

$$L = 2H = 2 \times 52.06 = 104.12 \text{ mm}$$

L=Length of vortex mm

B=10mm

B=Width of vortex mm

V.MODELLING

PARAMETER	ORIGINAL DIMENSION (mm)	SCALED DIMENSION (mm)
LENGTH	4440	621
HEIGHT	1495	195
WIDTH	1695	237

Table 1. Prototype Dimensions of the model

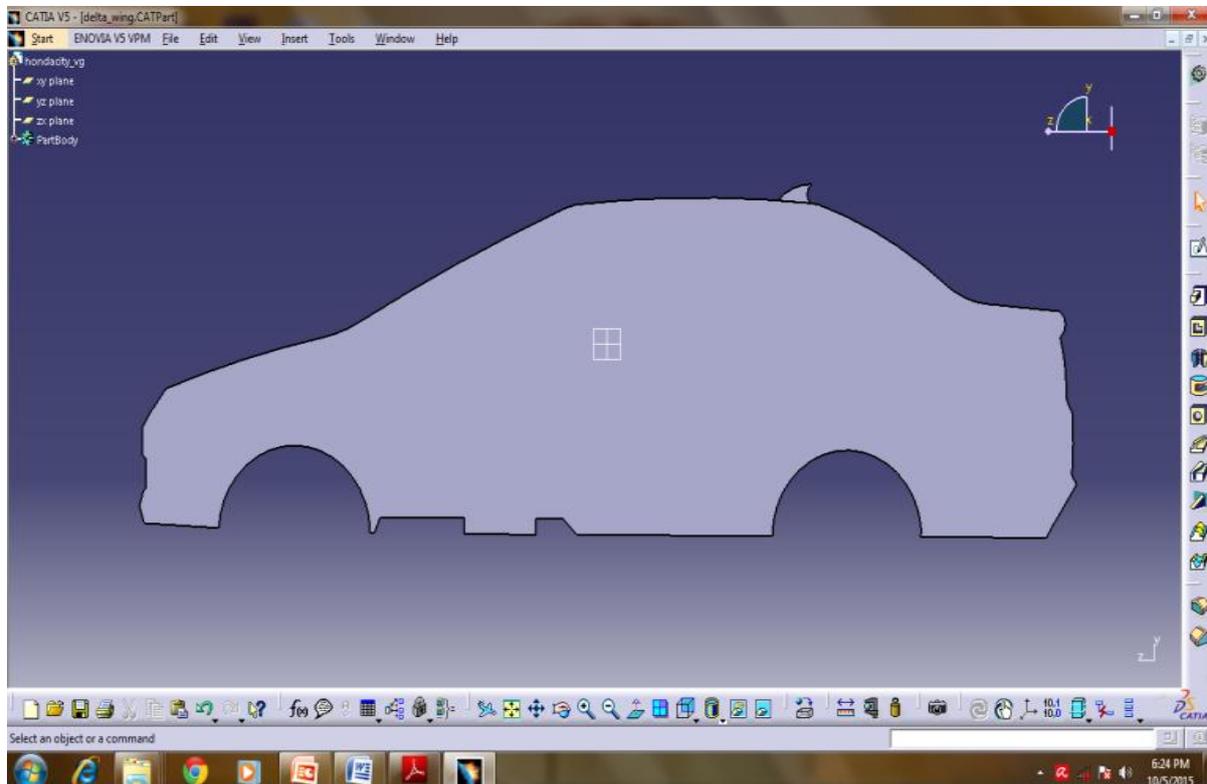


Fig. 2 Scale Ratio of Prototype model– 1:7

VI.RESULT AND DISCUSSION

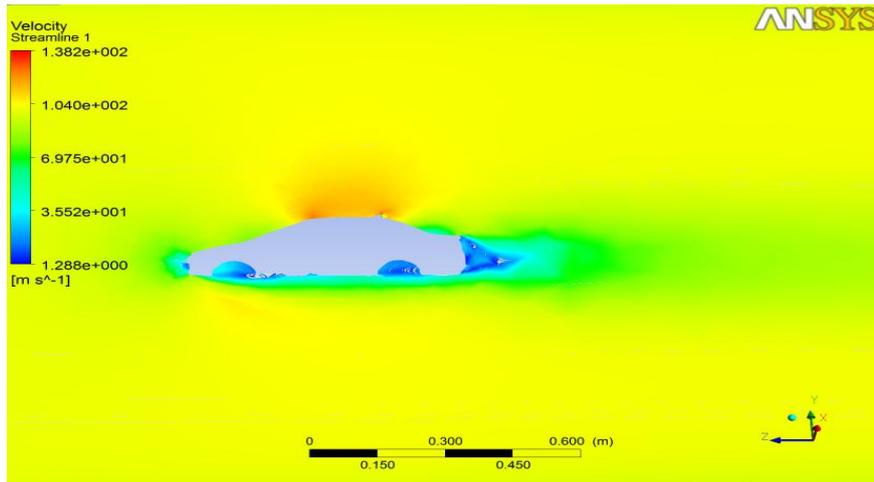
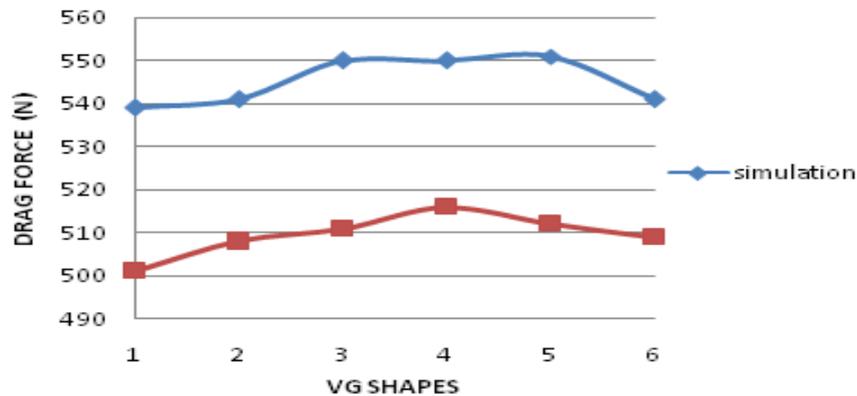


Fig. 3 Velocity Distribution



Graph 1. Relation between VG Shapes and Drag Force(N)

RESULT COMPARISON

S.No	Vortex generator shapes	Drag force (N)
1	Rectangular shapes	539.005
2	Delta wing shapes	541.504
3	Parabolic shapes	550.008
4	O give shapes	550.228
5	Gothic shapes	551.916
6	Triangular shapes	541.504

Table 2 Representation of Drag Force for various VG Shapes

International Journal of Innovative Research in Science, Engineering and Technology

An ISO 3297: 2007 Certified Organization

Volume 6, Special Issue 7, April 2017

5th National Conference on Trends in Automotive Parts Systems and Applications (TAPSA-2017)

30th & 31st March 2017

Organized by

Sri Krishna College of Engineering & Technology, Kuniamuthur, Coimbatore-641008, Tamilnadu, India

VII.CONCLUSION

The flow analysis over a car is studied for various vortex generators and results are discussed. From the results the rectangular profile has minimum drag compare than other profile vortex generators. Minimum Percentage of drag decrease is 0.4636%. compare with delta wing shapes VG. Maximum Percentage of drag decrease is 2.45% compare with triangular VG. If this type of VG used in car, the fuel economy may be increased. Thus the best VG is identified based on drag.

REFERENCES

- [1] C. H. K. Williamson. Three Dimensional Vortex Dynamics in Bluff Body Wakes.
- [2] Alam F. and Watkins S. 2004. Aerodynamics of Trucks and Drag Reducing Devices. Journal of Mechanical Engineering, Khulna University of Engineering and Technology, Khulna, Bangladesh.
- [3] Hucho W. H. 1998. Aerodynamics of Road Vehicles. 4th edition. Society of Automotive Engineers (SAE). ISBN 0-7680-0029-7, Warrendale, USA.
- [4] Cooper K R. 2006. Full-Scale Wind Tunnel Tests of Production and Prototype, Second-Generation Aerodynamic Drag-Reducing Devices for Tractor-Trailers. SAE Paper No. 2006-01-3456, USA.
- [5] Barlow J.B., Rae Jr., W. W. and Pope A. 1999. Low-Speed Wind Tunnel Testing, 3rd Edition. John Wiley and Sons, Inc. New York, NY, USA.
- [6] RAS 66029. 1981. Subsonic Performance Data for NACA Type Submerged Air Intakes. Royal Aeronautical Society.
- [7] Stephens A.V. 1957. Solid Boundary Surface for Contact with a Relatively Moving Fluid Medium. U.S. Patent No. 2, 800, 291. U.S. Department of Commerce, Washington DC, USA.
- [8] Kuethe A.M. 1970. Boundary Layer Control of Flow Separation and Heat Exchange. US Patent. 3, 74 1