EFFECT OF VORTEX GENERATORS ON THE EFFICIENCY OF A VEHICLE

Ratnajeet Wadile

Terna Engineering College, Nerul, New Mumbai, Maharashtra, India.

---***---

Abstract - *Nowadays, vehicles are not constrained to a specific speed limit. Most of the vehicles that we see today have maximum speed over 150 km/hr. Having such a high speed enormously affects the overall efficacy of the vehicle. At these high speed, aerodynamic forces, acting on the body, plays an important role performance and overall stability of the vehicle. Many ways to enhance the efficiency of vehicle are been proposed and implemented; aerodynamic shapes, curves over edges, etc. One external way to improve the aerodynamic forces is by using vortex generators. To demonstrated this, a typical anatomy of a hatchback car (Hyundai i20), which has coefficient of drag (Cd)= 0.63 and coefficient of lift (Cl)= 0.081, has been designed using Solidworks software and analyzed using Ansys 2019 R3. The analysis shows improvement in the values of coefficient of drag and coefficient of lift after the use of vortex generators.*

Key Words: **Vortex generators, Aerodynamics, drag force, lift force, CFD**.

1.INTRODUCTION

Aerodynamics is a branch of fluid dynamics concerned with studying the motion of air, especially when a body interacts with ai. Aerodynamics is also a subfield of gas dynamics, with much theory shared with fluid dynamics. Understanding the airflow over an object helps accumulate the data required to calculate the forces that are applied on the object by air. Typical properties calculated for a flow field include velocity, pressure, density and temperature as a function of position and time. By defining a control volume around the flow field, equations for the conservation of mass, momentum, and energy are often defined and used to solve for the properties. The use of aerodynamics through mathematical analysis, empirical approximation and wind tunnel experimentation from the scientific basis. Aerodynamics and its analysis are branched into two major sub-categories; external and internal aerodynamics. External aerodynamics is the study of flow around solid objects of various shapes. Evaluating the lift and drag on a vehicle or on an aeroplane, the shock waves that a rocket generates in ahead of itself or the flow of air above a wind turbine blade are some of the examples of external aerodynamics. On the other hand, internal aerodynamics is the study of flow through solid objects. For instance, internal aerodynamics subsumes the study of the airflow through a jet engine or through an air conditioning pipe and other internal flow conditions. The vehicle aerodynamic flow process is fall under three types (i) Flow of air surrounding the vehicle. (ii) Airflow through the object. (iii) Airflow within the vehicle machinery.

1.1 AERODYNAMIC FLOW FIELD OF A CAR BODY:

Whenever a body moves through a viscous fluid, the fluid tends to move over the body, which creates high pressure and, low pressure areas over different regions. The fluid flow exerts a pressure force on the boundary layer, which gets separated at the end of the vehicle. Fig 1.0 shows the streamline flow of air over the car body.

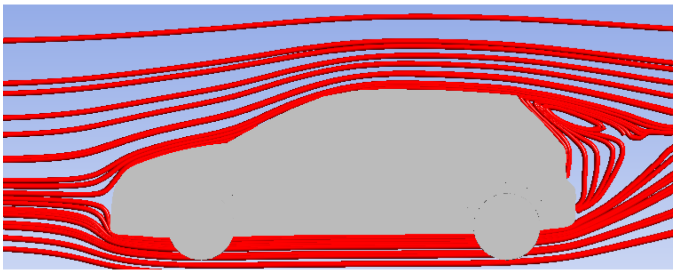

Fig.1.0 Streamline flow of air over a car

One of the phenomena which affects the flow of air over the car affecting the efficiency of the car, is 'wake'. Wake is generally created due to high pressure and, low pressure areas behind the car. When travelling at higher speeds, there is generation of low-pressure area at the back of the car, which bends the air flow into that low pressure zone, which makes the flow turbulent, generating the drag force. This drag force affects the efficacy as it acts in opposite direction as that of the car's motion.

1.2 DEFINITION OF COMPONENTS RESPONSIBLE FOR GENERATION OF FLUID FLOW:

1)BOUNDARY LAYER:

a boundary layer is the layer of fluid in the immediate vicinity of a bounding surface where the effects of viscosity are prominent. It is the flow close to the part of external surface of the vehicle.

There are two types of boundary layers.

i) Laminar boundary layer
ii) Turbulent boundary layer.

i) ***Laminar Boundary Layer***: The laminar boundary is a very smooth flow; it creates less skin friction, but it is less stable. Its thickness increases as it passes over the body.

ii)*Turbulent Boundary Layer*: The laminar flow breaks, at the end, to form a turbulent flow. It is the most responsible factor in exacerbating the efficacy of a vehicle as it creates opposing force.

2) SEPARATION: Air near the surface has lower velocity as compared to the upper layers of air due to pressure from outer layers. The velocity comes to stall at a point and start flowing backwards which is known as 'separation'. Separation of the boundary layer is more towards the end of the vehicle. This is one of the most important factors to be considered while studying the wake of the vehicle. Fig. 1.1shows the separation of air flow over the surface.

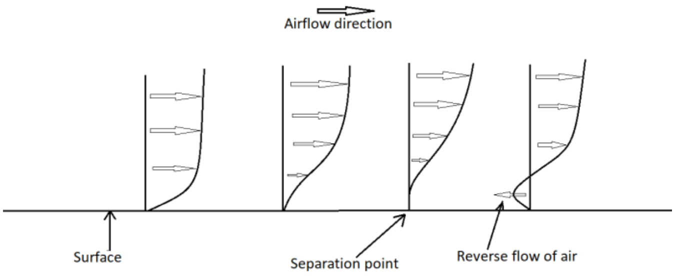

Fig.1.1 Separation of air flow

3)PRESSURE DRAG: Pressure drag is generated due to more compression of air particles at the front end of the car and more spaced out at the back. The layer of the air gets separated away from the surface and start to swirl- turbulent flow- due to the pressure difference generated at the respective regions.

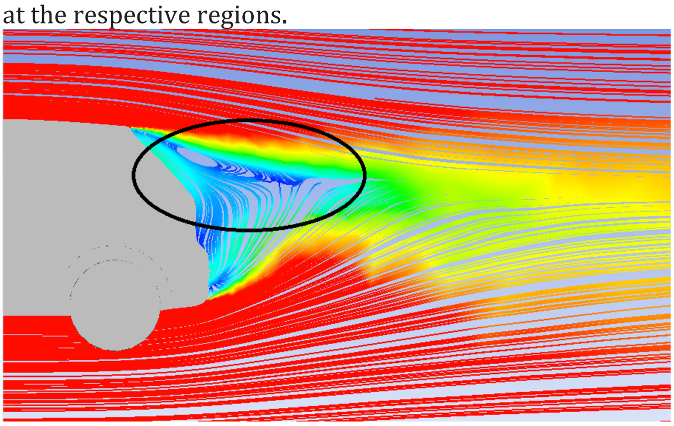

Fig.1.2 Pressure drag

4) FRICTION DRAG: Every material has different frictional properties, which resists then flow of air. This causes the considerable separation of the air flow to generate the drag. It is also known as 'skin friction drag'. It is directly proportional to the area of the surface which is in contact with the air and increases with the square of the velocity.

Turbulent flow creates more friction than laminar flow due to its greater interaction with the surface of moving body. The possible way to reduce the friction drag is by delaying the point at which laminar flow becomes turbulent.

2. FORCES ACTING ON A MOVING CAR:

Whenever a body moves through air, forces tend to act on the body because of this motion. As the air moves over the upper, curved part of the car, velocity of air increases respective to the air present at the bottom side. As velocity and pressure are inversely proportional the pressure at the top region decreases and air tends to move from lower, high-pressure area to the upper, low-pressure area, which generates a upward force called as lift force. This lift force affects the efficiency of the vehicle as it decreases the frictional force of the vehicle with the ground, decreasing the stability of the vehicle.

Another force that acts on the moving vehicle is drag force, which is generated at the back of the car due to swirling motion of air. This impedes the velocity of the vehicle, thus reducing the efficiency.

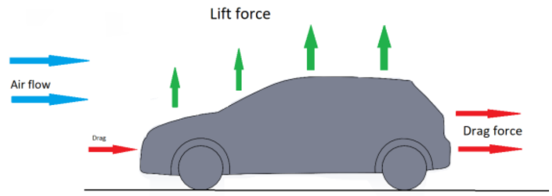

Fig.1.3 Lift and Drag force acting on a vehicle.

3. SCOPE OF AUTOMOBILE AERODYNAMICS:

Aerodynamic characteristics of a vehicle are responsible for the properties from energy efficiency linked to aerodynamic drag and safety to environmental influence of vehicles on their surroundings including air pollution and noise.

The reduction of vehicle drag is an important aspect for improvement of numerical and experimental tools. However, the other aspects such as crosswind stability, unsteadiness from passages of tunnels, platforms or other vehicles, ballast projection for high-speed trains, aero acoustics and soiling which require new improved approaches in flow predictions. Along with advanced facilities, experimental techniques have also been developed on the experimental side. While, Introduction of the time dependent simulation is the most significant improvements in the numerical vehicle aerodynamics. Bothe of these approaches are used in development of techniques for improvement of vehicle properties by flow control or aerodynamic shape optimization. Aerodynamics has so many tasks like it is used by the engineers to enhance the efficiency of engine by maintaining its temperature lower, improving comfort for drivers, stability of the car in external wind forces, etc. Despite of having so many tasks, this work concentrates on the external factor to improve the flow of air to maximise its effectiveness. Fig. 1.4 demonstrates the flow of air over a typical hatchback vehicle.

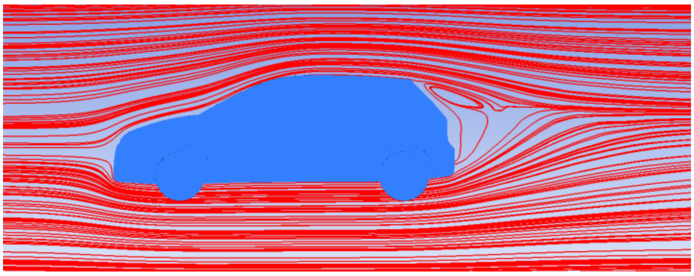

Fig.1.4 Flow pattern of air over a hatchback car

4. METHODS TO TEST AERODYNAMIC FLOW:

There are three main methods to test the vehicle's aerodynamic behavior; using wind tunnels, CFD, and on-track testing. On-track testing method is considered as best and most accurate as it consists the testing into real life environment, under the real-life conditions. It is overly efficient as well, but it has some shortcomings. There is a problem regarding isolating drag measurements rom rolling resistance, all the equipment, for calculations, is need to be attached to the vehicle, which causes practical problems. The other two methods which seems promising are Computational Fluid Dynamics (CFD) and Wind Tunnel Testing.

Computational Fluid Dynamics (CFD): Computational Fluid Dynamics (CFD) uses numerical method and algorithm to solve and analyze problems that involve fluid flow. Computers are used to determine the interaction of various fluids with the body. CFD is useful in plotting the force graphs, to understand where exactly the lift and drag forces are acting on the body. Using CFD we can examine the flow structure and can modify the necessary parts accordingly for better performance of the vehicle. Approximate solutions can (CFD) be achieved using high speed super computers.

Wind tunnel Testing:

In wind tunnel testing large number of parts can be tested effectively. Wide range of aerodynamic data can be collected depending on the yaw and roll conditions. If similar changes in conditions are made in the computer mode, whole simulation need to be performed again. Nevertheless, wind tunnel testing can be costly and time consuming as well. Small changes in the design can take a long time to implement on a prototype than it would in a computer model. The accuracy of the wind tunnel depends on other element and can be affected by several factors such as, blockage, scaling effects, moving road problems, and reliability and validation of the data need to be calculated in each case.

5. VORTEX GENERATORS:

What are Vortex Generators (VG):

A vortex generator is an aerodynamic device which is attached to the surfaces which generates lift. Whenever the body is in motion, these vortex generators generate vortices, which delays the airflow separation thereby improving the effectiveness of the vehicle.

Vortex generators are mostly used to delay the flow separation generating at the back of the vehicle. Similar to the car body, they also create high-pressure and low-pressure regions behind them, which makes air to move from area of high pressure to the aera where pressure is low. This generates the small vortices behind the car, which helps to delay the flow separation.

6. CAR MODEL:

An approximate model of a hatchback car (Hyundai i20) has been designed using Solidworks software, which is shown below.

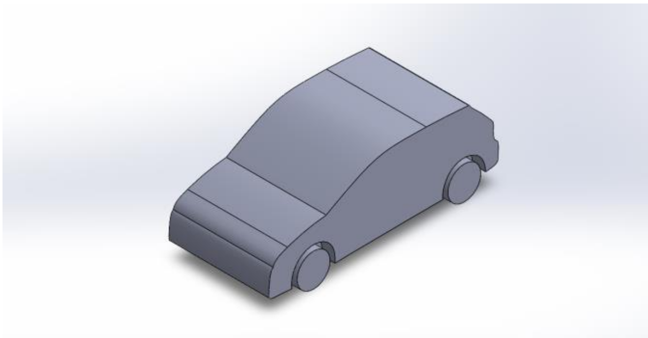

Fig.1.5 Model of the hatchback car without vortex generators

The same car model has been modified with the vortex generators at the top of the car.

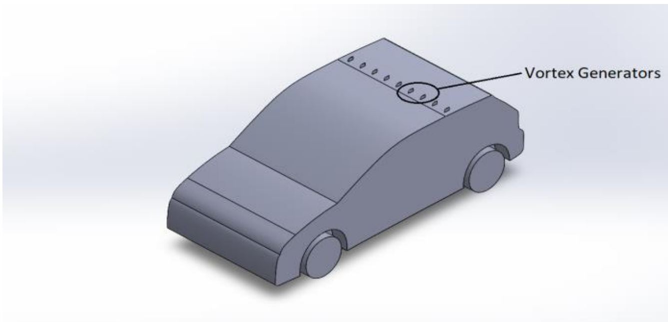

Fig.1.6 Model of the hatchback car with vortex generators

METHODOLOGY:

Initially, an approximate model of the car is designed using Solidworks modelling software. Which is then imported to ". igs" file, to be supported by the Ansys simulation software. Further all the processes were carried out in the Ansys software. In the next step, model in the required format is to be imported into the analysis software then required boundary conditions are to be applied. Finally, the problem is being initialized and an efficient, iterative scheme with solution algorithm was used to solve the problem. The analysis is to be carried out repeatedly until the optimal result is obtained the efficient yaw angle of the vortex generator. In order to reduce lift and drag co-efficient of selected car model.

RESULTS AND DISCUSSIONS:

The analysis report of the hatchback car (Hyundai i20) has been conducted based on two configurations, without vortex generators (normal car) and with vortex generators, which are mounted on the roof of the vehicle. The inlet air velocity is kept at 100 km/hr. (27.28 m/s) and outlet zone pressure is set as zero pascals. The representation of total pressure acting on the vehicle without and with implementation on vortex generators is shown in the following figures.

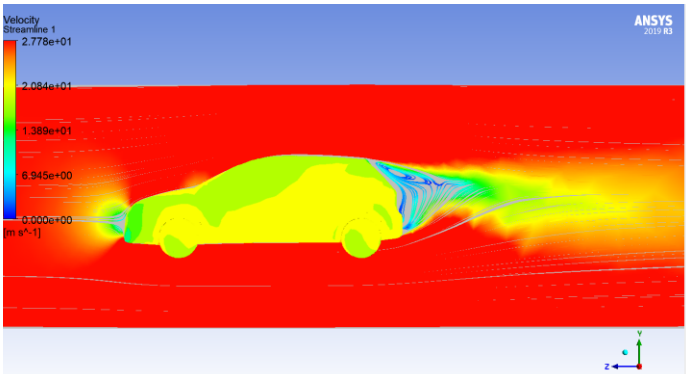

Fig.1.7 Flow of air over the car without VGs

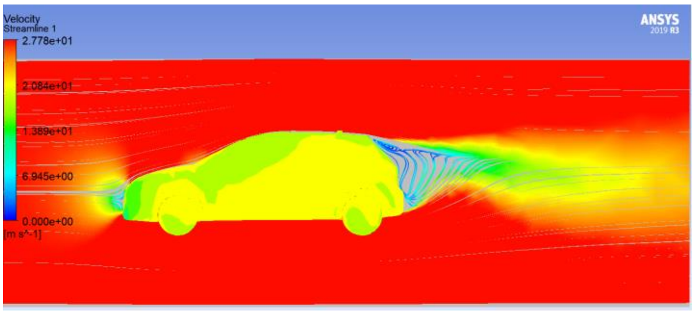

Fig.1.8 Flow of air over the car with VGs

It can be clearly seen from the results that vortex generators are able to delay the air separation to some extent, thereby reducing the drag force and also increasing the downward force applied on the vehicle by external air.

Table -1: Table 1.1 Results of a car model with and without vortex generators

Model configu-ration	Lift force (Fl)(N) (-Y)	Coeffi-cient of lift (Cl) (N)	Drag force (Fd)(N) (-Z)	Coeffi-cient of drag (Cd) (N)
Base model (Without VGs)	78.4546	-0.0814	612.524	0.63
Model with VGs	175.874	-0.1823	597.727	0.61

7. CONCLUSIONS

This project emphasizes on an efficient way to enhance the effectiveness of a car with the help of an external device- vortex generator- without modifying the actual anatomy of the vehicle. A typical design of car (Hyundai i20) is generated using a 3D modelling software (Solidworks 2021) and then analyzed with the help of a simulation software (Ansys 2019 R3). From the results there is a significant 2.41% decrease in the drag force and 124.17% increase in the downward lift force, without changing the actual structure of the vehicle.

REFERENCES

[1] Analysis of drag and lift performance in sedan car model using CFD by P.N.Selvaraju, Dr.K.M.Parammasivam, Shankar, Dr.G.Devaradjane.

[2] Angelis, W., D. Drikakis, F. Durst., W. Khier: Numerical and experimental study of the flow over a two-dimensional car model, Journal of wind engineering and industrial engineering. 62, 57-69 (1996).

[3] Introduction to aerodynamics by NASA.

[4] Ha, J., S. Jeong., and S. Obayashi.: Drag reduction of a pickup truck by a rear downward flap, International Journal of Automotive Technology, 12(3), 369–374 (2011)

[5] Drage, P., A. Gabriel., and G. Lindbichler.: Efficient Use of Computational Fluid Dynamics for the Aerodynamic Development Process in the Automotive Industry, Applied Aerodynamics 26, 1-15 (2008)