IMPROVEMENT IN FUEL EFFICIENCY OF BUS USING A ROOF FAIRING

[1]Yuvraj P. Pawar, [2]Swapnil D. Chaware
[1]M. Tech Automotive Engineer, [2]M. Tech Automotive Engineer
[1]School of Mechanical Engineering,
[1]Vellore Institute of Technology, Vellore ,India

Abstract*— Buses plays vital role in mass transportation around globe. As bus body is not designed aerodynamically therefore coefficient of drag is high. Bus fore body and back body are two important areas where you can make changes to reduce the coefficient of drag by changing the wind shield angle, rounding edges or by using the faring's at the top of bus. The work aims to develop a low-cost add-on drag reduction roof fairing which can be used on existing buses. A unique shaped aerodynamic roof faring is designed to reduce drag. Because of its shape it avoids the flow separation at top of body. Star CCM+ is used as virtual analysis tool to estimate various aerodynamic parameters. Analysis was performed on standard Ahmad body and Ahmad body with roof faring; this resulted in the drag reduction by 9.71%. Ratio of the height, width and length of roof faring curve was calculated so that it can be used for any bus.

***Index Terms*— Roof fairing, Aerodynamic, drag reduction, fuel economy, Bus.**

I. INTRODUCTION

There are three main resistances to vehicle motion, rolling resistance, aerodynamic resistance and gradient resistance. Out of which at higher speeds about 100km\hr. aerodynamic resistance is about 65% of total resistance [11]. As roads have been significantly improving from last few years, travelling time is reduced because of high speed travel. An average travel of an intercity bus is about 250~350 km/hr. And fuel consumption rate of these buses is about 2.5~3.5 km/liter at the cruising speed (100km/h) on a highway which is very high. To achieve high fuel efficiency and to keep operating cost low drag of buses should be low. Not only this but also to meet stringent Government Regulations for low emission force manufacturers to design buses with low drag and low fuel consumption rate [5, 7]. Ashok Patidar et. al [4] rounded leading edges and increased curvature of windshield in their simulation. Authors had performed all calculations with STAR-CD v3.10. It was found that coefficient of drag reduced by 30% and approximately 17% of fuel can be saved. Min-Ho Kim et al [1] did a numerical simulation on the drag reduction of large sizes bus using Rear-Spoiler at the end of the upper body. The simulation was carried out using CFD tool Fluent. The simulation resulted in 12% reduction in drag and 0.08% reduction in drag coefficient. Also negative lift force worked in direction of ground enhancing the driving stability at high cruising speed. A. Muthuvel et al [7] compared four different models of a bus which were modified from preceding model considering various aerodynamic aspects. Simulation was carried out using STAR CCM+ tool. Grid independent test is carried out for all the buses and 1.57 million cells were fixed for further analysis. Results showed that The percentage reduction in drag force of Bus No.4 from Bus No.1 is found to be 30%- 34%. The 30-34% deviation in drag force, fuel consumption of about 8% to 23% can be reduced from 80 Km/hr. to 115 Km/hr. Arun Raveendran et al [8] carried out extensive product study, market study, also aspirations and frustrations of travelers were recorded. CFD Simulation was done using ANSYS FLUENT commercial package from various studies and simulation a new design of bus with low floor height and appealing exterior with extemporized aerodynamics was developed. The modification in exterior design resulted in reduction in Coefficient of drag from 0.53 to 0.29 and over all drag reduction by 60%.

II. GEOMETRY

3 D model was created using PTC Creo parametric 3.0.

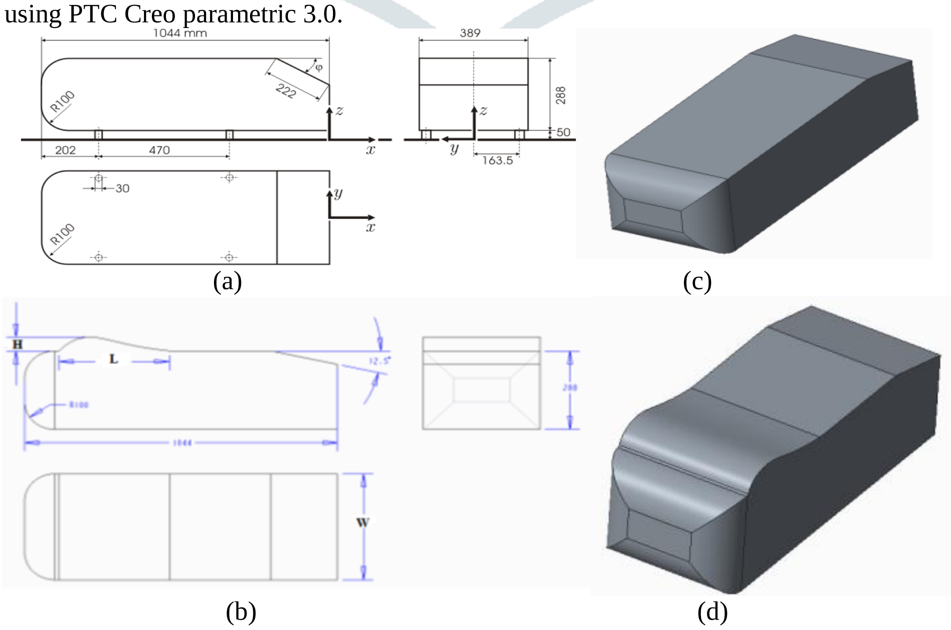


Figure 1: (a) 2D Ahmad body (b) 2D Ahmad body with fairing (c) & (d) 3D Ahmad body model.

III. CFD METHODOLOGY

Air Flow around vehicle body, wake region behind the body is important to know the drag resistance on the vehicle body. This can be done using the CFD simulation techniques. By changing the geometry, flow pattern, flow separation and wake strength of vehicle can be identified and resolve. To do so following CFD methodology is used in present work:

a) Mesh Generation:

A virtual wind tunnel of Width (W), Length (L), and Height (H) was created, which is shown in figure 2. The blockage ratio of 3% is assumed and Reynolds number 2 X10^6 was calculated. The surface mesh is created on the body and a volume mesh is generated between surface of body and wind tunnel domain. 3 different control volumes are considered as show in figure 3. They are considered because finer mesh is required near the vehicle body and there should be smooth transition from finer mesh to coarse mesh. As air flow passes the surface it crates boundary layer. Boundary layer thickness is calculated as 22mm and there are12 prism layers with growth rate of 1.15. Number of cell counts are 1.7 million. The prism layers are shown in figure 4. Polyhedral type mesh is used to generate the volume mesh because of its some advantages like faster converge with less iteration, robust convergence to lower residual values, faster solution runtimes. Mesh independency, when you solve any CFD problem the solution should not be affected by the size of grid. Grid independence is done to ensure that solution is intendant on the grid size. We try to solve problem with fine grid and see the variation in result.

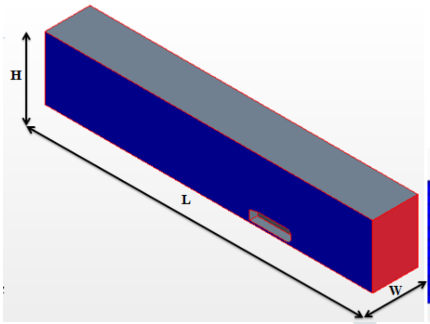


Figure 2.Vertual Wind Tunnel

Figure 3. Mesh with 3 different control voulme

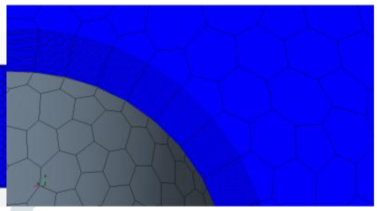


Figure 4. Mesh with Prism layer

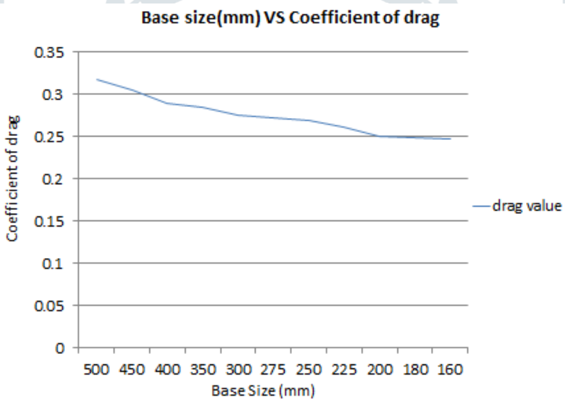


Figure 5: Mesh Independacy

b) Solver Setup

Star CCM+ CFD tool was used for this analysis.Reynold averaged navier stokes equations are used to solve for incompressible turbulent flow .A standard k-ϵ Turbulent model was used as it has advantages like robustness and easy converged. The simulation performed is steady state simulation. The solver type used was segregated flow. For one simulation CPU time Lenovo 2.8GHz windows based is approximately 12 hours. Following table no 1shows the boundary conditions used

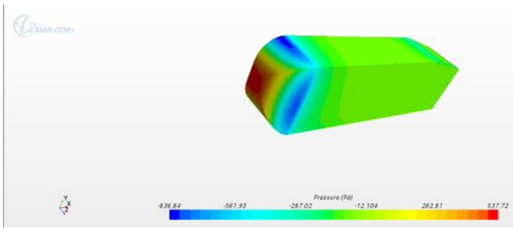
Table no1. Boundary condition

Boundary	**Boundary condition**	**Value**
Inlet	Constant Velocity Inlet, Turbulent viscosity ratio	V=30m/s I=10
Outlet	Pressure Outlet	Constant Pressure 0 Pa
Ahmad body	No slip Stationary Wall	-
Walls	Slip Stationary Wall	-
Symmetry Plane	Symmetry Plane	-

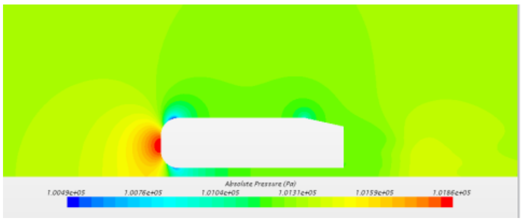
IV. RESULT AND DISCUSSION

a) Ahmad body

In this section the results obtained from the simulation are shown below:



(a)



(b)

Figure 6:(a)&(b)Standard ahmad body Pressure counter

The kinetic energy of air flow over an Ahmad body is converted into the pressure energy. Pressure counter shows the energy losses over body as shown in Figure 6. More resistance is observed over frontal area which offers resistance to motion of vehicle. Rear of body experiences the low pressure. The pressure difference creates the aerodynamic drag which offered resistance to motion of body.

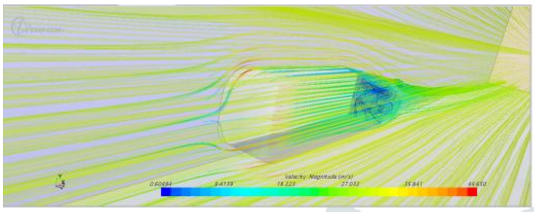


Figure 7:Standard ahmad body streamline

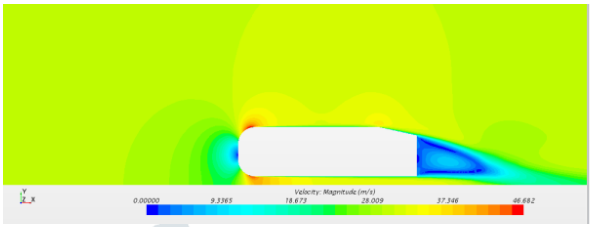
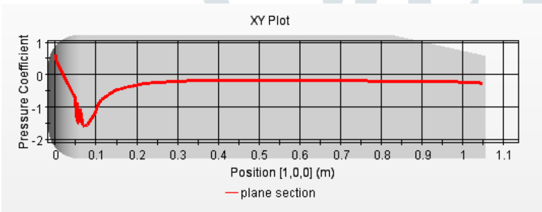
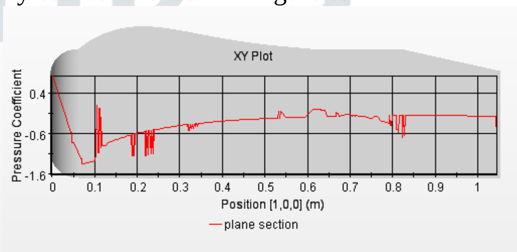


Figure 8:Standard ahmad body velocity counter

Streamlines are shown in the Figure 7. It shows the flow separation and wake region behind the body which creates the low pressure at rear of the body. This produces drag force opposite to vehicle motion. Velocity counter is shown in Figure 8.



(a)



(b)

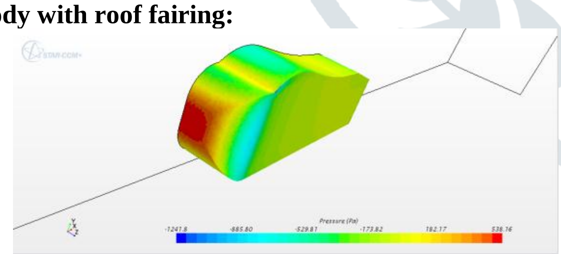
Figure 9: (a) & (b) Pressure coefficient

The pressure coefficient is shown in the Figure 9. The pressure coefficient is a dimensionless number which describes the relative pressures throughout the body length.

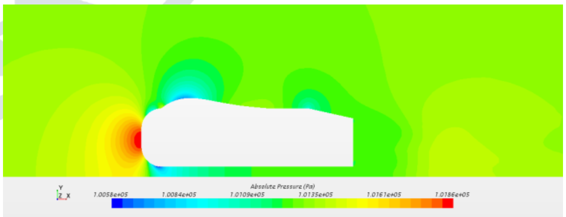
Pressure coefficient= $C_p= (P-P_{ref})/ (1/2\ \rho_{ref} V_{ref}^{2})$

Where, P, $P_{ref}, \rho_{ref}, V_{ref}$ are the working pressure, reference pressure, density, and velocity, respectively

b) Ahmad body with roof fairing:



(a)



(b)

Figure 10: (a)&(b)Standard ahmad bodywith roof fairing Pressure counter

Figure 10. Shows the Pressure counters around Ahmad body with roof faring.

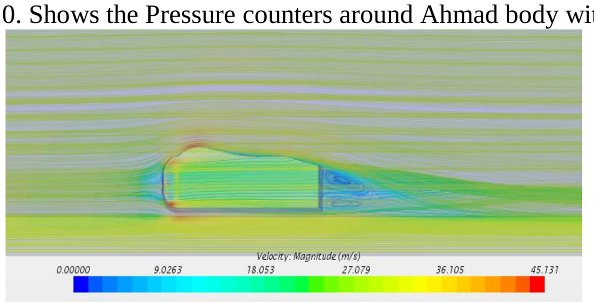


Figure 11: Standard ahmad bodywith roof fairing streamline

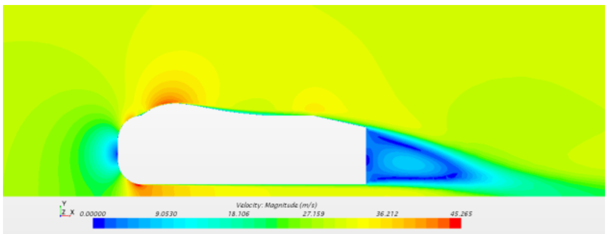
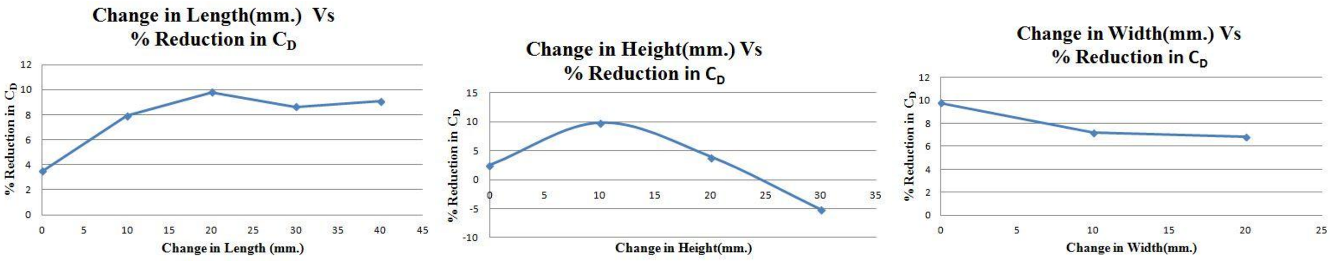


Figure 12:Standard ahmad body velocity counter

Figure 11. Shows the Streamline and Figure 12. Shows the velocity counters. Because of the unique shape of the roof faring the flow separation over the top of Ahmad body shifts toward the rear of the body. Because of which the wake region behind the body is reduced and hence the coefficient of drag is reduced.

V. GRAPHS AND PLOTS



(a) (b) (c)

Figure 13: Maximum %reduction in C_d for perticular (a)Length(b) Height and (c)Width

To find out the optimum shape of the roof faring number of simulations are perform by changing the Length ,Height ,Width of cure at each point as show in above Figure no13.

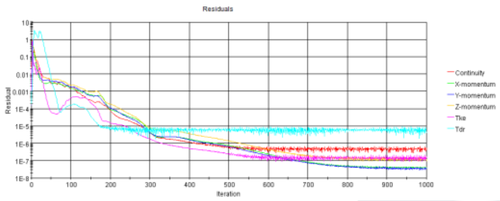


Figure 14: Residual

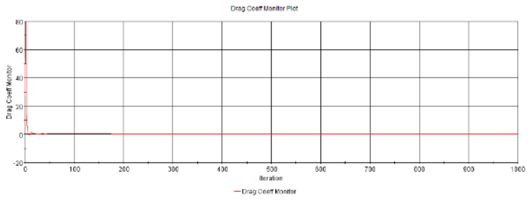


Figure 15: Coefficent of drag

Figure 16: Coefficent of drag

From the analysis it is found that coefficent of drag and coefficent of lift for ahemad body are 0.223and 0.046 respectively and for ahemad body with roof fairing are 0.247 and 0.049 respectively.This shows percentage drag reduction achived with roof faring is 9.71%.

VI. FUEL ECONOMY CALCULATIONS

The fuel economy of a vehicle can be related to the specific fuel consumption by the vehicle engine and the power required for the vehicle to move at a constant velocity U as follows:

$$Power = Drag \times U / 0.65$$

$$= 0.5\, \rho U^2 A\, C_D \times U / 0.65$$

To overcome the aerodynamic resistance at 100kmph about 65 % of power of vehicle is consumed. The specific fuel consumption is defined as the fuel mass consumed by engine to operate at one horsepower for an hour. It is denoted as 'bsfc' and its unit is lbm/(HP-hr). bsfc is expressed in terms of gallons.

$$bsfc_v = bsfc / \rho fuel$$

1 / $bsfc_v$ is the energy produced by the engine per one gallon of fuel and is related to the total power required and the fuel economy in MPG.

$$\frac{Power}{U} x\ MPG = \frac{1}{bsfc_v}$$

MPG is expressed in terms of $bsfc_v$ and C_D :

$$MPG = \frac{1.3}{bsfc_v\ x\ \rho U^2 A C_D}$$

Drag coefficient with and without roof fairing is given by

$$C_{D,Fairing} = (1 - RR) C_{D,without\ fairing}$$

Where, RR is drag reduction and. The value of RR is 9.48%. $bsfc_v$ is fixed for an engine, we can write the following expression

$$MPG_{fairing} = \frac{MPG_{without\ faring}}{1 - RR}$$

Putting the value of RR in the above equation, we get MPG $_{fairing}$ = 1.1047 x MPG $_{without\ faring}$, or a theoretical 11.047% increase in fuel economy.

If we considered 5.88MPG (2.5Kmpl) for bus, then MPG with Roof Fairing is 6.49MPG (2.76Kmpl) there for increase in fuel economy by 0.61MPG (0.26Kmpl).

VII. Conclusion

It is found that lot of work had done to reduce drag of race cars, trucks but very little work had done for buses. Authors would like to conclude that using CFD analysis as design tool at early stage of vehicle body design; tremendous improvement can be done in bus aerodynamics performance, which is still unexplored area. The work uniquely proves the possibility of refining of exterior of a bus for simultaneous improvement in aerodynamics and aesthetics from the point of view of product design. The goal of reducing the aerodynamic drag of bus is a worthy one. It is economically and socially valuable. The development of advanced CFD is also technically useful and will be of benefit in bus. Percentage drag reduction using the faring is 9.71% and hence increase in fuel economy by 0.61MPG (0.267Kmpl).

References

[1] Min-Ho Kim, Jong-YoungKuk and In-Bum Chyun *"A Numerical Simulation on the Drag Reduction of Large-Sized Bus using Rear-Spoiler".* SAE Technical paper series 2002-01-3070.

[2] Marella Vamsi Krishna, C. V. R. "Adjustable Roof Fairing for Truck Aerodynamics",*SAE International (*2011-26-0106).

[3] David Burton, Damien McArthur, John Sheridan, and Mark Thompson,24 September,2013. Contribution of Add-On Components to the Aerodynamic Drag. *SAE International(2013-01-2428).*

[4] Ashok Patidar, Umashanker Gupta, Ankur Bansal *"Fuel Efficiency Improvement of Commercial Vehicle by Investigating Drag Resistance"* SAE Technical Paper 2015-01-2893

[5] Chul-Ho KIM, *"A Streamlined Design of a High-Speed Coach for Fuel Savings and Reduction of Carbon Dioxide"*,International Journal of Automotive Engineering 2 (2011) 101-107

[6] Vignesh T. Shekar, Ayyappan ThuraiMoni, Sreedhar Reddy,Anil Cherukuri, Ashok Leyland, Ltd., *"A Holistic Approach to Aerodynamics of Intercity and Interurban Buses "* SAE Technical Paper 2014-01-0581.

[7] A.Muthuvel, M.K.Murthi, Sachin.N.P, Vinay.M.Koshy, S.Sakthi, E.Selvakuma *"Aerodynamic Exterior Body Design of Bus"* International Journal of Scientific & Engineering Research, Volume 4, Issue 7, July-2013

[8] Arun Raveendran1, D. Rakesh2, S. N. Sridhara . *"Exterior Styling of an Intercity Transport Bus for Improved Aerodynamic Performance"* , SAE Technical Paper 2009-28-0060 09/29/2015.

[9] Mandar Gophane, Gauri Salvi, Gandhi Pratik Pradeep, Dr. Ravi. K. , *"Effect of Aerodynamic Forces over the Bus Body and Design of Conceptual Bus for Enhanced Performance"* International Journal of Engineering Trends and Technology (IJETT) – Volume 11 Number, 4 - May 2014.

[10] Richard M. Wood, Steven X. S. Bauer, *"Simple and Low-Cost Aerodynamic Drag Reduction Devices for Tractor-Trailer Trucks"* SAE Technical Paper, 2015-01-2893, 2003-01-3377.

[11] Wolf-Heinrich Hucho, "Aerodynamics of Road Vehicles" Butterworth-Heinemann 1987.