

# Enhancement of Aerodynamic Drag Reduction of Passenger Vehicle using CFD analysis- Review

Shivaji S. Shinde<sup>1</sup>, Prof. M.D. Shende<sup>2</sup>

Research Scholar, Shreeyash College of Engineering & Technology, Aurangabad, Maharashtra, India<sup>1</sup>

Associate Professor, Dept. of Mechanical Engineering Department, Shreeyash College of Engineering & Technology, Aurangabad, Maharashtra India<sup>2</sup>

**ABSTRACT:** The rapidly increasing fuel prices and the regulation of greenhouse gasses to control global warming give tremendous pressure on design engineers to enhance the current designs of the automobile using the concepts of aerodynamics with the help of computational fluid dynamic. It is also used to understand the fuel consumption, stability of the automobile and passenger comfort. The air flow over automobile is analyzed and calculates the drag with the help of CFD simulation. Drag reduction is most important method for improving the aerodynamic effect as well as improving fuel efficiency in automobile industry. Computational fluid dynamics (CFD) technique are used for reducing drag. Thus, CFD technique is performed very important role in automobile industries for grate design and increasing aerodynamic effect. In this paper, review is taken to study the various passive flow control methods to reducing the drag in automobile industry and improving the performance of automobile as well as introducing the superior design in automobile sector with the help of CFD.

**KEYWORDS:** Aerodynamic, Automobile Design, CFD, Coefficient of Drag, Passive flow control

## I. INTRODUCTION

**A. AERODYNAMIC:** The subject "Aerodynamics" studies with air flow passing over the automobile and also the behavior of air flow with the help of experimental setup like as Wind Tunnel as well as theoretical (analytical or semi analytical) and computational fluid dynamics (CFD) approaches respectively. The term "air" is used in a generic sense. It basically means the flowing gaseous medium which could be air, helium, or any other gas for that matter depending on condition. Anything is passing through air then effect of aerodynamic is shown, like rocket launching, kite flying in sky. Even cars passing through air is also affected by aerodynamics effect [6]. CFD analysis is only efficient tool in order to evaluate specific design parameterization of a generic shape of automobile. Computational Fluid Dynamics (CFD) is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze the problems that involve fluid flows. The analysis of system is associated by means of computer based simulation. Nowadays consume fuel 50% for ground vehicle all over worldwide and affected 60% greenhouse gas affected by the emission of gases from consuming fuel [1]. In recent times increasing the greenhouse effect in global warming world to reduce CO<sub>2</sub> emission with the help increasing the performance of automobile by reducing the drag coefficient by using the various methods like passive flow control system ex. Ahmed body, Add-on Device, VG is conclude [2][3]. The effect of aerodynamic design is increasing fuel efficiency, reducing drag and also improving the stability of automobile in high speed is shown in survey taken by Lynette Cheah & John Heywood [4].

**B. COMPUTATIONAL FLUID DYNAMICS:** Computational fluid dynamics (CFD) is the branch of fluid dynamics providing a commercial tool means of simulation authentic flows by the numerical solution of the governing equations. Computational fluid dynamics (CFD) is the science which deals with the fluid flow, heat transfer, mass transfer,

**International Journal of Innovative Research in Science, Engineering and Technology**

An ISO 3297: 2007 Certified Organization

Volume 6, Special Issue 1, January 2017

**International Conference on Recent Trends in Engineering and Science (ICRTES 2017)**20<sup>th</sup>-21<sup>st</sup> January 2017

Organized by

**Research Development Cell, Government College of Engineering, Jalagon (M. S), India**

chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process. The “Computational Fluid Dynamic” is relates to realize air flow over the automobile and study various types of forces apply on exterior body [5]. However, the improvement of reduced forms of these equations is still an active area of research, in particular, the turbulent closure problem of the Reynolds-averaged Navier-Stokes equation. Computational techniques replace the governing partial differential equations with systems of numerical equations that are much easier to solve using supercomputer with the help of CFD Ansys software [8].

**C. COEFFICIENT OF DRAG:** coefficient of drag is dimensionless number that represents the drag and lift characteristics of the given body. Drag and lift Coefficient are primary function of shape of body but some case they also depend on aerodynamic pressure drag or friction drag force [9]. Improving the aerodynamic drag offers an economical solution to develop fuel efficiency and thus shape optimization for small drag becomes a fundamental part of the overall automobile design process. It has been found that 60% of the drag force is concentrated at the rear of the geometry [6]. In a modern vehicle, the overall shape contributes to 45% of drag while the shares of wheels, their arrangement, the bottom, and its design details, are 30% and 25%, respectively[10]. In optimization of automobile aerodynamics, more absolutely the reduction of drag coefficient ( $C_D$ ), which is mainly influenced by the outer shape or design of automobile, has been one of the main issues of the automotive research centers all around the world. Automobile power consumption decrease can be achieved by different method such as superior engine efficiency and aerodynamic drag reduction. From the early days, the designers recognized the importance of drag reduction and tried to streamline the design and get better the power in automobile sector [2]. Since the early 20-th century, large number of studies was carried out in this field and some device is given below with description [11].

**II. RELATED WORK**

**A. EFFECT OF UNDERBODY DIFFUSER ON AERODYNAMIC PERFORMANCE:** Diffuser is device used at rear end of car underbody which improves the car’s aerodynamic property by enhancing the change between the high-velocity airflow at the bottom of the car and the much slower free stream airflow of the ambient atmosphere. It works on the principal of Bernoulli’s principle, such that the pressure decreases while the velocity increases, therefore the air flowing under the vehicle increase in speed and drops in pressure, creating down force. Diffuser use to reducing aerodynamic drag of any type of car model it is very effective device. It is also extensive application in turbo-machine, in jet propulsion engines and in wind tunnels for the purpose of decelerating confined fluid flows as well as wide-angle diffuser is very effective for drag reduction it is directly effect on pressure recovery at rear end [12]. The rear diffuser has lot of jobs to do, firstly is acts as way of speeding of air flow (lowering pressure) and then slowing down air flow (Increasing pressure), in an aid to create to down force and smooth out turbulent air existing the rear of car, matching the outside high pressure air while minimizing drag. The diffuser use in car model at front and rear end of car model, the air flow like venturi tunnel from front side of car model is use diffuser the air enter at atmospheric pressure but it flow under body of car the pressure is increasing then velocity decreasing at middle point of car but at the rear end diffuser using to increasing wide angle & area the pressure is decreasing and velocity increasing at rear end. Conclude that at the end higher vacuum effect, more down force is generated and therefore the aerodynamic drag was reduced.

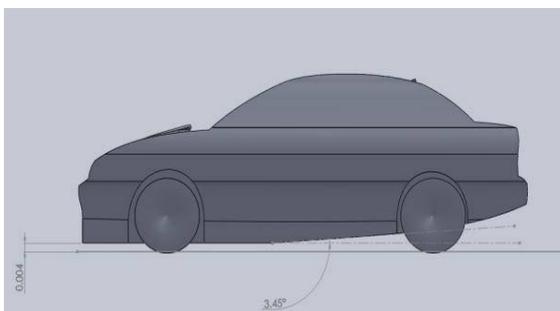
a) **Akshay Parab [11]** In this paper, the study on lift and drag coefficient of vehicle is available in India also it also compare the result getting from Ansys 14.5 to officially provide value of vehicle drag coefficient. In this paper improving fuel efficiency and improving stability of given model then focuses on the reducing lift coefficient with the help of diffuser type device at the rear end of underbody of vehicle. In this paper the using three type of angle of elevation for reducing lift is  $8^0$ ,  $10^0$ ,  $15^0$  coefficient. The addition of an 8 deg. diffuser helped reduce the lift considerably (34% reduction in  $C_L$ ) while only slightly increasing the drag coefficient (0.5% increase in  $C_D$ ). Addition of a diffuser showed an increase in drag and a corresponding decrease in lift in all three cases. Therefore such modifications could be carried out in the selected car to improve its handling capabilities at higher speeds thereby improving the overall safety of the vehicle.

b) **S.M. Rakibul Hassan [13]** is study on the improving aerodynamic drag coefficient of a racing car model with the help of rear end modification like by using diffuser or underbody sliced for different angle and also uses exhaust gas

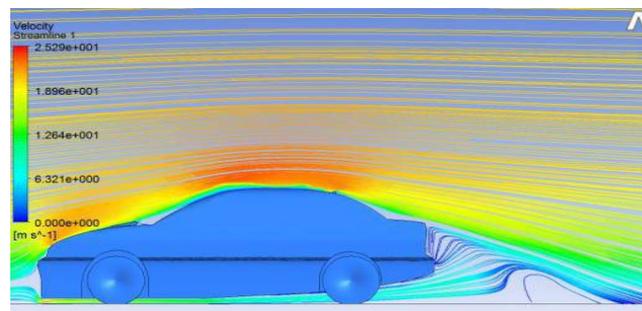
redirection system. Flow separation is dependable for the major portion of aerodynamic drag of racing cars. Therefore the design of racing car model is like streamline shape, reducing surface roughness and fewer joints of the body or avoiding sharp fillets. They are also the other factor like coefficient of lift is very important for designing racing car for driving safety in high-speed. Aerodynamic drag reduction by rear under body modification (rear underbody sliced) results in up to 22.13% and rear under-body diffuser results 9.5% reduction of drag coefficient. Exhaust gas redirecting is up to 3.3% drag coefficient can be reduced by this procedure considering the ideal exhaust gas composition. With the help of various type of modification and underbody diffuser using improving aerodynamic drag coefficient and lift coefficient it is very helpful for handling of racing car and increasing fuel efficiency.

**c) Jesper Marklund [14]** is told exterior body of a passenger vehicle, such as the upper and under-body, is responsible for more than 60 % of the drag. It is work on under body diffuser used in various type of model like square back model, large backlight angle, passenger vehicle, sedan vehicle. In this research work sedan vehicle greater potential to reduce drag by using a smooth floor and under-body diffuser than the wagon type model, because diffuser generates an upsweep and makes the wake more symmetrical shown in side view and it is wagon type model the diffuser generates unsymmetrical wake. Wake symmetry is very important for optimum pressure recovery of passenger vehicles it is also helpful to reducing aerodynamic drag. The diffuser angle for optimum drag reduction of sedan based and wagon based model is 8° & 5° respectively. The improvement for the sedan was approximately 4%, from 13 % reduction of the drag coefficient  $C_D$ .

**d) Ahmed .H & Chacko [15]** this research study on diffuser installed at the rear end of sedan based car. They also designing the diffuser with the help of angle of elevation and it start position because the effect of diffuser is depend on the angle of elevation and the effective elevation angle is the one that prevents the formation of the flow separation. Therefore they also find the how much downforce generated at the end underbody of given model. In this study the diffuser angle of elevation is 3.45° and start position at half way of sedan based car model and the ground clearance at the rear end is 0.08m. As shown in fig.(1) The effect of adding diffuser on sedan based car the shown CFD result the coefficient of drag is reduced from 0.377 to 0.3478 (8% reduction of drag coefficient) and lift coefficient is reduced from 0.2944 to 0.094 (31% reduction of lift coefficient). This is very good result in lift coefficient because the reducing the lift coefficient the improving the stability of car model it is also improve the driving stability at high speed. The stream line flow by using diffuser at rear end of sedan based car is shown in fig. (2)



**Fig(1) Ground effect of diffuser [15]**



**Fig(2) stream line flow when using design diffuser [15]**

**e) K. Ramesh [16]** This research develop an actively translating rear diffuser device to reduce the aerodynamic drag with the help of TATA INDOGO CS passenger car model at high speed driving condition. In this study, a movable arc-shaped semi-diffuser device, round in form, is designed to maintain the streamlined automobile's rear underbody configuration by using CFD analysis software. Seven types of rear diffuser devices whose positions and protrusive lengths and widths are different (with the basic shape being identical) were installed the main purpose of this study is to explain the aerodynamic drag reduction mechanism of a passenger car cruising at high speed via an actively translating rear diffuser device. Therefore the result are rear diffuser device slides out from under the rear bumper, the aerodynamic drag of the passenger car is reduced within the speed range of 70 km/h~160km/h. Among the various

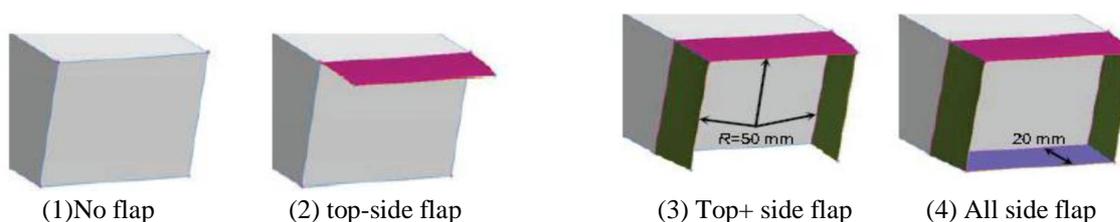
lengths of the diffuser, Case 6 (450 mm) type diffuser is best drag reduction performance, with an average reduction of more than 4%. Moreover, as the driving speed increases, the drag reduction effect also increases as well as the improvement of fuel efficiency of given passenger car model is travelling at a high speed.

**f) Hu [17]** conducted CFD investigation to study various diffusers with angles of 0°, 3°, 6°, 9.8° and 12° on a sedan type car model. The outcome of CFD investigation is the drag coefficient first reduced from 0° to 6° and then improved from 9.8° to 12° whereas the lift coefficient every time reduced from 0° to 12°.

**B. EFFECT OF FLAP AND GUIDE VANE ON AERODYNAMIC PERFORMANCE:** flap and guide vane is add on device is used for reducing the aerodynamic drag for various type of ground vehicle. They are installed at rear end of with some angle. They are used commonly for square back model for improving aerodynamic drag and aerodynamic lift for increasing fuel efficiency and better handling of ground vehicle. It is performance is depending on the length, shape and angle of installing flap on ground vehicle. It is very impotent device because the no modification is required for available design; it is only add for improving aerodynamic performance.

**a) E.M. Wahba [18]** is research on lateral guide vanes as a drag reducing device for ground vehicles is numerically investigated in the present study using computational fluid dynamics. They are use two types of ground vehicles like a simplified bus model and a simplified sport utility vehicle (SUV) model. In this present study bus model & SUV model is investigated with the help of CFD software with different turbulence model like k-ε & k-ω getting result at different guide vanes in following angle like 5, 10 & 15 degree simultaneously. Guide vane cross-section, chord length and angle of attack are various in order to obtain the best configuration for superior aerodynamic performance. The overall result is when the lateral guide vanes are used at rear end the coefficient of aerodynamic drag up to 18% for both models. Moreover, it is also shown that guide vanes with symmetric airfoil cross-sections result is getting better result than asymmetric airfoil cross-section for reducing aerodynamic drag coefficient. They also predicted frontal area of given model the  $C_D$  is affected up to 10%. Therefore the flap is very important for reducing drag of SUV ground vehicle or Bus model for controlling or driving very safety and increasing fuel efficiency.

**b) Kowata et al.[19]** is research on flap using at rear end a simplified car model for reducing aerodynamic drag with the addition an underbody slant similar to a diffuser setup. As shown in below fig. flap is arranged no flap (1), top side (2), top+side (3), All side (4) with 20mm length. It observation is shown the value of coefficient of drag is reduces by using underbody slant alone at low angles, the gains could be enlarged by adding flaps to reduce the development and detaching of trailing vortices into the wake. With the help of flap and slant low angle underbody is effect on aerodynamic drag co-efficient and also improvement of coefficient of lift for improving fuel efficiency of simplified car model and also improving stability in high speed.



**Rear Flap configuration for drag reduction [19]**

**c) Aider [20]** is study on the improvement of aerodynamic drag performance with the help of flap is attaching at rear end with best combination. In this present study is work on the flap combination for improving the performance. The drag reduction related with reducing the back angle from 30 degrees to the most favorable of around 12 degrees. The test velocity was 40m/s. From these outcomes it may be possible to create additional gains in drag reduction beyond the optimum back angle of 12 degrees using flaps in combination with other slant angles. Aerodynamic drag reducing up to

**International Journal of Innovative Research in Science, Engineering and Technology**

An ISO 3297: 2007 Certified Organization

Volume 6, Special Issue 1, January 2017

**International Conference on Recent Trends in Engineering and Science (ICRTES 2017)**20<sup>th</sup>-21<sup>st</sup> January 2017

Organized by

**Research Development Cell, Government College of Engineering, Jalagon (M. S), India**

25% with best combination of flaps, it is very good result for improving performance of bluff model and also improving efficiency as well as stability of model in higher speed.

**C. EFFECT OF SPOILER ON AERODYNAMIC PERFORMANCE:** The drag force is produced by relative motion between air and Vehicle and about 60 % of total drag at the rear end. A spoiler is device is use in sedan based car model at rear end for improving aerodynamic performance. Spoiler means the “spoil unfavorable air movement across a car body in motion” it is every time effect on aerodynamic performance. Spoiler is installed in front side of car and rear side of car, the front side spoiler is called Air dams & rear side of spoiler is also called rear wing. The front side spoiler (Air dams) is directing airflow they also reduce the amount of air flowing underneath the vehicle, which generally reduces aerodynamic lift and drag. Spoilers are installed at the rear end of sport car for improving performance in car race like better car handling is possible with the help of spoiler. Spoiler is also helpful to delay flow separation as well as the higher pressure in front of the spoiler can help reduce the lift on the car by creating down force. This may reduce drag in certain instances and will generally increase high-speed stability due to the reduced rear lift. . The goal of many spoilers used in passenger vehicles is to reduce drag and increase fuel efficiency.

**a) AHMED. H & CHACKO [15]** In this research paper is investigated wing (spoiler) is device is used inverted position at the rear end of car model for producing negative lift and improving aerodynamic performance. This type of wing is very popular & it is very effective for producing good amount of down force is very important for improving aerodynamic performance. It is located at the rear end of passenger car model of chord length 0.2m and height from boot is 0.3m similarly the angle is 0° inclination. With the help of CFD simulated this type of model the result is drag coefficient is increased up to 5% & lift coefficient is reduced up to 7%. But when installed the wing at the rear end of car model is back turbulence is totally vanished and air flow over car body very stream line.

**b) Ram Bansal and R. B. Sharma [3]** Spoiler is 15cm long is installed at rear end of car at the inclination of angle 12°. It is follow the important role for delay the separation of flow that normally occurs at the rear end of the car. When the spoiler is installed on passenger car at rear end the percentage of reduction on coefficient of drag in compare with baseline (without installing spoiler) model is 2.02% & coefficient of lift is 6%. Hence drag force and lift force is simultaneously reduced. One of the best results is obtained when the spoiler is installed with the combination of vortex generator at the rear end of car the coefficient of drag is decreased up to 4.35% and coefficient of lift in reduced up to 18.83%. This is very good result for improving performance of car model. Therefore the performance of car model is also improved like increasing fuel efficiency as well as increasing handling performance.

**c) V. Naveen Kumar & K. Lalit Narayan [5]** in this investigation the two different type of rear spoiler designs are used on sedan based car at the rear end. He was study the most of sedan based car 60% drag force are produced in between relative motion between air and at the rear end of car. So this type of drag force are reduced with the help of rear spoiler are used on sedan base car and the result is the second spoiler better than first spoiler. The result is the coefficient of drag reduced up to 2.18% than the base line model. This analysis is conducted with the help of CFD Fluent solver. The second type of rear spoiler are used at the rear end of car model is increases the tires capability, stability of car at high speed & improve the breaking performance and give better traction for avoid accident.

**d) Xu-xia Hu & Eric T.T. Wong [21]** In this research paper work on the rear spoiler of passenger car and simulated the result with the help CFD software. By using standard k-ε model is selected to numerically simulate the external flow field of the simplified Camry model with or without a rear-spoiler. Through comparing the results of 24 cases, a rear spoiler of better performance has been developed. The high speed passenger car with new rear spoiler shows that the aerodynamic drag is reduced by 1.7% and also it increases negative lift. The high speed sedan car with a spoiler shows a drag reduction of 1.7%, which would result in fuel saving. With the help of using rear spoiler it also improving tires capability, braking performance & stability with better traction. When reducing drag with the of rear spoiler then also improving the fuel efficiency of sedan based passenger car

e) **Cakir Mustafa [22]** in this thesis present a numerical simulation or flow analysis of passenger car with spoiler situated at the rear end using computational fluid dynamic software ANSYS FLUENT®. It is investigation Aerodynamic characteristics of a racing car are of significant interest in reducing car accidents due to wind loading and in reducing the fuel consumption. It will be used different type of spoiler located at different position at the rear end of vehicle and the simulation will be run on CFD and to investigate the aerodynamic effect of spoiler. One of the design goals of a spoiler is to reduce drag and increase fuel efficiency. Adding different type of spoiler at the rear end of car model makes the air slice longer, gentler slope from the roof to the spoiler, which helps to reduce the flow separation. Reducing flow separation decreases drag, which increases fuel economy. In this study the use most appropriate turbulence model for external flows around the car body is  $k-\epsilon$  model. The first type rear spoiler (wing style spoiler as shown in fig 3.) is installed at the rear end of passenger car aerodynamic drag coefficient reduced up to 17% as well as the aerodynamic lift increased up to 7%. The second type of rear spoiler design it is located on the rear end without leaving gap between spoiler surface and car rear end. As shown in fig 4. it provides 6% of coefficient of drag reduction but negative impact lift coefficient is increased up to 17%. In between two type of spoiler are simulated and getting result but the most suitable is for higher performance is first type of rear spoiler because the reducing drag coefficient is maximum than increasing lift coefficient. This is better condition for driving safely with high speed. It is also improve the performance of braking system as well as increasing tires capability to produce cornering force.

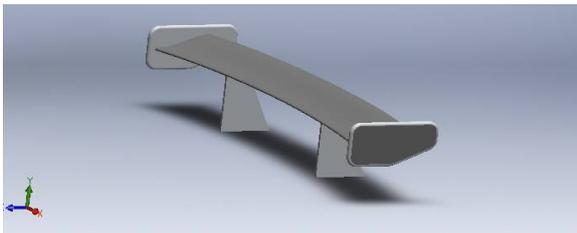


Fig.3 wing style rear spoiler [22]

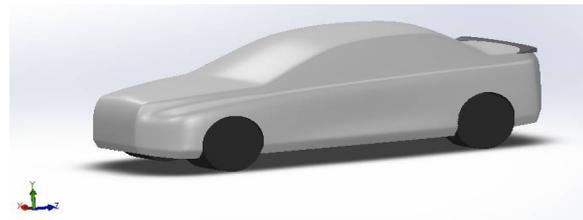


fig.4 assembly 2<sup>nd</sup> rear spoiler on car model [22]

f) **Chien-Hsiung [23]** Tsai in this present investigation the main propose find out the effect of wing type rear spoiler on passenger car on aero-acoustics system. When the wing type of rear spoiler installed at the rear end of passenger car model an appropriate angle of attack can reduce the aerodynamic lift coefficient. This study is totally depending on CFD software with the help of investigates the effect of spoiler on aero acoustic properties and to reduced the noise produced form wind travel on car body. When lift coefficient proportional to aero acoustic effect. It is clear that the vertical stability of passenger car and its noise elimination can be improved. In this present study the total 6 cases with the help of CFD software and final result the case 4 is very effective than other cases. When the 4 case type of spoiler is add on the passenger car the lift coefficient is decreasing but slightly the drag coefficient is increasing but the objective of this paper is improved the aero acoustics properties with the help of reducing lift coefficient.

g) **SUN Qi & CHEN Ling-shan [24]** The main objective is about the relationship between the rear spoiler and car body, to make calculation easy, so the styling of cases has been simplified. The reducing lift coefficient with the help of 9-case rear spoiler is applying on different car body like flat, racing and common in various angles. They also work on the length of spoiler in different condition. This are all work on completed with the help of Solid work, Hypermesh and CFD software at constant speed 120km/h. Choose standard  $k-\epsilon$  turbulence model. The result is the best combination is flat type, 150mm chord length and 15 degree horizontal angle. This is good for reducing the lift force of car, increasing the ground adhesion of car, saving the time of designing, seizing new market easily and quickly.

h) **G. Ganesh [25]** is investigated the Honda Civic car 2009 aerodynamic effect adding rear end spoiler at different location with the help of AUTODESK flow design software. Then main aim of this paper is increasing fuel efficiency with the help of reducing aerodynamic drag coefficient. when the applying the spoiler at the rear end of Honda Civic car model the reducing flow separation decreses the drag. Analysis with the help of AUTODESK the result is with spoiler & without spoiler is coefficient of drag is 0.21 & 0.28 respectively. Therefore the aerodynamic drag reduced up to 25% this is very effective value for improving the performance of Honda Civic model.

i) **Dileep P. Menon [26]** is research on the Hatchback model for increasing the fuel efficiency with the help of reducing aerodynamic drag by adding rear roof spoiler. The main aim of this paper the study the rear roof spoiler effect on hatchback model with the help of 12 different case of slant length and radius of curvature, the simulation perform on the CFD software. The results are obtained positively the case 11 & 12 showed the highest drag reduction up to 5 to 6% when attaching a rear roof spoiler on hatchback model. They also show that the improving aerodynamic performance for better stability when spoiler slant length increasing and decreasing curvature angle.

j) **Vishal Shukl [27]** is investigated the passenger vehicle for reducing the drag with the help of splines with rear spoiler. By using the Hypermesh and Fluent software analyze the various type of passenger vehicle. The result are obtained when the installed rear end rear spoiler with splines the coefficient of drag is 0.51 and the coefficient of lift is 0.22. The percentage reduction in drag coefficient in comparison of base line car is 15% and in coefficient of lift is 12%. When reducing the car aerodynamic drag with the help of rear spoiler with splines for improving fuel efficiency & stability of passenger car.

k) **P Ramya [28]** is work on the two types of passenger cars like Sedan & SUV models for reduction of aerodynamic drag with the help of spoiler for improving fuel efficiency and stability of passenger car model. The main objective of this work is to study the flow over modern passenger cars and determine the drag force exerted over them. The analysis was done at varying vehicle velocities on two types of models. With the help of CFD ANSYS CFX software they perform analysis for sedan based and SUV car model from 30Kmph to 120Kmph. When the spoiler is installed on sedan based & SUV car model at rear end the result are obtained the reduction of drag force up to 15% & 5%. The coefficient of drag is reduced up to 16% & 14% for sedan based vehicle & sport utility vehicle (SUV). This is very important result obtained for improving aerodynamic performance for better handling of passenger car model.

**D. EFFECT OF VG ON AERODYNAMIC PERFORMANCE:** Vortex Generator (VG) is most important device used in automobile industry for reducing aerodynamic drag. They are installed on the roof of the any type of vehicle. They also working very effectively with single piece or in group are installed on the roof of the car. The VG function is “trip the airflow” passing over the ground vehicle and convert large vortex into small size vortex therefore the large vortex is converted into small vortices. The vortex generator follows the theory of exchange of momentum between upstream and downstream flow [15]. This is very important for reducing aerodynamic drag at the end of car. The purpose of VG is to control flow separation at the roof end of a sedan based car. The two different VG design are available for study like Delta wing Design and bump-shaped design but the delta shape design is very effective than bump-shaped design [15].

a) **Ram Bansal and R. B. Sharma [3]** is study on the VG installed on passenger car and it effect of drag coefficient with the help of CFD software. Two different VG designs will be studied, which include a delta wing design and bump-shaped design for reducing aerodynamic drag force for improving the fuel efficiency as well as better handling for driving safety. The VG are installed at the backside roof of passenger car model hence the effect of aerodynamic coefficient of drag is reducing up to 1.17% as well as reducing coefficient of lift is up to 9.8%. It is very beneficial for automobile engineering, because not any changes in design of car model the only add on the roof and reducing drag force and improving aerodynamic performance. It is very economical and low cost increasing performance method. Now days the new passenger vehicle is already installed Single VG on passenger vehicle is shown on road. In this paper the one addition work is the VG in installed with combination of spoiler and the result is obtained very best than the use only VG. The coefficient of drag reduced in combination of VG+ spoiler is 4.35%.

b) **AHMED. H & CHACKO [15]** In this research paper is investigated VG aerodynamic performance on passenger car with the help of CFD simulation. When VG is installed at the rear wind screen on passenger vehicle the exchange of momentum between upstream and downstream of the flow reduce the high pressure region. The VG is reducing the drag and also creates a drag because it's presence. The performance is depending on the ratio between the drag generated to drag reduced by the VG. The efficiency of VG is depends on the various design parameter like the Height (H), Width and their numbers as shown in fig.6 . In this research paper the height of the VG is equal to Boundary layer thickness, its length is 2H and distance between adjacent tow can be 100mm. In this research paper the result are

obtained the total drag is be reducing by a great extend hence they are considered to be performing well. The coefficient of drag has been reduced from the value of 0.4103 to 0.39 Near about 5% reduction.

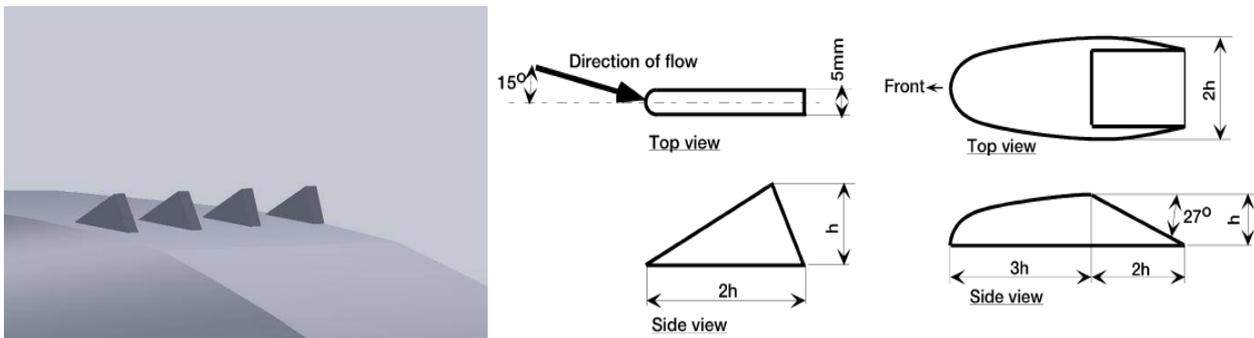


Fig. 5 Arrangement of VG on Sedan car based [15]

Fig.6 Design of VG in Delta & Bump-shape[29]

e) **Masaru KOIKE [29]** in this research paper presents the optimization result by using VG on passenger vehicle for Mitsubishi Lancer model. The VG height is almost equal to the thickness of boundary layer thickness (15 to 25mm) & its arrangement most favorable row type in lateral direction 100mm distance between them shown in above fig 5. The result are obtained reduced drag coefficient and lift coefficient when apply the VG on the roof of Mitsubishi is 0.6%. From this result the research show that the pressure of the vehicle's entire rear surface to increase therefore decreasing drag, also the velocity of rear spoiler to increase and lift to decrease. In this research paper is obtained the main important result the Delta wing Shaped VG is more effective than the Bump-shaped Design.

d) **Uendra S. Rohatgi & Volodymyr Sirenko [30]** in this present study the small scale model of General Motor SUV is tested experimentally in wind tunnel with various type of drag reduction technique is used. One of the most technique is VG are used on front edging and rear vortex generators. They are tested the three type of VG are used like VG1, VG2 & VG3 in some pattern. Different factor are considering to installing the VG on SUV model. The result are obtained when the installing VG on SUV model the coefficient of drag reduction up to 1.24%.

e) **P.N.Selvaraju [31]** in this paper investigated the aerodynamic performance of vehicle and its stability at high speed the reducing of drag by using VG at rear portion of the vehicle in 0 degree yaw angle and different location. The vehicle are used for installing VG is like sedan based (Hyundai Elantra) vehicle and the simulation done with the help of CFD software it is most important tool for reducing drag. The main objective of this paper is without changing the outer profile of car body the add VG and reduced the Coefficient of drag & coefficient of lift. When the installing VG at different position and the result are obtained is reduced coefficient of drag up to 6%.

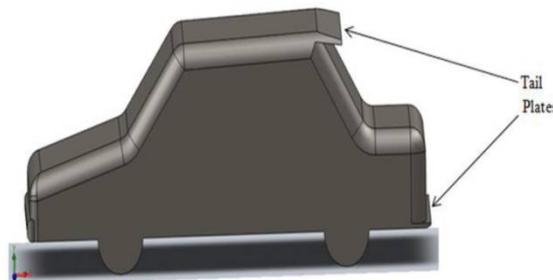
f) **Gopal P. and Senthilkumar T.[32]** In this research work the use VG for different three yaw angles are 10°, 15° and 20° and perform the experimental work with the help of wind tunnel by using scale model is 1:15 for the utility vehicle in different type of velocity. The experiments results are very good for aerodynamic drag force reduction with adding VG at the rear end of utility vehicle. The experimental result are  $C_D$  is reduced by 90% with the addition of VG at a velocity of 2.42 m/s and a minimum of 20% reduction in drag is obtained for VG with a yaw angle of 10°. It is observed that VG with a yaw angle of 15° will be useful at lower velocity. The value of Coefficient of lift is remains constant when VG with varying yaw angles at higher velocity, but decrease with increasing in velocity with and without VG.

h) **Beaudoin J.-F. and Aider, J-L [20]** in this research paper is most study on parameters of VG installed on bluff body at different location in longitudinal direction like complete line, VG on each side, VG in the center. The effect on aerodynamic performance of installing VG is reducing aerodynamic drag coefficient 12% and lift coefficient

60%. These are very important results obtained in VG. Coefficient of lift is reduced up to 60% therefore the stability of car model in higher speed.

**E. EFFECT OF TAIL PLATE ON AERODYNAMIC PERFORMANCE:** The tail plates are also used to lower wind drag, improve traction which leads to better handling. It is also reduced drag with the help of tail plate added at the end of passenger vehicle. This type of device is very cheap and installing very easily for available model. So this method is very popular seen in present survey. Present scenario the every hatch back model used Tail plate at rear end of car model or on roof. It is very effective method for old and new model of passenger vehicle.

a) **Ram Bansal and R. B. Sharma [3]** in this research paper the work on the tail plate installed on the rear end of sedan based passenger vehicle. The simulation done with the help of CFD ANSYS Software at given boundary condition. When the tail plate is installed on the passenger car model at underneath half-foot long plate was attached to the floor of vehicle is like diffuser and 5cm long plate was attached to the top at rear side of roof as shown in fig. 7 both the plate angle of inclination is 12 degree. The obtained result for CFD simulation is coefficient of drag is 0.3376 and Coefficient of lift is 0.1926. Therefore the percentage of reduction of aerodynamic drag coefficient is comparing with base line model is 3.87% and in coefficient of lift is 16.62%



**Fig.7 Tail plate attached on rear roof & tail bumper.[3]**

b) **Dongri Kim [33]** in this research paper the work on AMD (Automatic Moving Deflector), is added on the 3D model of vehicle for reducing aerodynamic drag coefficient & lift coefficient. This idea inspired by the movement of secondary feathers on bird's wing suction surface: i.e., secondary feathers pop up when massive separation occurs on bird's wing suction surface at high angle of attack, which increases the lift force at landing. Slanted surface consider the angle 25 degree the drag coefficient is highest because of at the end of model the separation bubble are present & this reason the highest drag force are generated. With the help of AMD prevented the formation of the separation bubble above the slanted surface by delaying the separation point to the trailing edge of AMD, and suppressed the strength of the longitudinal vortices. By using of AMD of different size and material are with respective longitudinal length is tested and result is getting to reduction of drag up to 19%. This concept is applicable on passenger car like sedan based the slant angle from 20 to 27 degree & Hatchback and SUV models having similar slant angles it is very beneficial for drag reducing.

c) **Manigandan. R [34]** in this research paper the main objective to reduced aerodynamic drag coefficient of hatchback car model with the help of tail plate using at rear end by using CFD Ansys Software to conduct the simulation. The tail plates are placed at back side of the roof and at the tail bumper of the passenger car at 12 degree inclination angle. The aerodynamic drag reduced from 0.430 to 0.352 at velocity 40km/h means the coefficient of drag reduced up to 0.078%.

d) **G. Siva and V. Loganathan [35]** in this paper the work on the Toyota Fortuner model is used to reduce aerodynamic drag without changing automobile performance. The work model creates in 2D and 3D & simulation is performing on ANSYS with the help of CFD Fluent software. The main objective of this paper is to improve performance of car without modification in car shape. Therefore with help modification in angle of tailgate and C-post is getting better result. The simulation is conducted at velocity 100km/h at an angle of tailgate is 35 & 45 degree. The

original angle of tailgate of C-post is 45 degree and modification angle is 35 degree. The result are show the improvement of aerodynamic of drag coefficient up to 14.21%. This is very important for improving performance of car without change in shape of car.

e) **Alamaan Altaf [36]** The main objective of research paper to study various type of flap (tail Plate) in different geometry like elliptical, Triangular, Rectangular attaching at the rear end of bluff body vehicle like trucks and buse & its effect on aerodynamic performance on a given model. The three type of geometry shape are considered is equal in surface area & simulation on perform the STAR CC++ software and finding better result. They are also work on the perforated plate with different diameter of hole on different geometry tail plate. When the plan flap or Tail plate are used at the rear end of model the better result is getting for elliptical plate the coefficient of drag reduction up to 11.11% when flap or tail plate mounting angle is 50 degree and its equivalent length is 0.12m. Similarly the result are getting for perforated tail plate of circular diameter is 0.02m and is reduced aerodynamic drag up to 8.8%. Means the elliptical plate is better performance in all three geometry.

### III. SUMMARY

The aerodynamic performance of all research work and the value of Coefficient of drag and coefficient of lift are noted in below table.

**Table 1 Research paper work on aerodynamic performance summary.**

Various type of Device is improving Aerodynamic Performance		
Device	Coefficient of drag reduction in %	Coefficient of lift reduction in %
1. Under Body Diffuser	0.5[11]*,22.13[13],4-13[14],8[15],4[16]	34[11],0[13],0[14],31[15],0[16]
2. flap and Guide	18[18],25[20]	0[18],0[20]
3. Spoiler	5[15],4.35[3],2.18[5],1.7[21], 6-17[22],25[25],5 to 6[26],15[27],14-16[28]	7[15],18.83[3],0[5],0[21],17-7*[22], 0[25],0[26],12[27],0[28]
4. Vortex Generators (VG)	1.17[3],5[15],0.6[29],1.24[30],6[31],10[32], 12[20], 4.35#[3].	9.8[3],0[15]0[29],0[30], 0[31],0[32], 60[20], 18.83#[3].
5. Tail Plate (AMD, flap, C-post)	3.87[3],19[33],0.078[34],14.21[35],8.8- 11.11[36]	16.62[3],0[33],0[34],0[35]

\*increasing drag and lift coefficient, # combination VG with Spoiler.

### IV. CONCLUDE REMARK

1. In Automobile industry the Aerodynamic performance is very important factor developing various type of model. The aerodynamic performance is important for all old model also. Therefore the improve coefficient of drag without modification of original model of ground vehicle the use add on devices like spoiler, tail plate, VG, flap.
2. The two factors are considered for improving aerodynamic performance one is Coefficient of drag and another is coefficient of lift. In summary table the coefficient of drag is very important factor for improving performance of any ground vehicle. More Literature is not present on Coefficient of lift the research work done on coefficient of drag maximum time means the aerodynamic drag coefficient is very important factor in aerodynamic performance.
3. The aerodynamic drag is affected on fuel efficiency up to 50% as well as CO<sub>2</sub> emission reduction up to 60% it is very important for green house effect. The various type research work completed with the help of CFD Ansys Software is very helpful for studied of coefficient of drag is increasing or decreasing.

## REFERENCES

1. Ramesh K. Agrwal " Sustainable ground transportation – Review of technologies, challenges and opportunities", *International Journal of Energy & Environment*, Volume 4, Issue 6, 2013 pp.1061-1078.
2. Chul-Ho KIM "A Streamlined Design of a High-speed Coach for Fuel Saving and Reduction of Carbon Dioxide", *International Journal of Automotive Engineering 2 (2011) 101-107*.
3. Ram Bansal and R. B. Sharma "Drag Reduction of Passenger Car Using Add-On Devices", *Hindawi Publishing Corporation Journal of Aerodynamics*, Volume 2014, Article ID 678518, 13 pages.
4. Lynette Cheah & John Heywood, "Factor of Two: Halving the Fuel Consumption of New U.S. Automobiles by 2035", *laboratory for Energy and the Environment, Massachusetts Institute of Technology, Publication No. LFEE 2007-04 RP*
5. V. Naveen Kumar " Investigation of Drag and Lift Forces over the Profile of Car with Rear spoiler using CFD", *International Journal of Advances in Scientific Research* 2015; 1(08): 331-339. ISSN: 2395-3616
6. M. Rouméas, P. Gilliéron & A. Kourta " Drag Reduction by Flow Separation Control on a Car After Body", *Int. J. Numerical Methods Fluids* 60, 2008 . ISSN 0271-2091
7. Ferziger, J.H. and M. Perić, *Computational Methods for Fluid Dynamics* (Springer London, Limited, 2002 )
8. Huminic A. and G. Huminic, "CFD Study concerning the Influence of the Underbody Components on Total Drag for a SUV", *SAE International*, 2009.
9. Yunus A. Cengel, John M. Cimbala, *Fluid Mechanics :Fundamentals and Application 3<sup>rd</sup> Edition in SI units* (McGraw Hill Education (India) Private Limited, 2014)
10. Abdollah Khalesi Doost, "Green Nature and Reducing of Air Pollution with Vehicle Drag Coefficient Correction" *Advances in Energy Engineering (AEE) Volume 1 Issue 2, April 2013, pp.28-33*
11. Akash Parab, " Aerodynamic Analysis of a Car Model using Fluent- Ansys 14.5", *International Journal on Recent Technologies in Mechanical and Electrical Engineering (IJRMEE) ISSN: 2349-7947 Volume: 1 Issue: 4,2014, pp.007– 013*
12. M. D Shende, "An Experimental Investigation to Control the Flow Emerging From a Wide Angle Diffuser", *IOSR Journal of Engineering (IOSRJEN) ISSN: 2250-3021 ISBN: 2878-8719 PP 27-32*.
13. S.M. Rakibul Hassan, "Numerical Study on Aerodynamic Drag Reduction of Racing Cars" *Science Direct, Procedia Engineering 90 ( 2014 ) 308 – 313*
14. JESPER MARKLUND, "Under-body and Diffuser Flows of Passenger Vehicles", *Department of Applied Mechanics, Chalmers University of Technology, Sweden, ISBN 978-91-7385-866-3*.
15. AHMED. H & CHACKO, "Computational Optimization of Vehicle Aerodynamics", *Annals of DAAAM for 2012 & Proceedings of the 23rd International DAAAM Symposium, Volume 23, No.1, ISSN 2304-1382*
16. K. Ramesh, " Design and Analysis the Effect of Rear spoiler and Rear diffuser on Aerodynamic Forces using CFD", *International Journal of Research & Engineering, Science & Technology (IJRESTS) ISSN 2454-5392. Volume: 1 Issue:8, pp.29-34*
17. Hu, X., Zhang, R., Ye, J., Yan, X., and Zhao, Z., "Influence of Different Diffuser Angle on Sedan's Aerodynamic Characteristics", *Physics Procedia, Volume 22, 2011, Pages 239-245*.
18. E.M. Wahba, "Aerodynamic Drag Reduction for Ground Vehicles using Lateral Guide Vanes", *CFD Letters Vol. 4(2) 2012 ISSR Journals, PP: 68-79*
19. Kowata, S., Ha, J., Yoshioka, S., Kato, T. and Kohama, Y., "Drag Force Reduction of a Bluff- Body with an Underbody Slant and Rear Flaps", *SAE Technical Paper 2008-01-2599*.
20. Beaudoin, J.-F. and Aider, J.-L., "Drag and lift reduction of a 3D bluff body using flaps", *Exp Fluids (2008) 44:491–501*.
21. Xu-xia Hu, Eric T.T. Wong, "A Numerical Study On Rear-spoiler Of Passenger Vehicle", *World Academy of Science, Engineering and Technology, International Journal of Mechanical, Aerospace, Industrial, Mechatronic and Manufacturing Engineering Vol:5, No:9, 2011*.
22. Cakir Mustafa, "CFD study on aerodynamic effects of a rear wing/spoiler on a passenger vehicle", (2012). Santa Clara University, Mechanical Engineering, Masters Theses. Paper 1.
23. Chien-Hsiung Tsai, Lung-Ming Fu, Chang-Hsien Tai, Yen-Loung Huang, Jik-Chang Leong, " Computational aero-acoustic analysis of a passenger car with a rear spoiler", *Elsevier Inc., Applied Mathematical Modeling 33 (2009) 3661–3673*
24. SUN Qi, CHEN Ling-shan, GU Qing, " Car's Aerodynamic Characteristics at High Speed Influenced by Rear Spoiler", *International Journal of Research in Engineering and Science (IJRES) ISSN (Online):2320-9364, Volume 2 Issue, 12 December. 2014, PP.01-04*
25. G.Ganesh, "Analysis of Effects of Rear Spoiler in Automobile Using Ansys", *International Journal of Scientific & Engineering Research, Volume 6, Issue 6, June-2015, ISSN 2229-5518, PP: 762-766*.
26. Dileep P. Menon, " To Improve the aerodynamic performance of a model hatchback car with the addition of a rear roof spoiler", *16<sup>th</sup> Annual CFD Symposium, August 11-12, 2014, Bangalore*.
27. Vishal Shukl & Gaurav Saxena, "Computational Drag Analysis of Passenger Car Using Splines and Spoiler", *International Journal of Engineering Trends and Technology (IJETT) – Volume 21 Number 1 – March 2015, ISSN: 2231-5381*
28. P Ramya, "Analysis of Flow over Passenger Cars using Computational Fluid Dynamics", *International Journal of Engineering Trends and Technology (IJETT) – Volume 29 Number 4 - November 2015, ISSN: 2231-5381, PP: 170-176*
29. Masaru Koike; Tsunehisa Nagayoshi and Naoki Hamamoto, " Research on Aerodynamic Drag Reduction by Vortex Generators", *Mitsubishi Motors Technical papers, (2004) No. 16,pp: 11-16*
30. Upendra S. Rohatgi, Volodymyr Sirenko, " Methods of Reducing Vehicle Aerodynamic Drag", *Brookhaven National Laboratory, BNL-98039-2012-CP, Presented at the ASME 2012 Summer Heat Transfer Conference Puerto Rico, USA July 8-12, 2012*
31. P.N.Selvaraju, "Analysis of Drag & Lift Performance In Sedan Car Model Using CFD", *Journal of Chemical and Pharmaceutical Sciences, JCHPS Special Issue 7: 2015, ISSN: 0974-2115 PP: 429-435*

**International Journal of Innovative Research in Science, Engineering and Technology**

An ISO 3297: 2007 Certified Organization

Volume 6, Special Issue 1, January 2017

**International Conference on Recent Trends in Engineering and Science (ICRTES 2017)**

20<sup>th</sup>-21<sup>st</sup> January 2017

Organized by

**Research Development Cell, Government College of Engineering, Jalagon (M. S), India**

32. Gopal P. and Senthilkumar T, "Aerodynamic Drag Reduction In A passenger Vehicle Using Vortex Generator With Varying Yaw Angles", *ARPJ Journal of Engineering and Applied Sciences*, VOL. 7, NO. 9, SEPTEMBER 2012, ISSN 1819-6608, PP: 1180-1186
33. Dongri Kim, "A bio-inspired device for drag reduction on a three-dimensional model vehicle", 2016 IOP Publishing Ltd, *Bioinspir. Biomim.* doi:10.1088/1748-3190/11/2/026004
34. Manigandan. R, "CFD Simulation over a Passenger Car for Aerodynamic Drag Reduction", *IJSRD - International Journal for Scientific Research & Development*, Vol. 3, Issue 02, 2015, ISSN (online): 2321-0613, PP:1894-1897
35. G. Siva and V. Loganathan, "Design and Aerodynamic Analysis of a Car to Improve Performance", *Middle-East J. Sci. Res.*, 24 (Recent Innovations in Engineering, Technology, Management & Applications): 2016, ISSN 1990-9233, PP: 133-140.
36. Alamaan Altaf, "Passive drag reduction of square back road vehicles", *Science Direct, J. Wind Eng. Ind. Aerodyn.* 134 (2014) 30-43