International Journal of Science and Research (IJSR)
ISSN: 2319-7064
Impact Factor (2018): 7.426

Drag Reduction: Design & Analysis of Vortex Generator

Ashish Domal[1], VinayAhuja[2], Pankaj Ande[3]

[1]BTech Degree in Mechanical Engineering, NMIMS University, Mumbai, Maharashtra, India
ashishdomal.nmims[at]gmail.com
[2]BTech Degree in Mechanical Engineering, NMIMS University, Mumbai, Maharashtra, India
vinayahuja.nmims[at]gmail.com
[3]MS Degree in Automotive Engineering, Staffordshire University, England
pankajande0910[at]gmail.com

Abstract: ***In initial flow simulation, Audi a4 car model is imported in CHAM PHOENICS-CFD software and tested under different speeds, then airflow, pressure and velocity value are analysed. After that in modified flow simulation, vortex generators are carefully designed and installed at the rear end of the vehicle and same cycle is repeated. After getting results from both the flow simulations, differences in aerodynamics characteristics are studied. Drag forces are calculated in both initial and modified flow simulations, to understand the effect of vortex generator i.e. to check whether drag of the vehicle is reduced or not after installing vortex generator, as ultimate aim of the vortex generator is to reduce drag properties at the rear end of the car by reducing flow separation and turbulence of air.***

Keyword: CHAM Phoenics- CFD, Vortex generator, Drag force, Flow separations, PTC Creo-CAD

1. Introduction

Aerodynamics is a branch of fluid mechanics, dealing with fluids like gases and liquids under motion. It includes mainly, study of fluid flow: observation of energy and volume conservation in a fluid flow and fluid resistance [1]. In vehicle or Automotive aerodynamics, air is considered as fluid. It comes under aerodynamics, includes examination of aerodynamics features of road vehicles like car and its main intend is to reduce drag and noise of wind. The main objectives of improvement in flow past vehicle bodies are to improve fuel economy, comfort and driving characteristics.

1.1 Drag Force:

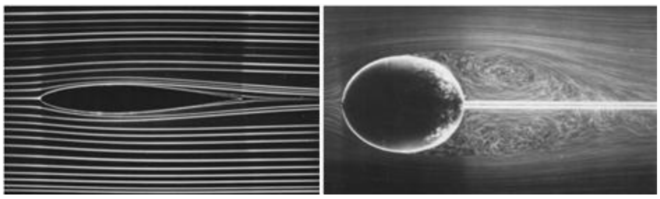


Figure 1: Streamlined and bluff bodies [2]

When air interacts with the object, it creates mechanical force on the surface of the object, this mechanical force is known as drag. In stream line bodies flow is attached flow while bluff bodies create flow separation and bubbles. Shear stresses contribute major part to cause drag in streamlined bodies while drag caused due to pressure forces in case of bluff bodies. Cd coefficient of drag is smaller than Cd of bluff bodies because pressure forces contribute small share in the drag force [2].

1.2 Vortex Generator:

While designing a car, one should consider many factors in mind like fuel efficiency, aerodynamic properties, aesthetic considerations etc. In recent year designers are trying to reduce the drag of the vehicle by using different techniques. Reason behind the reduction in drag is to increase the fuel efficiency and to avoid the effect of dirty rear screens. Vortex generator is one of the techniques used to reduce drag of the vehicle by controlling flow separation at the rear roof end of a car, as VGs are placed 100mm in front of the rear roof end of a car as shown in fig 2. The purpose of using VGs in car is similar to the aircraft VGs, therefore while designing shape and size of vortex generator, aircraft VGs data is referred [3].

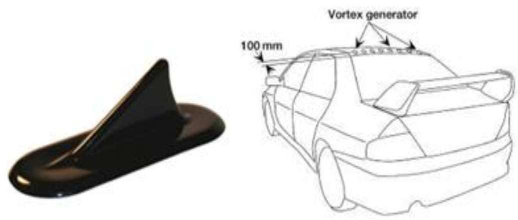


Figure 2: Vortex generator and location of vortex generators [3]

2. Literature Review

A vortex generator is one of the techniques used to reduce the drag of a vehicle. A vortex generator is an aerodynamic device, which consist of small vane and it is attached to the part of the vehicle like car. Aerodynamic characteristics of the vehicle can be improved by carefully installing vortex generators at the flow separation point. Moreover, VGs reduce drag properties at the rear roof end of the car by reducing the separation of airflow and turbulence of air. Now days designer trying their best to decrease the drag force of a vehicle because it will results in reduction of fuel consumption. There are different types of drags acts on the vehicle in motion namely, induced, skin friction, pressure, wave etc.

2.1 Advantages behind the reducing drag of a vehicle are

1)To increase fuel efficiency

2) To avoid the effect of dirty rear screens
3) To protect the global environment

2.2 Design Consideration

Following design considerations designer should keep in mind, which will be helpful for the reduction in drag of vehicle;

1) Aerodynamic drag reduction achieved by carefully designing of streamlines flow
2) Ideal drag coefficient of vehicle is considered as 0.16.
3) Total drag mainly depends on the rear part of the vehicle
4) For better results designer try to make screen movable and smooth

2.3 Selection of Vortex Generator

Vortex generators are used to prevent flow separation; they create drag by themselves and also reduce drag of the vehicle by disturbing downstream flow separation. The net effect of vortex generators is calculated by summation of positive and negative effects. Two types of vortex generators commonly used for the application of car model vehicle in order to prevent flow separation and therefore to reduce drag of the vehicle [4].

1) Delta wing shaped vortex generator
2) Bump shaped vortex generator

Delta-wing-shaped vortex generators are having frontal projection area very small, which results they create smaller drag by themselves. Furthermore, at the edge of delta-wing-shaped vortex generators, vortex generation takes place in such way that it will keep its strength in the downstream flow edge. While in case of bump shaped vortex generators, vortex generation takes place at the point closer to the downstream flow edge which results in vortex interfere with bump and loses its energy. Therefore, delta wing shaped vortex generators are used in following experiment as effectiveness i.e. net effect of bump shaped vortex generator is less compared to delta wing shaped vortex generator [4].

2.4 Shape & Size of vortex generator

The main objectives while designing the shape and size of the vortex generator are, it should create stream wise vortex and have least drag by itself. Based on the assumptions, boundary layer thickness is approximately equal to the height of the vortex generator. The boundary layer thickness at the separation point is about 5mm; as a result optimum height of the vortex generator will be taken 5mm [4].

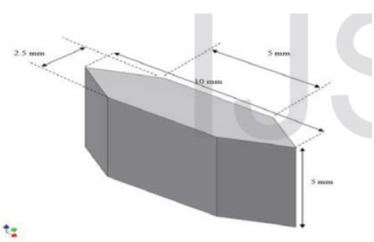


Figure 3: Vortex generator with dimensions in mm [4]

2.5 Computation Fluid Dynamics (CFD)

CFD is a technique under fluid dynamics used to obtain real flows simulation by the numerical solution, obtained by solving governing equations i.e. Navier- Stokes equations. For some practical applications, solving of flow governing equations became very complicated and sometimes impossible using analytical method. CFD techniques replace the analytical method i.e. governing partial differential equations with algebraic equations, which is very much easy to solve with the help of computers and also testing of conditions is very much easier with the help of computational techniques while it is extremely difficult with the experimental and analytical method [5]. Thus, CFD is best and cheaper technique for testing fluid flows compared to experimental and analytical method.

2.6 CHAM Phoenics

Phoenics is one of the most used CFD software developed by the British company CHAM. CHAM is abbreviated as Concentration Heat and Momentum and major product of CHAM is Phoenics CFD code, which is used for the simulation of airflow over the vehicle and to see visually results obtained after solving the simulation. Major advantage of this software, CAD files can be loaded to the geometries of object. In CHAM Phoenics software it is very easy to import the vehicle into the Phoenics-VR and to do flow simulation and get desired results.

3. Research Methodology

3.1 Initial simulation without vortex generator:

In initial flow simulation, Audi a4 car model without vortex generators is simulated in CHAM PHOENICS-CFD software. From fig 4, dimensions of the car are converted in meters ‘m’ are, Length 4.726m, Width 1.842m&Height 1.427m.

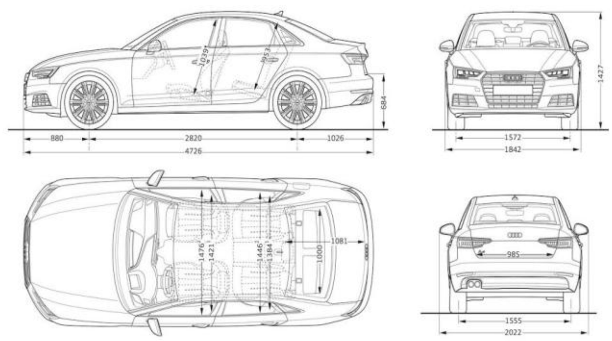


Figure 4: Front view, side view and top view of the Audi A4 car with all dimensions in mm

3.1.1 Pre-processor

1) Domain: Test Geometric dimensions of the domain should be more than that of the model car. So, dimensions of the domain in ‘m’ is taken as follows

Length X=19m (greater than the length of the car)
Width Y=4.7m (greater than width of the car)

Height Z=3.44m (greater than the height of the car)

Figure 5: Domain

2) Inlet Object: Size of the inlet object- X-size=0, Y-size=4.7m & Z-size=3.44m

Placement i.e. position of the inlet object is defined as,
X-position=Y-position=Z-position=0
Simulation done with three different velocities, 30mph, 50mph and 70mph
Inlet velocity i.e. velocity in X-direction= 30mph
Converting mph to m/s= 30(4/9) = 13.33m/s
Similarly for 50mph and 70mph, inlet velocity is given as 22.352m/s and 31.2928m/s respectively

Figure 6: Inlet Object

Similarly for outlet and ground object,

3) Outlet Object: Size of the inlet object- X-size=0, Y-size=4.7m & Z-size=3.44m

Placement i.e. position of the inlet object is defined as,
X-position=19m & Y-position=Z-position=0

Figure 7: Outlet Object

4) Ground Object (Solid Floor): Size of the inlet object- X-size=19m, Y-size=4.7m & Z-size=0

Placement i.e. position of the inlet object is defined as,
X-position=Y-position=Z-position=0

Figure 8: Ground Object

5) Importing Audi A4 Car Model: stl files can be imported to the CHAM software, in initial simulations stl file of Audi A4 car model is imported and simulated in PHOENICS without vortex generator.

Size of the object (Audi A4 car model)- X-size= 4.726m, Y-size= 1.842m & Z-size= 1.427m
Placement i.e. position of the object is defined as, X-position= 6.33m, Y-position= 1.38 & Z-position= 0

Figure 9: Audi A4 car model

6) Meshing: Meshing is set using concept of geometric progression. Area of interest i.e. around the surface of car model mesh is made very fine to obtain detailed result in this interested area.

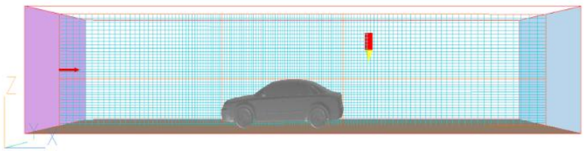


Figure 10: Meshing

7) Probe Position: To achieve better and accurate results, probe is placed at following position;

X- position = 12.05960m
Y- position = 2.374680m
Z- position = 1.956309m

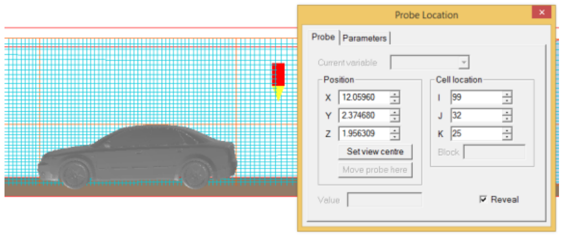


Figure 11: Position of probe

3.1.2 Solver

After pre-processing next step is solving the model using solver. Model problem is solved using KENCHEN model. Then 500 number of iterations are given. Following graphs of different velocities are obtained after solver solving the Audi A4 car model.

1) Velocity at 30mph

Coefficient of drag= CDx= 0.4436179
Drag force= Fx= 123.1785N

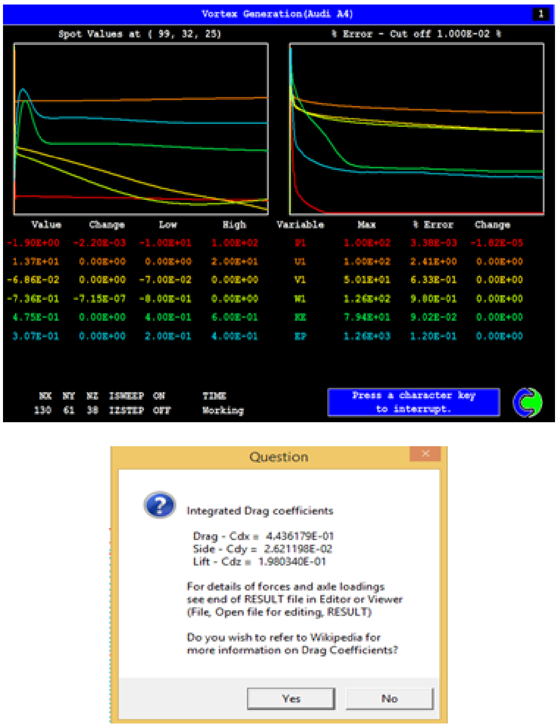


Figure 12: Spot values, % error - cut off and drag coefficients at 30mph velocity

2) Velocity at 50mph

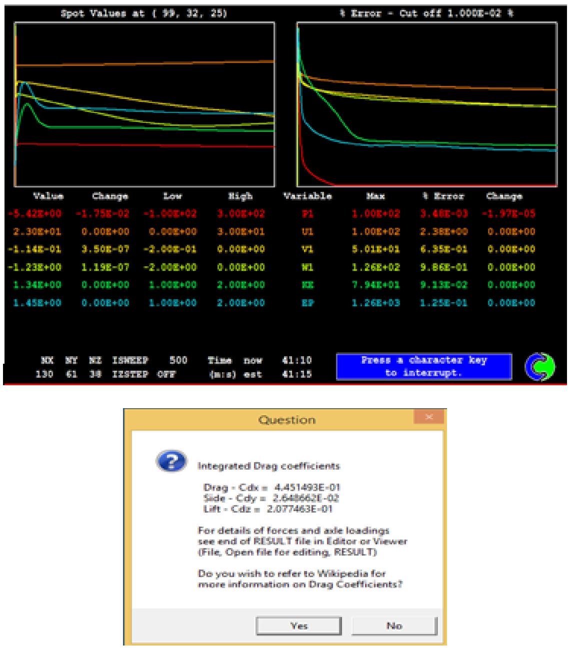


Figure 13: Spot values, % error - cut off and drag coefficients at 50mph velocity

Coefficient of drag= CDx= 0.4451493
Drag force= Fx= 347.5392N

3) Velocity at 70mph

Coefficient of drag= CDx= 0.4460641
Drag force= Fx= 682.5768N

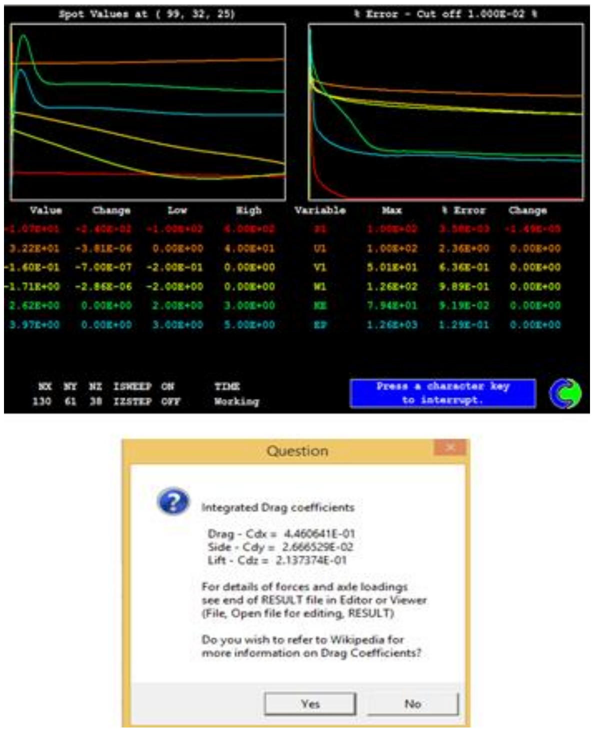


Figure 14: Spot values, % error - cut off and drag coefficients at 70mph velocity

After carefully observing all the graphs of figure 12, 13 and 14 it is clear that, first all errors converged and then all error lines settled and formed a steady and straight line. That indicates results obtained are accurate.

3.1.3 Post Processor

To understand the effect of increasing velocity on car model, flow simulation is carried out with three different velocities and differences in airflow, pressure, velocity and drag is studied. The results obtained from simulations are then viewed and values of pressure and velocity are studied.

At three different speeds 30, 50 and 70mph, the velocity and pressure distribution of the car model are shown in the following figures.

1) Velocity at 30mph

Figure 15: Velocity distribution at 30mph

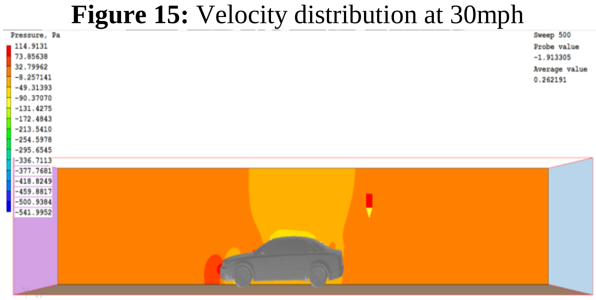


Figure 16: Pressure distribution at 30mph

In figure 16, at the rear end of car model, we can see boundary layer separation. This boundary layer separation creates drag at the top of rear end of car model by creating low pressure region in this area. For this particular simulation coefficient of drag value obtained is 0.4451493, which is high. This effect of boundary layer separation is undesirable and this effect can be reduced by inserting vortex generator at this area. Therefore, it is necessary to install vortex generator at the rear end of the car and now simulation is done again with vortex generator to check whether drag reduced or not.

2) Velocity at 50mph

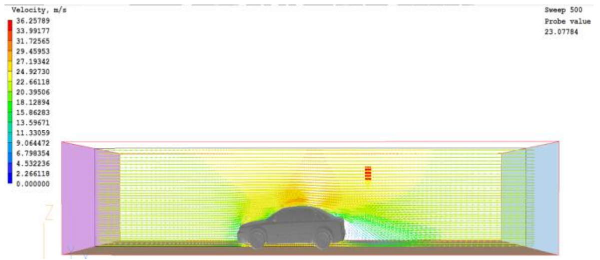


Figure 17: Velocity distribution at 50mph

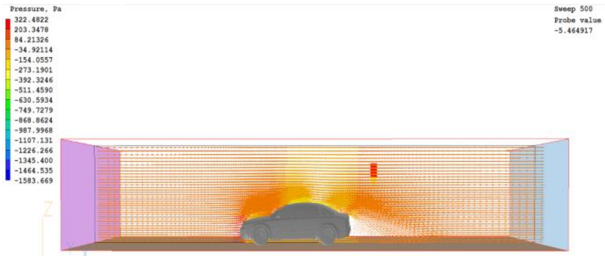


Figure 18: Pressure distribution at 50mph

3) Velocity at 70mph

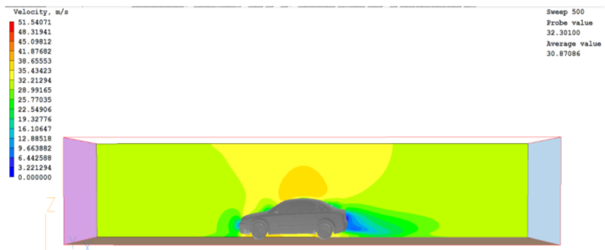


Figure 19: Velocity distribution at 70mph

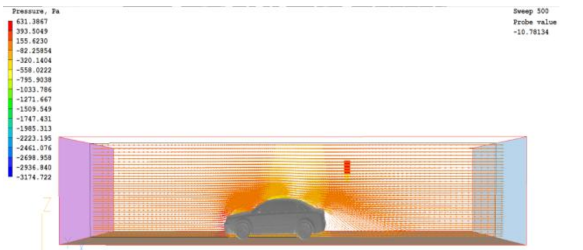


Figure 20: Pressure distribution at 70mph

For initial 30mph velocity simulation drag coefficient value is 0.4436179 while for final 70mph velocity simulation coefficient of drag value obtained is 0.4460641. From obtained results, it is clear that drag forces increases with increase in velocity of the car. So by importing vortex generator to the model, further modified simulation is done to check value of drag reduced or not.

3.2 Modified Simulations

In modified flow simulation vortex generators are carefully designed and eight vortex generators installed at the rear end of the vehicle perpendicular to the airflow and same cycle is repeated.

3.2.1 Design of vortex generator

Using PTC Creo software, vortex generator is designed.

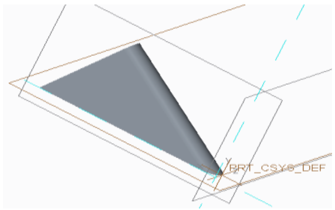


Figure 21: Vortex generator model

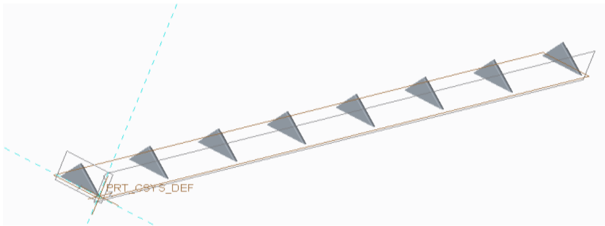


Figure 22: Eight Vortex generators

3.2.2 Pre Processor

1) Importing and placing vortex generators: Similar procedure is followed to import stl file of vortex generator (designed in PTC creo) and then this vortex generators are placed 100mm in front of the rear roof end of a car.

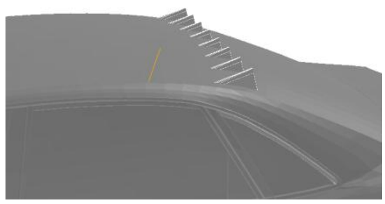


Figure 23: Vortex generators, placed at the rear end of the car model

2) Remeshing: As vortex generator is placed in the simulation, so we need to remesh the domain, now around the vortex generator meshing is done finer to get detailed effect of vortex generator. For this concept of geometric progression is used.

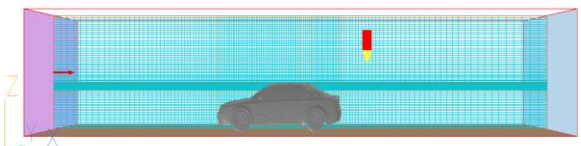


Figure 24: Remeshing

3.2.3 Solver

Similar to the initial simulation, to solve the problem of model KENCHEN model is used and this time number of iterations is set to 350.

1) Velocity at 30mph

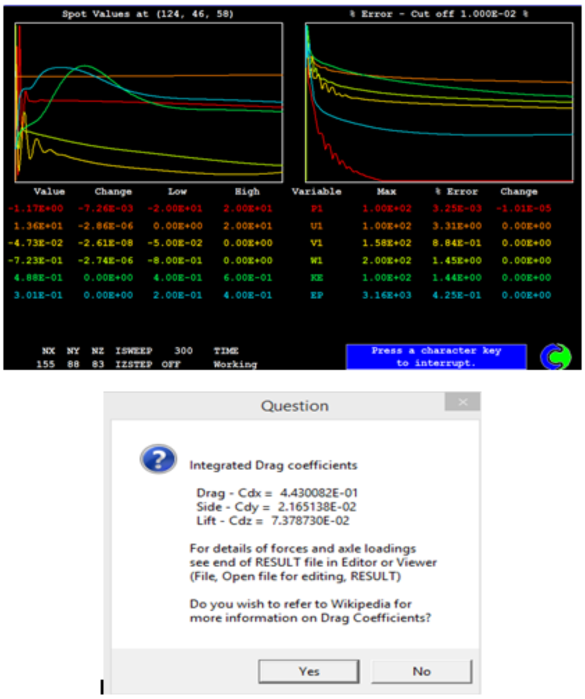


Figure 25: Spot values, % error - cut off and drag coefficients at 30mph velocity with VG

Coefficient of drag= CDx= 0.4430082
Drag force= Fx= 123.4036 N

2) Velocity at 50mph

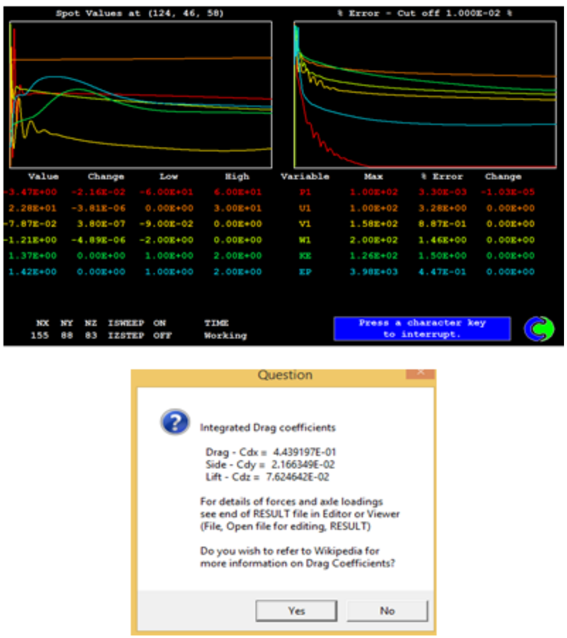


Figure 26: Spot values, % error - cut off and drag coefficients at 50mph velocity with VG

Coefficient of drag= CDx= 0.4439197
Drag force= Fx= 347.6909 N

3) Velocity at 70mph

Coefficient of drag= CDx= 0.4444087
Drag force= Fx= 682.2245N

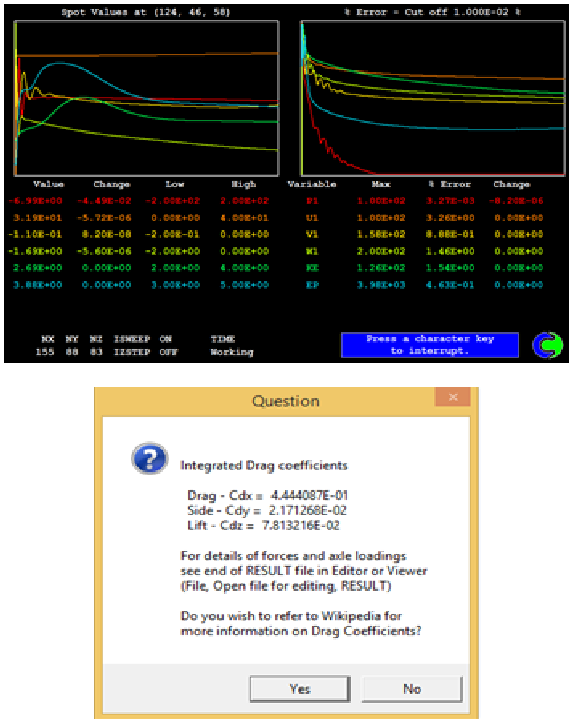


Figure 27: Spot values, % error - cut off and drag coefficients at 70mph velocity with VG

3.2.4 Post Processor

1) Velocity at 30mph

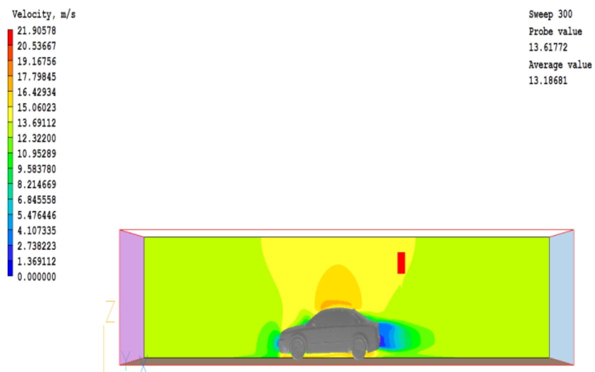


Figure 28: Velocity distribution at 30mph

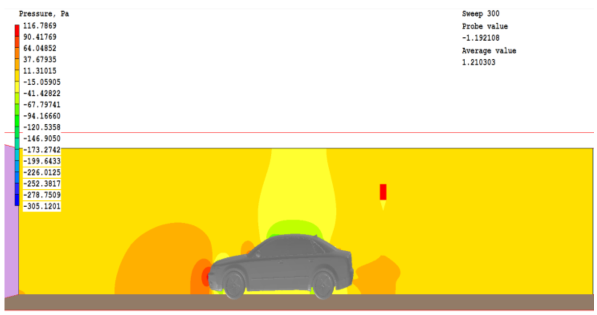


Figure 29: Pressure distribution at 30mph

After looking at results figure 28 & 29, it can be concluded that boundary layer separation is reduced. As we got expected results after installing vortex generator to the model for initial simulation with 30mph. Therefore, further simulation is carried with higher velocities.

2) Velocity at 50mph

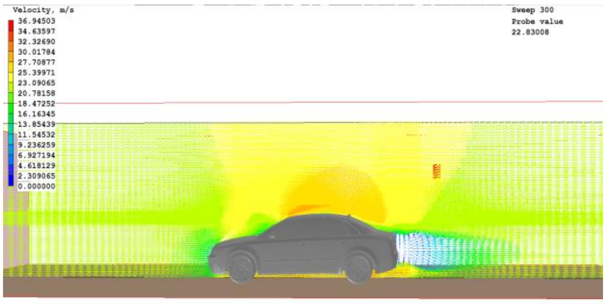


Figure 30: Velocity distribution at 50mph

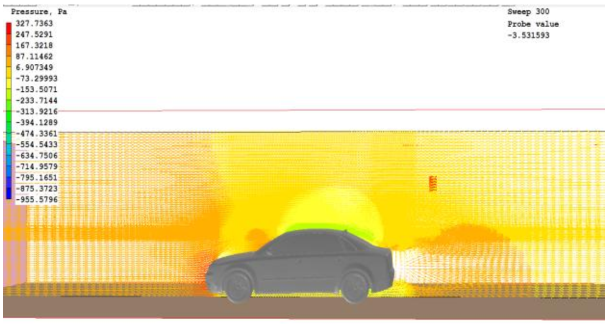


Figure 31: Pressure distribution at 50mph

3) Velocity at 70mph

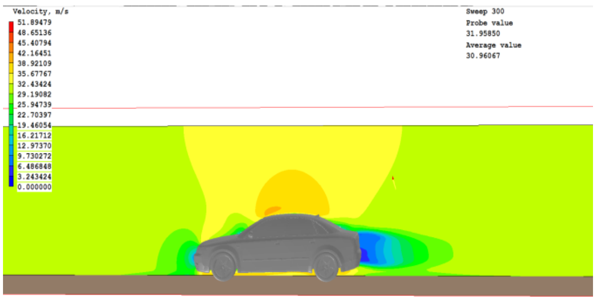


Figure 32: Velocity distribution at 70mph

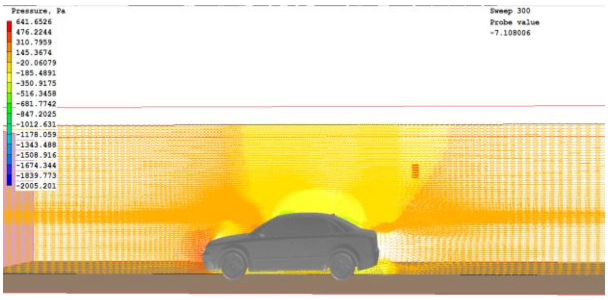


Figure 33: Pressure distribution at 70mph

Finally, for 70mph velocity (initial simulation) without vortex generator, drag coefficient value was 0.4460640 and after importing vortex generator drag coefficient value decreased to 0.4444087. Therefore, with installation of vortex generators drag is reduced.

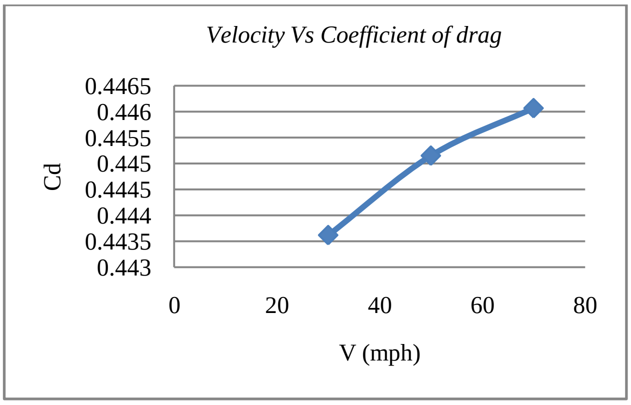
4. Results & Discussion

From the obtained simulation results, following drag comparison table is prepared. With fixed velocities 30, 50 and 70 mph, corresponding Cd values of without and with vortex generator are described in below table.

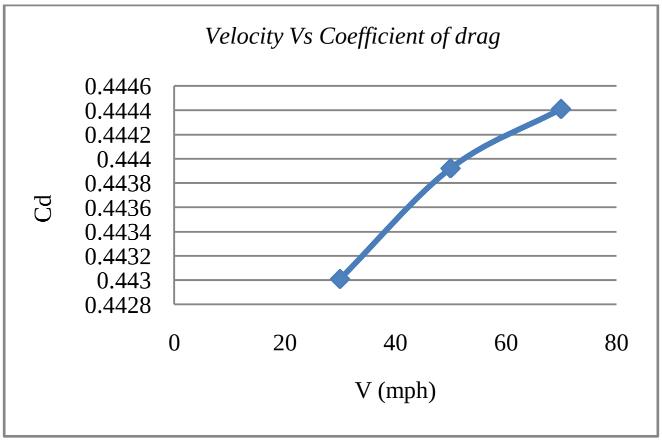
Table1: Drag Comparison

Velocity (mph)	Coefficient of drag (C_d) without vortex generators	Coefficient of drag (C_d) with vortex generators
30	0.4436179	0.4430082
50	0.4451493	0.4439197
70	0.4460641	0.4444087

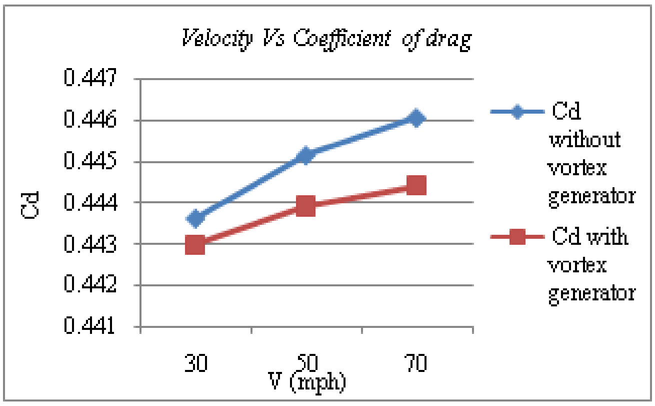
From drag comparison table, following graphs are plotted and comparison is explained briefly.



Graph 1: Velocity VS Coefficient of drag without Vortex Generator



Graph 2: Velocity VS Coefficient of drag with Vortex Generator



Graph 3: Comparison of C_d with and without Vortex Generator

From Graphs 1&2, it can be seen that Cd value is increasing with increment in the velocity of the airflow in case of both the cases with and without vortex generator. Graph 3 shows comparison, from this graph it can be observed that, for fixed velocity, coefficient of drag values with vortex generator is less than, compared to coefficient of drag values without vortex generator. From this, it can be conclude that drag is reduced after installing vortex generators.

4.1 Velocity Comparison

From Fig19, we can see that drag is very high at the rear of the model. From simulation, for 70mph coefficient of drag value without vortex generators is 0.4460641. The vehicle performance is reduced with the effect of drag force. In this case Cd value is higher, for better performance of vehicle this has to be reduced. From Fig 32, velocity distribution plot velocity is increased at the rear end of the vehicle, which results in reduction of drag. From simulation, for 70mph coefficient of drag value with vortex generators is 0.4444087.

Air resistance on car reduces with reduction of drag and which results in higher top speed and then it reduces power required. Therefore, finally this has a great impact on the increment of fuel efficiency.

4.2 Pressure Comparison

From the figure plots16& 29, in case of car model without vortex generators, we can see low pressure area at the bottom rear end of the car and boundary layer separation is larger at the end of the rear side of the model. While, in case of after installing vortex generator, we can see low-pressure region at bottom of the model, therefore which increases the velocity at the bottom of the model and finally which cause to increase the down force. Because of this more down force, vehicle has good grip on the road track and dynamics characteristics like handling and cornering ability of car will be improved.

5. Conclusion

Vortex generators are installed to introduce energy back into the boundary layer, which effect to stick the airflow better to the body surface of the vehicle from roof to the rear end (window) of the vehicle. From simulations figures it is clear that without vortex generator, flow is separated at the rear end of the car. While, when vortex generators placed at its rear end, flow separation is controlled. From the results and comparison of simulations, it is clear that after installing vortex generators, drag and boundary layer separation is reduced while downward force is increased. Therefore, finally we can conclude that designed vortex generators have met their purpose of increasing fuel efficiency and improving aerodynamics stability of vehicle by reducing the drag.

References

[1] Anon (2016). Aerodynamics - Introduction to the science of air flow. [Online]. 2016. Explain that Stuff. Available from: http://www.explainthatstuff.com/aerodynamics.html. [Accessed: 10 January 2019].
[2] Lajos, T. (n.d.). Basics of vehicle aerodynamics. 1st Ed. [Online]. Available from: http://www.ara.bme.hu/oktatas/letolt/Vehicleaerodyn/Vehicleaerodyn.pdf. [Accessed: 3 January 2019].
[3] Koike, M., Nagayoshi, T. & Hamamoto, N. (n.d.). Research on aerodynamics drag reduction by vortex generators. p. 6.
[4] Islam, M., Hossain, M., Mashud, M. &Gias, M. (2013). Drag reduction of car by using vortex generator. International journal of scientific and engineering research. 4 (7). p. 6.
[5] Sayma, A. (2009). Computational fluid dynamics.
[6] User, S. (2016). PHOENICS. [Online]. 2016. Safesolutions.com.br. Available from: http://www.safesolutions.com.br/en/software-4/phoenics-en. [Accessed: 6 January 2019].
[7] Anon (2016). [Online]. 2016. Hitechcfd.com. Available from: http://www.hitechcfd.com/cfd-knowledgebase/seven-stages-of-a-typical-cfd-simulation.html. [Accessed: 1 January 2019].
[8] Rohatgi, U. (2012). Methods of reducing vehicle aerodynamic drag. 1st Ed. [Online]. Available from: https://www.bnl.gov/isd/documents/78582.pdf. [Accessed: 18 December 2018].
[9] Naveen. S., Vipin Prakkash, & Sukanth Kannan, (2016). Design of a custom vortex generator Optimisation of vehicle Drag and Lift Characteristics. International journal of Innovative science, Engineering and technology. 2 (9). p. 7. Available from: [Accessed: 4 December 2018]