ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 4, April 2017

Investigation of Drag Force of Car Body by Changing Rear Slant Angle Using Computational Fluid Dynamics Simulation Software

[1] Pavankumar [2] R Sonawane, [3] Dr. M. Chandrasekaran

VEL UNIVERSITY

Abstract: -- **Drag force had significant contribution in the fuel consumption. The main objective of this paper is to determine the aerodynamic drag force for Car body by using computational fluid dynamics simulation software. In this paper the model of Car is created with the help of Computer Aided Design software. The magnitude of drag forces are determined by simulation run for rear slant angle of Car body. Afterward rear slant angle is varied by the span of 10 degrees. This time simulation run is carried out on these CAD car models to predict the drag forces for different velocities like 11.11 m/s, 16.66 m/s and 22.22 m/s. Wind tunnel experimentation is cost associated and time consuming work. This analysis shows that change in rear slant angle of the Car body from 55 to 45 degrees corresponds to reduction in drag force up to 16 %. The reduction in drag will lead to saving in fuel consumption.**

Index Terms— **Computational fluid dynamics, drag force, rear slant angle, simulation, fuel consumption.**

I. INTRODUCTION

Most of power of vehicle is utilized in overcoming the resistance due to drag force. Secondly the fuel consumption is the product of brake specific fuel consumption and power re-queried for the vehicle to overcome the aerodynamic drag [7]. In this paper attempt is made to minimize the drag by changing the rear slant angle of the Car. Also the effect of vehicle velocity on the drag force is also studied. In the aerodynamics of vehicle front half portion as well as rear half portion also has the importance.

In order to perform aerodynamic analysis the original equip-mint manufacturers often simulate wind turbulence with simulation software applications to make the necessary changes to a model design, so aerodynamic improvements are possible before the prototype get manufactured.

• Pavankumar R Son wane is currently pursuing Ph. D degree program in Mechanical Engineering in Veils University, Chennai, India. PH- 09503305104. E-mail: pavankumar.sonawane@dyptc.edu.in

• Dr.M. Chandrasekaran is currently Director, Department of Mechanical Engineering in Vels University, Chhennai, and India.PH-9962506351 E-mail:director.semc@velsuniv.org.

II. LITRATURE SURVEY

Hugo G. Castroa et.al has in his work suggested that, the experimentation or the computational analysis of real vehicle is seems to be complex that is why simplified vehicle geometry called Ahmed body was suggested as a reference model. Thus, with this model as a reference model general flow characteristics can be studied [4].

L. Joseph Bachman ET. Al in his work concluded that reducing aerodynamic drag force and tire rolling resistance result in increases in fuel economy and decreases in NOx releases [8].

Emmanuel Guilmineau had investigated numerically the flow around the Ahmed body for slant angles 25° and 35°. He has observed the two-dimensional behavior of the flow, for the slant angle 35∘, is well predicted, whereas the transition of the wake to a fully three-dimensional, for the slant angle 25°, is not reproduced. Therefore, the flow for the 25° slant angle can be considered an open challenge for turbulence modeling [1].

Mahmoud Khalid et. Al had conducted experiment in which aerodynamic force measurements are taken on a simplified vehicle model. Tests were performed in wind tunnel for changed airflow configurations. In different configurations in which the overall drag coefficient can be reduced by 2%, the aerodynamic cooling drag coefficient is reduced by more than 50% and the lift

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 4, April 2017

coefficient by 5% From the literature survey done, we have identified the research gap. In alomost all paper the Ahmed model is assumed as the refrance model to find the aerodynamic characteristics of vehicle. But in actual practice the geometry of actual car body is far differ in shape as well in size. So in this article efforts are taken to investigate the fluid past characteristics over the approximate car model and to dermine the significance of rear slant angle over the drag force.

III. PROBLEM FORMULATION

The tentative model of car body is created by using CATIA V5 R-17 software. While modeling this Car the streamlines, flow lines, corners, radius, front and back lights, windows and wheels are not taken into the consideration so this model represent only overall dimensions of the Ambassador Diesel DX Car body. The measurements of the Ambassador Car body are taken with measuring tape. The Rear slant angle is measured by taking actual photograph of Car body by digital camera in side view position.In previous study the simplified Ahmed body model was considered to found out aerodynamic characteristics[7].

CFD Fluent 12.0 is used for calculating the drag over the model of Ambassador Diesel DX Car body whose rear slant angle found 55 degrees approx. Meanwhile we want to predict the aerodynamic performance at different vehicle speed so we run simulation by considering vehicle stationary and giving speed to air. For this we considered the Computational flow domain. Previously such computational flow domain was used for the analysis of Ahmed model. We consider the same domain for the purpose of simulation. Here all dimensions are the function of body length L.The width of domain is 2 L, height is as 1.5 L & Length is considered as 10 L. From the entrance of domain Ambassador Car model is place at distance of 2.4 L and it is 6.6 L far away from the rear end of domain. Ground clearance for the Car model is taken as 0.4 Meters.

Air is made to flow with different velocities like 11.11 m/s, 16.66 m/s and 22.22 m/s. The rear slant angle is varied by the span of 10 degrees. Again the simulation run is carried out on different CAD model to predict the drag forces for different velocities like 11.11 m/s, 16.66 m/s and 22.22 m/s.

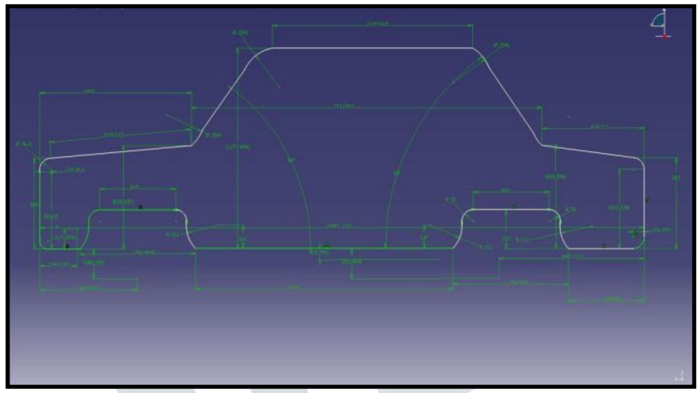


Fig. 1 Dimensions of the Ambassador Car Body [slant Angle 55 degree]

MODEL OF AMBASSADOR CAR BODY FOR SIMULATION WORK

We consider the a flow domain for the purpose of simulation [7]. Here all dimensions are the function of body length L. The width of domain is 2 L, height is as 1.5 L & Length is considered as 10 L. From the entrance of domain Ambassador Car model is placed at distance of 2.4 L and it is 6.6 L far away from the rear end of domain. Ground clearance for the Car model is 0.4 Meters. The Length of Ambassador car measured is 3.986 m.

COMPUTATIONAL FLOW DOMAIN & BOUNDARY CONDITION:

Assumption in CFD geometry: Symmetric model used for calculation due to available computational resources and simplification process.
Input Condition:
Analysis Type: Steady state simulation

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 4, April 2017

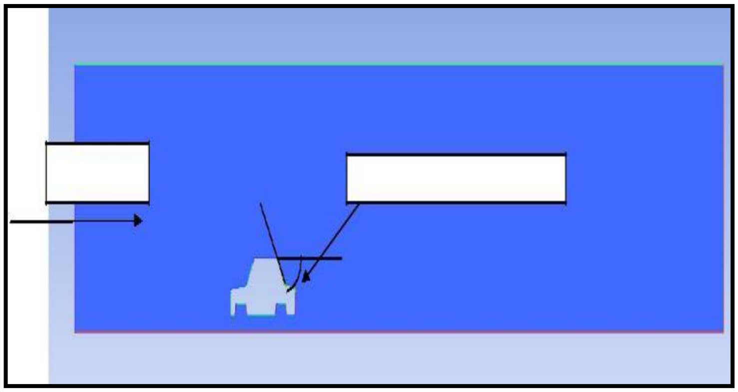


Fig. 2 Computational Flow Domain

Turbulence Model: K-epsilon model
Element Type: Hexa elements at input of Computational flow domain: Air with Velocity as 11.11 m/s, 16.66 m/s and 22.22 m/s.
At output of Computational Flow Domain: Atmospheric Pressure condition
Turbulent Intensity: 5 %
Length scale ratio: 5-10 %
Elements: 3, 00,000 - 5, 00,000.
Air Properties: Standard at 27 degree Celsius

IV. CFD FLUENT 12.0 AS SOLVER

A CFD Fluent 12.0 is used for calculating the drag over the Car body for the slant angle 55 degree. Along with this different contours are plotted are also like

1] Pressure Contours
2] Velocity Contours

This simulation is carried out at different air velocities like 11.11 m/s, 16.66 m/s and 22.22 m/s.

B] In the next step this slant angle is varied from 55 degree to 45 degree with a span of 10. The slant angle plays an important role by decreasing the drag over the body. Different Contours are also evaluated with respect to different slant angles. Again the drag force is determined for three different air velocity like 11.11 m/s, 16.66 m/s and 22.22 m/s.

C] Now the same simulation run is repeated for the changed slant angle of 65 degree. Air velocities are taken same as that of mentioned above.

D] As we get all values of drag forces for Ambassador Carbody and trial Slant angle value 45 & 65 degrees. From this the reduction in power to overcome the drag will be determined.

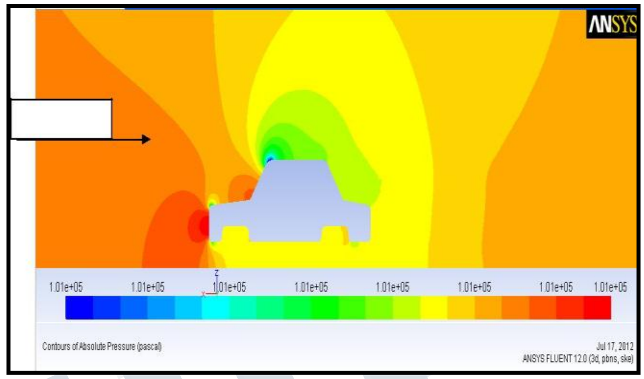


Fig. 3 Pressure contour for 55 degree Slant Angle At velocity 11.11 m/s

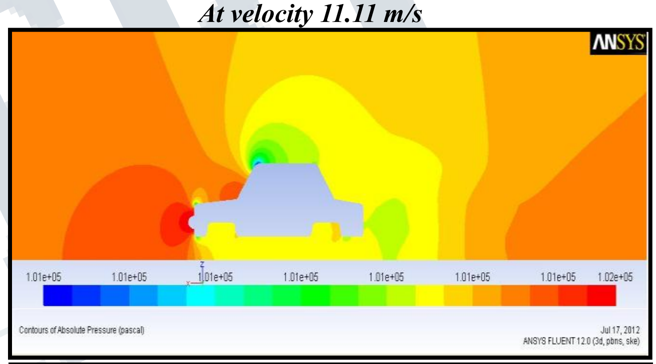


Fig. 4 Pressure contour for 55 degree Slant Angle At velocity 11.11 m/s

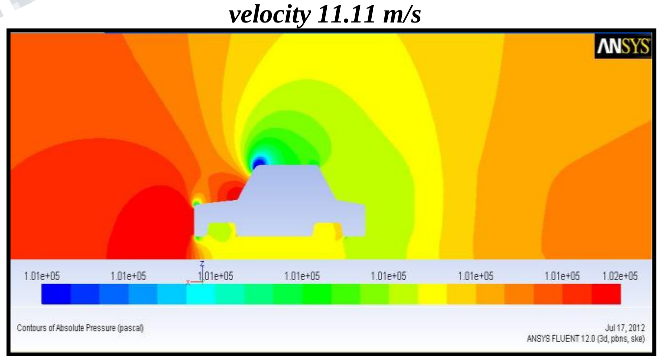


Fig.5 Pressure contour for 55 degree Slant Angle At velocity 22.22 m/s.

IFERP
connecting engineers... developing research
ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 4, April 2017

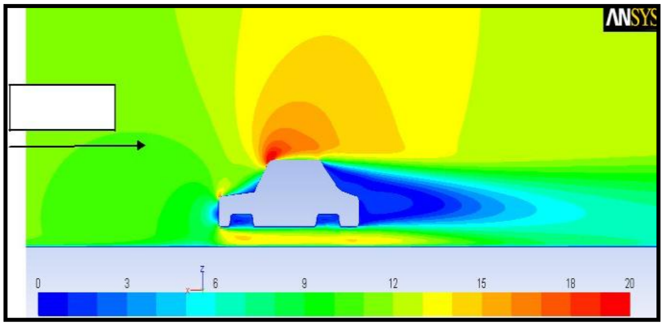


Fig . 6 Velocity contour for 45 degree Slant Angle At Velocity 16.66 m/s

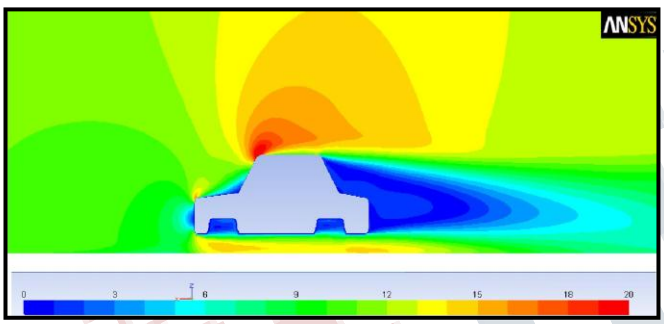


Fig . 7 Velocity contour for 55 degree Slant Angle At Velocity 16.66 m/s

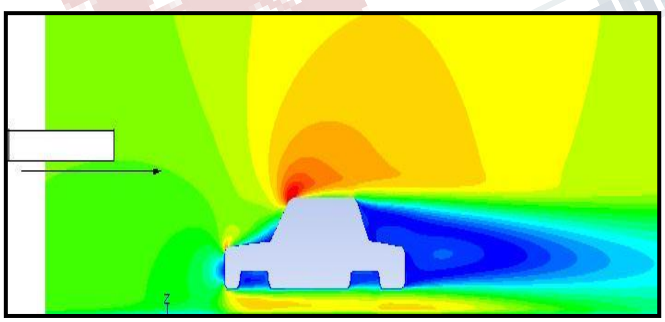


Fig . 8 Velocity contour for 65 degree Slant Angle At Velocity 16.66 m/s

V. RESULT AND DISCUSSION

TABLE 1

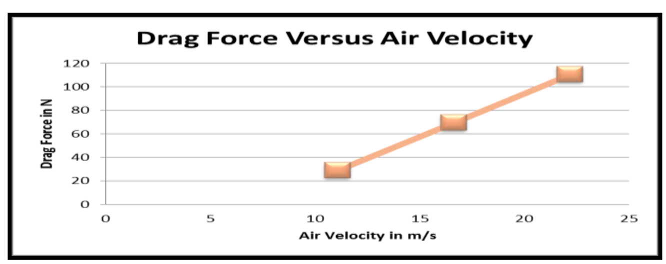
Drag Force Values obtained with simulation software

Air Velocity in m/s	Drag Force in N	Drag Force in N	Drag Force in N
11.11	25.96	29.15	29.16
16.66	58.07	69.68	65.41
22.22	98.71	110.65	114.71
	45	55	65
	Slant Angle in degrees	Slant Angle in degrees	Slant Angle in degrees

Characteristics of flow past over the bodies includes the flow past over the front, side walls and roof,undebody gap and behind the rear wall.As we are changing the rear slant angle of vehicle, because of this flow characteristics at rear wall alters significantly.

Case I [Car body, slant angle 55 degree] – An simulation is carried out for the slant angle of Car body which measures 55 degree. The drag force is observed 29.15 N for the velocity of 11.11 m/s. It reaches up to the 69.67 N for the flow velocity 16.66 m/s. Drag forces have the magnitude of 110.64 N when air flows with velocity of 22.22 m/s.

Fig. 9 Drag force versus velocity for rear slant angle 55 degrees



ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 4, April 2017

The statistics obtained from the Fig. 9 plotted between increase over the drag force with respect to velocity, when the velocity is increased from 11.11 m/s to 16.66 m/s it means 50.07 % of increase in air velocity causes 139 % increase in drag force. Further when we will make the air velocity to flow at 22.22 m/s, it means 33.37 % increase of air velocity causes 58.79 % increase in the drag force**.**

Case II [Car model of slant angle 45 degree] –

As in this case the rear slant angle is changes to 45 degrees and dues to this depression on the rear end is reduces as because of increase in pressure. In overall drag the base drag has also important role to play.

From the Fig. 10 - drag force versus velocity we found the drag force 25.96 N for the air velocity of 11.11 m/s. This magnitude of drag force becomes 58.07 N when air velocity 16.66 m/s is considered. Drag forces reaches up to 98.71 N for air velocity of 22.22 m/s.

4] When we compare the case I and Case II, we can say that there is reduction in magnitude of drag force from 10.79 % to 16.66 % .The maximum reduction is observed is 16.66 % when air flows with velocity of 16.66 %.

Case III [Car Model of slant angle 65 degree] - It is clear from the simulation run that drag force found for the simulation run is 29.16 N for air velocity 11.11 m/s. This force reaches to the 65.41 N for velocity of 16.66 m/s and become 114.71 N at velocity of 22.22 m/s. It has been clear that there is no noteworthy increase is detected in the drag for magnitude.

1] At the velocity 11.11 m/s the drag force almost remain same when compared to case I.

2] At the velocity of 16.66 m/s the drag force is observed 65.41.It is less when compared to case I.

3] At the velocity of 22.22 m/s the magnitude of drag force observed is 114.71 N, so percentage increase is 3.53 % in comparison with case I. And percentage increase is 16.20 % when compared with the Case II.

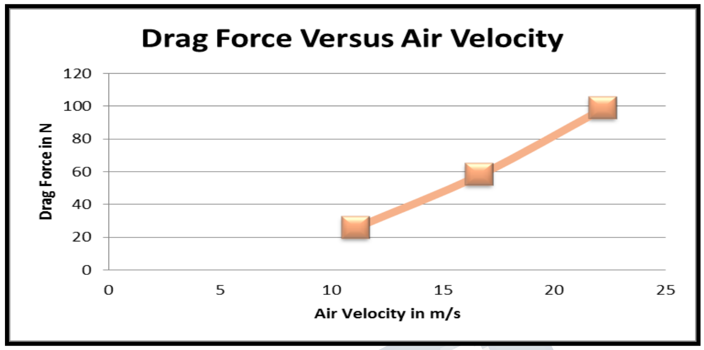


Fig. 10 Drag force versus velocity for rear slant angle 45 degrees

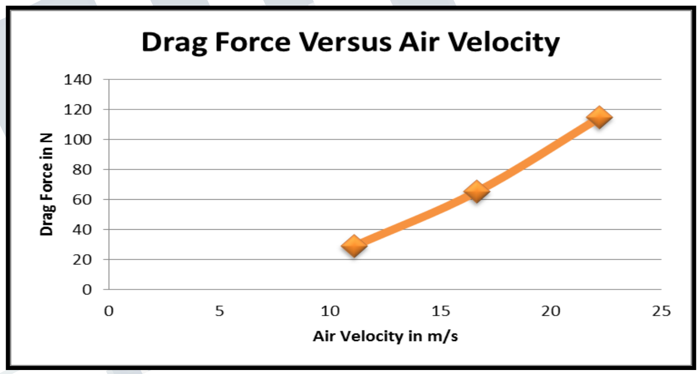


Fig. 11 Drag force versus velocity for rear slant angle 65 degrees

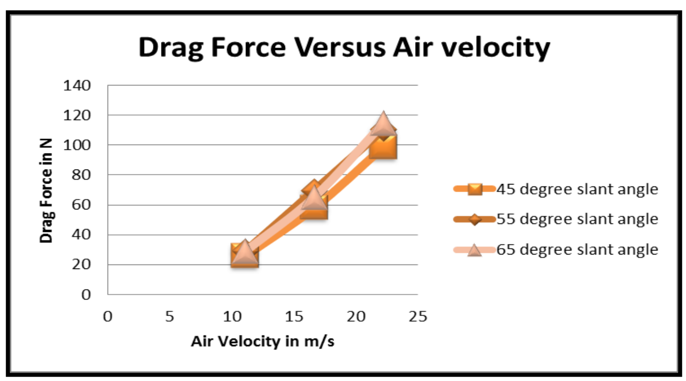


Fig. 12 Drag force versus velocity for rear all slant angles.

ISSN (Online) 2456-1290
International Journal of Engineering Research in Mechanical and Civil Engineering (IJERMCE)
Vol 2, Issue 4, April 2017

From the Fig. 12 - drag force versus velocity at different rear slant angle we can conclude that when slant angle is changed from 55 degree to 45 degree there is maximum decrease in drag force found 16.66 %. At the same time when we change rear slant angle from 55 degree to 65 degree, then the maximum increase observed is 3.53 %. Here we should notice that, as aerodnamic drag is reduced further it causes the reduction in fuel consumption [11].

V.CONCLUSION

Simulation and result analysis clearly indicates that change in rear slant angle of the Car body from 55 degree to 45 degree, there is reduction of aerodnamic drag force up to the 16.66 %. So obviously the power spends by vehicle to overcome the aerodnamic drag is reduced. Further it causes the reduction in fuel consumption.

Here we able to prove the potential of rear slant angle over the flow past characteristic, rear wall. From the aerodynamic analysis of car we concluded that, when the velocity is increased up to 16.66 m/s from 11.11 m/s, it means for 50.07 % of increase in air velocity causes 139 % increase in drag force. When velocity is increased up to 22.22 m/s from 16.66 m/s, it means for 33.37 % increase of air velocity causes 58.79 % increase in the drag force. Fuel consumption is direcly dependent on the drag force resistance acting over the car body.

REFERENCES

[1] Emmanuel Guilmineau,“Computational study of flow around a simplified car body” Journal of Wind Engineering and Industrial Aerodynamics, Volume 96, Issues 6–7, June–July 2008, Pages 1207–1217.

[2] Conan, B., Anthoine, J. & Planquart, “Experimental aerodynamic study of a car-type bluff body” P. Exp Fluids (2011) 50: 1273. doi:10.1007/s00348-010-0992-z, May 2011, Volume 50, Issue 5, pp 1273–1284.

[3] Boris Conan, Jerome Anthoine, Philippe Planquart, “Some innovative concepts for car drag reduction: A parametric analysis of aerodynamic forces on a simplified body” Journal of Wind Engineering and Industrial Aerodynamics, Volumes 107–108, August–September 2012, Pages 36–47.

[4] Hugo G. Castro, Rodrigo R. Paz, Mario A. Storti, Victorio E. Sonzogni, Jorge O. Marighetti and Mario E. De Bortoli, “ Experiment and Numerical Study of the Aerodynamic Behavior of A Simplified Road Vehicle. ” Asociation Argentina,de Mecanica Computational Vol. XXIX, pags.3291-3303 (articulo complete) Eduardo Dvorkin, Marcela Goldschmit, Mario Storti (Eds.) Buenos Aires, Argentina, 15 - 18 November 2010.

[5] Harinaldi, Budiarso, Warjito, Engkos Achmad Kosasih, Rustan Tarakka, Sabar Pangihutan Simanungkalit, and I Gusti Made Fredy Lay Teryanto, “ Modification of Flow Structure over A Van Model by suction flow control to reduce Aerodynamic Drag. ” Makara, Teknologi, Vol. 16, NO. 1, April 2012 : 15 - 21.

[6] Lamond, Andrew and Kennedy, Johnathan J. and Stickland, M. T., “ An investigation into unsteady base bleed for drag reduction in bluff two-box SUVs.” In: 4th European Automotive Simulation Conference, EASC 2009, 6 - 7 July 2009, Munich, Germany.

[7] G. Frank,N. Nigro, M. Storti and J. D Elia, “ Numerical Simulation of the Flow around the Ahmed Vehicle Model. ” Latin American Applied Rsearch, 39 : 295 - 306 2009.

[8] L. Joseph Bachman, Anthony Erb, Cheryl Bynum, Brent Shoffner, Hector De La Fuente, “ Fuel Economy Improvements and NOx Reduction by Reduction of Parasitic Losses: Effect of Engine Design. ” U.S. Environmental Protection Agency ,Southwest Research Institute Carl Ensfield. Paper Number 2006 – 01 - 3474.

[9] Bruce Storms, kambiz Salari, Alex Babb “Fuel Saving & Aerodynamic Drag Reduction from Rail Car Covers.”

[10] Reducing Aerodynamics Drag and Fuel Consumption, Fred Browand Aerospace and Mechanical Engineering, Viterbi School of Engineering University of Southern California for *Global Climate and Energy Project Workshop on Advanced Transportation October 10 - 11, Stanford University.*

[11] Manan Desai, S A Chaniwala and H J Nagarsheth, “ A Comparative Assessment of Two Experimental Methods for Aerodynamic Performance Evaluations of Car ” Journal of Scientific & Industrial Research, Vol. 67 July, 2008.