A Review on "Reduction of Drag Force using ADD-ON Devices"

Shashank Arya[1], Pankaj Goud[1], Shubham Mathur[1], Vishal Sharma[1], Sourabh Tripathi[1], Vishal Shukla[2]

[1]*Research scholar, Mechanical Engineering, IITM Gwalior*
[2]*Assistant Professor, Dept. of Mechanical Engineering, IITM Gwalior M.P, India*

---***---

Abstract - *Now a days as we see rate of fuel increases tremendously due to shortage of energy resources as a result of which there is a chance of doping of fuel with some unnecessary elements, which influence green house gas emission from vehicle. In order to increase vehicle efficiency there is much pressure over automobile designers to enhance vehicle design, performance by optimizing vehicle shape more aerodynamically. Aerodynamic drag is one of the main obstacles to accelerate a vehicle when it moves in the air. The aerodynamic drags of a road vehicle is responsible for a large part of the vehicle's fuel consumption and contribute up to 50% of the total vehicle fuel consumption at highway speeds. Review on the research to control air flow separation around vehicle using add-on devices to reduce aerodynamic drag is reported in this paper. The review mainly focuses on the methods employed to prevent or delay air flow separation at the rear end of vehicle. This review intends to provide information on the current approaches and their efficiency in reducing pressure drag of ground vehicles. Researches carried out by a number of researchers with regard to control flow separation by modification of external vehicle geometry attaching spoilers, VGs, diffuser, tail-plate, guide vanes etc. Addition of add-on devices significantly improves aerodynamic efficiency of the passenger car. Hence, the drag force can be reduced by using add-on devices on vehicle and fuel economy, stability of a passenger car can be improved.*

***Key Words*: Aerodynamic Drag, Pressure Drag, Spoilers, VGs, Drag coefficient, Diffuser, Passenger car, CFD.**

1.INTRODUCTION *(Size 11 , cambria font)*

Today time is very crucial according to the automobile industry, car designers which designs the car have tremendous pressure to make a car which standing over different aspects such as performance , fuel economical, aerodynamic shape etc. Aerodynamic manufacturers are therefore, looking for new ways and developing new technologies to reduce fuel consumption and improve vehicle efficiency.

In terms of vehicle efficiency, drag is an important factor which is why vehicle aerodynamics is such an active area of research for automobile manufacturers. These researches were done profoundly using wind tunnel testing . With improvement in computer technology, manufacturers are looking toward computational fluid dynamics instead of wind tunnel testing to reduce the testing time and keep the cost of R&D low [1]. A car design can only be acceptable if its form drag reduced. Many researchers have made use of CFD techniques [3-6] to perform numerical simulations related to automobile [2]. In optimization of car aerodynamics, more precisely the reduction of associated drag coefficient (C_D) and lift coefficient (C_L), which is mainly influenced by the exterior profile of car, has been one of the major issues of the automotive research centers all around the world [7]. It has been found that 40% out of total drag force is concentrated at the rear end of the car and any geometry. To control the flow separation at the rear end, various techniques are used to reduce aerodynamic drag . these technique, consisting in modifying the shape of the vehicle or attaching add-on devices to reduce the aerodynamic drag, appears as the easiest to implement but unfortunately it only dedicated for particular application [8].

2. Some Researches- Review

In recent years, a number of researchers have reported the aerodynamic study of vehicle body using CFD. Here we focus on the effect of shape of vehicle body on the performance of vehicle

JOSEPH KATZ [9], the author of book named "Race Car Aerodynamics: Designing for Speed" has explained following theory for aerodynamic of Vehicle body.

2.1 PRESSURE DISTRIBUTION ON AN AUTOMOBILE SHAPE:

When the flow is turn by concave surface, then speed slows down and the pressure increases.

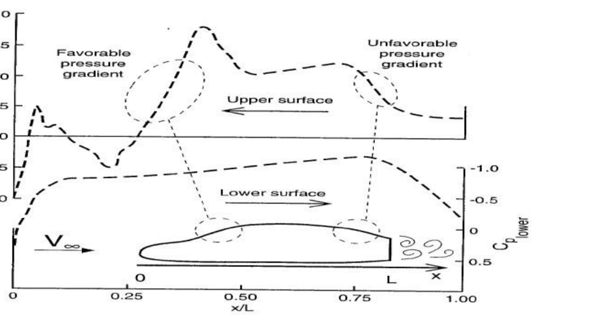

Fig -1: Distribution of pressure coefficient over two dimensional automobile shapes[9]

On the other hand when flow turns around a convex surface, then the speed increases & pressure goes down. Now considering the fig as shown above, We can understand how body shape affects pressure distribution along the center line. The basic features of these pressure distributions are similar to those as shown above. At the front there is the stagnation point & Cp=1.0, since the surface shape is concave. The flow then accelerates over the hood and Cp becomes –ve, since the surface shape is convex. At the roots of the windshield the flow slows down again because of concave shape & the pressure increases. The flow reaccelerates over the top of the vehicle because of convex, where the lowest pressure is observed. Across the back side of the vehicle the whole sequence is reversed. Here, of course the shape of the front is different from the aft portion in this vehicle. However the pressure at the back of the vehicle does not reach the Cp=1.0 as it was in the attached flow case. This is because the flow separates behind the vehicle. In this fig the pressure distribution on the lower side of the vehicle is also shown .herein this case the pressure at the back doesn't recover to the stagnation pressure level. As a result of flow separations, the pressures at the back of the car are lower than at the front, which effect creates drag. We can call this component of the drag force which results from flow separations, the form drag

2.2 FAVORABLE AND UNFAVORABLE PRESSURE DISTRIBUTION:

Based on the pressure distribution we can identify areas on the vehicle where the pressure is decreasing along a streamline. This condition is called favorable pressure distribution. Another area can clearly be seen near the front stagnation point. The opposite pressure distribution near the rear window is called unfavorable pressure distribution since the pressure increases along the streamline.

2.3 WAKES:

The track of disturbed flow left behind a body moving through an otherwise undisturbed fluid is called a wakes. Typical examples for wake flows include the vortex wakes visible behind airplanes flying in humid air, or the dust clouds which continue to roll behind a truck, long after it has passed by.

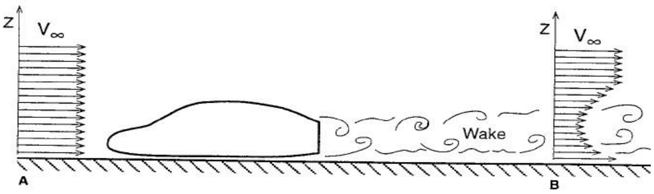

Fig 2: Wake produce at back side of vehicles[9]

This local disturbance in the flow pattern behind the vehicle actually causes a momentum loss (or form drag) which extends far behind the vehicle and is described schematically by fig. Suppose we measure the velocity distribution, at various heights z, in the symmetry plane ahead of the vehicle at point A. Then, if the measurement is taken at a reasonable distance ahead of the vehicle (e.g., more than one car length) the velocity profile indicates a near uniform velocity distribution. Now, if the same measurement is made behind the vehicle, even at a relatively large distance of 10 to 20 body lengths, then a velocity deficiency will be detected, as shown at point B. if the flow separates behind a bluff body, then such a wake will result, and in the wake area the flow seems to be dragging behind the vehicle. the flow inside the wake is moving into the vehicle's direction, another vehicle moving closely behind the first one can use the drafting effects of this separated flow. In many forms of racing those effects are noticeable and drafting behind a lead car is common practice in stock-car racing.

The discussion on the effect of aerodynamics on vehicle performance clearly indicates that the typical objectives of a good aerodynamic design are to reduce drag and to increase the downward force. With these objectives it is very important for one to investigate how some very generic changes in a body's geometry can affect its aerodynamic lift and drag.

The information found on this topic in the open literature can be further divided into two subcategories. The first group identifies typical flow field over generic bodies with quite sharp corners, resembling a variety of road vehicles. The second categories include additional generic shapes, more relevant to race cars, which have the potential to generate down force with reasonably low drag. The following two subsections describe these two groups of generic body shapes.

2.4 Flow Field over Generic Ground

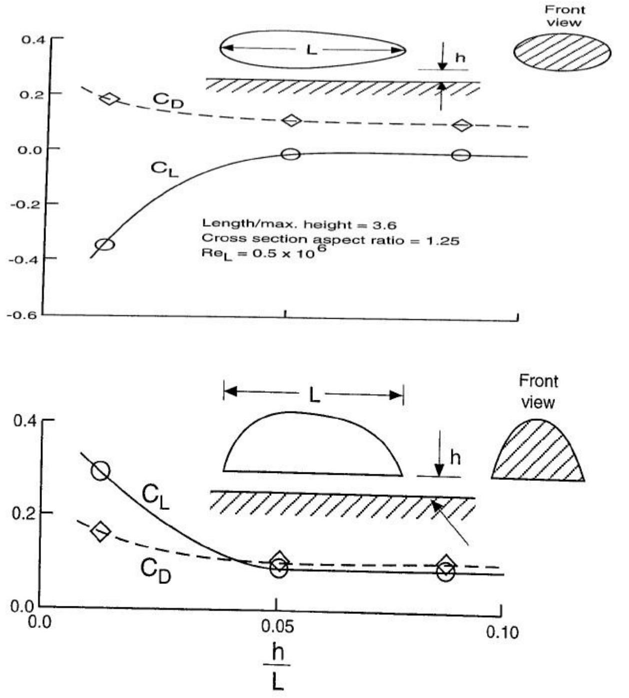

Fig 3: Effect of aerodynamic lift & drag on generic ellipsoids[9]

$$C_D = \frac{D_A}{\frac{1}{2}\rho V^2 A}$$

where D = drag force
A = frontal area of vehicle
ρ = air density(1.202 kg/m³ bosch automotive hand book)
V = velocity of vehicle

Now here in given figure two basic ellipsoidal shapes with dimensions reminiscent of the ratios used on road vehicles are shown. The important conclusion which id to be drawn from this figure is that both positive and negative lift can be generated by bodies when placed close to the ground. Drag however, is primarily a result of the blunt rear end shape, which creates local flow separation as shown in figure. while the first type of design as shown in figure will focus on highly streamlined shapes, with minimum rear end flow separations, flow separation may appear in difficult locations on vehicles with more angular geometries, and vortex dominated flows can exist on a variety of road vehicles.

In order to reintroduce this concept of vortex dominated flows, Let us return to the discussion of a flat plate lifting surface as described below. In this case, a thin, low aspect ratio flat plate is placed in a free stream at an angle of attack larger than ten degree. The main aspect of this flow field, relevant to the present discussion, is the formation of the two concentrated side edge vortices which dominate the nearby flow field. Those two vortices induce a large velocity on the plane, creating strong suction forces which considerably increase the lift of the flat plate wing.

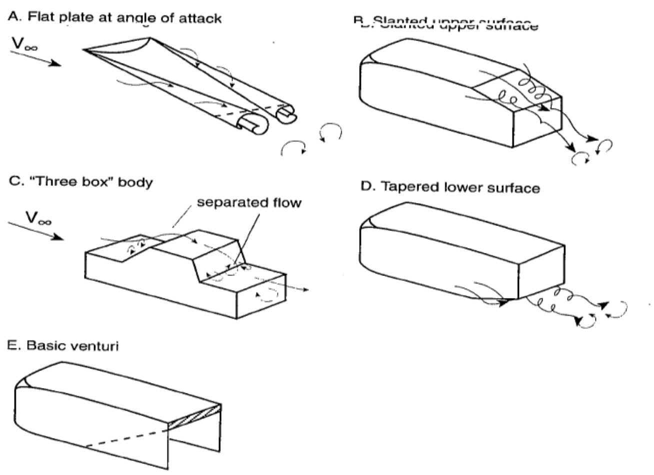

Fig 4: Different shapes of blunt body[9]

Interestingly, a similar situation develops when slanting the rear, upper surface of generic body. This vortex dominated flow is present in a slant angle range of greater than ten degree and less than thirty degree, as indicated by the lift and drag data as shown in region 1.at the larger angle, the flow over the whole rear base area is separated as on a typical bluff body.

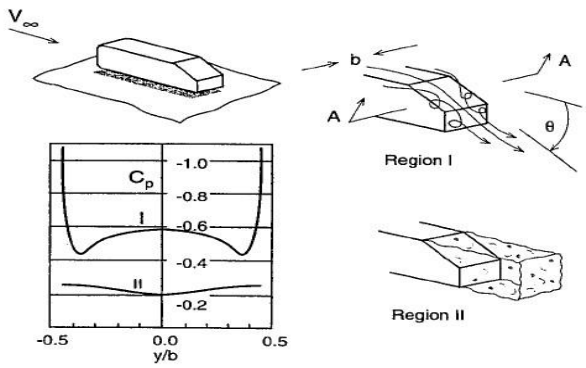

Fig 5: Drag lift & Lateral pressure distribution along the sloping rear end of generic body[9]

This fully separated case is indicated as region 2.the figure as shown above indicates that the corresponding pressure distribution is evenly distributed. In region 1, the two concentrated side vortices attach the flow near the body's longitudinal centerline effectively creating a lifting flow. The pressure distribution for this case shows the large negative pressure peaks created by the vertices at the side of the slanted rear surface resulting in a force acting normal to the slanted surface which can be resolved in to lift & drag.

As the slant angle is increased from zero, a positive lift will developed, which increases up to θ=300.at slant angles larger than 100 the rearward projection of this negative pressure causes quite a large increases in drag as shown in fig. The most interesting feature of this data is that above a critical angle (close to θ=300) the vortex structure breaks down & the drag & lift contribution of the slanted surface is much smaller. This fact has an effect on hatchback automobile design, where rear window inclination angle more then 350 or less then says 250.

Also, note that in this case, the basic body (with θ=00) has negative lift due to ground effect, similar to the case with the ellipsoid as discussed earlier.

3. CFD (Computational Fluid Dynamics)

"CFD (Computational fluid dynamics) is a set of numerical methods applied to obtain approximate solution of problems of fluid dynamics and heat transfer"[10]

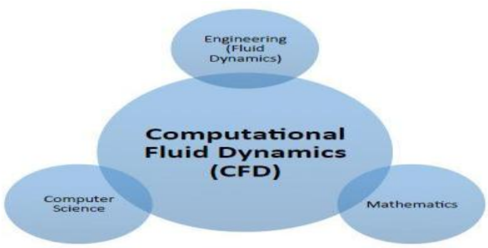

Fig 6: The different disciplines contained within computational fluid dynamics[10]

According to this definition, CFD is not a science by itself but a way to apply methods of one discipline (numerical analysis) to another (heat and mass transfer)[10].Computational fluid dynamics(CFD) has come out as a modern alternative for reducing the use of wind tunnels in automotive engineering. CFD is now being intensively applied to various stages of aerodynamic design of automobiles [11].

4. Drag Reduction Method :

Aerodynamic drag is the main obstacle to accelerate a vehicle when it moves in the air. About 50 to 60% of total fuel energy is lost only to overcome this adverse aerodynamic force. Reduction of aerodynamic drag has become one of the prime concerns in vehicle aerodynamic [12]. this article is concentrated on review of different aspects analysis of aerodynamic drag reduction using spoilers, diffusers, VGs, tail-plates etc of a passenger car under computational fluid Dynamics.

4.1 Drag reduction: using spoiler

The spoiler is used as a tool to minimize unfavorable air movement around the vehicle and can be divided into the front spoilers and the rear spoilers. A front spoiler, connected with the bumper, is mainly used to direct air flow away from the tires to the underbody. A rear spoiler is commonly installed upon the trunk lid of a passenger vehicle. The added spoiler can diffuse the airflow passing a vehicle, which minimizes the turbulence at the rear of the vehicle, adds more downward pressure to the back end and reduces lift acted on the rear trunk [13]. A researcher **vishal shukla et el [14]** researches over passenger car using rear spoiler and splines to reduce drag force. First of all he prepared a baseline model of passenger car without rear spoiler and compared it with that car model attaching rear spoiler on it. In the case of splines and the rear spoiler 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%. Hence drag force & lift force on the passenger car is reduced as proportional to drag coefficient and lift coefficient respectively.

BASELINE MODEL: The base line model of generic passenger car is designed in CATIA. Then after, this model has been analysed for drag coefficient and forces under the HYPERMESH (FLUENT) module and values of drag coefficient, lift coefficient.

SUMMARY: This report summarizes the results of an external aerodynamic CFD analysis performed by Altair's Virtual Wind Tunnel, leveraging AcuSolve's CFD technology. The first section provides a brief overview of the run and its results.

Table 1: Problem Information[14]

Simulation Type	"transient"
Element Count	532321
CPU Time	0.029h
Inflow Velocity	30m/s
Drag Coefficient, C_d	0.604
Lift Coefficient, C_l	0.25

DIMENSIONS: This sections contains geometric dimensions related to the wind tunnel and the body.

Table 2: Geometric Dimensions[14]

Wind tunnel, bounding box	[0.000,7.000],[2.000,2.000],[0.000,3.000]
Body, bounding box	[1.661,4.140],[-0.540,0.540].[0.0391.224]
Wind tunnel dimension	7.000m x 4.000m x 3.000m
Body dimension	2.478m x 1.080m x 1.185m
Frontal ref area, Aref	1.0363m^2
Blockage ratio	8.63583333333
Distance inflow- body	1.661m

Boundary Conditions and Solution Strategy : In this section the boundary conditions and the setup for the CFD run are listed.

Table 3: Boundary conditions[14]

Inflow velocity	30m/s
Outflow	Pressure outlet
Slip walls	Top, right, left faces of wind tunnel
No- slip walls	Wind-tunnel ground, body, wheels, heat exchange

RESULTS: In this section the results of the CFD run are reported.

Table 4: Coefficients[14]

Surface	Drag coefficient	Lift coefficent	Cross coefficent
Part 1	0.60381	0.25075	0.00867
Total	0.6038082846	0.250752907	0.008668037

PASSENGER CAR WITH SPLINES AND REAR SPOILER:

The model of generic passenger car is designed in CATIA. Then after, this model has been analysed for drag coefficient and forces under the HYPERMESH (FULENT) module and values of drag coefficient, lift coefficient.

SUMMARY: This report summarizes the results of an external aerodynamic CFD analysis performed by Altair's Virtual Wind Tunnel, leveraging AcuSolve's CFD technology. The first section provides a brief overview of the run and its results.

Table 5: Problem information[14]

Simulation type	"transient"
Element count	1054672
CPU time	0.050 h
Inflow velocity	30 m/s
Drag coefficient, C_d	0.511
Lift coefficient, C_l	0.003

DIMENSIONS: This sections contains geometric dimensions related to the wind tunnel and the body.

Table 6: Geometric Dimensions[14]

Wind tunnel, bounding box	[0.000,7.000],[-2.000,2.000], [0.000,3.000]
Body, bounding box	[1.897,4.376],[-0.540,0.540], [0.102,1.298]
Wind tunnel dimension	7.000m x 4.000m x 3.000m
Body dimension	2.478m x 1.080m x 1.196m
Frontal ref area, aref	1.0462 m^2
Blockage ratio	13.0775
Distance inflow- body	1.897m

Boundary Conditions and Solution Strategy : In this section the boundary conditions and the setup for the CFD run are listed.

Table 7: Boundary conditions[14]

Inflow velocity	30 m/s
Outflow	Pressure outlet
Slip walls	Top, right, left faces of wind tunnel
No-slip walls	Wind tunnel ground, body, wheels, heat exchange

RESULTS: In this section the results of the CFD run are reported

Table 8: Coefficients[14]

surface	Drag coefficient	Lift coefficient	Cross coefficient
Part 1	0.51125	0.22700	0.00328
Total	0.5112506609	0.22700205839	0.003279146

Table 9: Drag and Lift coefficient of baseline Passenger car model with a model fitted with Splines and Rear Spoiler[14]

configuration	drag coefficient	%reduction from base model	Lift coefficient	%reduction from base madel
Basemodel	0.60	0	0.25	0
Splines and rear spoiler	0.51	15%	0.22	12%

4.2 Drag Reduction : Using diffuser

A diffuser, in an automotive context, is an arc shaped section of the car underbody. The diffuser improves the car's aerodynamic properties by enhancing the transition between the high-velocity airflow underneath the car and the much slower free stream airflow of the ambient atmosphere. It works by providing a space for the underbody airflow to decelerate and expand so that it does not cause excessive flow separation and drag, by providing a degree of wake infill or more accurately, pressure recovery. When a diffuser is used, the air flows into the underbody from the front and sides of the car, accelerates and reduces pressure. There is a suction peak at the transition of the flat bottom and diffuser. The diffuser then reduces this high velocity air back to normal velocity and also helps fill in the area behind the car making the whole underbody a more efficient downforce producing device by reducing drag on the car and increasing downforce [15].

Fig.7: Rear Diffuser[15]

In order to reduce the pulling effect created by Low pressure zone at the rear of the car . A unique idea is to slice the rear under body at certain angles as shown in fig. 8(a). which actually directs some flow from the under-body to the low pressure zone. This reduces the effect of vortices and low pressure effect. Another popular way to reduce the rear end separation is to use under-body diffuser as shown in Fig. 8(b) which also adds elegance in aesthetics of the car, but it has less flow area as the rear under-body is not fully sliced out. So less reduction of drag is experienced then similar degree of rear under-body slicing[15]. In Fig.9 a pressure cut plot for base of the car at 50 m/s free steam velocity shows a certain envelope of pressure range of 100800 Pa to 102000 Pa. Another small envelope inside this envelop for β=0, shows further pressure reduction in the ranges of 100700 Pa to100799 Pa. With the increase of slicing angle β, the low pressure area decreases and the coefficient of drag decreases too. As the coefficient of drag obtained of the car from the relation between the drag force and the Reynolds number at the free stream velocity 50 m/s is found to be 0.3233. now the analysis is done over a car with rear under body modification.

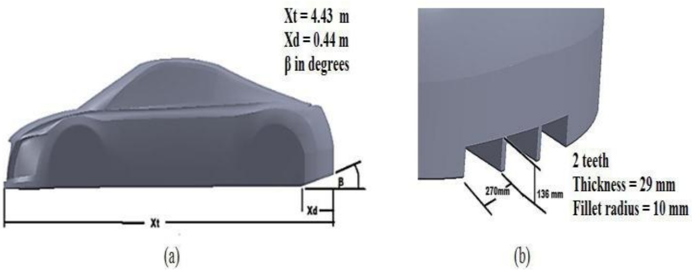

Fig.8.(a)Rear under-body slicing at angle β degree[15]
(b) Rear body diffuser[15]

Table 10 . For rear under-body modification, drag reduction at flow velocity 50 m/s[15]

modifications	Description of modification	C_d	C_d	% of reduction of C_d
None	-	0.3233	-	-
Modification1	$\beta = 2.5^0$	0.3083	0.0150	4.639
Modification2	$\beta = 5^0$	0.2962	0.0271	8.373
Modification3	$\beta = 10.0^0$	0.2694	0.0538	16.58
Modification4	$\beta = 12.5^0$	0.2517	0.0716	22.13
Modification5	$\beta = 12.5^0$	0.2926	0.0307	9.5

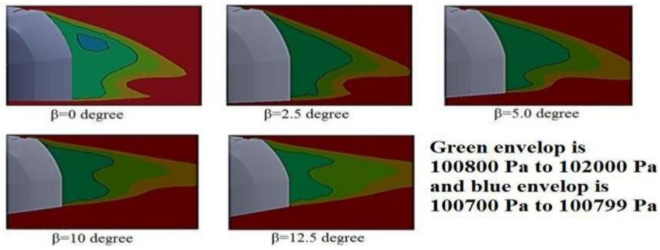

Fig.9. Pressure contour at the rear end (base) of car at different β, showing reduction of low pressure zone (green) with increasing β[15]

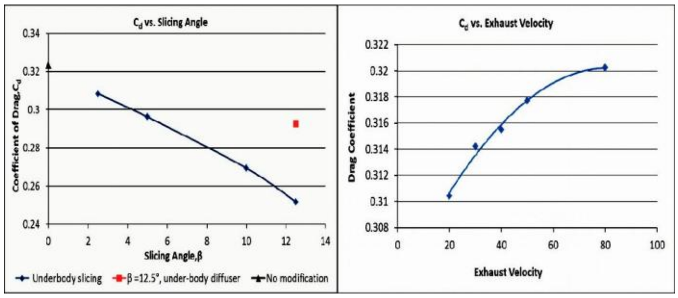

Chart -1: Change in Cd due to different modification [15]
Chart -2: Change in Cd with exhaust gas velocity[15]

As the slicing angle increases in case of under body slicing, more air is permitted to flow to the low pressure region. As a result the size of low pressure zone is reduced as shown in Fig. 8 which indicates the increase in pressure at that region. So normal force along x axis i.e. the drag force decreases and hence the Cd decreases too. From Fig.9 it can be seen that Cd decreases almost linearly with the increase in slicing angle. For under body diffuser, though the drag force decreases, the reduction is not as high as that for same degree of under body slicing. The difference is shown for β= 12.50 in fig.9

Another research scholar using rear diffuser at car ends, **K. Ramesh et el[16]** investigated that the use of rear diffuser at high speed helps to reduce drag force . He also investigated that only at high speed (above 70km/hr) does the rear diffuser device slip out and therefore change the automobile's rear flow pattern. He prepared a base model in CATIA and import it into CFD simulation for found out the influence of Rear diffuser of different lengths. To simulate a passenger car moving on an actual road, The dimension of the computational domain is chosen such that the aerodynamic force is not affected by the domain size.

Boundary condition for CFD analysis

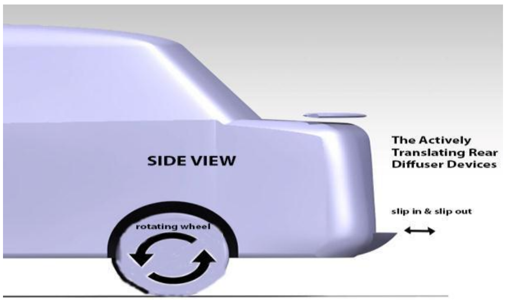

Fig 10: Translating rear diffuser[16]

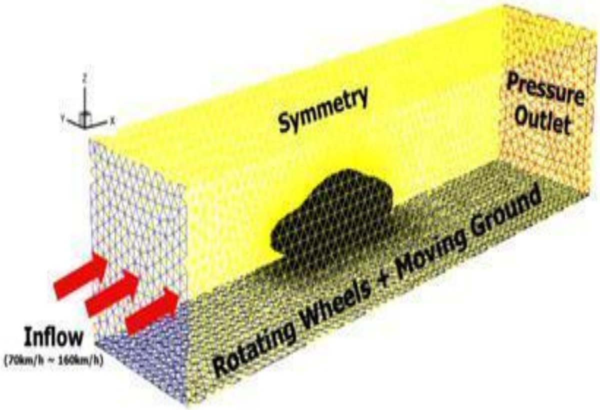

Fig 11: Boundary conditions for the CFD[16]

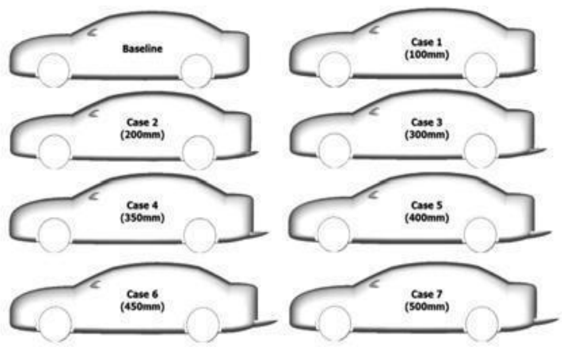

Fig 12: Seven different types of rear diffuser[16]

Seven types of diffuser devic es with differing positions, protrusive length, width and height (but the same basic shape) were constructed for assessment in a CFD analysis.

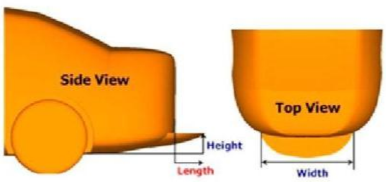

Fig 13: Side view and Top view of car with diffuser[16]

Table 11: Specifications of the seven rear diffusers[16]

Seven cases	Length (mm)	Width (mm)	Height (mm)
Case 1	100	1518.2	144.5
Case 2	200	1318.2	154.5
Case 3	300	1218.2	174.5
Case 4	350	1218.2	194.5
Case 5	400	1218.2	204.5
Case 6	450	1158.2	224.5
Case 7	500	1078.2	244.5

4.2.1 Drag Reduction Mechanism Induced by the Rear Diffuser Device

To investigate why the drag reduction mechanism is induced by the rear diffuser device, the pressure distribution contour of the baseline condition was analyzed. The pressure of the upper flow is the highest, with that of the side flow second and that of the under flow third. the most evident characteristic of the streamline pattern in the baseline condition is the strong up wash flow that soars up the rear side of the trunk. However, the extracted rear diffuser device blocks out the up wash from the bottom. Therefore, the drag reduction phenomenon is explained in terms of a blocking out of the low-pressure air in its position behind the trunk surface, allowing for relatively high-pressure air from the side and the upper position of the trunk. Finally, the base pressure is increased, which results in the reduction of aerodynamic drag.

4.3 Drag Reduction : Using VGs

One of the other method to reduce drag force in car is using vortex generators (VGs) at the outer end surface of roof of car. The main function of vortex generator is to delay the air separation. They are small and fit inside the boundary layer, shaped like triangular fins (see in fig.14) and are especially effective in speeds in excess of 100 kmph [17].

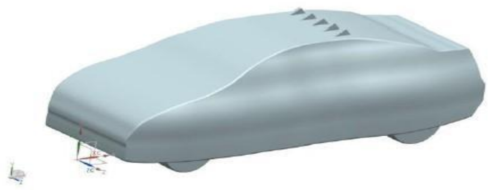

Fig 14: car with VGs[17]

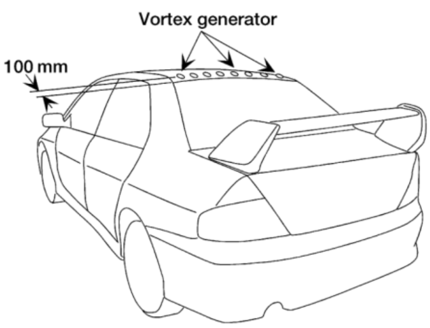

Fig.15: location of vortex generator[18]

Katz et el [19,20], graphically represented his data of work published regarding with incorporation of VGs onto the ends of under body of an Indy car which shows that the lift of the car was very low at low ride heights about and the drag coefficient was reduced from 0.2 to 0.18 with addition of vortex generator. Flow visualizations indicate that with reduced ground clearance the vortex strength increases and the vortices untangle and get closer to the vehicle's surface increasing suction force.

4.4 Drag Reduction : Using Tail-Plates

Ram Bansal et el[21] numerically investigated the use of tail- plates as a drag reducing device for ground vehicle, used baseline model of passenger car with and without tail-plates then simulated in CFD. First of all prepared a baseline model of passenger car without tail-plate in SOLIDWORKS software and this generic model is import into the ANSYS FLUENT to do the simulation of the coefficient of drag and coefficient of lift in the wind tunnel which is generated in the design module of the ANSYS FLUENT.

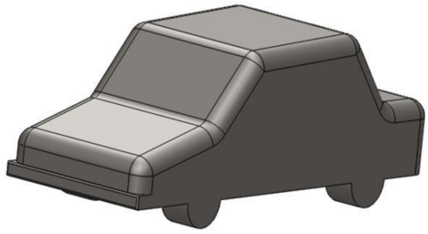

Fig16: solid-work model of car without VGs[21]

After simulation of car in ANSYS FLUENT coefficient of drag obtained was 0.351 and coefficient of lift obtained was 0.231. then next step was put tail- plate at back side of the roof and at the tail bumper of the passenger car at 12^0 inclination angle. The arrangement of them is shows in the figure 17

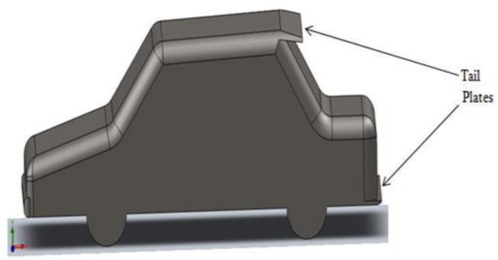

Fig 17: car model with tail-plate[21]

After simulation found that in case of tail plates are applied on the rear bumper and rear side of roof of the base line car respectively with inclination angle 12^0. The coefficient of drag is 0.3376 and the coefficient of lift is 0.1926. The percentage reduction in drag coefficient in comparison of base line car is 3.87% and in coefficient of lift is 16.62%. Hence drag force & lift force on the passenger car is reduced as proportional to drag coefficient and lift coefficient respectively. The comparative results between the baseline car and car with tail-plate are shown in table below:

Table12: comparison of drag & lift coefficient of baseline passenger car model with a model attached with tail plates[21]

configuration	Drag coefficient	%C_d reduction from baseline	lift coeficient	%C_l reduction from baseline
Baseline	0.351	0	0.231	0
Tail plates	0.337	3.87	0.192	16.6

5. CONCLUSIONS

Two experimental approaches for the investigation of external aerodynamics of different car models were presented by different authors. Combining wind tunnel experiments and CFD computation, he has shown how integration of both methods can lead to a better aerodynamic design. For each project, the requirements and the methodology were different, but the final goal was to obtain an optimized aerodynamic design[22]. This paper is mainly focusing on different add-on devices which is used to reduce aerodynamic drag. Many researchers used different drag reduction devices to overcome the problem of drag force which resist car motion and enhance vehicle shape for better fuel efficiency. When We use Rear spoiler as device at the trunk of vehicle Drag reduced to 15% and lift reduced to 12% , drag coefficient and lift reduced from 0.60 to 0.51 and 0.25 to 0.21 respectively. When we use diffuser at different angles 2.5,5,10,12.5 degrees % drag reduction was 4.6,8.3,16.58,22.13% respectively. while using VGs as devices does not make much effect on drag force, but when we use tail-plates as devices drag reduction was 3.87% and lift reduction was 16.6% very effectively. So the use of devices depend upon vehicle geometry, design etc to make car more efficient and more aerodynamically.

ACKNOWLEDGEMENT

We would like to express our special thanks of gratitude to our guide Asst. Prof. Vishal Shukla for helping me throughout.

REFERENCES

[1] Akshay Parab, Ammar Sakarwala, Bhushan Paste, Vaibhav Patil, " Aerodynamic analysis of a car model using Fluent- Ansys 14.5," International Journal of Recent Technologies in Mechanical and Electrical Enginering (IJRMEE) Volume:1 Issue:4.

[2] R. B. Sharma, Ram Bansal, "Aerodynamic Drag Reduction Of a Passenger Car using Spoiler with VGs." International Journal of Engineering Research and Applications (IJERA) ISSN: 2248-9622.

[3] Gilhaus, R. Hoffmann, Directional Stability, Aerodynamics of Road Vehicles, in: W.H. Hucho (Ed.), SAE International, Warrendale, PA, 1998.

[4] J.R. Callister, A.R George, Wind Noise , Aerodynamic of Road Vehicle, in: W.H.Hucho (Ed), SAE International, Warrendale, PA, 1998.

[5] F.R. Bailey, H.D Simon, Future Direction in Computing and CFD, AIAA paper 92-2734, 1992.,

[6] H. Taeyoung, V. Sumantran, C. Harris, T. Kuzmanov, M. Huebler, T. Zak, Flow Field Simulations of three Simplified Vehicle shapes and comparisons with experimental measurements, SAE Transactions 106(1996) 820-835

[7] Jaspinder Singh, Jagjit Singh Randhawa, "CFD Analysis of Aerodynamic Drag Reduction of Automobile Car- A Review", International Journal of Science And Research(IJSR), ISSN(online): 2319-7064, Impact Factor (2012): 3.358

[8] Mohd Nizam Sudin, Mohd Azman Abdullah, Shamsul Anuar Shamsuddin, Faiz Redza Ramli, Musthafah Mohd Tahir," Review of Research on Vehicle Aerodynamic Drag Reduction Methods", International Jouranal of Mechanical & Mechatronics Engineering IJMME-IJENS VOL:14 NO:02.

[9] JOSEPH KATZ, "Race Car Aerodynamics: Designing for Speed" (Ch-2: Aerodynamic Forces and Terms (Page No:40- 44) & Ch- 6 :Aerodynamic of Complete Vehicle(Page No:179-185)".

[10] V. Naveen Kumar, K. Lalit Narayan, L. N. V. Narasimha Rao and Y. Sri Ram,"Investigation of Drag And Lift Forces over the Profile of Car with Rear spoiler using CFD".

[11] Kobayashi, T. and Kitoh, K., "A Review of CFD Methods and Their Application to Automobile Aerodynamics," SAE Technical Paper 920338, 1992, doi:10.4271/920338.

[12] S.M. Rakibul Hassan, Toukir Islam, Mohammad Ali, Md. Quamrul Islam, "Numerical Study on Aerodynamic Drag Reduction of Racing Cars", 10th International Conference on Mechanical Engineering, ICME 2013.

[13] Mohd Nizam Sudin, Mohd Azman Abdullah, Shamsul Anuar Shamsuddin, Faiz Redza Ramli, Musthafah Mohd Tahir." Review of Research on Vehicle Aerodynamic Drag Reduction Methods", International Jouranal of Mechanical & Mechatronics Engineering IJMME-IJENS VOL:14 NO:02.

[14] Vishal Shukla, 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.

[15] Harish Kulkarni, Omprakash.D.Hebbal, "CFD Analysis of Rear Diffuser in a Sedan Vehicle to Reduce Drag Force", IJSRD - International Journal for Scientific Research & Development| Vol. 3, Issue 06, 2015 | ISSN (online): 2321-0613.

[16] K.Ramesh, "Design and Analysis the Effect of Rear spoiler and Rear diffuser on Aerodynamic Forces using CFD", International journal of Research in Engineering, Science And Technology(IJRESTs)(2016) VOl:1 No:8

[17] Asif Ahmed, Murtaza M A, "CFD Analysis of car Body Aerodynamic including effect of Passive Flow Devices – A Review", IJRET: International Journal of Research in Engineering and Technology eISSN: 2319-1163 | pISSN: 2321-7308.

[18] Masaru KOIKE ,Tsunehisa NAGAYOSHI, Naoki HAMAMOTO, "Research on Aerodynamic Drag Reduction by Vortex Generator".

[19] Katz J, Garcia D. 2002, Aerodynamic effects of Indy car components, SAE J. PassengerCars: Mech. Syst., pp. 2322-30. SAE 2002-01-3311.

[20] Katz J, Garcia D, Sluder R. 2004, Aerodynamics of race car liftoff, SAE 2004-01-3506.

[21] R. B. Sharma, Ram Bansal, "CFD Simulation for Flow over Passenger Car Using Tail Plates for Aerodynamic Drag Reduction", IOSR Journal of Mechanical and Civil Engineering (IOSR-JMCE) Volume 7, Issue 5 (Jul. - Aug. 2013), PP 28-35.

[22] Hardik Panchal, Krishna Kumar, Raybahadursinh chauhan, "A Review on Aerodynamic Study of Vehicle Body using CFD", Conference Paper · April 2014.