MAPLE TREE
PUBLISHING HOUSE (INDIA)

INTERNATIONAL RESEARCH JOURNAL OF AUTOMOTIVE TECHNOLOGY (IRJAT)

http://www.mapletreejournals.com/index.php/IRJAT
Received 22 June 2018
Accepted 29 July 2018
Published online 30 July 2018
ISSN 2581-5865
2018; 1(4);85-90

Aerodynamic Design Improvement for an Intercity Bus

J.Vasanthe Roy[1*] ,T.Santhosh Kumar[2] ,R.Ashish Kumar[2] ,S.Jeeva[2],V.Vinoth Kumar[2]

[1]Assistant Professor, Department of Mechanical Engg, Velammal Institute of Technology, Chennai-601204.

[2]UG Students, Department of Mechanical Engineering, Velammal Institute of Technology, Chennai-601204.

**Corresponding author E-Mail ID: vasantheroy18@gmail.com*

ABSTRACT

Intercity buses travel about 250 to 350 km in a stretch and usually are of sleeper coach mode. The exterior styling, sleeper comfort and aerodynamically efficient design for reduced fuel consumption are the three essential factors for a successful operation in the competitive world. The bus body building companies prioritizes the exterior looks of the bus and ignore the aerodynamic aspect. Scientific design of sleepers for increased comfort of the passengers is seldom seen. The overall aim of this project was to redesign an intercity bus with enhanced exterior styling, reduced aerodynamic drag and increased comfort for the passengers. Principles of product design were used to analyze the styling and comfort. The benchmarked high floor bus was redesigned with low - floor for reduced aerodynamic drag. The exterior was redesigned with emphasis on improvised aerodynamic performance and appealing looks. The interior was modified to meet aspirations of the commuters. The results of the redesigned exterior body showed a reduction of about 45% in coefficient of drag and overall aerodynamic drag reduction by 60% due to combined effect of reduced coefficient of drag and frontal area.

Keywords— Aerodynamics, CFD, Conceptual Bus, Drag Reduction, Comfort.

1. 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. 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.

2. 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 (Cd). Drag force of the moving vehicle is given by,

Drag Force (D) = 0.5*ρAV2Cd
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 (Cd). 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 Creo and the standard dimensions of Bus are obtained from urban bus specifications. Basic model is further modified to reduce fuel consumption rate 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.

2.1 Modified 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 100 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 150 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.

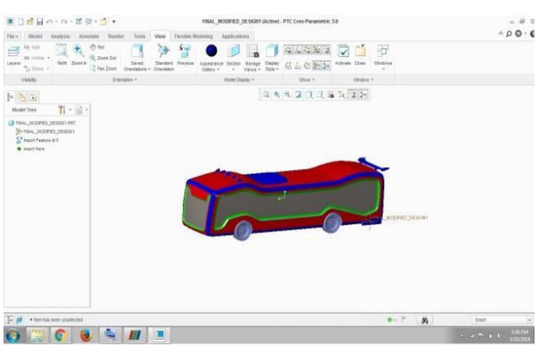


Fig. 1 Solid model of conceptual bus

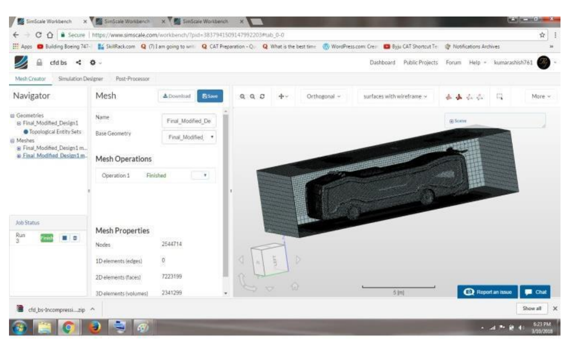


Fig. 2 Meshing of the conceptual bus

2.2 CFD Methodology

CFD solver ANSYS Fluent 14.5 is used for analysis and calculations. K-□ turbulence model is used. The courant number was set to 50 and the relaxation factors were taken as0.25. 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.

2.3 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. Minimum element size was 1mm and number of grid elements is around 910931.

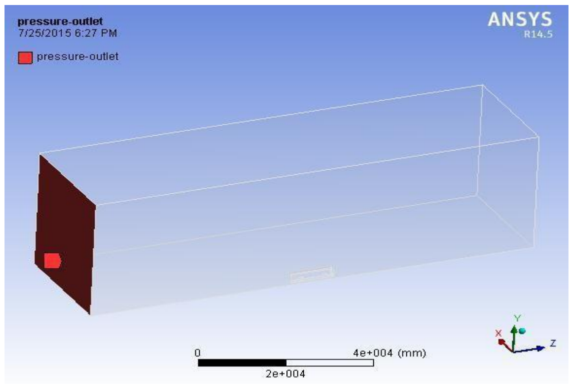


Fig.3 Application of boundary conditions Boundary Conditions for CFD Simulation

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

Table 1. List of Boundary Conditions

SPEED (KM/H)	BASELINE MODEL(F_D) Area =2500 x 3330 mm^2	MODIFIEDMODEL(F_{D1}) Area =2500 x 3000 mm^2
10	33.434	22.702
20	132.393	88.049
30	293.031	193.793
40	517.986	340.416
50	805.129	529.017
60	1155.279	755.222
70	1562.937	1020.625
80	2045.478	1326.758
90	2548.061	1674.099

3. RESULT

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.

3.1. Co-efficient of Drag

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. It also provides the attached flow for the streamline reducing the drag resistance.

3.2 Co-efficient of Lift

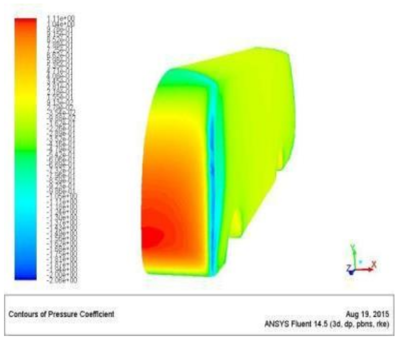


Fig 4.Co-efficent of lift

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. 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 provides nozzle effect

Table 2

Kmph	Speed in m/s	Model 1	Model 2	Model 3
85	23.61	-178.97	-23.28	-238.23
100	27.78	-254.34	-47.84	-749.12
115	31.94	-405.63	-234.89	-1251.7

4. OVERVIEW

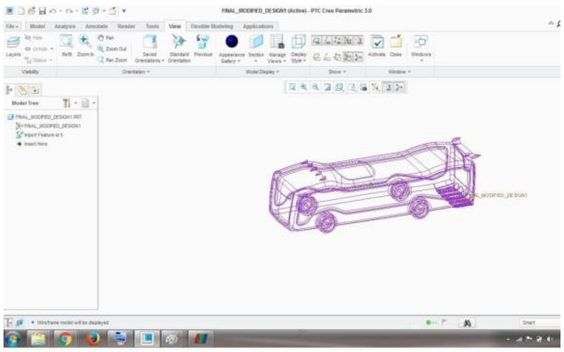


Fig 5-Wire frame model

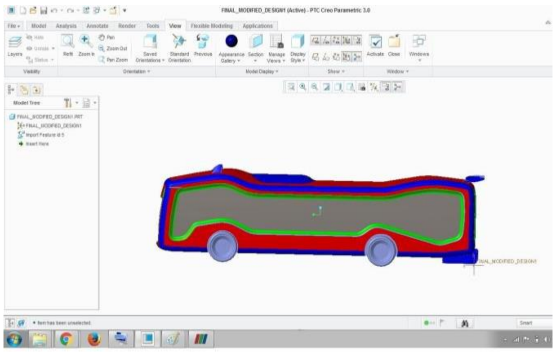
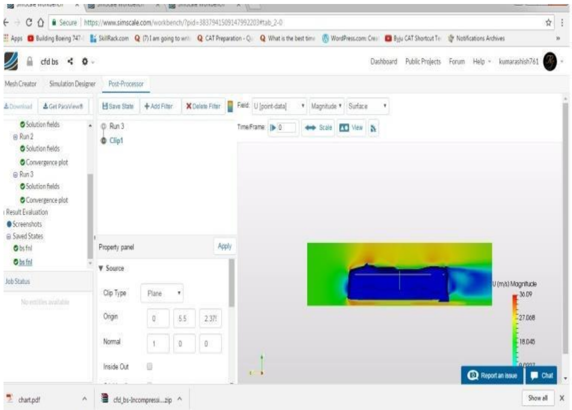


Fig 6- Modified model

4.1 Velocity Contours



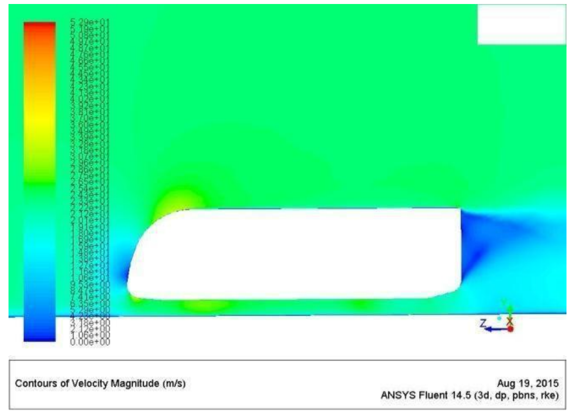


Fig. 14, *15 Velocity contour for model 1, model 2 and conceptual bus respectively*

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 front face

5. CONCLUSION

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. Improvement of bus aerodynamic shape by drag reduction techniques of vehicle i.e. chamfering, rounded corners, tapered rear end etc. and import this model in Flow Design for drag analysis. The flow lines pass over the body and free vortex create near the rear side of the bus which increases with the speed of vehicles. The frontal area is curved as compared to the original styling. At a distance from a halfway of the bus the back area of the bus is declined to an angle of 15°. Reduced drag force improves fuel consumption and stability.

REFERENCES

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

2. P.Krishna Teja, G. Moorthy, S.Kaliappan, "finite element analysis of propeler shaft for automotive and naval application" , international research journal of automotive technology (IRJAT) Volume 01-Issue 1, January 2018, PP.8-12.

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

4. S.Kaliappan, J.Lokesh, P.Mahaneesh, M.Siva, " Mechanical Design and Analysis of AGV for Cost Reduction of Material Handling in Automobile Industries" , international research journal of automotive technology (IRJAT) Volume 01-Issue 1, January 2018, PP.1- 7.

5. Urban Bus Specifications – II, Ministry of Urban Development, Government of India

6. Raj Kamal M.D Socrates.S Kaliappan S (2015) "Aerodynamic Effects on Formula One Car Using CFD" , International Journal of Applied Engineering Research (IJAR) Vol. 10 No.33,, PP.28164-28172.

7. 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.

8. R.H. Barnard, "Road Vehicle Aerodynamics", Third edition.

9. S.Kaliappan, J.Lokesh, P.Mahaneesh, M.Siva, " Mechanical Design and Analysis of AGV for Cost Reduction of Material Handling in Automobile Industries" , international research journal of automotive technology (IRJAT) Volume 01-Issue 1, January 2018, PP.1- 7.

10. Sivaram S Kaliappan S Raj Kamal M.D Suresh kumar R. BalamaniKanda Suthan K " Performance Analysis of Gasifier on CFD ",International Journal of Applied Engineering Research, Vol. 10 No.33, PP- 25880- 25889.

11. W.H. Hucho, "Aerodynamics of Road Vehicles"

12. Min-Ho Kim, Jong-Young Kuk and In-Bum Chyun, "A Numerical Simulation on the Drag Reduction of Large-2002-01-3070

13. Kumaragurubaran.J Raj Kamal M.D Kaliappan S (2015) " Investigation of jet noise reduction using fan flow deflectors on CFD ", International Journal of Applied Engineering Research, Vol. 10 No.33, PP- 26003- 26010.