IMPROVEMENT IN FUEL EFFICIENCY OF BUS USING A ROOF FAIRING

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

Abstract— Buses play a vital role in mass transportation around the 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 fairing 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 fairing; this resulted in the drag reduction by 9.71%. Ratio of the height, width and length of roof fairing 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.

II. GEOMETRY

3D model was created using PTC Creo parametric 3.0.
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 $2 \times 10^5$ 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 are 12 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 intendent on the grid size. We try to solve problem with fine grid and see the variation in result.

![Figure 2. Virtual Wind Tunnel](image)

![Figure 3. Mesh with 3 different control volume](image)

![Figure 4. Mesh with Prism layer](image)

![Figure 5: Mesh Independacy](image)

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 $k-\varepsilon$ 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>
<thead>
<tr>
<th>Boundary</th>
<th>Boundary condition</th>
<th>Value</th>
</tr>
</thead>
<tbody>
<tr>
<td>Inlet</td>
<td>Constant Velocity Inlet, Turbulent viscosity ratio</td>
<td>V=30m/s, I=10</td>
</tr>
<tr>
<td>Outlet</td>
<td>Pressure Outlet</td>
<td>Constant Pressure 0 Pa</td>
</tr>
<tr>
<td>Ahmad body</td>
<td>No slip Stationary Wall</td>
<td>-</td>
</tr>
<tr>
<td>Walls</td>
<td>Slip Stationary Wall</td>
<td>-</td>
</tr>
<tr>
<td>Symmetry Plane</td>
<td>Symmetry Plane</td>
<td>-</td>
</tr>
</tbody>
</table>

IV. RESULT AND DISCUSSION

a) Ahmad body

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

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.

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.

\[
P_c = \frac{P - P_{\text{ref}}}{\frac{1}{2} \rho_{\text{ref}} V_{\text{ref}}^2}
\]

Where, \(P\), \(P_{\text{ref}}\), \(\rho_{\text{ref}}\), \(V_{\text{ref}}\) are the working pressure, reference pressure, density, and velocity, respectively.

b) Ahmad body with roof fairing:

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

Figure 11. Shows the Streamline and Figure 12. Shows the velocity counters. Because of the unique shape of the roof fairing 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

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

To find out the optimum shape of the roof fairing number of simulations are performed by changing the length, height, width of cure at each point as shown in above Figure no13.

Figure 14: Residual

Figure 15: Coefficient of drag

Figure 16: Coefficient of drag

From the analysis it is found that coefficient of drag and coefficient of lift for Ahmed body are 0.223 and 0.046 respectively and for Ahmed body with roof fairing are 0.247 and 0.049 respectively. This shows percentage drag reduction achieved with roof fairing 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:

$$\text{Power} = \text{Drag} \times U / 0.65$$

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

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.

$$\text{bsfc} = \frac{\text{bsfc}}{\rho_{\text{fuel}}}$$

1 / bsfc, 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.

$$\frac{\text{Power}}{U} \times \text{MPG} = \frac{1}{\text{bsfc}_{\text{g}}}$$

MPG is expressed in terms of bsfc, and $C_D$:

$$\text{MPG} = \frac{1.3}{\text{bsfc}_{\text{g}} \times \rho U^2 A \text{C_D}}$$

Drag coefficient with and without roof fairing is given by

$$\text{C_D}_{\text{Fairing}} = (1 - \text{RR}) \text{C_D}_{\text{Without Fairing}}$$

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

$$\text{MPG}_{\text{Fairing}} = \text{MPG}_{\text{Without Fairing}} \times \frac{1}{1 - \text{RR}}$$

Putting the value of RR in the above equation, we get MPG $\text{fairing} = 1.1047 \times \text{MPG}_{\text{without fairing}}$, 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


