ICMIEE20-011

A Numerical Study on the Drag Reduction of Sedan Car using Vortex Generator

Rahul Dev Paul[*], Abdullah Al- Faruk and Pranto Karua
Department of Mechanical Engineering, Khulna University of Engineering & Technology, Khulna-9203, BANGLADESH

ABSTRACT

Petroleum fuel reserve has been decreasing at a very high rate. Almost all the automobiles rely upon IC engine which is run by petroleum fuels. As the number of automobiles is increasing in this modern world, it is necessary to reduce the fuel consumption. One of the ways to do this is to reduce the drag of the car. Among a numerous process of reducing drag, using Vortex Generators is one. Delta shaped vortex generators are used at the rear trunk of the sedan type car where the flow separates. K-epsilon turbulent model in ANSYS-Fluent 16.2 (student version) software is used to simulate the airflows. The drag and lift coefficients depends upon the velocity, frontal area, number of vortex generators used and force exerted on the body. In this work, number and spacing between successive vortex generators were investigated based on the comparison of drag coefficient values with each other. The contours of static pressure and velocity magnitude were also investigated for each models. Velocity vectors were scrutinized to detect the flow separation regions. Vortex generators were attached just before the beginning of flow separation. When the vortex generators were attached, the pressure coefficients at the rear trunk began to increase, that also confirmed the increment in back pressure. Hence, the increase of back pressure indicates the reduction in drag coefficient. It has been found that a combination of 8 vortex generators is the optimum solution. After that, the effectiveness of the vortex generators has been measured at different velocities. The devices work better at higher velocity than the lower velocity without creating any effect in the car stability.

Keywords: Sedan Car, Vortex Generator, Back Pressure, Drag Coefficient, Fuel Efficiency

1. Introduction

Whenever a body or an object moves through the air, a resultant force acting upon the respective body or object is exerted. The obtained resultant force is split into two forces. These two forces are called drag force and lift force. The drag force acts axially with the free stream velocity whereas the lift force acts perpendicularly with the free stream velocity. So, it can be said that, the drag force is accountable for resisting the vehicle movement to the front. If this drag force can be eliminated or reduced by some margin, the minimum force to move a body through air can also be reduced.

In this era, a huge application of engines is seen in the automobile sector. During the past few years, the amount of vehicles have increased extensively. Most of these vehicles are run by the power of petroleum fuel. But fuel is a scarce matter. The amount is not increasing rather the reserve of fuel in the world is decreasing day by day. So, it is a great concern for us to save fuel by any means. The fuel efficiency of vehicles can be increased by shaping them aerodynamically. Most of the vehicles energy is consumed by the drag force that is generated by the vehicle while running. Aerodynamic drag is of two kinds. One of them is skin friction drag and other is pressure drag. Between these two kinds of drag, pressure drag is more accountable for this fuel wastage. More than 80% of the total drag is generated by this pressure drag and highly dependent on the geometry of vehicle due to boundary layer separation from the rear windshield surface. It has been estimated that the consumption of fuel is around 77% by the ground vehicles in the United States of America. Among the consumers, the largest of them is automobiles (34%). The next is light trucks which consumes 25% and the least is heavy duty trailers and trucks which consumes around 18% of domestic as well as imported fuel [1]. Reducing this drag offers an inexpensive solution to improve fuel efficiency and thus shape optimization for low drag become an essential part of overall vehicle design process [2]. Around 40% of the drag force is generated at the rear of the geometry [3]. Besides, the greenhouse effect in today's era is an alarming factor. This greenhouse effect can be caused by burning of extensive amounts of fuels. This is another reason why reduction of drag is needed in case of road vehicles because, they emit a huge amount of CO_2 which causes greenhouse effect. Aerodynamic drag force is proportional to the square of the speed of the car. So, we can easily understand that, if the speed of car increases, the drag force will generate at a high rate. That will cause more fuel to burn and fuel efficiency will be hampered. That is why aerodynamic shape of the vehicles is much needed. It is the cheapest way to improve the performance of the car because, it needs no change in internal settings. It only needs the outer geometry to be changed [4].

The drag coefficient of a typical sedan type of car varies between 0.2~0.5 depending upon the shape and design of the vehicle [5]. The flow control using passive way is one of the significant ways through which the drag can be reduced. Vortex generator is such a passive device which is used in particular flow separation region for the extension of the flow. As the flow is extended, i.e. gets more power, so a greater pressure rise happens there which eventually results in reduction of drag. The more the drag reduced, the more the fuel saved. The rear trunk is such a place where flow separation occurs even

* Corresponding author. Tel.: +88-01934587268
E-mail addresses: rahuldevpaul31@gmail.com[1], alfaruk@me.kuet.ac.bd[2], karuapranto@gmail.com[3],

in a streamlined type sedan car. Vortex generator was not attached at this section to investigate the reduction of drag coefficient in any experiments. So, Vortex Generators can be placed at that point which would help in increasing back pressure and reducing drag. Among different types of vortex generators, delta shaped ones are most useful.[6] So, delta shaped vortex generators are used here to carry out the investigation.

In this paper, the reduction of drag coefficient through the attachment of vortex generators on the car body where flow separation occurs at different spacing between the vortex generators has been investigated. Then the simulation of the airflow around the car model without vortex generator and with vortex generators under certain boundary conditions have been carried out. Thus the drag coefficient and lift coefficient generated for both models was compared. And finally, the stability of the car has been ensured.

2. Method

Toyota Camry 2015 is used as the base model to be worked on and triangular delta wing shaped vortex generators are selected to be attached on the model. The reason behind choosing delta wing shape is, it is capable of reducing drag coefficient up to 0.006 where the upper surface of the car body is not streamlined [6].

2.1 Working Proceduce

The steps that has been followed is given below:

2.1.1 Geometry formation in Design Modular:

The model is designed using a commercial designing software named Solidworks 2015. For simplification, the wheels are excluded and the model is scaled down to 10:1. After designing the base model, the vortex generators are designed. The height of the vortex generators is chosen to be 3mm in accordance to the 10:1 scale down. The optimum height of these devices varies from 20mm-30mm. [5] The thickness is 0.5mm and the half delta angle is 30°. They are attached symmetrically at the rear trunk.

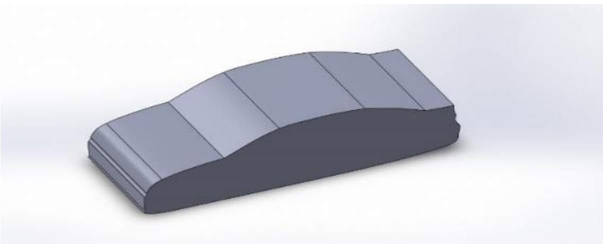

Fig.1 Isometric view of simplified car body

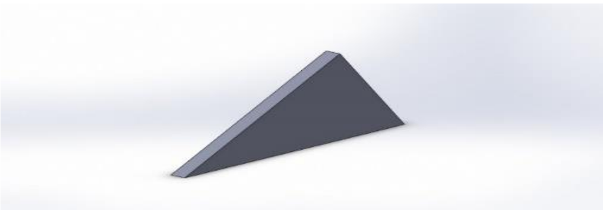

Fig.2 Isometric view of vortex generator

Then an enclosure has been made to carry out the simulation process which acts as an air domain. The car model is then placed into the enclosure.

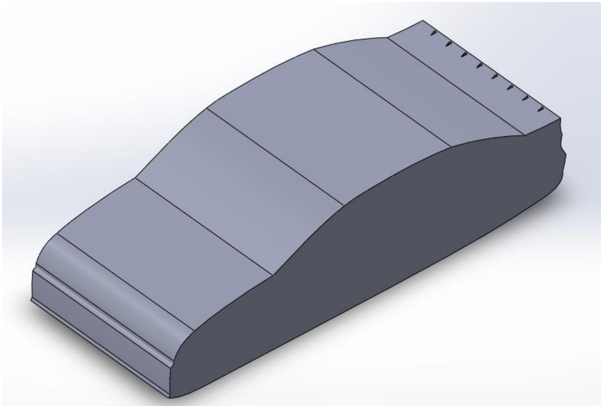

Fig.3 Car body attached with vortex generators

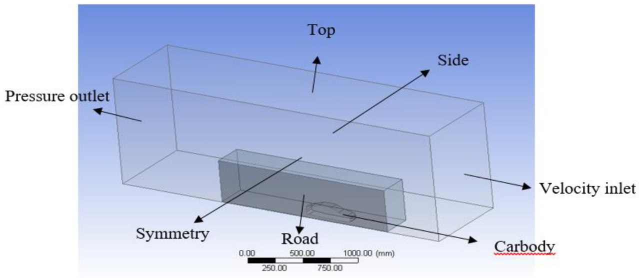

Fig.4 Enclosure formation in Design Modular

2.1.2 Meshing

After the geometry formation is done, the air domain is made to go through the meshing process.

2.1.3 Setup

In this section, the boundary conditions related to the simulation process is inputted. In a steady state condition, Realizable k-epsilon turbulent model with non-equilibrium wall functions is used. Velocity has been varied from 10 ms^{-1} to 50 ms^{-1} at inlet with a turbulent intensity of 1%. At the outlet, the gauge pressure is selected 0 pa with a backflow turbulent intensity of 5%. The density, ρ and the viscosity of working fluid (air) are taken as 1.225 kgm^{-3} and 1.7894e-05 $kgm^{-1}s^{-1}$ respectively. Operating pressure is atmospheric.

2.1.4 Solution

The coupled scheme is selected at standard pressure. Second order upwind functions are used. Turbulent viscosity is taken to be 0.95. Then the initialization has been done in hybrid initialization and thus the calculation process has been started.

2.1.5 Results

After the calculation process, the results have been extracted for further analysis.

2.2 Governing Equations

The continuity and momentum equations with a turbulence model are used to solve the airflow. The equations are:

$$\frac{\partial u}{\partial v}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0$$
(1)

$$u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{1}{\rho}\left(\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{xz}}{\partial x}\right)+B_x \quad (2)$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{1}{\rho}\left(\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{yz}}{\partial x}\right)+B_y \quad (3)$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\frac{1}{\rho}\left(\frac{\partial \tau_{xz}}{\partial y}+\frac{\partial \tau_{yz}}{\partial x}\right)+B_z \quad (4)$$

Where u is x-component of velocity vector, v is y-component of velocity vector and w is z - component of velocity vector. ρ is density of air, p is static pressure, τ is shear stress and B_x, B_y, B_z are the body forces [7].

For turbulent kinetic energy, k:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial t}(\rho k u_i)=\frac{\partial p}{\partial x_j}\left[\left(\mu\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho_\varepsilon-Y_M-S_k$$
(5)

For dissipation, ε:

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial t}(\rho\varepsilon u_i)=$$
$$\frac{\partial p}{\partial x_j}\left[\left(\mu\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1e}\frac{\varepsilon}{k}(G_k+C_{3e}G_b)+C_{2e}\rho\frac{\varepsilon^2}{k}+S_\varepsilon$$
(6)

In these equations, the generation of turbulence kinetic energy due to the mean velocity gradients is represented by G_k and the generation of turbulence kinetic energy due to buoyancy is represented by G_b.

3. Mesh Generation

The generated mesh is found as follows:

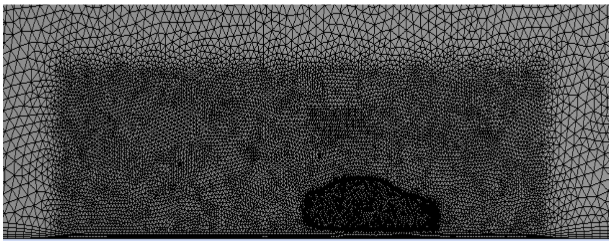

Fig.5 Meshing of the model

The meshing has been done using proximity and curvature sizing functions. The smoothing is kept high. The minimum and maximum size and maximum face sizes are 0.0003 m, 0.06 m and 0.03 m respectively with a growth rate of 1.20. A 5 layer program controlled inflation layer is generated which included the car body and the road. At the car body, a face sizing is done using element size of 0.006 m. For a finer result, a body sizing has been done around the car body at a body of influence using element size of 0.01 m.

3.1 Mesh dependency test

A mesh dependency test is done on the model without any vortex generators with respect to drag coefficient to find out the optimum number of meshing elements.

Table 1 Mesh dependency test for the model without any vortex generators.

No. of Elements (In millions)	Coefficient of Drag (C_{d})
0.52	0.32251
0.98	0.3259
1.4	0.32076
1.6	.3204
2.0	0.319277
2.4	0.31772
2.9	0.316588
3.3	0.31618
3.7	0.315816

The values from Table 1 can be plotted as follows,

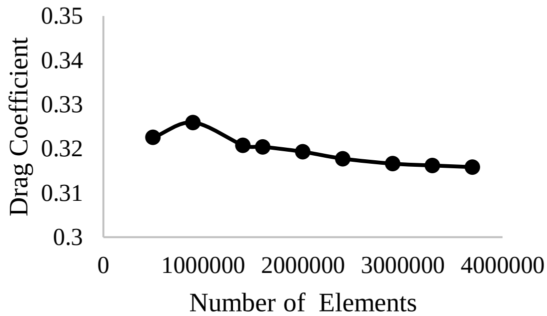

Fig.6 Drag coefficient vs number of elements curve

From Figure 6, it can be concluded that, the value of C_d decreases with respect to the number of elements. After it passes the 2.9 million mark, the value becomes nearly constant. So, it can be said that, the model becomes mesh independent above 2.9 million elements. The further analysis of this study is based on this meshing property.

4. Finding optimum number of vortex generators

The model has been simulated attaching 0 to 9 vortex generators gradually to find out the optimum number. The findings are given as follows.

Table 2 Details of Drag force, Lift force, C_d and C_l with respect to number of vortex generators at 50 ms^{-1}.

Number of vortex Generators	Drag Force (N)	Lift Force (N)	Co-efficient of Drag (C_d)	Co-efficient of Lift (C_l)
0	5.78753	-7.3991	0.31658	-0.40474
1	5.77990	-7.4266	0.31617	-0.40625
2	5.77121	-7.4726	0.31569	-0.40876
3	5.75824	-7.5187	0.31498	-0.41128
4	5.75779	-7.5141	0.31469	-0.41103
5	5.74002	-7.4982	0.31398	-0.41016
6	5.72481	-7.5149	0.31315	-0.41107
7	5.72162	-7.5191	0.31298	-0.41130
8	5.72073	-7.5195	0.31293	-0.41133
9	5.71557	-7.5338	0.31265	-0.41211

The values from Table 2 can be plotted as follows,

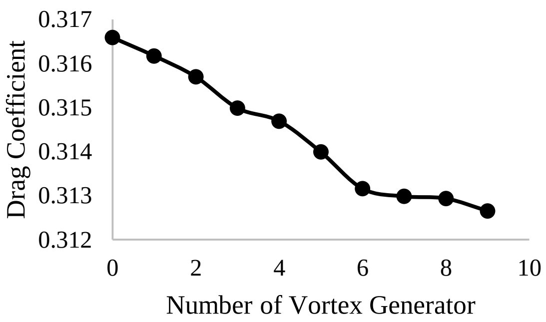

Fig.7 Plot of C_d vs number of vortex generators for inlet velocity of 50 ms^{-1}

The Figure 7 clearly indicates that, C_d decreases with the increase of vortex generators. After attaching 6 to 8 vortex generators, the value of C_d becomes more or less constant. So, the optimum number of vortex generators is taken to be 8 and further analysis associated with vortex generators are done on the basis of 8 vortex generators which are 19.72 mm apart from each other.

5. Post Processing

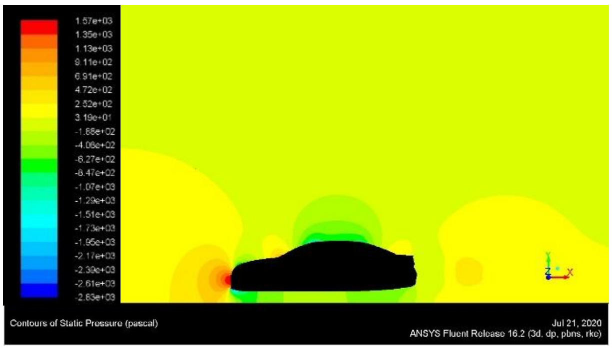

Fig.8 Static Pressure contour of model without any vortex generator at 50 ms^{-1}

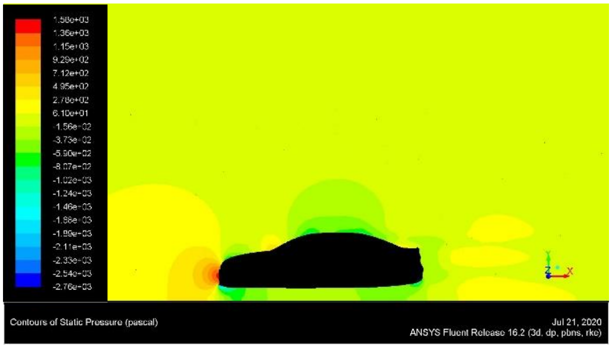

Fig.9 Static Pressure contour of the model with 8 vortex generators at 50 ms^{-1}

From Figure 8 and Figure 9, it can be concluded that, the pressure distribution between the both models is similar at the front end. But at the rear end the static pressure is high in case of the model attached with vortex generators. This phenomenon is termed as back pressure rise.

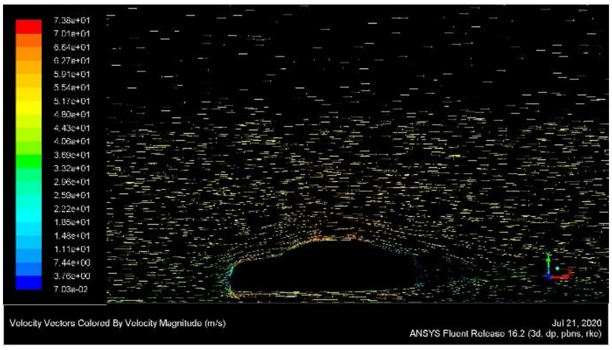

Fig.10 Velocity vector of the model without vortex generator at 50 ms^{-1}

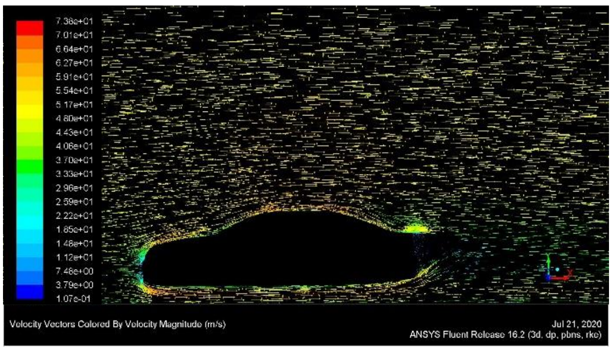

Fig.11 Velocity vector of the model with 8 vortex generators at 50 ms^{-1}

From Figure 10 and Figure 11, it is clearly seen that, the velocity at the rear end is suddenly increased which helps to reduce drag force as well as coefficient of drag.

6. Result and Discussion

Then the values of C_d and C_l has been calculated for the car models with and without Vortex Generators. Starting from 1 to 9, Vortex Generators has been attached at the rear trunk of the car. Taking C_d as the standard of comparison, the optimum number of Vortex Generators are found to be 8. Each of the Vortex Generators are 19.72 mm apart. The values of C_d and C_l have been plotted for the optimum number of Vortex Generators at different velocities varying from 10 ms^{-1} to 50 ms^{-1}. The minimum value of C_d is found to be 0.31293 for the model attached with vortex generators at 50 ms^{-1}. It seems that, the greater the values of velocity, the greater the reduction of C_d. The highest drag reduction has been found as 1.15% which is great for any aerodynamically shaped cars. Also, the reduction of C_l is found to be 1.60% at this velocity when using the vortex generators. It confirms the stability of the car on road as well.

The plots drawn from the whole process are discussed below:

Table 3 Values of C_d and C_l without vortex generators and with vortex generator.

Velocity, ms^{-1}	Without Vortex Generator		With Vortex Generator	
	C_d	C_l	C_d	C_l
10	0.34104	-0.42029	0.33862	-0.42057
20	0.32478	-0.40082	0.32275	-0.40866
30	0.31920	-0.40026	0.31670	-0.40753
40	0.31777	-0.39939	0.31503	-0.40662
50	0.31658	-0.40474	0.31293	-0.41133

The values from Table 3 can be plotted as follows,

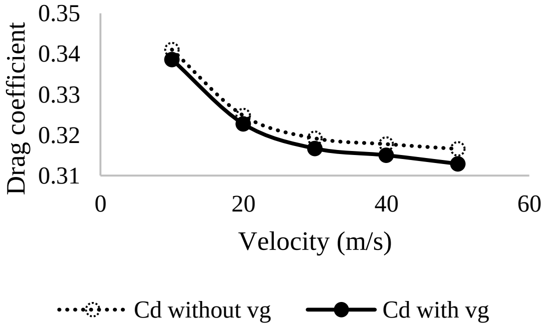

Fig.12 Comparison of C_d for both models with respect to velocity (ms^{-1})

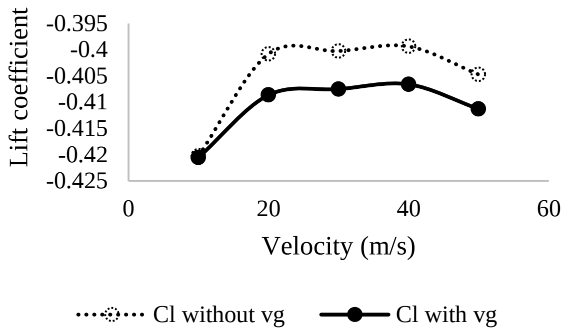

Fig.13 Comparison of C_l for both models with respect to velocity (ms^{-1})

From Figure 12 and Figure 13, it is seen that, the values of C_d and C_l decrease with the increase in velocity.

Table 4 Values of reduction percentage of C_d and C_l with respect to velocity.

Velocity, ms^{-1}	Reduction in C_d, %	Reduction in C_l, %
10	0.71	0.07
20	0.62	1.92
30	0.78	1.78
40	0.86	1.78
50	1.15	1.60

The values from Table 4 can be plotted as follows,

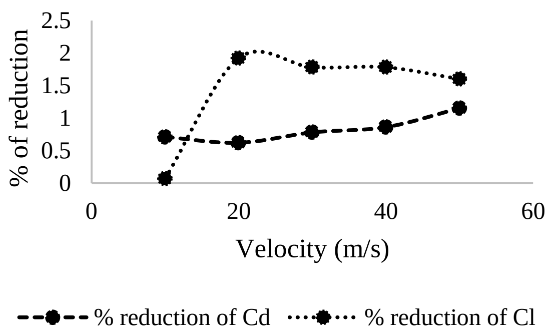

Fig.14 Reduction percentage of C_d and C_l with respect to velocity (ms^{-1})

From Figure 14, it can be said that, C_d depends upon the velocity rise. As the velocity rises, the reduction of C_d increases which helps to reduce the fuel consumption. Apart from that, the reduction of C_l is also noticed. That indicates the stability of the car is not compromised rather it is enhanced.

7. Conclusion

The following conclusion is derived from this study.

1. Depending upon the velocity, the variation of reduction percentage of C_d is noticed. As the Vortex Generators can reduce C_d more at high velocity, the use of them should be in the high velocity cars.

2. It can also be used in cars with a pop up mechanism. While the car moves in the highway, the velocity becomes high. Using the pop up mechanism would actuate the Vortex Generators to pop up after achieving a minimum velocity and be helpful in reducing fuel consumption by some margin. But in a busy city like Dhaka where traffic jam is a common affair, it would not bring much change in fuel consumption using Vortex Generators on cars.
3. While reducing drag, the lift is reduced simultaneously. So, the stability isn't hampered using Vortex Generators.

8. References

[1] M. Sudin, M. Abdullah, S. Shamsuddin, F. Ramli and M. Tahir, "Review of Research on Vehicle Aerodynamic Drag Reduction Methods," *International Journal of Mechanical & Mechatronics Engineering IJMME-IJENS,* vol. 14, no. 2, pp. 35-47, 2014.

[2] W. Meyer and G. Wickern, "The new audi A6/A7 family-aerodynamic development of different body types on one platform," *SAE International Journal of passanger cars- Mechanical Systems,* vol. 4, no. 1, pp. 197-206, 2011.

[3] Chailani, A., Perera, N., "CFD Investigation of airflow on a model radio control race car," in *WCE*, London, 2008.

[4] Cho, J., Kim, T., Kim, K., Yee, K., "Comparative investigation on the aerodynamic effects of combined use of underbody drag reduction devices applied to real sedan," *International Journal of Automotive Technology,* vol. 18, no. 6, pp. 959-971, 2017.

[5] Kumar, M.M.J; Dubey, A; Chheniya, S; Jadhav, A, "Effect of Vortex generators on Aerodynamics of a Car: CFD Analysis," *International Journal of Innovations in Engineering and Technology (IJIET),* vol. 2, no. 1, pp. 137-144, April 2013.

[6] M. Koike, T. Nagayoshi and N. Hamamoto, "Research on Aerodynamic Drag Reduction by Vortex Generators," *Mitshubishi Motors Technical Review,* no. 16, pp. 11-16, 2004.

[7] ANSYS Fluent, Release 16.2: Help Topics.

NOMENCLATURE

C_d : Coefficient of drag
C_l : Coefficient of lift
F_{d} : Drag force, N
F_{l} : Lift force, N
V : Velocity, ms^{-1}
ρ : Density of air, Kgm^{-3}