Aerodynamics: Basic Concept and Rear End Application in Small Vehicles – A Review

Dr. Pravin M. Ghanegaonkar[1], Hrushikesh M. Hiwase[2]
1. Professor in Mechanical Engg.Deptt., Pad.Dr.D.Y.Patil Institute of Engg. & Tech., Pimpri, Pune, 2.Workshop Superintendent, PICT College of Engg. , Pune

***Abstract* – The basic concept of Boundary Layer and its understanding on flat plate is important as prerequisite for the study of aerodynamics of small vehicles. The concept of separation of flow plays vital role in designing small vehicles. The small vehicle design is based on the concept of streamline bodies and needs clear understanding. The effects of Wake formation is also studied in detail.**

***Keywords* - Vehicle, Aerodynamic, Drag, Ahmed body, Spoilers**

I. INTRODUCTION

The subject 'Aerodynamics' relates to the study of relative flow of air past an automobile or any other object of interest like aircraft, train, buildings etc. 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 situation. There are three components in modern aerodynamic studies. They are experimental, theoretical (analytical or semi analytical) and computational fluid dynamics (CFD) approaches respectively. Each approach has its own advantages and disadvantages. Usually the most effective approach is amalgamate both experimental and theoretical/CFD investigation in a most rational manner to solve a particular problem.

The aerodynamic drag of a vehicle is among the most crucial parameters in new car development. This drag has been decreased over the years by more than 33%, but beyond that further improvement has become difficult and challenging for car manufacturers."[7]

The rear-end shape of a car is one of the most crucial parts from the aerodynamics point of view. It governs aerodynamic characteristics of the car, especially drag and rear lift. In this paper, the review of the relation of rear-ends shape to drag and lift is studied by consideration of the wake structure. Finally, new rear shape which reduces rear lift without increasing drag and front lift is discussed."[11]

II. BOUNDARY-LAYER SEPARATION

Separation of flow is said to occur when the direction of the flow velocity near the surface is opposed to the direction of the free stream velocity, which means (du/dy) ≤ 0. Such a situation arises when there is a negative pressure gradient near the wall (Refer Fig.1). An example is subsonic diffuser or rear portion of the car. In the direction of flow the pressure increases which causes decreases in velocity and creates condition for separation to occur.

It is important to note that, the skin-friction coefficient reaches the value zero. At this point the wall shear stress vanishes is called the separation point. Reverse flow takes place beyond point of separation. Due to reverse flow, irregular eddies are formed and lot of energy is dissipated. This region is called 'Wake'. The pressure distribution in the wake is quite different from that on the remaining boundary and this gives rise to an additional drag force on the body. Wake also leads to lateral vibration to the object which may be harmful. Also large amount of K.E. is lost.

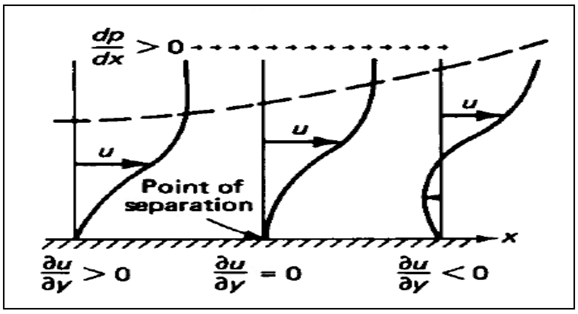

Fig. 1 – Boundary-Layer Separation[7]

In the turbulent boundary layer, some of the energy is dissipated in friction, slowing airflow velocity, resulting in a pressure increase. If the increase in pressure is gradual, the process of turbulent mixing will cause a transfer of energy from the fast moving eddies to slower ones in the turbulent boundary layer. If the rate of change in pressure is too great, for example in sharp corners, the mixing process will be too slow to push the slower air molecules moving. When this happens, the boundary layer flow stops following the contours of the surface, resulting in separation. Air particles downstream of the separation region will then move towards the lower pressure region in the reverse direction to the main flow, the separation region will reattach. In the region between separation and reattachment points, air flow is circulating and this is called the 'separation bubble'.

Separation will normally occur if the resultant flow encounters a sharp edge and that is why it is always important for ground vehicles to have smoothly rounded edges everywhere. Each type of separation can form a separation-bubble zone either by reattaching itself downstream to the flow or being transmitted into a wake, where the separation bubble re-circulates frequently. Hucho [5] named this frequent circulation as “dead water” zone.

Flow separations that lead to a pressure drag can be divided in two different groups, according to Hucho [5]. If the separation line is located perpendicular to the flow direction as shown in Fig. 2, the vortices generated will have the axis perpendicular to the outer flow and parallel to the line of separation. Fig. 3 shows that a symmetrical flow exists only for low Reynolds number. For larger Reynolds number, periodic vortex shedding occurs, and the flow in the separated region is unsteady. The kinetic energy of the vortex field is rapidly dissipated by the turbulent mixing and irreversibly converted into frictional heat [5], and it leads to considerable total pressure loss in the region behind the body and the corresponding deficit in kinetic energy is equal to the work needed to overcome the pressure drag. Behind the body a wake is formed in which, time averaged, relatively uniform suction and very low flow velocities are present.

The second type of flow separation is characterized by separation line inclined with respect to the flow as shown in Fig. 3, the vortex generated have axis nearly parallel to the line of separation with vortex shedding [5]. In this case a well-ordered steady three dimensional flow separation is found and on the rearward surface of the body and the separated flow induces suction which leads to pressure drag. On the inclined surface the flow is attached and behind the body only relatively small total pressure losses are observed. The flow field of the concentrated vortices, however, contains a lot of kinetic energy which corresponds to the work necessary to overcome pressure drag.

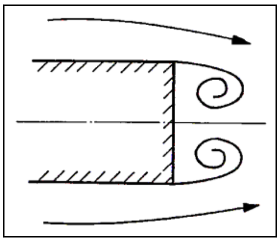

Fig.2 Flow Direction

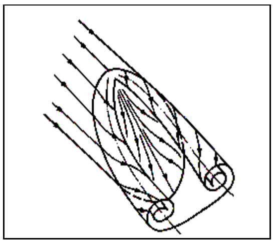

Fig.3 Flow Direction

A) Centerline Flow Field of a Typical Passenger Car

- The centerline flow field around a passenger car is characterized by separations and reattachments.
- The flow character can be directly related to the pressure gradients

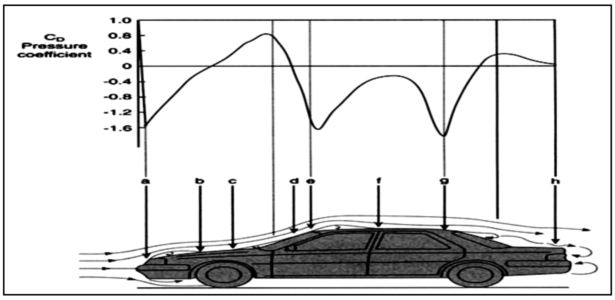

Fig. 4 a) Separation above grille, b) Reattachment on hood, c) Separation in front of windshield, d) Reattachment to top of windshield, e) Separation at roof corner, f) Reattachment downstream on roof, g) Separation at end of roof line, h) Unsteady attachment/separation over, i) trunk

III. AERODYNAMICS OF SMALL VEHICLE

Whenever a solid body is placed in a moving viscous fluid, the body will experience a net force ‘F’. The magnitude of this force depends on many factors – certainly the relative velocity ‘V’, but also the body shape and size, and fluid properties (ρ, μ, etc.). As fluid flows around the body, it will generate surface stress on each element of the surface, and it is leads to the net force. The surface stresses are composed of tangential stresses due to viscous action and normal stresses due to the local pressure. These forces are quite difficult to evaluate except for simple body shape. The flow separation plays very important role while evaluation this drag. Flow separation causes a wake, which not only creates a low-pressure region usually leading to large drag on the body, but radically changes the overall flow field and hence the in-viscid flow region and pressure distribution on the body. For these reasons, we usually resort to experimental method to determine the net force for most body shapes (although CFD approaches are improving rapidly). Traditionally the net force ‘F’ is resolved into the drag force, F_D, defined as the force component in the direction of motion, and the lift force, F_L, (if it exists for a body), defined as the component of the force perpendicular to the direction of motion. The lift force F_L, is generated due to the occurrence of pressure difference between top and bottom of car. The velocity below the car is comparatively more than the velocity of air above the roof of car, which generates downward force on the car. More the clearance of car floor from the ground reduces downward force on the car [12].

The power loss due to the aerodynamic drag depends on the speed of the vehicle. The power required to overcome the drag becomes significant approximately for speeds above 50km/hr. The split of the co-efficient of drag (i.e. front end, front windshield, hood, rear end, windshield rear, rear roof, and cowl) has been shown in Table 1. which is calculated by numerous amounts of tests and numerical calculations. Thus, for various body styles, we can predict the drag depending on the body shape. As seen in Fig. 5 the drag coefficient for car has decreased continuously over the years.

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 3 Issue 7, July - 2014

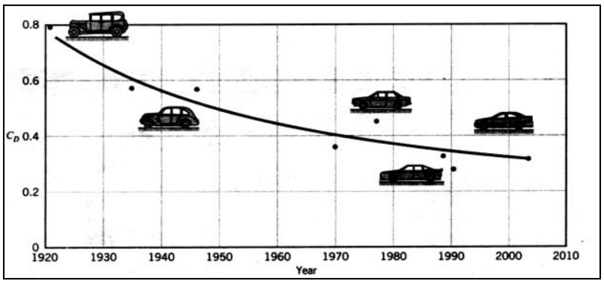

Fig. 5: The historic trend of streamlining automobiles to reduce their aerodynamic drag and increase their fuel economy. [12]

IV. IMPROVEMENT OF AERODYNAMIC PERFORMANCE OF CAR

The flow over an automobile can be macroscopically seen as shown in Fig. 6. The performance of the car can be judged with following aspects:-

- Reduction of Fuel consumption.
- Minimizing the drag and lift by studying the aerodynamics of the vehicle
- Stability and good maneuverability of the vehicle in various conditions
- Studying the flow over various vehicle components like wheels, rear and frontal windshields as well as cooling of the passenger compartment.

Lot of importance is given to aerodynamic design of the car shapes. The study of flow characteristics on the operation of vehicles becomes very relevant with this contest. The effect of C_D on speed of the car is shown in Fig. 4. It is very clear that, lot of fuel saving can be done by improving coefficient of drag [9].

The influence of flow characteristics over various portion of car plays vital role to determine C_D. Over the period lot of changes were made on the body of the car and have been examined for the performance by numerical analysis as well as full-size wind tunnel test.

Table 1. - Components of the drag of a brick shaped bluff body [8]

Drag type	% of C_D	Caused by	Way of reduction and measures
Forebody drag	65	Overpressure on the front face	Reduction of overpressure by accelerating the flow: rounding up of upper horizontal and vertical leading edges, slanting the front face.
Base drag	34.9	Depression on the rear end	Increase of pressure: boat-tailing, tapering the rear part of the body, rounding up of trailing edges.
Side wall, roof and under body drag	0.1	Shear stresses over the walls, roof and under body	Decrease of shear stresses: reduction of roughness, decreases of the velocity in the under body gap.

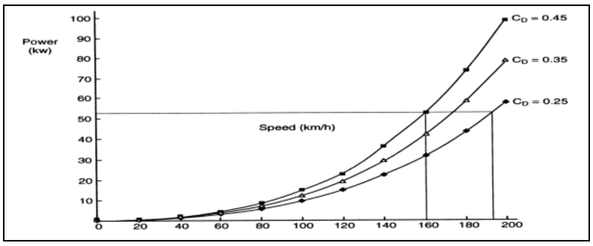

Fig. 6 – Influence of drag on velocity of car[9]

V. ANALYSIS OF REAR END AND WAKE REGION:

A) Comparative Study of Rear End Drags of Notchback, Fastback and Semi-Fastback Models

Notchback Fastback Semi-Fastback

Fig. 7: Various rear end profiles

Fig. 7. shows different types of rear end profiles of passenger cars. Santer[1] have conducted various tests to compare the C_D in all the models. "In the range of Backlight angles around 60° from vertical (backlight is defined as the rear windshield), the backlight treatments showed extremely high drag and lift levels, even with detail tuning treatment at either the leading or trailing edges of the backlight surface. The best drag levels were obtained using the backlight surfaces which were more nearly horizontal, and used deck edged treatment approximation what is usually reffered to as a 'deckspoiler'. Comparison of the the above profiles showed that for minimum drag, the best choice was the semi-fastback profile followed by the fastback and the normal notchback"[1]

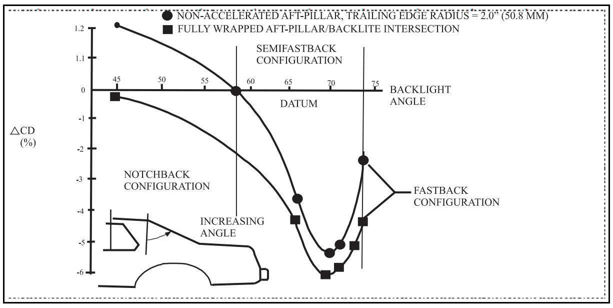

Fig. 8: Generalized effect of notchback, semi-fasback backlight angle on relative C_D levels for non accelerated and accelerated backlight planview tapers.[1]

In the graphs seen above, the variation of C_D is shown relative to the datum (58° backlight angle) with variation of backlight angle. Bifurcations are made depending on the backlight angle by dividing cars into Notchback, Semi-Fastback and Fastback(without deck). From the graph, the lowest value of C_D is seen in the Semi-Fastback configuration.

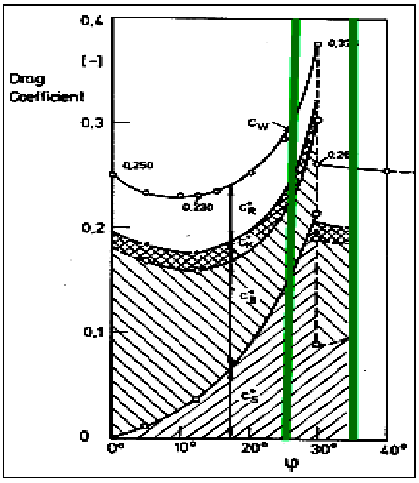

Fig.9. Characteristic drag coefficients for the Ahmed body for various slant angles, after
Ahmed et.al [3]

From Fig. 9, it can be seen that the drag value reached a maximum when the rear slant angle was 30°. Also, it was observed that as the base slant was varied slightly (i.e. beyond 30°) the drag coefficients dropped significantly and remained steady thereafter. Hence, flow visualization studies were performed by Ahmed et.al in order to analyse these influences of rear slant angle on drag coefficient. It was observed that for base slant of P 30°, the shear layers that begin from the rear slant rolled up into strong three dimensional C-pillar vortices. Also, these shear layers after separating at the leading edge of the rear slant, reattached at the lower end of the rear slant, before reaching the base. Thus, the flow created a separation bubble between the separation and reattachment point on the rear slant surface. The flow on reaching the bottom edge of the rear slant gets separated from the top and bottom edges into two separate recirculatory flow regions A and B, one above the other and in opposing directions, called the quasi static two dimensional vortices, as shown in Fig.10.

Ahmed et.al. [4] proposed the time-averaged flow structures, based on the surface flow patterns shown in fig.10. It was found that the longitudinal C-pillar vortices and the upper two dimensional vortex A was highly influenced by the rear slant angle; the lower vortex B depends on the under body flow and the ground clearance of the model. Fig.10. High Drag and Low drag regimes for 30° Slant fast back proposed by Ahmed et.al. [4]. At 30° rear slant angle, the separation bubble E was observed to reach its maximum size, thus forming the horseshoe vortex on the slanted surface. This in turn increased the strength of the C-pilar vortices and promoted the flow to reattach before separating at the base into two quasi-static two dimensional vortices. Additionally, the separation bubble which was a region of flow reversals reduced the pressure on the base slant and ultimately increased the pressure drag of the model. Based on the skin friction patterns performed for 12.5°, 25° and 30° configuration of Ahmed model.

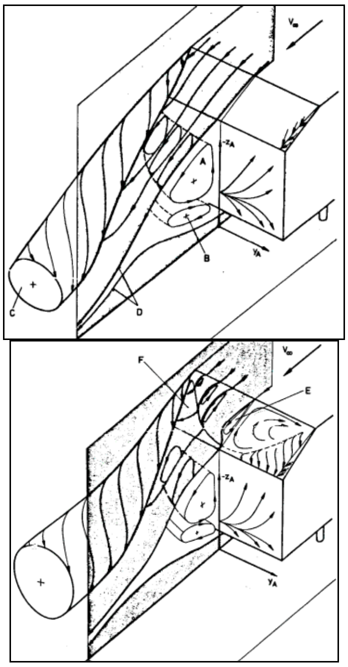

Fig.10High Drag and Low drag regimes for 30° Slant fast back proposed by Ahmed et.al.[4]

Ahmed et.al. [4], it can be seen that, the flow separation at the leading edge of the slanted surface exists even for angles lower than 30°. The patterns clearly exhibit the presence of C-pillar vortices and the horse shoe vortex on the slant surface. For rear slant angles greater than 30°, the flow that separated at the roof backlight junction would no longer reattach at the base of the backlight (i.e. flow region E) and functions as part of flow region B (see Fig.10). Thus, the separation bubble on the rear slant is broken and the C-pillar vortices lose their strength because of insufficient supply of flow from the sides of the model, therefore resulted in improved pressure recovery and reduced the drag coefficients.

Vino et.al.[10] examined the critical rear slant angle (30°) of Ahmed model geometry experimentally, to study the time averaged and time dependant nature of the wake flows at critical rear slant angle. His time averaged results showed excellent agreement with Ahmed et.al. [4], as the surface skin clearly indicated the formation of C-pillar vortices, which was observed identically by both of the previous authors.

Rohatagi [13] observed that use of rear screen allows a reduction in aerodynamic drag of the vehicle model under consideration by up to ~6.5%.Efficiency of rear screens from point of view of drag reducing equally depends on configuration, dimensions and arrangement of screens as well as on model's rear part configuration. It is also observed that rear fairing as part of flow separation area behind the vehicle can reduce aerodynamic drag of the vehicle with investigated configuration, by up to 26%.

B) With Deck Model Analysis (Semi Fastback)

In cars with decks, the most important two parameters are H_d (Deck leading edge height above bumper and H_b (Backlight header height above bumper) as shown in Fig. 11. Deck height ratio (H_R)is defined as the ratio of H_d and H_b. H_R=(H_d/

H_b). Variation of this Deck Height Ratio with the backlight angle were studied by Santer[1]

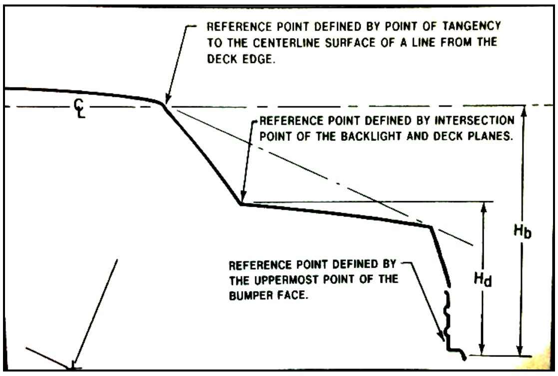

Fig. 11

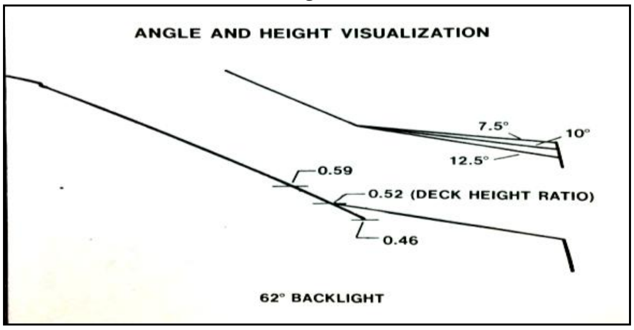

Fig. 12

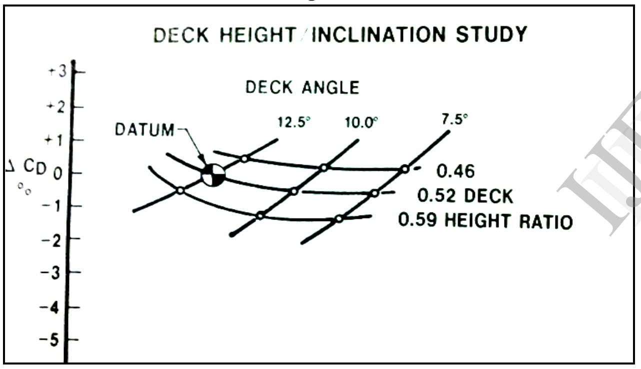

Fig. 13

Fig. 11, 12, 13: C_D as a function of deck angle and height for a 62° condition [1]

Various experiments were done on the model to plot the following data in Fig.13. The graph is plotted at various H_R and Deck angles to see the change in C_D)[1]

The results showed that C_D reductions of almost 1.3% could be achieved by just playing with the parameters of deck angle and Height ratio. So, this method is quite important in automobile aerodynamic drag reduction. This method is very effective in the design but can disturb the conventional aesthetics of the automobile.

Studies were also caried out with spoilers and how they would affect the drag and lift of the rear end. The aerodynamics of rear shapes and their effect on the C_D and lift C_{RL} was studied by H Fukuda et al.[6].

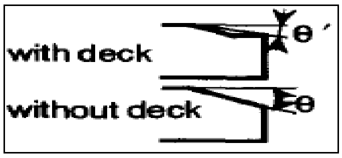

Fig. 14 Rear end of car with and without deck[6]

Experimental study in the wind tunnel shows that with the change in the rear angle (with and without deck), it is found that at a certain optimum angle; minimum value of C_{RL} is obtained whereas the other coefficients remain almost constant. This reduction in C_{RL} is is observed due to increase the static pressure on the rear inclined surface.

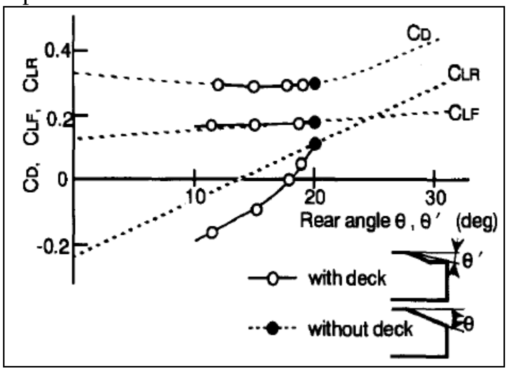

Fig. 15 - Effect of rear angle on C_D, C_{RL} and C_{FR} (Ahmed Model)[6]

Rear angle variation with and without deck is shown in Fig. 15, where the C_D and C_{LF} don't vary much with the change in the rear angle. However, with decrease in the deck angle(θ') and inclination angle θ (without deck), the C_R value decreases. These results were obtained from wind tunnel testing [6].

Also as seen in the Fig., for decreasing the rear lift, models with a deck are better than without decks.

Before getting into details about the rear end characteristics of the Ahmed Model, we understand the various basic rear end models developed over the years and study their characteristics.

C) Using A Spoiler: Analysis With The Ahmed Model

Spoiler effects: A spoiler is primarily used in cars to reduce the Coefficient of rear lift C_{RL} so that better maneurability can be achieved at high speeds. Using it in cars reduces the C_{RL} and C_D on the inclined surface (backlight) but in turns increases the C_D on the rear surface. This is because it stops the downwash and avoids the formation of spiral vortices on the inclined surface, it increases the vorticity of the the spiral rings on the rear surface. If an equivalent increase in height is obtained by raising the deck, almost the same increase in C_{RL} is obtained but the C_D at the rear surface does not increase much. This is the usual method adopted in passenger cars as shown in Fig. 16. [6]

We see from the pressure distribution that on the inclined part the pressure distribution is better in case of the spoiler. C_{RL} is small. However, the pressure at the rear end drops as compared to the "deck" case which increases the C_D component at the rear end. The spoiler has its own drag which adds on to the total C_D. Thus, the cumulative effect with the spoiler is: To decrease the C_D and C_{RL} on the inclined surface and increase the C_D on the rear surface in addition to the spoiler drag itself. The net result is an increase in C_D. This is shown by the pressure distribution in Fig 17. This is the reason why spoilers are used usually in high speed sport cars where the lift becomes very crucial at high speeds [6]. And raising the deck is preferred in passenger cars because reducing the C_D is

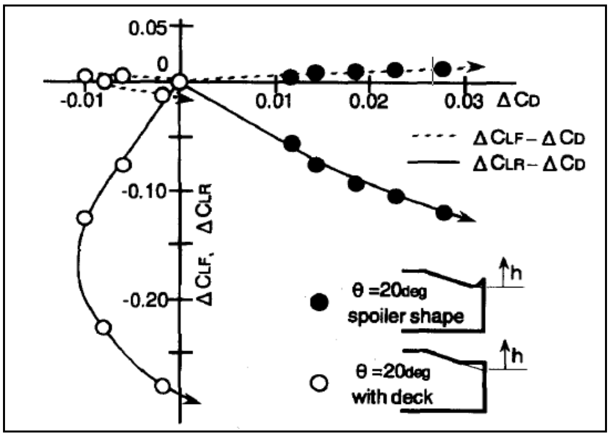

Fig. 16: Effect of deck and spoiler on C_D, C_{RL}, C_{FL} (Ahmed Model)[6]

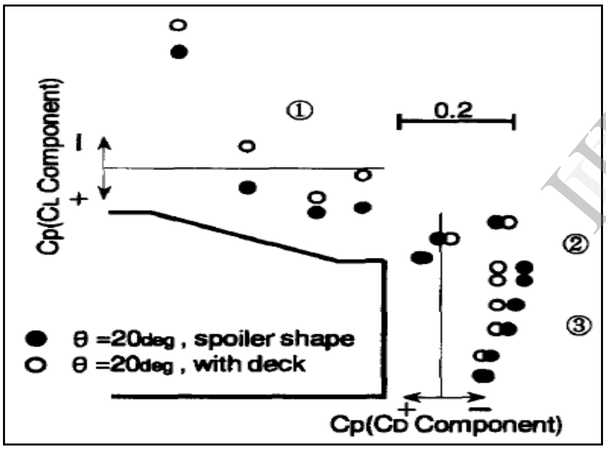

Fig. 17: Pressure distribution on surface(Ahmed Model)[6]

more important as the threat of rear lift is less due to lower speeds.

D) Application of a New Type of Spoiler Shape

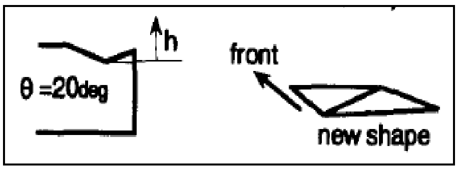

Fig. 18: New spoiler shape development (Ahmed Model)[6]

Based on the various tests carried out in the wind tunnel, a new type of spoiler shape was devised as seen in Fig 18. This is a trigonal pyramid and is formed by centrally raising the rear deck [6].

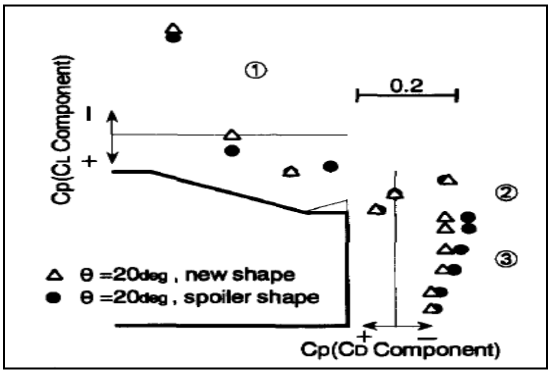

Fig. 19: Pressure distribution on surface(Ahmed Model)[6]

This shape is in many ways better than the conventional spoilers as the downwash from the upper surface is weakened and it also relieves the spiral vortices at the rear surface. The net effect being decrease in the C_{LF} and also a slight restrain on the increase on C_D. The resulting pressure distribution comparing the original spoiler shape and the new trigonal shape is shown in Fig. 19.

This is a shape that can be more easily used in passenger (low speed) automobiles where reduction of drag is more crucial as compared to rear lift.[6]

Thus, the spoilers still fail to reduce the aerodynamic drag as effectively as they reduce the Rear Lift. So, their application has still been more or less limited to the high speed sport car segment.

CONCLUSIONS

- The shape of an automobile has improved over the years but still there are various phenomena about the wake region which are not completely understood and we still have to depend on experiments and hit and trial methods to make any considerable improvements.
- The basic effects of spoilers are studied in this paper. Spoilers are used mainly in sports cars or high speed cars and their ***major function is to reduce the lift***. ***Spoilers should be studied to reduce Drag in the rear region rather than to just reduce the rear lift.*** If this happens, we can have various spoiler designs even in the common passenger cars which will not disturb the aesthetics of the vehicle but will produce the same effect as obtained by raising the deck height, or can get rid of the small vortex generators as been used by some current passenger cars. This is an area where a lot of work can be done.
- The phenomena of turbulence can be better understood. CFD analysis can improve in that direction. Simulations to solve the Navier Stoke's equations especially in the wake region accurately can be one of the best ways to understand drag and to reduce it. Also with that, better experimental procedures to validate the results should be developed. If the wake region phenomena is understood clearly and can be modified, we can reduce the car body drag by a considerable amount thereby increasing fuel efficiency.

REFERENCES

1. Santer, *"Vehicle Aerodynamics Improvements"* SP 545, Society of Automotive Engineers, 1983
2. Blevins, R. D., *"Applied Fluid Dynamics Handbook"*, Van Nostrand Reinhold, New York, 1984.
3. Ahmed, S.R., *"Influence of Base Slant on the Wake Structure and Drag of Road Vehicles"*, Transactions of the ASME, Journal of fluids engineering, Vol. 105, pp 429-434, 1984
4. Ahmed, S.R., Ramm, G. and Faltin, G., *"Some Salient Features of the Time-Averaged Ground Vehicle Wake"*, SAE Paper 840300, 1984
5. Hucho, W. H., *"Aerodynamic of Road Vehicles"*, Butterworth – Heinemann, 1987.
6. *"Improvement of vehicle aerodynamics by wake control"* Hitoshi Fukuda, Kazuo Yanagimoto, Hiroshi China, Kunio Nakagawa: JSAE Review 16 (1995) 151-155
7. *"Boundary Layer Theory"*, 8th revised and enlarged edition: H. Schlichting, K.Gersten, Springer (India) Private Limited, 2000
8. Tamas Lajos, Budapest University of Technology and Economics, *"PPT Presentation 2002"*.
9. Guido Buresti, University of Pisa (Italy), *"PPT Presentation"*, Place - KTH Stockham, March 2004
10. Vino, G., Watkins, S., Mousley, P., Watmuff, J., Prasad, S., *"The Unsteady Near-Wake of a Simplified Passenger Car"*, 15th Australasian Fluid MechanicsConference, Sydney, 2004.
11. *"Advanced Engineering Fluid Mechanics"*, 2nd Edition: K Murlidhar, G Biswas, Narosa Publication, India, 2005
12. Munson, Young, Okiishi, Huebsch – *"Fundamental of Fluid Mechanics"*, 6th edition, John Wiley & Sons, Inc., UK, 2008
13. Upendra S. Rohatgi "*Methods of Reducing Vehicle Aerodynamic Drag"*Presented at the ASME 2012 Summer Heat Transfer ConferencePuerto Rico, USAJuly 8-12, 2012
14. Some innovative concepts for car drag reduction: *"A parametric analysis of aerodynamic forces on a simplified body"*, Mahmoud Khaled , HichamElHage, FabienHarambat , HassanPeerhossaini, 2012