Effect of Vortex generators on Aerodynamics of a Car: CFD Analysis

Mohan Jagadeesh Kumar M
School of Mechanical and Building Sciences
VIT University, Vellore, Tamil Nadu, India

Anoop Dubey
School of Mechanical and Building Sciences
VIT University, Vellore, Tamil Nadu, India

Shashank Chheniya
School of Mechanical and Building Sciences
VIT University, Vellore, Tamil Nadu, India

Amar Jadhav
School of Mechanical and Building Sciences
VIT University, Vellore, Tamil Nadu, India

Abstract- Flow separation at the vehicle's rear end is the major cause of an aerodynamic drag in a car. In order to delay the flow separation at the rear, bump-shaped vortex generators at the roof end of a car are tested in this paper for two different types of car models Sedan and Hatchback. The aerodynamic analysis is carried out using GAMBIT and FLUENT for Sedan and Hatchback models. Vortex Generators are found to be not very sensitive for the designing parameters. CFD analysis confirms that the use of Vortex Generators reduces both the drag and lift coefficients.

Keywords – Vortex generators, Aerodynamics, Drag, Lift, Flow separation, CFD

I. INTRODUCTION

Reducing the fuel consumption is the primary concern in an automotive development, for energy resources conservation and protecting the global environment. Reduction of drag force is an essential process in vehicle aerodynamics for improving the fuel consumption as well as the vehicle driving performance.

In a passenger car, in addition to minimum required space for its engine and other components, there must be enough space to accommodate passengers as well as luggage. Because of this it becomes extremely difficult to design an aerodynamically ideal body shape for a car. Thus a passenger car has a body shape that is rather aerodynamically bluff for which flow separation occurs at its rear end. The aerodynamic bluffness of a passenger car body when expressed by the drag coefficient, CD is generally between 0.2 and 0.5, for more bluff cubic objects it is greater than 1.0 and for the least bluff bullets it is less than 0.1 [1]. The two elements that have major influence on CD of a bluff object are the roundness of its front corners and the degree of taper at its rear end. Because of the presence of a trunk at the rear, the flow separates at the roof end and then spreads downward. As a result, the flow around the car is similar to that around a streamline-shaped object with a taper at the rear [1].

Flow-separation control remains extremely important for many of the practical applications in fluid mechanics, due to large energy losses often associated with the boundary-layer separation [6]. It has been shown that 40% of the drag force is concentrated at the rear of the geometry [7]. By controlling the flow separation; a significant increase in system performance can be obtained resulting in energy conservation as well as weight and space savings. A correct understanding of the physical phenomenon that connects a car body with the related aerodynamic forces can provide a breakthrough for the better design of a car body [5]. The numerical simulation of the airflow around a car is usually complicated and difficult; therefore, use of CFD technique is an effective tool as it provides the detailed, quantitative data at any point in the flow field.

II. FLOW SEPERATION MECHANISM OF VORTEX GENERATORS

The Air Flow velocity over the car centre line plane near the roof end is shown in Figure 1. As the height of the car at rear end reduces the flow area increases, due to which the air expands and thus its velocity drops and pressure increases. This increased downstream pressure creates the force in opposite direction generating the reverse flow at point ‘C’, acting against the air movement. There is no reversal of air flow at point ‘A’ which is upstream of point ‘C’ since momentum of the boundary layer is prevailing over the pressure gradient (dp/dx) [1]. At point ‘B’ the momentum of the boundary layer and pressure gradient balance each other. The airflow near the lower end and close to the vehicle surface, within boundary layer losses its momentum due to the viscosity.

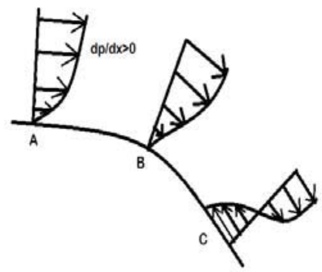


Figure 1. Schematic of velocity profile around rear end

The Vortex Generators placed just before the separation points, supply the loss in momentum by generating stream wise vortices. Thus separation point will be shifted further into the downstream and allows the expanded airflow to persist proportionally longer and hence the velocity of flow at the separation point reduces with an increase in static pressure. This static pressure reduces the control of overall pressure in the entire flow separation region. As a result of increased back pressure, the drag force is reduced. Thus shifting the separation points provides advantage in drag reduction first 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. A combination of these two effects reduces the drag acting on the vehicle. But the Vortex Generators itself produces the drag. So the total effect is calculated by subtracting the drag produced by itself from the reduction in drag caused by shifting of the separation point downstream. Larger the size of Vortex Generators larger is the effect. But the effect will be optimized for a certain size of the Vortex Generator.

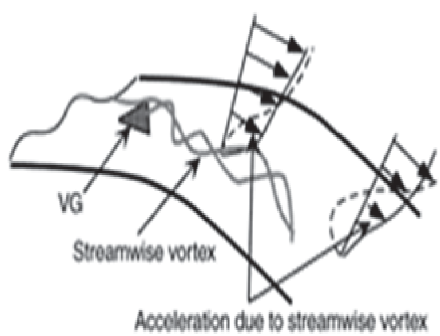


Figure 2. Schematic of flow around vortex generator

III. OPTIMUM VORTEX GENERATORS

The size and the thickness of the boundary layer is measured based on the assumption that the optimum height of the Vortex Generator (VG) would be nearly equal to the boundary layer thickness. The boundary layer thickness at the roof end immediately in front of the flow separation point is about 30 mm [1]. Therefore, the optimum height for the VG is found to be up to approximately 30 mm. The shape of the VG selected for the analysis is a bump-shaped piece with a rear slope angle of 25 to 30°. As to the location of Vortex Generators, a point immediately upstream of the flow separation point exists and a point at an optimum distance of 100 mm in front of the roof end was selected as shown in Figure 3. The front half contour of the bump-shaped VG was smoothly curved to minimize drag and its rear half was cut in a straight line to an approximate angle of 27° for maximum generation of a stream wise vortex. Three bump-shaped Vortex Generators similar in shape but different in height (15, 20 and 25 mm) to those shown in Figure

4 are examined. Experimental analysis shows that the drag coefficient was lower at the height of 20 to 25 mm, so a height in this range was considered to be optimum for the VG [1]. However, a taller VG might cause a decrease in the lift. The rather small change in drag coefficient resulting from change in height can be accounted for as follows. As mentioned before, an increase in height of the VG simultaneously causes two effects: one is reduced drag resulting from delayed flow separation and the other is increased drag by the VG itself. These two effects are balanced when the VG's height is between 20 and 25 mm. From these results, a reduction of CD is 0.003 with this bump-shaped VG when the shape and size are optimized [1].

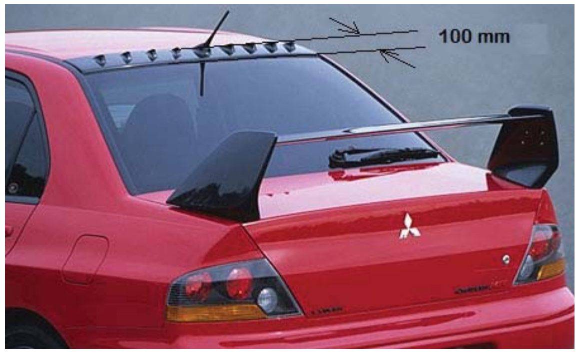


Figure 3. Position of Votex generators at the rear end of the roof

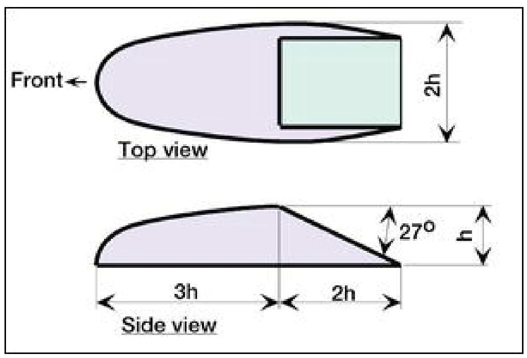


Figure 4. Dimensions of a bump shaped vortex generator

IV. CFD ANALYSIS OF SIDE CONTOUR OF A CAR

Aerodynamic features of a vehicle and the resultant turbulences can be analyzed using the wind tunnel or by Computational Fluid Dynamics (CFD). The efficiency and the financial aspect of CFD make it a better solution. New turbulence models and the increasing computing power make CFD a significant tool to use. Finite volume method (FVM) was used in CFD analysis of airflow around a side car contour. GAMBIT was used as a pre-processor for the modeling and FLUENT was used as a solver and postprocessor. The first step in FVM is to discretize the analysis area. With discretization, finite volumes are forming in way that they are touching each other (no overlap) and fills up the area of analysis. Finite volume set is also called as geometrical mesh. Geometrical mesh can be structured and unstructured. Numerical analysis using FVM is simple as in the case of structured meshes [3]. System matrix of discretized equations in the case of structured meshes is usually diagonal, while in unstructured meshes it is not. FVM shows more accurate results in the case of structured meshes than in the case of unstructured meshes (for the same number of volumes). So for the problem in this paper, taking complexity of geometry into consideration, mesh generated is structured to the greatest extent. FVM is an integral method based on the integration of the conservative form of transport equations by finite volumes which are discretizing the area of analysis [3]. Speed of changing the content of physical properties inside the finite volume is proportional to the speed of flow of those physical properties through the boundaries of the finite volume and speed of emergence or disappearance of those physical properties inside of a finite volume [3]. After the application of numerical scheme in sense of approximation of normal derivative at the faces of the finite volumes using only the nodal values, the embedding of boundary conditions is next step. So, in the system of discretized equations it is necessary to incorporate boundary conditions. Boundary condition defines the flow through the faces of the finite volumes. After the boundary conditions are set, next step is to solve linear algebraic equations system. In the case of the linear problem, solution is reached by solving only one linear algebraic equations system. And in case of nonlinear problem, solving process starts from a presume solution based on which coefficients in discretized equations, and the equations are solved. With the obtained solutions, coefficients in the system matrix are calculated again and the system is solved again. The procedure is repeated until the coefficients in the system matrix and the solution is stopped changing in the number of significant digits which is prescribed in advance [3].

V. DISCRITIZATION USING FINITE VOLUME METHOD (FVM)

Finite Volume Method is a finite differential method put forward by S. V. Patankar [8]. The difference between the FVM and the general finite differential method is that the density of the general flow quantity of a variant is calculated by the interpolation of the basic function, which is the result of the one-dimensional convection and diffusion equation. Then the discrete equation is derived on the basis of conservation principle [5]. Geometry and mesh was generated using GAMBIT. Quadrilateral Mesh was selected for meshing. The dimensions of the physical domain are shown in Figure (5). In Figure (5), the value of L is considered to be 4400 mm for Sedan and 4030 mm for Hatchback and the dimensions are scaled down to 1:22. As mentioned earlier, two models of car geometry Sedan and Hatchback were analysed. For Sedan geometry there is steeped rear end body profile and for Hatchback rear slope angle is 50^{o} to 25^{o}.

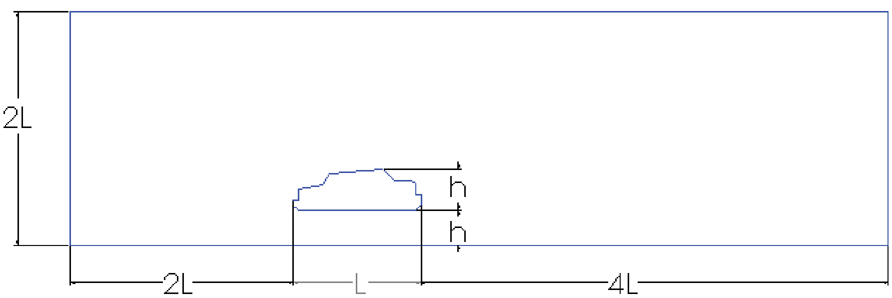


Figure 5. Physical model and its dimensions

VI. BOUNDARY CONDITIONS

The boundary conditions were configured in GAMBIT. Velocity of the air at the inlet is considered to be 27.7 m/s (100 km/h) and with a temperature of 300 K. The boundary condition at the outlet is considered to be pressure outlet with a gauge pressure of 0 Pa. The boundary condition for the car contour and top and bottom of the virtual wind tunnel is considered as wall boundary condition. The density of air is set as 1.225 kg/m^3 and viscosity of air is 1.7894 X 10^{-5} kg/(ms). To get the accurate results within the identical conditions, mesh is discretized with same density in both cases of the geometry [3]. The CFD analysis for the turbulent flow is carried out considering k – ε model.

VII. RESULTS AND DISCUSSION

The meshed geometry is analyzed in FLUENT by applying the specified boundary conditions and the variation of the flow parameters are plotted and studied. The variation of static pressure over the car geometry is shown in Figure 6.

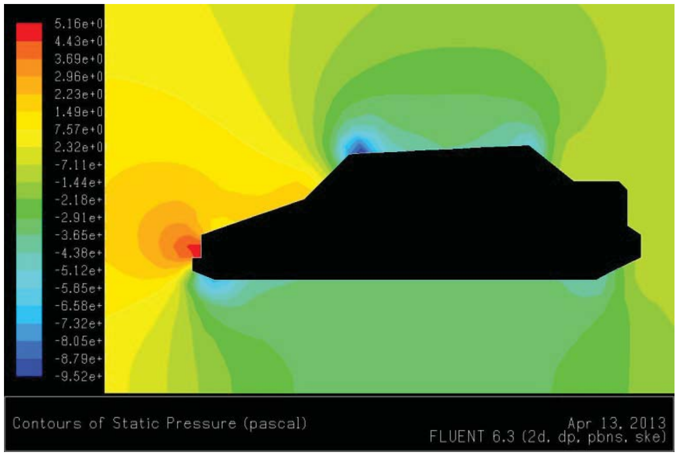
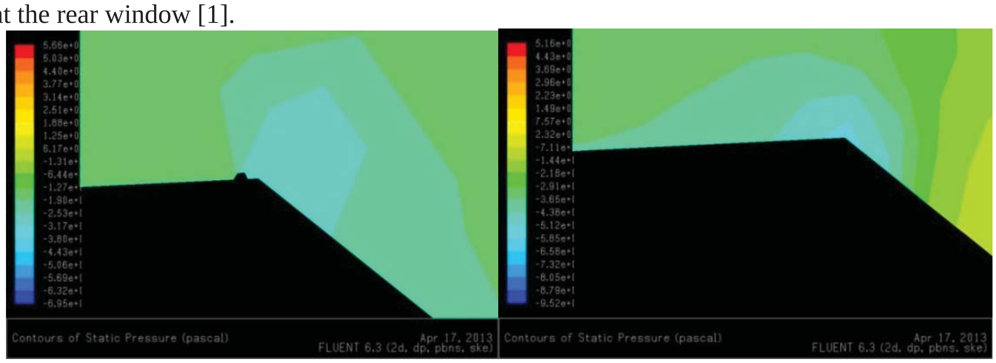


Figure 6. Static pressure contour over a Sedan model

The variation of static pressure contours for the Sedan model over the rear end of the roof is shown in Figure 7 (a) with Vortex Generator and (b) without Vortex Generator. It is found from Figure 7 (a) and (b) that the addition of Vortex Generators gives the effect of increasing the surface pressure over a wide area ranging from the rear window with a reduction in drag. Negative pressure region around the Vortex Generators indicates the Vortex Generators themselves causes the drag. Such changes in airflow can be attributed to Vortex Generators to suppress flow separation at the rear window [1].



(a) (b)

Figure 7. Static pressure over the rear end of the roof of a Sedan model (a) with Vortex Generator (b) without Vortex Generator

Detailed airflow analysis was carried further to verify the suppression of flow separation at the rear window. The distribution of Vorticity behind the VG is presented in Figure 8 for Sedan model. It is found from Figure 8 that the formation of streamwise vortices behind the VG confirms our estimation of shifting the separation point down the stream.

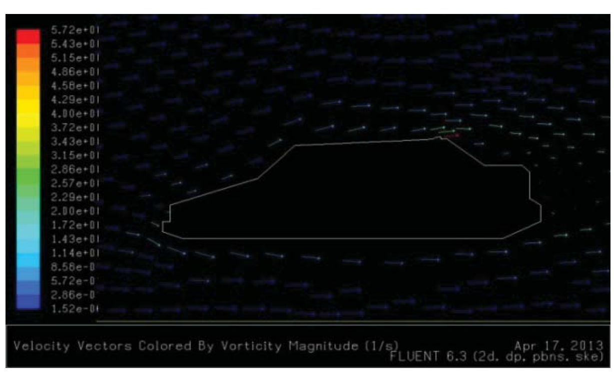
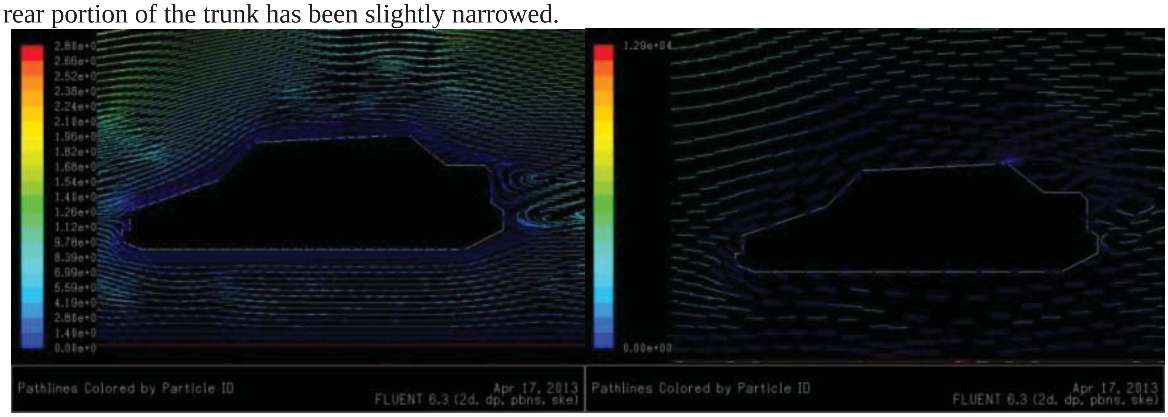
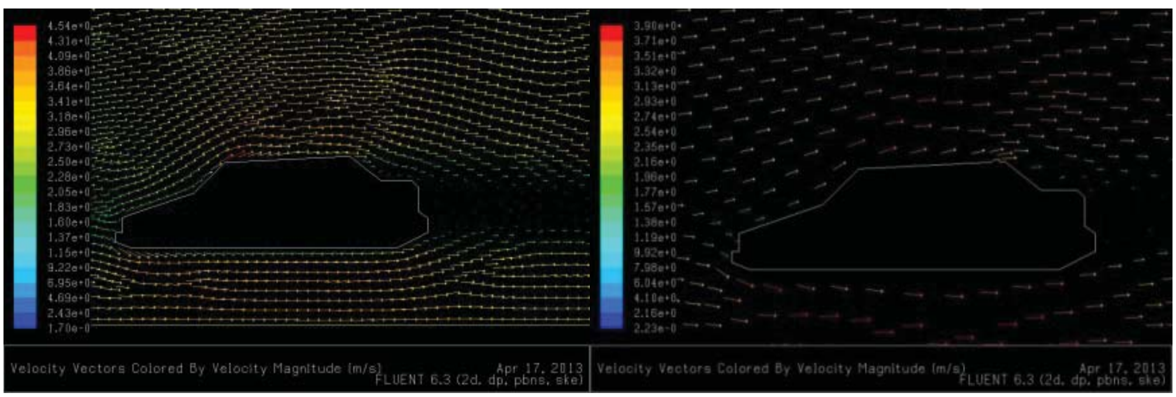


Figure 8. Vorticity distribution behind Vortex Generator over a Sedan model

Flow field near the flow separation point for Sedan model in the form of velocity vectors is shown in Figure 9 (a), (c) without VG and (b), (d) with VG. Similarly velocity field distribution is shown in Figure 10 (a) with VG and (b) without VG. It is found from Figure 9 that the flow separation is taking place further down the stream with a VG than in the Sedan model car without VG. The difference in low velocity region which is narrowed with the use of a VG is clearly observed from Figure 10 and the change in the value of drag coefficient (i.e. $\Delta C_D = -0.009$) and lift coefficient (i.e. $\Delta C_L = -0.042$) are in good agreement with the experimental results [1]. It can also be observed from Figure 10 that the lift is decreased, i.e. down-force increases and flow separation region, i.e. low velocity region at the rear portion of the trunk has been slightly narrowed.

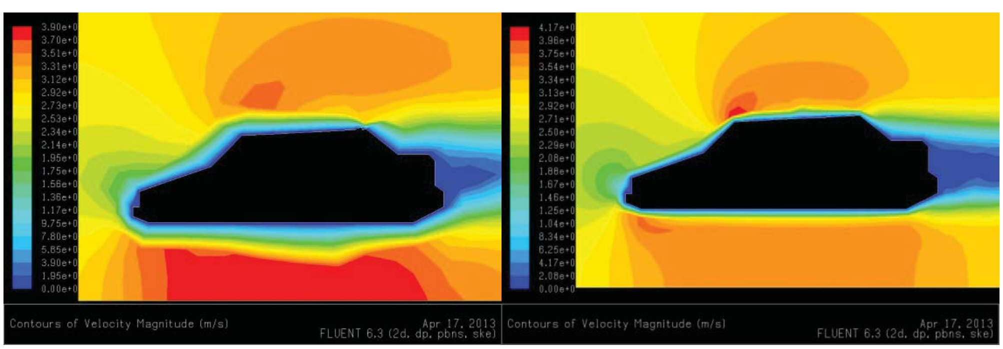


(a) (b)



(c) (d)

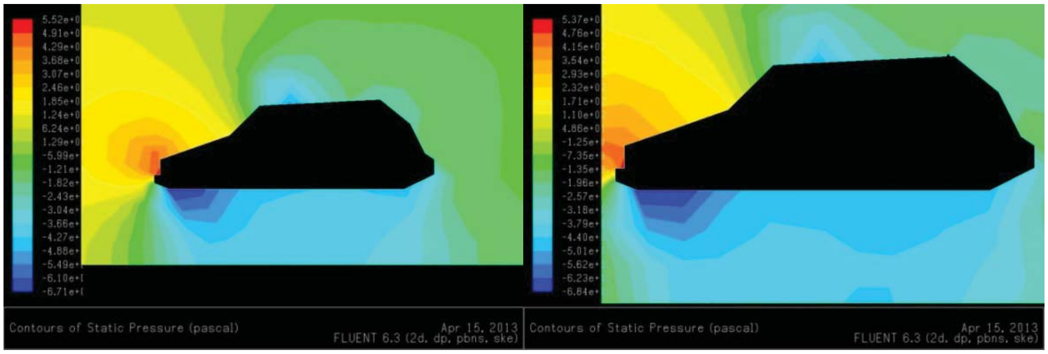
Figure 9. Pathlines and velocity vectors over a Sedan model (a), (c) without Vortex Generator (b), (d) with Vortex Generator



(a) (b)

Figure 10. Velocity distributions over Sedan model (a) with Vortex Generator (b) without Vortex Generator

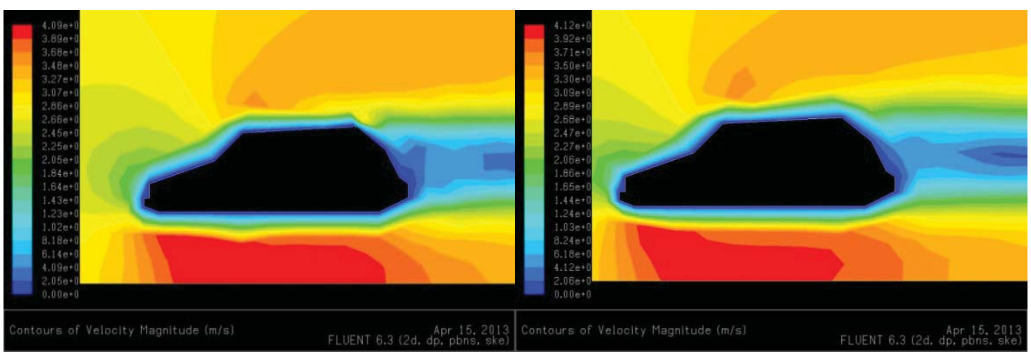
The static pressure distribution over the Hatchback car is presented in Figure 11 (a) without VG and (b) with VG. It is observed from Figure 11 that addition of VG's wide area (ranging from rear window to the back of the car) is increasing the surface pressure thus reducing the drag. Such airflow movement is observed because of the use of VG as it suppresses the flow separation near the rear window.



(a) (b)

Figure 11. Pressure distribution over hatchback (a) without Vortex Generator (b) with Vortex Generator

Similar to the results obtained for Sedan model it is observed from Figure 12 that the low velocity region near the roof is narrowed, which again changes the drag and lift coefficients. It can also be found from Figure 12 that the lift is decreasing, which in turn increases the downforce of the hatchback model car. In the case of Hatchback model the change in the value of drag coefficient (i.e. $\Delta C_D = -0.008$) and lift coefficient (i.e. $\Delta C_L = -0.046$) is observed from Figure 12. Comparing Figure 10 and Figure 12 it is clear that the narrowing of low velocity region is more prominent in case of Hatchback than in Sedan model and hence increase in downforce will be more for the hatchback model.



(a) (b)

Figure 12. Velocity distribution over hatchback (a) with Vortex Generator (b) without Vortex Generator

VIII.CONCLUSION

The following conclusions are made from the numerical analysis on Sedan and Hatchback models.

1.) Vortex Generators are to be installed immediately at the upstream of the flow separation point in order to delay the airflow separation above the rear window of the two models Sedan and Hatchback considered. As a result the aerodynamic characteristics are found to be improved. The optimum height of the Vortex Generators is considered to be equal to the thickness of the boundary layer, i.e. 15 to 25 mm and they should be placed almost 100 mm upstream of the roof. But the Vortex Generators are not very sensitive to these parameters and hence the optimum parameters are considered to be changing in a wide range.
2.) Application of Vortex Generators is reducing both the drag and lift coefficients for both Hatchback and Sedan models. Since the change in lift coefficient is more for Hatchback model more downforce will be produced with the use of Vortex Generators than in the Sedan model.

3.) Pressure and velocity distributions are obtained over the two model geometries with the help of Computational Fluid Dynamics. It is found that the stream wise vortices are formed because of Vortex Generators thus shifting the flow separation point down the stream and also narrowing flow separation region.

Thus, we can conclude that Vortex Generators increases the pressure at the vehicle's entire rear surface and hence decreasing the drag and increasing the downforce. Because of these advantages Vortex Generators are suggested to be used in commercial applications of passenger vehicles like Sedan and Hatchback model cars.

REFERENCES

[1] Masaru Koike, Tsunehisa Nagayoshi, Naoki Hamamoto, "Research on Aerodynamic Drag Reduction by Vortex Generators", Mitsubishi Motors Technical Review, pp. 11-16, No.16, 2004.

[2] K. Sai Sujith, G.Ravindra Reddy, "CFD analysis of sedan car with vortex generators", International Journal of Mechanical Engineering applications Research – IJMEAR, pp. 179-184, Vol 03, Issue 03, July 2012.

[3] Darko Damjanović, et al., "Car design as a new conceptual solution and CFD analysis in purpose of improving aerodynamics", Josip Juraj Strossmayer University of Osijek, Croatia.

[4] Manan Desai, S. A. Channiwala, H.J. Nagarsheth, "Experimental and Computational Aerodynamic Investigations of a Car", WSEAS Transactions on Fluid Mechanics, pp. 359- 368, Issue 4, Volume 3, October 2008.

[5] Gu Zheng-qi, et al., "Numerical Simulation of Airflow Around the Car Body", 2001-01-3086, SAE Technical paper series, International Body Engineering Conference and Exhibition Detroit, Michigan,October 16-18, 2001.

[6] John C. Lin, "Review of research on low-profile vortex generators to control boundary- layer separation", Progress in Aerospace Sciences, pp. 389-420, No. 38, 2002.

[7] Chainani. A, Perera. N, "CFD Investigation of Airflow on a Model Radio Control Race Car", Proceedings of the World Congress on Engineering, Vol 2, July 2-4 2008.

[8] Suhas V. Patankar, "Numerical Heat Transfer and Fluid Flow", Taylor and Francis.