Effect of Aerodynamic Forces over the Bus Body and Design of Conceptual Bus for Enhanced Performance

Mandar Gophane[#1], Gauri Salvi[#2], Gandhi Pratik Pradeep[*3], Dr. Ravi. K. [**4]

[#]M. Tech., Automotive Engineering, SMBS, VIT University, Vellore – 632014, Tamil Nadu, India.
[*]M. Tech., CAD/CAM, SMBS, VIT University Chennai Campus, Chennai – 600048, India.
[**]Assistant Professor, SMBS, VIT University, Vellore – 632014, Tamil Nadu, India.

***Abstract*—Buses are preferred as the major mode of mass transportation round the globe. In recent times hike in fuel price is meticulously observed and pertaining to current day's strict government norms, current buses are on track of being much more inefficient in terms of fuel costs. Aerodynamically efficient and accordingly redesigned bus body may improve their fuel economy. Even fuel used being from natural source, saving it will be a boon for mankind. To achieve better economy and good performance, it is necessary to redesign a vehicle with minimum drag resistance. This study relates three redesigned and retouched model of buses among themselves and their analysis for obtaining better aerodynamic design through ANSYS Fluent 14.5 package. CFD's aerodynamic simulations can give the detailed idea of the various parameters for all models which assist in redesigning the bus. This work depicts how the modifications could reduce the drag and the lift forces acting on the vehicle. Study includes the effect of variation of different parameters such as floor panel height, diffuser angle as well as modifying the edges for a bus body over the resisting forces. Attached streamline flow as being ideal flow was obtained for a model. CFD analysis for redesigned models shows remarkable results for the reduction in drag values and lift values. Drastic reduction in the pressure at front and wake region was obtained. Redesigning the overall body shape also improves vehicle stability and handling.**

***Keywords*— Aerodynamics, CFD, Conceptual Bus, Drag Reduction, Pressure co-efficient,**

I. INTRODUCTION

Buses are the major mode of mass transportation all over the globe, despite of the rail network. Buses are inefficient in term of fuel consumption, thus in order to decrease the fuel consumption of vehicles, improvement in the aerodynamics of bus shapes will add to the value. It becomes essential to thoroughly design a vehicle for its aerodynamics, as it directly relates to the fuel economy and resisting forces, which further this, become a parameter for mankind to purchase the vehicle. More precisely the reduction of their drag coefficient becomes one of the main topics of the automotive research. Decreased resistance to forward motion allows higher speeds for the same power output or lower power output for the same speeds.

Aerodynamics being the aid to form a body shape that maximizes the down force, the negative lifts and minimizes the force that opposes the forward movement and the drag forces. The aerodynamically efficient design of the bus reduces the drag force improving the fuel efficiency. In a moving vehicle, the engine power is used to overcome tractive resistance, which is the combination of rolling and aerodynamic resistance. The rolling resistance will be dominant over the aerodynamic resistance at lower speeds. Aerodynamic resistance (drag) amounts for more than three fourth of total engine power while operating at higher speeds, since the drag increases as the square of the speed [1]. Thus the maximum power generated by the engine is utilized to overcome the aerodynamic resistance. Due to this the engine load increases substantially which further raises the fuel consumption rate.

II. METHODOLOGY

Aerodynamic drag is the force that resists the forward movement of a solid object through a fluid, here air. There are two components for the drag force; pressure drag (perpendicular to the surface) and friction drag (along the surface). The aerodynamic drag of any shape is standardized by a dimensionless number called as the drag coefficient or the coefficient of drag (C_d) [2]. Drag force of the moving vehicle is given by,

$$\text{Drag Force (D)} = 0.5*\rho A V^2 C_d$$

Where,
A = Projected Frontal Area
ρ = Density of the Fluid Medium
V = Velocity of Vehicle Relative to the Fluid

From the equation it can be noted that, drag force acting on the vehicle depends mainly on the projected frontal area (A) of the vehicle and co-efficient of drag (C_d). Reduction in these values will directly reduce drag force exerted over the vehicle. But drag force cannot be simply minimized by reducing the frontal projected area or by reducing dynamic pressure because reduction of dynamic pressure will reduce the velocity and will increase the transit time of vehicle which will further lead to slow and uneconomical transportation.

In this work, three models of buses are redesigned and modeled using well known CAD package Pro-E and the standard dimensions of Bus are obtained from urban bus specifications. Basic model is further modified from the aerodynamic perspective and further two retouched and redesigned buses are modeled. This is done for optimization of the contribution of each modification on drag force, lift force and pressure co-efficient. CFD simulation is done for the three models properly.

A. Modifications in Bus

First one is the standard model which has a flat front surface and sharp corners. Hence, it has more resistance to air which raises the drag force. Also it has higher floor panel height of 0.6m from the ground, due to which stability is very low at higher speeds.

The second bus is a designed model with smooth rounded corners and diffuser angle of 10^0 at the rear end. It has comparatively lower floor panel height of 0.5m from the ground which gives better handling and stability to the bus.

The third bus is a conceptual model which is aerodynamically designed with its front surface tapering towards the rear end. It has a larger diffuser angle of 15^0 at rear end and floor panel height is further reduced to 0.4m. Modified shape of the bus successfully helps to achieve attached streamline flow.

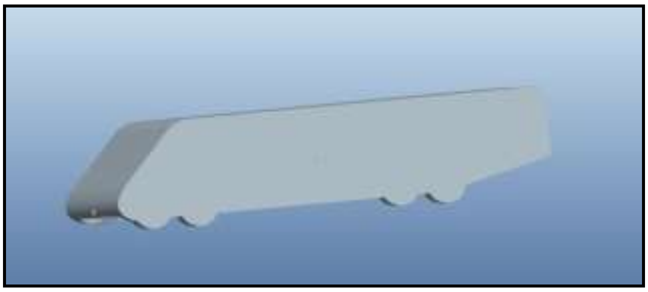

Fig. 1 Solid model of conceptual bus

B. CFD Methodology

CFD solver ANSYS Fluent 14.5 is used for analysis and calculations. K-ε turbulence model is used. The courant number was set to 50 and the relaxation factors were taken as 0.25. For inlet condition the turbulence intensity was set to 1% and turbulent viscosity ratio as 10. For outlet, they were set to 5% and 10 respectively.

C. Meshing

In order to reduce the computational time only half of the bus model is analyzed. This is fair enough because a vehicle has symmetry in vertical plane along its longitudinal axis. Quadrilateral mesh is created in order to obtain an unstructured grid. Program controlled inflation layer is used to capture boundary layer effects close to the body. Minimum element size was 1mm and number of grid elements is around 910931.

Fig. 2 Meshing of the conceptual bus

D. Boundary Conditions for CFD Simulation

Boundary Conditions were applied on meshed models using ANSYS Fluent 14.5 and were analyzed in moving road and rotating wheel conditions [3]. In this simulation, straight wind condition was considered at the vehicle speed of 25m/s. Constant velocity inlet condition and zero gauge pressure at the outlet was applied. Operating condition was set to atmospheric pressure. Blue and red faces indicate velocity inlet and pressure outlet respectively. White represents wall whereas yellow represents symmetry conditions. All the boundary conditions used in the analysis are listed below;

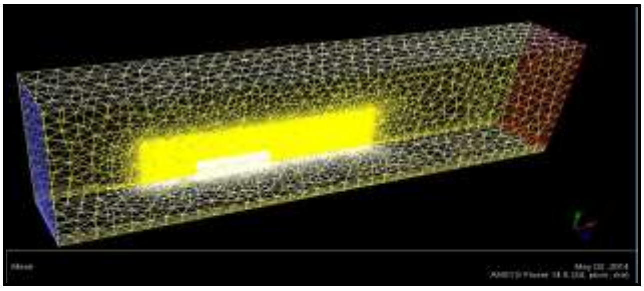

Fig. 3 View after applying boundary conditions

TABLE NO I
LIST OF BOUNDARY CONDITIONS

Boundary	Boundary Type	Values
Inlet	Velocity Inlet	25m/s
Outlet	Pressure Outlet	0 Gauge Pressure
Top	Symmetry	-
Side	Symmetry	-
Bottom	Road	-
Bus Body	-	-

III. RESULTS

CFD analysis of flow over the bus is carried for the speed of 25 m/s for all three models. Results are obtained for five different velocities for both the models and graphs are plotted.

A. Co-efficient of Drag

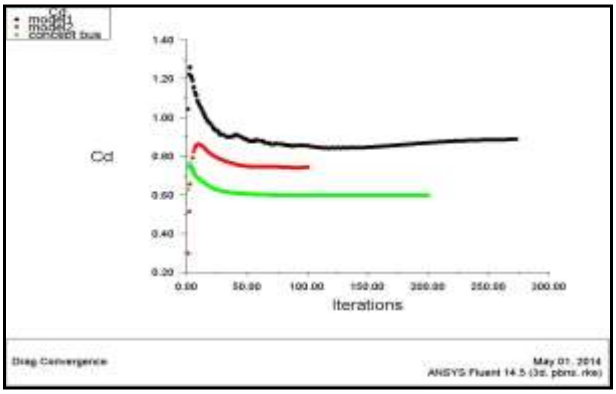

Fig. 4 Comparision of drag coefficient for model 1, model 2 and conceptual bus

Co-efficient of drag always depends on shape of the vehicle body. In this study, shape of the standard model of bus is modified by redesigning the front end of the bus, shaping the corners and providing the diffuser angles. From the above three graphs it can be observed that C_d for the two modified buses, is lower, compared to the standard bus. C_d for the conceptual bus is found to be 0.6 as the front end of the bus is modified such that it tapers downwards the rearend. It also provides the attached flow for the streamline reducing the drag resistance.

B. Co-efficient of Lift

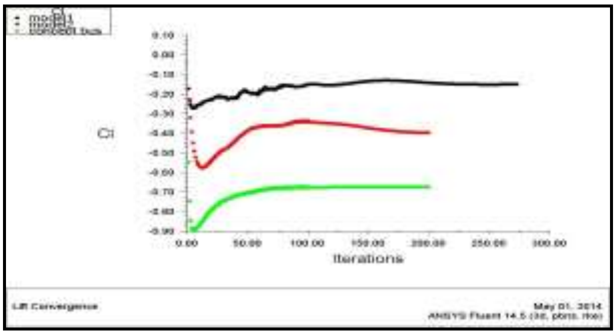

Fig. 5 Comparison of lift coefficient for model 1, model 2 and conceptual bus

Negative lift is the down force which pushes the vehicle closer down to the ground. Underside of the bus is responsible for creating the lift or down force. In order to maximize the down force, floor panel height of the bus should be reduced. From the above graphs, it can be observed that co-efficient of lift is reduced from -0.18 for the standard bus to the value -0.7 for the conceptual bus. It is because the floor panel height of the conceptual bus is 0.4m which is considerably lower than the other two buses. Reducing the lift ultimately assists to achieve vehicle stability.

C. Static Pressure Contours

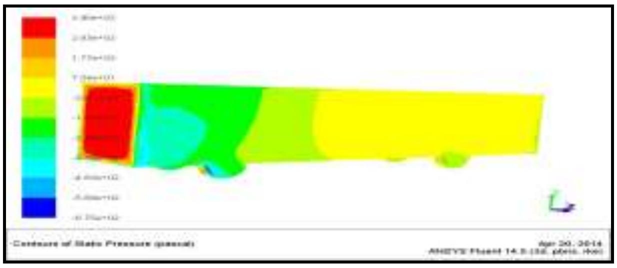

Fig. 6

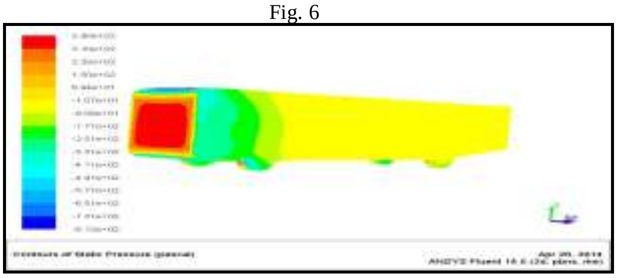

Fig. 7

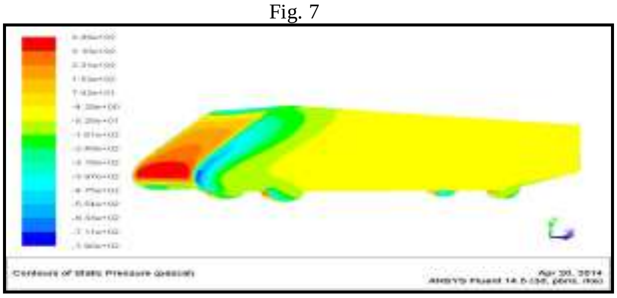

Fig. 8

Fig. 6, Fig. 7, Fig. 8 Pressure contours for model 1, model 2 and conceptual bus respectively

Since the current standard models have flat front ends and sharp edges, more air flow impinges on frontal area which leads to rise in pressure. For 2nd model, rounded edges provide nozzle effect which accelerates the flow at the edges. This raises the velocity of air and lowers the pressure at the rounded edges. For the conceptual bus, more pressure is developed at the stagnation regions [5]. As the aerodynamic shape provides more nozzle effect at frontal surface, pressure reduces tremendously towards the leading edge.

D. Total Pressure Contours Along Symmetry

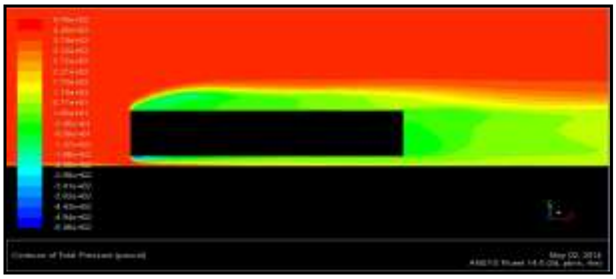

Fig. 9

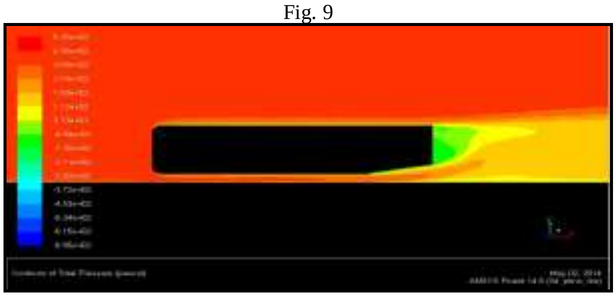

Fig. 10

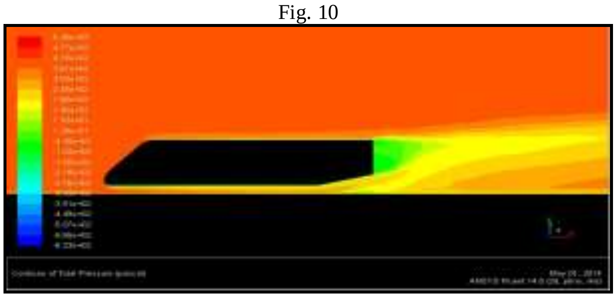

Fig. 11

Fig. 9, Fig. 10, Fig 11 Total pressure contour along symmetry for model 1, model 2 and conceptual bus respectively

Rounded edges at the undercarriage accelerate the air flow, but to obtain the optimum velocity, floor panel height for the conceptual designed bus is reduced. This provides lesser scope for the air to flow through undercarriage which ultimately maintains the pressure [6]. This provides lesser air resistance, and simultaneously gives more cooling effect and reduces friction losses at the undercarriage components. From the graph, it is observed that pressure variation for the concept bus is less.

E. Pressure Co-efficient

Pressure co-efficient depends on the local as well static pressure values. Change in shape of the vehicle body affects the local pressure. As the standard model has sharp edges and flat ends, so there is less difference between static and local pressure due to which pressure coefficient shows less variations over the body.

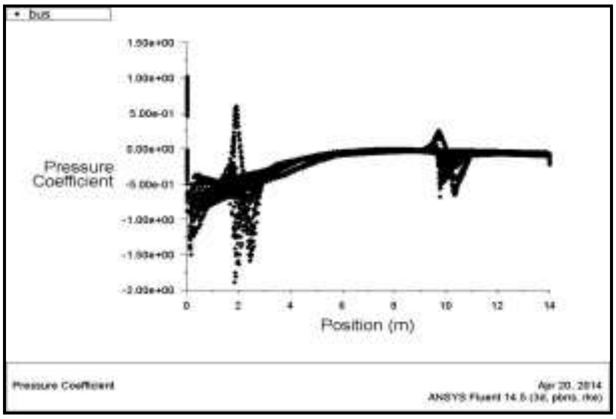

Fig. 12 Co-efficient of pressure along length (Conceptual Design)

In case of concept bus, edges are rounded, front end is aerodynamically shaped and larger diffuser angle is provided due to which local pressure shows variations which further leads to large variations in pressure co-efficient [4]. Similarly for model 1, due to sharp edges and flat front surface lower pressure co-efficient variation is observed. Moderate variations in pressure co-efficient are obtained for model 2.

F. Velocity Contours

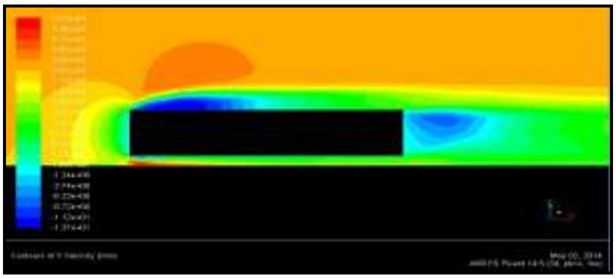

Fig. 13

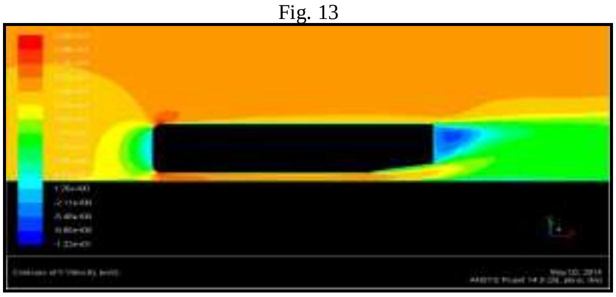

Fig. 14

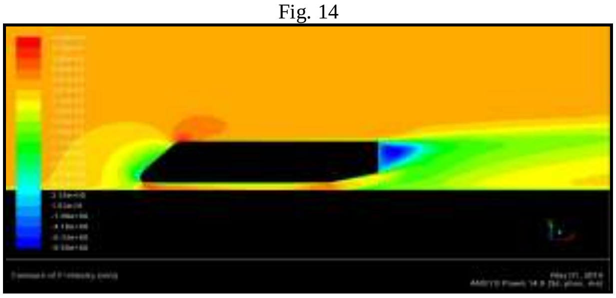

Fig. 15

Fig. 13, Fig. 14, Fig. 15: Velocity contour for model 1, model 2 and conceptual bus respectively

From the above velocity contours, it is observed that, velocity of air increases at leading edge of modified bus due to its streamlined shape. But in case of standard model, air flow is obstructed due to the flat front face.

IV. Conclusions

To obtain economical and performance advantages, an attempt is made to design a conceptual bus with improved aerodynamic performance, Comparative study is done on three bus models by carrying out CFD simulations. Aerodynamically shaping the front end, rounding of the corners, providing optimum diffuser angle and lowering the floor panel height leads to reduction of drag and lift for the modified models. Drag co-efficient is found to get reduced from 0.9 for the standard bus to 0.6 for the concept bus whereas negative lift is increased from -0.18 for standard bus to -0.7 for the conceptual bus. The pressure at front side is found to be reduced for the modified buses due to aerodynamic shape, where flow remains attached and due to the nozzle effect at the leading edge velocity is increased and thus overall pressure at the front end is reduced to 231 Pa for the concept bus. Whereas at the rear end wake region is reduced to -81 Pa for the concept bus due to the increased diffuser angle and overall modified shape of the body. Reduced floor panel height improves vehicle handling and stability.

References

[01] Arun Raveendhram, S.N. Sridhara, D. Rakesh, S. R. Shankapal, " *Exterior styling of an intercity transport for improved aerodynamic performance*", SAE 2009-28-0060.

[02] Chitrarth Lav, " *Three dimensional CFD analysis on aerodynamic drag reduction of a bluff tractor trailer bodies using bluff generators*", SAE 2013-01-2458.

[03] Urban Bus Specifications – II, Ministry of Urban Development, Government of India

[04] A.Muthuvel, M.K.Murthy, Sachin.N.P, Vinay.M.Koshy, S.Sakthi, E. Selvakumar, "*Aerodynamic exterior body design of bus*", International Journal of scientific and engineering research, Volume 4, Issue 7, July 2013.

[05] R.H. Barnard, "*Road Vehicle Aerodynamics",* Third edition.

[06] W.H. Hucho, "*Aerodynamics of Road Vehicles*"

[07] Min-Ho Kim, Jong-Young Kuk and In-Bum Chyun, "*A Numerical Simulation on the Drag Reduction of Large-Sized Bus using Rear-Spoiler*," 2002-01-3070, doi: 2002.