Vortex Generator for Drag Reduction in Hatchback Vehicles

Dhumal Hrushikesh S.
Institute of Knowledge College of Engineering Pimple Jagtap, Pune
dhumalhs@gmail.com

Shelar Kashinath V.
Institute of Knowledge College of Engineering Pimple Jagtap, Pune
kashinathshelar16@gmail.com

Lakule Suhas J.
Institute of Knowledge College of Engineering Pimple Jagtap, Pune
lakulesuhas89@gmail.com

Hande Tushar D.
Institute of Knowledge College of Engineering Pimple Jagtap, Pune
tushar101094@gmail.com

Kodarkar Nakul M.
(Assistant Professor.*) Institute of Knowledge College of Engineering Pimple Jagtap, Pune*
nkodarkar@gmail.com

Abstract— *Any physical body being propelled through the air has drag associated with it. In aerodynamics, drag is defining as the force that opposes forward motion trough the atmosphere and is parallel to the direction of the free-stream velocity of the airflow. Drag must be overcome by thrust in order to achieve forward motion. Drag is generated by motion of air particles over the aircraft. A vortex generator is an aerodynamic surface, consisting of a small vane or bump that creates a vortex. Vortex generator can be found on many devices, but the term is most often used in aircraft design vortex generators delay flow separation; they improve the effectiveness of control surface. The boundary layer normally thickness as it moves along the car surface , reducing the effectiveness of trailing-edge control surfaces, vortex generators can be used to remedy this problem, among others, by "re-energizing the boundary layer". The goal of project is to delay flow separation on the downstream side of the roof of car on account of decreasing pressure difference between the upstream and downstream side by creating vortex at the rear end of the car roof.*

Keywords— Drag force; boundary layer

I. INTRODUCTION

Vortex generators can be found on many devices, but the term is most often used in aircraft design. All vehicles don't have vortex generator. So this project is basically implemented to do study on several parameters related with vortex generator, like vortex generator location and their size . That's why we can do CFD analysis of vortex generator and evaluation drag reduction. Same way it will be verified by the wind tunnel. Hatchback vehicle don't have vortex generator, so it also deal with vortex generator analysis with hatchback car.

The presence of road vehicles is increases each day. This situation forced the manufactures to improve their vehicle's design. Some of the improving features on the vehicles are aerodynamic design, ergonomic design and the power of the engine. Feature such an aerodynamic design need to be improve after concerning to the attractive looks, fuel consumption and passengers safety regarding to the vehicle's stability. It is an important feature of aerodynamic design to give safety condition for the passenger.

The body of a car is design via studies of air flows through the surface. Drag coefficient of a car is refers to the dimensionless coefficient of drag force of a certain shape. The reduction of drag is essential to improve driving performance, fuel consumption and aerodynamics design that can gain attractive. The body car is design in such ways to allow air flows through the body. The sedan car like Proton

Saga car body's bluffness, when expressed by the drag coefficient is generally between 0.2 and 0.5. Two elements that have major influence on the drag coefficient of a bluff object are the roundness of its front corners and the degree of taper at its rear end. Presence of the trunk at the rear end cause the flow to separates at the roof end and then spreads downwards.
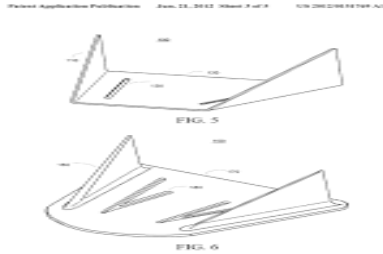

Fig no.1 Actual diagram of VG

II. LITERATURE SURVEY

In the literature [1], study aims at distinguishing the individual influence of the passive flap & the control system .In this paper show the effect of flow control on 2 dimensional bluff body & show relation between static pressure at rear end of modal & characteristics of work flow. The bluff body depends on numerous parameters such as the Reynolds number & characteristics of the incoming boundary.

G. Pujals and team[2] deals with the potential of separation control based on large scale coherent streaks is experimentally investigated. Such streaks are generated by means of a carry cylindrical roughness element a 3D flow subject to an adverse pr. Gradient due to geometrical constraints.In many applications boundary layer separation is associated with a large loss of performance making separation control of great importance. Active closed loop system has to be effective in reducing the drag of bluff bodies. Passive (open-loop) separation control is very attractive because as it is easier to implement and to handle it has very low production and maintenances cost. The most common passive device is vertex generator.

Literature [3] present's the optimization result & effect of vortex generation in the flow field. Also it explains the mechanism by which this effect takes place. The car obliged to have a body shape that is rather aerodynamics bluff, not ideal streamline shape as seen on fish and birds. Such a body shape is inevitably accompanied by flow separation at the rear end. The drag coefficient is generally between 0.2 and 0.5.It work's to reduce drag by increasing the back pressure. However the VG's that are installed for

generating stream wise vortices bring drag by it. Evaluation of the effectiveness of VG's and optimization were conducted using MMC's full scale wind tunnel. VG's mechanism phenomenon was explained in detail using CFD analysis. Star –CD was used as the solver and RNG K-E model as the turbulence model in this system. In this paper we seen that the optimum height of VG's almost equivalent to the thickness of the boundary layer and the optimum method of placement is to arrange them in the row in the lateral direction, 100mm upstream of the roof end at intervals of 100mm.

S.A.Isaev, Y.F.Gortyshov, V.M.Gureev, Yu.S.Opara, I.A.Popov [4] work on front & stern vertex creator boards are used of reduced drag & increases fuel economy of the truck. Truck with front & stern vertex generators have 25% of less fuel consumption as compare to truck with no vertex generator.

Study of Gopal P. & Senthilkumar T. [5] on a vertex generator with different yaw angles stated that VG'S are boundary layer manipulator .Their function is to reenergize an adverse pressure gradient boundary layer that is about to separate by transporting high momentum fluid from outer part of boundary layer down to the low momentum zone. In this experiment V.G. used have height nearly equal to boundary layer, length is taken twice to the height. Internal is six times of height between VG's. Which provide better result at yaw angle of 15^0. Where pressure coefficient is increased by 17% and it's also observed that additional VG increases dynamic pressure.

As per the literature [6], in vehicle body design & development the reduction of drag is essential since the primly concern of automotive industry is fuel consumption and protection of global environment. The most practical way of drag reduction at the rear end to us an effective flow control technique. Taper with trunk shape leads to delay in flow separation & hence lesser drag force. The VG4 configurations re-energize the flow significantly while comparing with other VG profile. From simulated result the VG4 is more effective in drag reduction while comparing with other configuration. The CFD results that the S-A modal predict the wake patterns and flow patterns significantly as well as quantitatively a reasonable drag reduction has been predicated by changing the VG drag force is being reduced by using VG4. CFD used with press outlet, velocity inlet, simple algorithm &second order upwind scheme.

III. WORKING AND DETAILS OF VORTEX GENERATOR

A. Working

Vortex generators work by mixing high-energy fluid (free stream air in this case) with the lower energy fluid found near the skin surface, called the boundary layer. This process is called re-energizing the boundary layer. The higher energy fluid is now more resistant to separation and allows for higher performance of the aircraft. Technically speaking, a vortex generator increases the mean stream wise momentum of the boundary layer by drawing in high momentum fluid from the free stream. Vortex generator (VG) is an aerodynamic surface, consisting of a small vane or bump that creates a vortex. Vortex generators delay flow separation and aerodynamic stalling, thereby improving the effectiveness of wings and control surfaces. We have chosen vortex generator as our aerodynamic device to be implemented in the sedan car to enhance the performance of the car. VGs were developed for the aircraft sector; this technology has made its way into car design.

The main function of this device is to delay air flow separation. Air flow separation is when the airflow of an object detaches from the surface and creates eddies and vortexes. So vortex generator over the rear of the roof effectively helps to reduce drag.

The fig shows a flow velocity profile on the vehicle's centerline plane near the roof end. Since the vehicle height in this section becomes progressively lower as the flow moves downstream, an expanded airflow is formed there. This causes the downstream pressure to rise, which in turn creates reverse force acting against the main flow and generates reverse flow at downstream Point C.
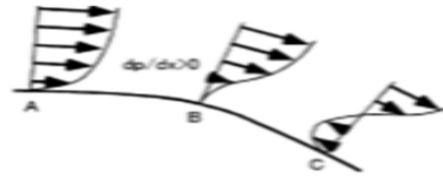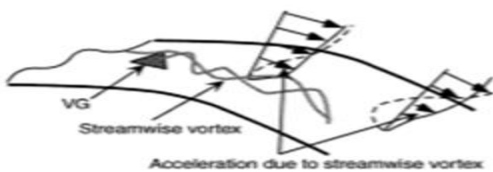

Fig.no2. Velocity profile around rear end

No reverse flow occurs at Point A located further upstream of Point C because the momentum of the boundary layer is prevailing over the pressure gradient (dp/dx). Between Points A and C, there is separation Point B, where the pressure gradient and the momentum of the boundary layer are balanced. Here the airflow quickly loses momentum as it moves downstream due to the viscosity of air.

The purpose of adding VGs is to supply the momentum from higher region to lower region. This allows the separation point to shift further downstream. So it enables the expanded airflow to persist proportionately longer. It reduces drag by increasing g the back pressure. It provides dual advantage in drag reduction: one is to narrow the separation region in which low pressure constitutes the cause of drag another is to raise the pressure of the flow separation region. As to the location of VGs, a point immediately upstream of the flow separation point was assumed to be optimum, and a point 100 mm in front of the roof end was selected.
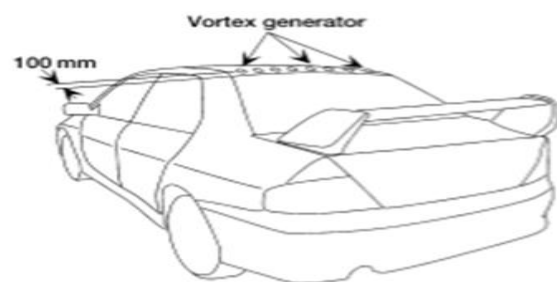
B. Location Of Vortex Generator

Fig.no.3 Location of the vortex generator

In connection with the size, the thickness of the boundary layer is measured base on the assumption that the optimum height of the VG would be nearly equal to the boundary layer thickness. Fig. shows the velocity profile on the sedan's roof. From this fig. the boundary layer thickness at the roof end immediately in front of the separation point is about 30mm.

Consequently, the optimum height for the VG is estimated to be up to approximately 30mm. As to the shape, a

bump shaped piece with a rear slope angle of 25 to 30 deg. is selected. Also to the location of VG's, a point immediately upstream of the flow separation point was assume to be optimum and a point 100mm in front of the roof end was selected.

C. Vortex Generator Design

As it was mentioned above, firstly the numerical simulation of non-controlled case was performed. According to existing model of NACA 63A421 with simple flap, the 3-D geometry with flap deflection 45° at angle of attack $\alpha = 0°$ and subsequently computational structured mesh were created in program Gambit. Inlet boundary conditions are in Tab. 1 and correspond to the boundary conditions of the experiment. The symmetric conditions were set on the upper, bottom and both side faces of the computational domain; pressure outlet condition was set at the outlet of the domain. Second order upwind discretization scheme was selected with respect to the mesh used.

Table no.1.Inlet boundary conditions

Results of the simulation provided information about location of flow separation, which occurred exactly on the flap edge. Location of the separation was a start parameter for vortex generator design. The design parameters of VG are defined in Tab. 2 and Because the position of VGs (ΔXVG) is determined from flow separation location and also from height of vortex generator (h), which depends on boundary layer thickness (δ) and this thickness in turn depends on the position on surface, the iteration method was used to define vortex generator position and height. Consequently other parameters were computed.

Table no.2. Design parameters of VG

VGs	h/δ	ΔXVG/h	e/h	L/h	Δz/h	β [°]
Triangular vanes	0.37	57	2	2.5	6	18
Rectangular	0.2	10	4	-	9	25

In order to find a viable configuration, one must first identify the important variables for vortex generator design. In order to reduce the degrees of freedom, most of the variables were fixed based on either analysis or recommendations of previous researchers. A Single vane type delta (triangular) shaped was chosen. Due to their simplicity and widespread usage, the low drag device than any other type makes the vane type more suitable for attaching on the vehicle body. Delta shaped VG's were most commonly used on aircraft wings.

In connection with the height, the thickness of the boundary layer is measured based on the assumption that the optimum height of the VG would be nearly equal to the boundary layer thickness. Figuire-1 shows the velocity profile on the vehicle's roof. From Figure-1, the boundary layer thickness at the roof end immediately in front of the separation point is found to be about 2 mm. Consequently, the optimum height for the VG is estimated to be up to approximately 2 mm. The thickness of VG was fixed at 0.5 mm uniform throughout so as to make a stiffened structure.
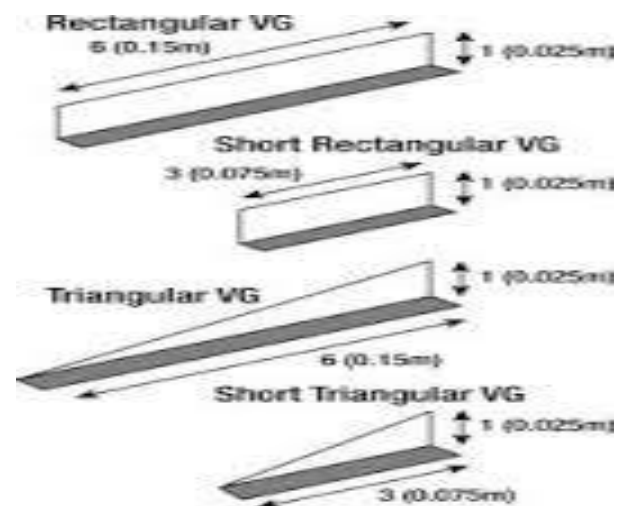
Free stream velocity v∞ [m/s]	Turbulence intensity Tu[%]	Length scale L [m]	Reynolds number Re [-]
14.3	0.25	0.005	200000

Fig.no.4. Types of VG
IV. EXPERIMENTAL SETUP

A) Wind Tunnel Testing

For the experimentation triangular delta shaped V.G. of height 30mm, length of 60mm(twice of the height) and width 5mm is selected and made wooden models to test in the wind tunnel.

The test model of car needs to be with a scale ratio of 1:10. The scale model of the vehicle will be shown in Figure-4. The length, breadth and height of the scaled model are was 0.360 m, 0.160m and 0.120m respectively.

The Vortex generators were cut into pieces from the sheet metal and they were fixed on to a base plate by gas welding process. The base plate with VG was fastened to the roof of scaled model by means of bolt and nut. To measure the static pressure on the body, 0.3 mm diameter holes were drilled on the centre line of the vehicle body starting from the front end along the roof to the rear end of the vehicle model. 10 pressure tapings are used. Out of which three of them are on the roof, two after the VG towards end ,three are on the front end and remaining two are at the rear of the vehicle. Pressure tubes are fixed from inside of the holes. Pressure tapings are connected to micro manometer using pressure tubes.
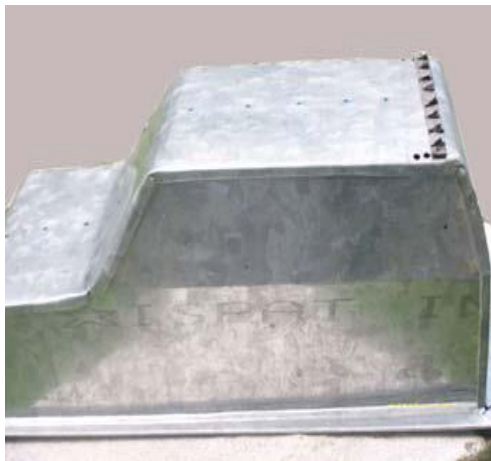
Fig.no.5. Scaled model of Vehicle

Wind tunnel has characteristics such as,
-maximum speed of air flux in working section without model installed is 15 m/sec;
-degree of initial turbulence of air flux in working section without model, defined using ball method, is 0.9%;
-length of working section is 400 mm;
-overall dimensions of working cross-section are: 200mm width, 250m m height

-wind tunnel is equipped with manometers test facility
-also has test facility for lift and drag.

B) CFD Testing

Model making is the first step for CFD testing. So CATIA model is prepared with the help of actual model of car (Model name swift VDi).
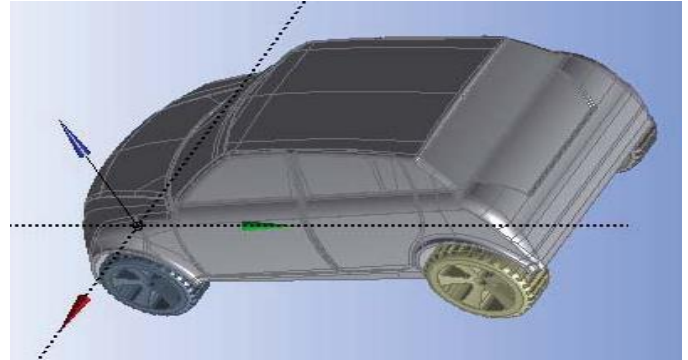
Fig.no.6. 3D model of Hatchback car

Model is imported in the fluent interface of the Ansys and for the analysis firstly domain is made with six time's greater dimension of the model in length and two times greater in width and height. Domain with vehicle model is shown in fig.no.7.
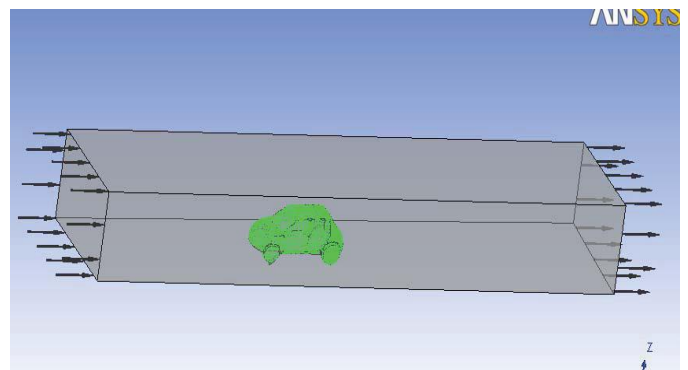
Fig.No.7. Domain Setup in Ansys Fluent

After the setup phase boundary conditions like velocity input of 20m/s , atmospheric pressure output and wall boundary condition is to be applied. Meshing will be done to the domain part with the help of hex meshing option.

V. RESULT AND DISCUSSION

A) Wind Tunnel Testing

In the wind tunnel testing when plotting of pressure distribution around car body done on drawing sheet, its observed that negative pressure gradient reduction is happen when V.G. is installed on the back of the vehicle.

It is seen that the value of CD decreases due to the addition of VG. This can be attributed due to the avoidance of flow separation with the help of VG. For instance at a velocity of 10 m/s the coefficient of drag is reduced by a maximum of 25% when VG with is used when compared to the values obtained without. Similarly at same velocity a minimum of 10% variation in reduction of drag is obtained for VG with a different yaw angles, this will be another parameter for further study.

B) CFD Testing

Once the simulation (without V.G.) was achieved forces acting in the horizontal and vertical direction were calculated. The horizontal axis and the vertical axis represent drag and lift respectively. The coefficient reported for the individuals are shown in Table

Table no.3

Parameter	Obtained value
Drag Coefficient	0.45
Lift Coefficient	0.27

The positive sign for the drag represents the force in an opposite direction to the motion of the car whereas for the lift coefficient presents the vertically upward acting force.

After the application of V.G. it is observed that by an exchange of momentum between upstream and downstream of the flow there is reduction in the high pressure region at the rear wind screen.
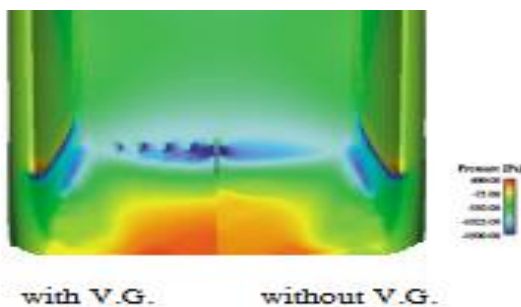

Fig.no.8 Impact of VG on the behaviour of rear end pressure

It is also proved that value of coefficient of drag and lift also reduced as shown in the table.

Table No.4

Parameter	Obtained value
Drag Coefficient	0.36
Lift Coefficient	0.20

Acknowledgment

References

[1] Sebastien Chaligne, Thomas Castelain, Marc Michard, D. Juve, "Active control of the flow behind a two dimensional bluff body in ground proximity", Elsevier march 1990.

[2] G. Pujals, S. Depardon, C. Cossu, "Drag reduction of a 3D bluff body using coherent stream wise streaks", Springer Exp. Fluids (2010)4a:1085-1094.

[3] M. Koike, Tsunehisa Nagayoshi, N.Hamamoto, "Research on aerodynamic drag reduction by vortex generation", Mitsubishi Motor's technical papers, 2004

[4] S.A.Isaev, Y.F.Gortyshov, V.M.Gureev, Yu.S.Opara, I.A.Popov,"A method of decreasing the drag of a heavy duty truck with the use of front & stern board generators of large scale vertices", Journal of engineering Physics & Thermophysics, vol.88. No.1, January 2015.

[5] Gopal P. & Senthilkumar T, "Aerodynamic drag reduction in a passenger vehicle using vortex generator with varying yaw angles", ARPN Journal of engineering & Applied Science, Vol.7, No: 9, Sept 2012.

[6] Raju Govindharajan, Dr. K.M.Parammasivam, S.Sathya Narayanan, "Design of vertex generators for light transport vehicle (Ltvs) using CFD", 8th Asia-Pacific conference on wind engineering, Dec2013, Chennai