2019-26-0071 Published 19 Jan 2019

Modifications in Minibus Design for Drag Reduction

Vivek Waman Nagose College of Engineering, Pune

Citation: Nagose, V.W., "Modifications in Minibus Design for Drag Reduction," SAE Technical Paper 2019-26-0071, 2019, doi:10.4271/2019-26-0071.

Abstract

The main objective of this paper is used to reduce drag of the minibus vehicle so as to reduce fuel consumption (power consumption). As Mini-bus is designed as H_2O_2 fuel cell and electric bus, no emissions are created to the environment. Different modified Computer Aided Design (CAD) model of minibus was used and performed Computational Fluid Dynamics (CFD) on it. While performing CFD on it, various process of simulations are done are importing CAD file from CATIA, designing of wind tunnel in its geometry and using Boolean operations (subtracting the body from tunnel block), Meshing in ANSYS and solving it. Models like k-ε Standard and k-ε Realizable models are used for good results. After observing the results, due to the changes in front and rear part of the minibus, there is reduction of almost 32% of the drag coefficient for Dolphin Bus and 42% of the drag coefficient for Aero-Dolphin Bus which results in a power savings of about 13.5% for Dolphin Bus and 18% for Aero-Dolphin Bus. The name given is Dolphin as it's front shape is like a Dolphin who has high IQ. So, Advanced Driver Assisted Steering system (ADAS) including Adaptive cruise control system and various future smart technologies can be used in these Dolphin and Aero-Dolphin Buses.

Introduction

With the growth in transport industry, there is huge need of buses so as to reduce fossil fuel consumption rather than personal vehicles. The usage of BUS reduces traffic in the city. The Vehicle has to overcome rolling resistance, gradient and aerodynamic load. The BUS body made of composite material reduces the weight of the vehicle, thus reducing fuel consumption or power consumption for electric vehicles. With changing the exterior design of the vehicle body, the good aerodynamic designed vehicle is made which reduces drag coefficient thereby reducing drag force and it acts in opposite direction of driving force.

The Aerodynamic drag and lift forces can be calculated as follows:

- Drag force (F_d)
  $F_d = C_d \rho A_{pv} V^2$
- Lift force (F_l)
  $F_l = C_l \rho A_{ph} V^2$

Where,
C_d – Drag co-efficient of the vehicle.
C_l – Lift co-efficient of the vehicle.
ρ – Density of air.
A_{pv} – Vertical projected area.
A_{ph} – Horizontal projected area.
V – Velocity of air with respect to vehicle.

Need and Justification

Due to increase in pollution and because of some environmental factors, it becomes necessary to develop vehicles consuming less fuels and with less emissions. Drag force is usually formed due to pressure difference in front and rear part of the vehicle. Many modifications are to be done to decrease pressure difference between front and rear part of the vehicle. The reduction in drag force eventually reduces fuel consumption and gives vehicle a better mileage.

Flow separation takes place at high speeds because of the reduction of pressure from upper front edge part of vehicle body to the rear edge. At high speeds, less pressure developed in the upper part of the body results into lift forces. For the commercial vehicles like trucks, buses, etc., lift force developed is negative. Lift in vehicle decreases traction and thus it decreases stability of the vehicle. As large wake region is developed at back side of heavy commercial vehicles because of eddies developed. The formation of 3D vortices generates eddies which results into development of back pressure in the wake region. More eddies results into more drag in vehicle body.

CFD uses mathematical modeling to solve the problems which includes fluid flows. Softwares like Ansys, OPenfoam, Autodeskflow design, etc. are used to do CFD analysis on complete body of the vehicle to find drag and lift coefficient of the vehicle.

Previous Study Conducted in Aerodynamics of Commercial Vehicle

Carregari [1] conducted both computational and experimental analysis of commercial bus scale model design with focusing on more reducing fuel consumption and efficient engine cooling. Used k-epsilon RNG turbulence model while doing CFD analysis and conducted test in wind tunnel. Carregari commented that there is possibility of changing design of front and rear for reducing drag in commercial Buses.

Videira [2] in his study observed experimental and computational results, realized that the cost to modify bus design on the basis of aerodynamics is less costly relatively to the modifications to be done in engine for reduction in fuel consumption.

Some features can be modified as follows:

- Rounding the lateral surfaces and edges.
- Varying front inclination angle or giving curvature to front part.
- Modifications in diffuser rear part of the bus.

Drag coefficient of Commercial buses varies from 0.4 to 0.7.

Hucho [3] studies says that model bus of cuboid type with sharp corners has $C_D = 0.88$ and with rounding the sharp corners has $C_D = 0.36$. According to Hucho, to reduce drag coefficient significantly, radius of curvature of 150 mm is taken for average vehicle speed.

Rodrigues, Antonio Flavio Aires, et al. [4] has studied minibus vehicle aerodynamics. Their purpose of the study was achieved because it was obtained a model of bodywork that offered a significant value in reducing the drag coefficient. The study had great complexity, by reason of the construction of CAD geometry and its simplification to the resolution of the analysis. The creation and development of mesh discretization of the computational domain.

From the results obtained, they observed that with small changes in the body geometry was possible to attain optimal values of the drag co-efficient for a minibus, where this results in considerable fuel savings.

Vignesh S, et al. [5] established that RANS with SST provides reliable results for ventilation studies, the same was used to study methods for improving the ventilation inside the compartment. Literature study was carried out to identify possible design modifications to improve ventilation inside a bus. They suggested the provision of additional inlet and exhaust structures on the bus body to increase the air circulation inside the passenger compartment. The positioning of such structures required a detailed study on the external air-flow and pressure profile of the bus body. [5]

They conclude RANS with SST model give reliable results for ventilation studies. The internal flow in the compartment was found to be from the rear to the front of the bus as found in the experimental studies and LES simulations found in literature. Bus ventilation can be improved by appropriately placing roof-vents. [5]

Mario, et al. [6], conducted computational simulations of 10 models with variable trailer design with changing space or volume available in it. Variable trailer design concept can be used in Mini-Buses as a variable coach design body. Different geometrical modifications of the trailer rear end have been evaluated regarding possible reduction of aerodynamic resistances and thus a potential reduction of fuel consumption under steady state conditions on highway traffic. For the evaluation of results, the fuel consumption of each variant was calculated by use of computational fluid dynamics simulations under steady state conditions. The possibility to adjust the trailer outer contour according to the required transportation volume combines both, advantageous aerodynamic behaviour and improved transportation efficiency. [8]

Callen [7] in his experiments found that after the removal of the mirror the drag of the vehicle reduces by 4.5%. This study says that the mirror produces flow separation and recirculation.

Patil [8] in their study suggested that addition of spoilers and panels at rear portion gives good optimization of BUS. Without changing its internal space or volume of the body, drag can be reduced.

According to Ahmed [9] the structure of the wake region that is formed in the backside of the vehicle depends on the back shape of the BUS, if it verifies the existence of a recirculation region and forming a pair of longitudinal vortices behind the vehicle.

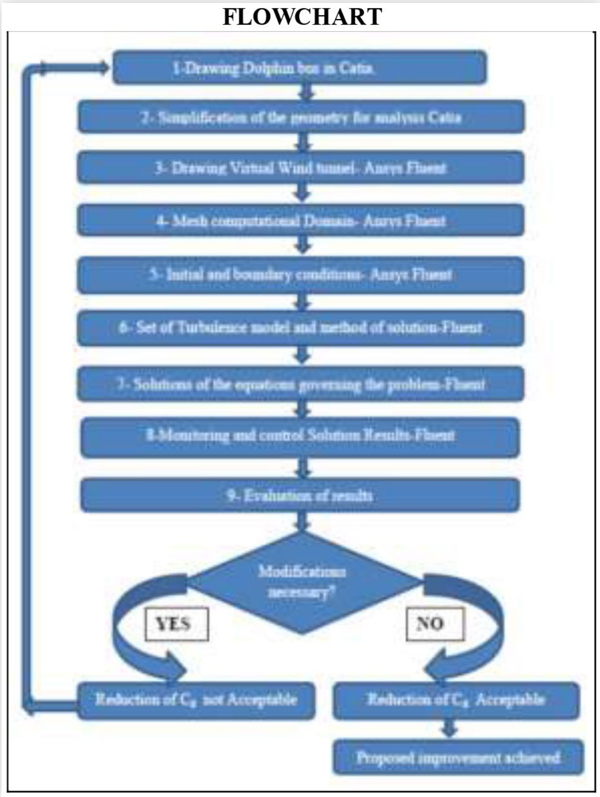
Ahmed [10] observed that at an angle of 15° from the rear surface forms a recirculation region at the back surface. With an angle of 10°, the flow was still stuck on that surface, concluding that the lowest drag was obtained with an angle of 12.5

The main parts that contain the exterior of an intercity bus are the windshield, front bumper, headlights, indicators, wipers, side windows, passenger doors, luggage space, rear bumper and grille the radiator. For commercial vehicles, more constraints are there to modify its body than for cars. Apart from aspects of styling and manufacturing the functional requirements are dominant. Any modification that would reduce the usable interior space has less chance of being adopted in practical bus design.

Methodology Used in This Study

The methodology used incudes design of CAD geometry of the Minibus in Catia and the design of virtual tunnel in ANSYS. Tunnel body is meshed by using Boolean function (subtracting Minibus from Tunnel body) in ANSYS. CFD analysis is done by using ANSYS fluent solver. 5 CAD geometric models are used for CFD analysis. Among which 2 models are selected on the basis of more space availability.

FIGURE 1 Flowchart of methodology used in this study.



Project Design

Geometric CAD Model

The vehicle used is micro bus i.e., collective transport vehicle accommodating 30 passengers, designed to have the best performance in transportation such as intercity and chartering. In this transport model must be agile, because there is great difficulty in traffic between major cities and in the mixed displacements between urban and road [5].

Taking Space into account, only compromise has been done in driver cabin of Minibus. The geometries for the proposed studies will be briefly presented so that the changes made in each analysis remain clear.

Stage 1 Bus

FIGURE 2 Geometrical model of 1st stage Mini-BUS

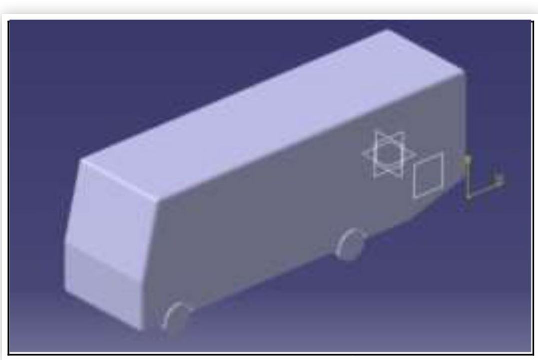
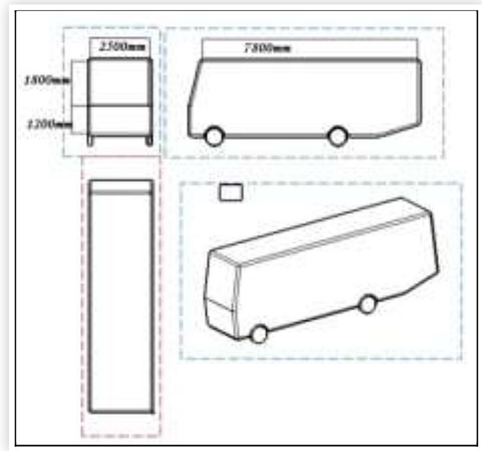


TABLE 1 Dimensions of the Mini-BUS.

Height	Length	Width
3350 mm	9000 mm	2500 mm

FIGURE 3 Catia Drawing/Drafting of Stage 1 BUS



Stage 2 Dolphin Bus

FIGURE 4 Geometric color model of Dolphin BUS.

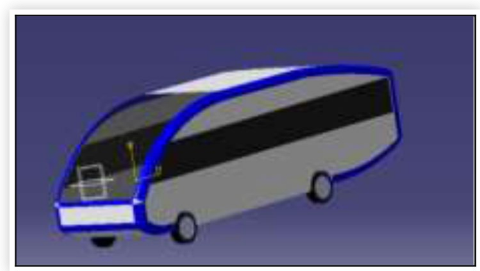
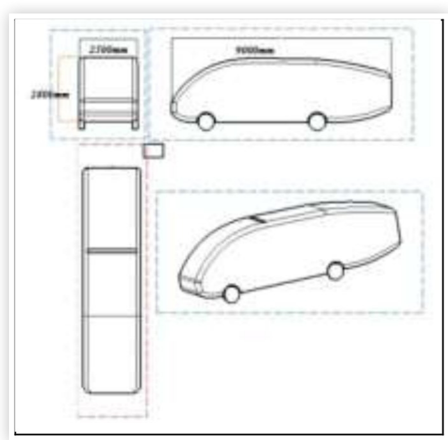


TABLE 3 Dimensions of the Dolphin and Aero-Dolphin BUS

Height	Length	Width
3070 mm	9000 mm	2500 mm

FIGURE 5 Catia Drawing/Drafting of Dolphin BUS.



Stage 3 Aero-Dolphin BUS

FIGURE 6 Geometric color model of Aero-Dolphin BUS.

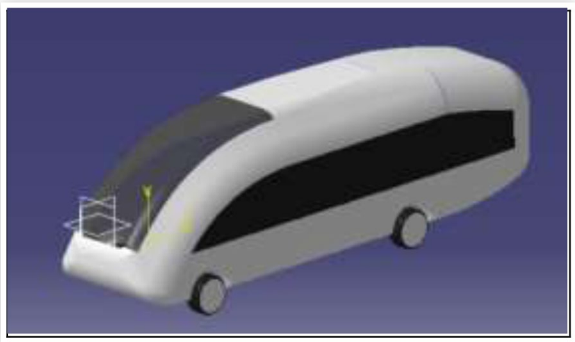
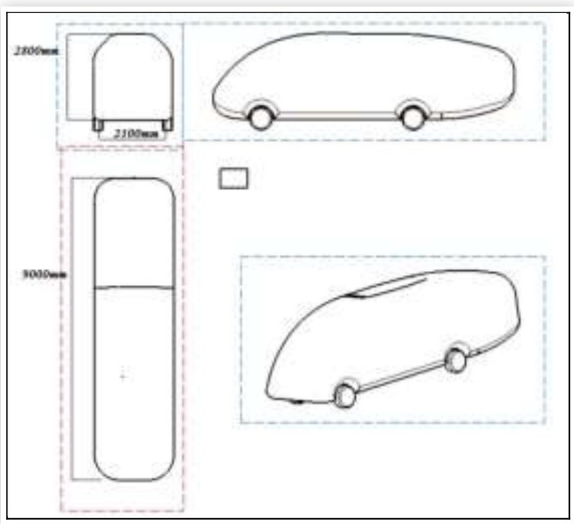


FIGURE 7 Catia Drawing/Drafting of Aero-Dolphin BUS.



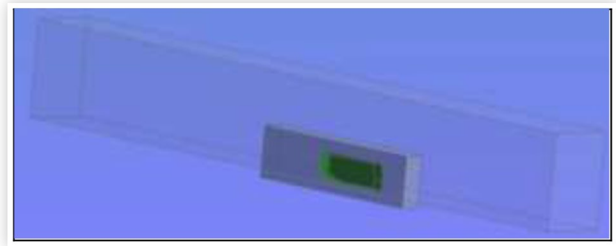
Virtual Wind Tunnel Design Based on the geometry of the body, the virtual wind tunnel was developed in ANYS Fluent. The dimensions of the tunnel are proportional to the Minibus are calculated as follows:

Here, length = 9500 mm,
Width = 2500 mm,
Height = 3350 mm.
Length of tunnel to +ve Xaxis = 3(length) = 28.5 m
Length of tunnel to -ve Xaxis = 5(length) = 47.5 m
Now Aspect ratio = 0.7
H/W = 0.7

$$\text{Blockage ratio} = A/A_d = 0.04$$
$$= 6.87/A_d = 0.04$$
$$A_d = 171.675 \text{ m}$$
$$171.675 = W*H$$
$$171.675 = 0.7* W^2$$

W= width of tunnel = 15.66 m
H = Height of tunnel = 10.96 m

FIGURE 8 Wind tunnel created in ANSYS.



For reducing the no. of elements, half body was inserted under analysis which effectively reduces time of solver and gives the same result as for full body. Figure 8 shows the 3D design of the bodywork and the tunnel, with proportional dimensions, developed for studies.

Numerical Analysis

Mesh Generation

The meshing of the computational domain was performed using the ANSYS fluent, from the geometry created in Catia software. While meshing, different types of elements are used such as tetrahedrons and prisms to the region of the boundary layer.

The dimensions for the initial mesh are based on the work done by Rodrigues are as follows:

Triangular elements = 20 mm and
Inner block = 5 mm (region near to bus body)
Height with growth rate = 1.2 [5]

FIGURE 9 Meshed body of tunnel with Mini-BUS extracted from it.

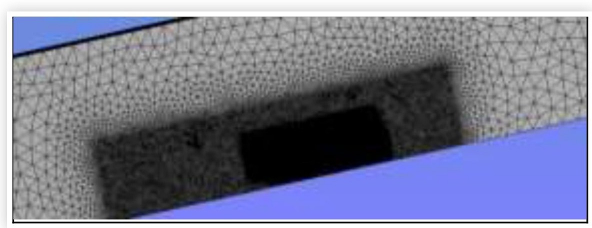
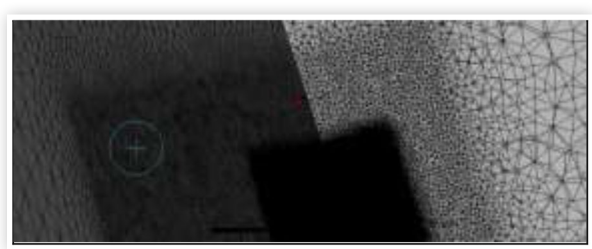


FIGURE 10 Symmetrical cut view to observe inside meshing elements.



Boundary Conditions

For the analysis we applied boundary conditions for the boundaries of the computational domain and the region containing the fluid. The virtual wind tunnel is made containing different sections such as tunnel entrance and exit, side walls, top and bottom face. The boundary conditions applied are as follows:

- Entrance section -Velocity Inlet (22 m/s = 80 km/h)
- Exit section - Pressure Outlet
- Ground - Stationary(wheels are not moving with respect to ground, so assumption is taken as rolling resistance is neglected)
- Symmetry plane - All the simulation results of pressure and velocity contours are projected on the symmetry plane

The front face of the BUS is faced to wind tunnel entrance as shown in fig towards right. [5]

With any simulations, there are some assumptions listed below:

- Flow is steady and incompressible
- Working fluid: air at 27°C
- All downward projections of gear trains, differential, etc. are neglected.
- Rolling resistance of rotating tires are neglected as tires are stationary.

Numerical Scheme and Strategy

The program has its numerical formulation based on the finite volume method. Run 300 to 700 iterations till the graph gives constant value of the parameters with waste reduction, model k-ε Realizable with 'Functions Non-equilibrium Wall' was applied. Due to the nonlinearity of the equations of conservation, it was necessary to use the sub-relaxation values, which were also modified. The values used are the result of observations from the advance of analysis regarding the convergence time of each interaction as well as the time to reach convergence with different criteria, articles and forum discussions CFD – FLUENT.

FIGURE 11 Boundary condition for tunnels for solving it in ANSYS.

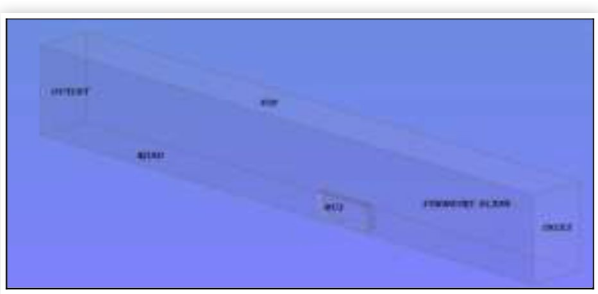


TABLE 4 Factors sub-relaxation. [4]

Variables	Standard	Modified
Pressure	0.3	0.1
Density	1.0	1.0
Momentum	0.7	0.5
Body forces	1.0	1.0
Turbulent Viscosity	1.0	1.0
Turbulent kinetic energy	0.8	0.6
Turbulent kinetic energy dissipation rate	0.8	0.8

Results

Data collected in the computational analysis will be presented through the contours of pressure and velocity, velocity vectors, lines of trajectory and the coefficient of drag and lift (C_d and C_l).

Stage 1 BUS

FIGURE 12 Pressure contour of stage 1 BUS seen on symmetric plane

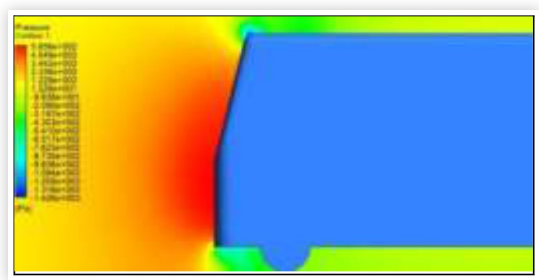


FIGURE 13 Pressure contour of stage 1 BUS

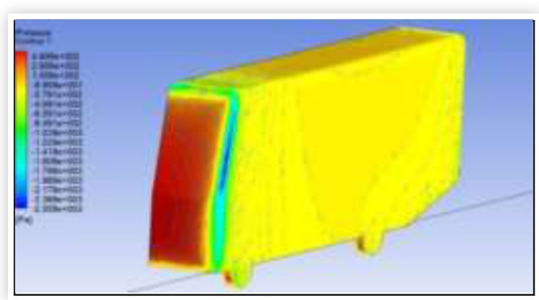
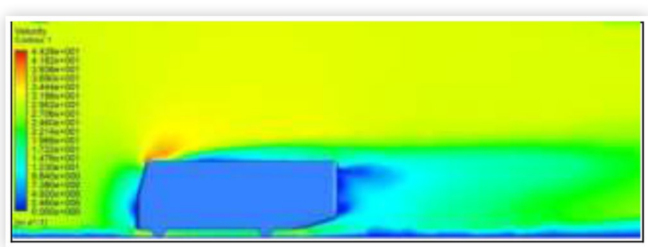


FIGURE 14 Velocity contour of stage 1 BUS



Dolphin BUS

FIGURE 15 Pressure contour of Dolphin BUS on symmetric plane.

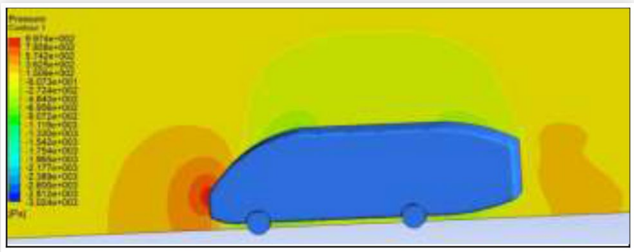


FIGURE 16 Pressure contour of Dolphin BUS

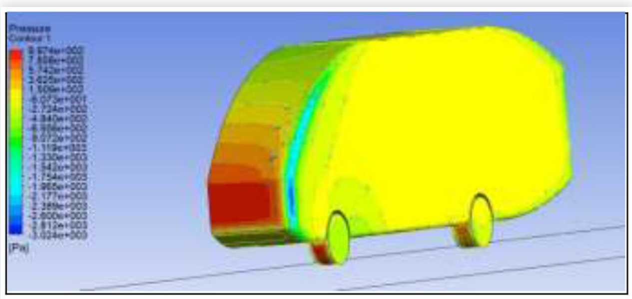


FIGURE 17 Velocity contour of Dolphin BUS

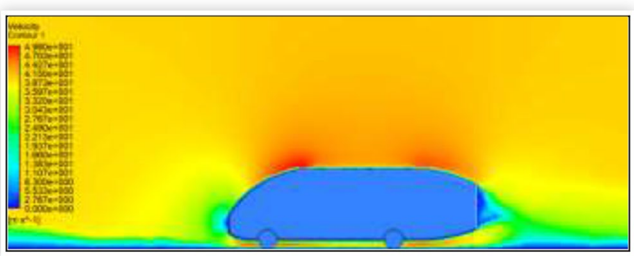


FIGURE 18 Pathline pressure of Dolphin BUS

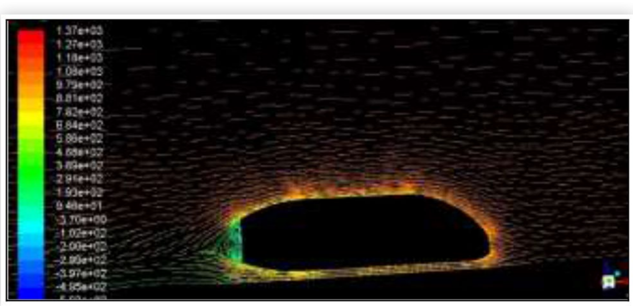


FIGURE 19 Particle path velocity of Dolphin BUS

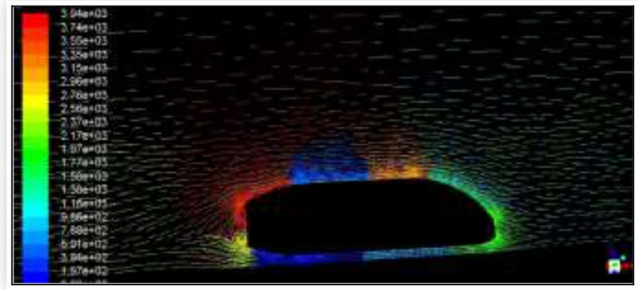
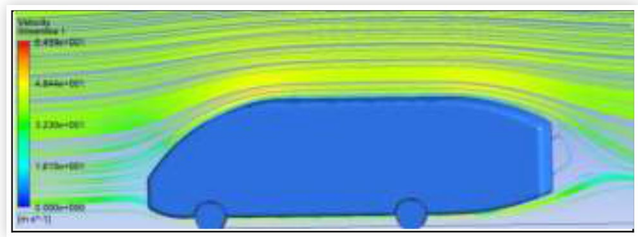


FIGURE 20 Streamline flow of Dolphin BUS



Aero-Dolphin BUS

FIGURE 21 Pressure contour of Aero-Dolphin BUS

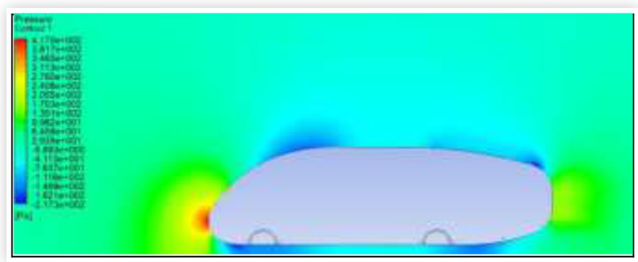


FIGURE 22 Velocity contour of Aero-Dolphin BUS

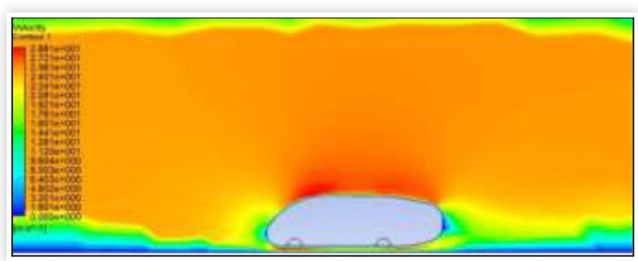
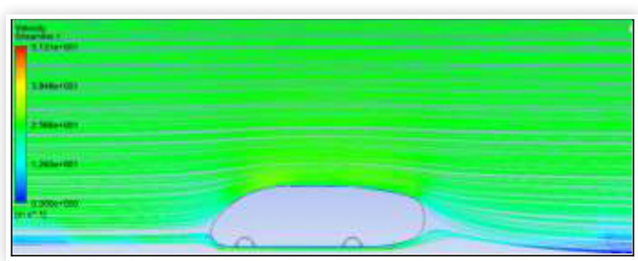


FIGURE 23 Streamline flow of Aero-Dolphin BUS



Drag coefficient and lift coefficient of 3 geometric models

FIGURE 24 C_d for stage 1 BUS.

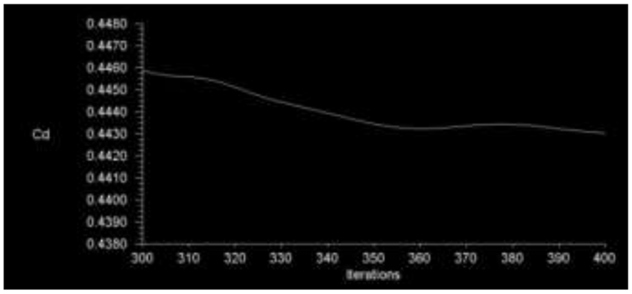


FIGURE 25 C_l for stage 1 BUS.

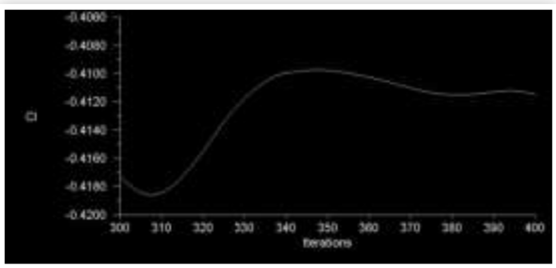


FIGURE 26 C_d for Dolphin BUS.

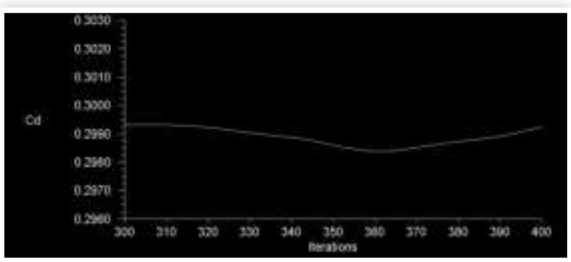


FIGURE 27 C_l for Dolphin BUS.

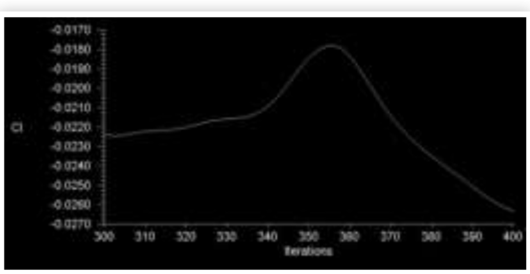


FIGURE 28 C_l for Aero-Dolphin BUS.

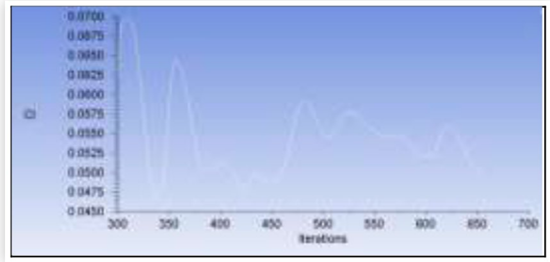


FIGURE 29 C_d for Aero-Dolphin BUS.

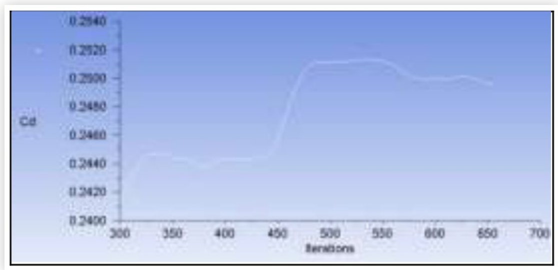


FIGURE 30 Graph of reduction in fuel consumption with respect to reduction in drag coefficient [4]

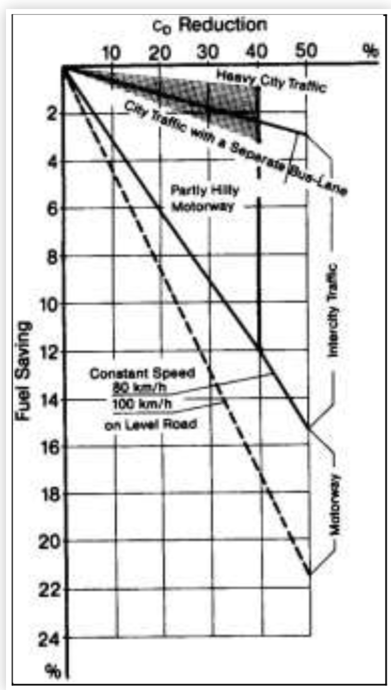


TABLE 5 Result table for drag and lift coefficient.

Geometric Model	Drag coefficient (C_d)	Lift Coefficient (C_l)	Drag reduction	Power/fuel Saving
Stage 1 BUS	0.44	-0.411	-	-
Dolphin BUS	0.30	-0.0263	31.81%	80 kmph – 9% 100 kmph – 13.5%
Aero-Dolphin BUS	0.25	0.0254	42%	80 kmph – 13% 100 kmph – 18%

Conclusion

1. The main objective of this study was achieved because it was obtained a model of bodywork that offered a significant value in reducing the drag coefficient. The study had great complexity, by reason of the construction of CAD geometry and its simplification to the resolution of the analysis.
2. The creation and development of mesh discretization of the computational domain, which includes the volume between the model and the wind tunnel. This proved to be the most difficult step in the analysis process, in view of the great importance of this, as this determines the resolution and accuracy of the results. It was soon that required more time and effort.
3. From the results obtained it can be observed that with small changes in the body geometry was possible to attain optimal values of the drag coefficient for a minibus, where this results in considerable fuel savings.
4. Drag is drastically reduced from (C_d = 0.44 to C_d = 0.25 i.e., 43% reduction in drag) Mini-BUS to Aero-Dolphin BUS. So Power Consumption is also reduced to 3% for separate lane BUS, 13% for BUS at highway speed of 80 kmph and 20% BUS at highway speed of 100 kmph.

Future Scope

1. The Dolphin and Aero-Dolphin BUS can be used as Electric Buses, H_2-O_2 fuel cell vehicle with solar panel installed on it.
2. Dolphin Buses can also be used for BRTS system and highway roads in India.
3. Aero-Dolphin Buses are luxurious in nature with less seating capacity used for official and corporate uses.
4. Aero-Dolphin Buses seating capacity and interior designing of BUS is decided by the customer and manufacturer will manufacture according to it.
5. As these Buses are electric in nature, no engine is used, so less vibrations and noises are produced. Due to some pressure fluctuation, vehicle at high speed, low intensity noises are produced. These noises are removed by using good aerodynamic design of vehicle. So, passenger's comfort level increases in Aero-Dolphin BUS.

References

1. Carregari, A.L., "Estudo do escoamento do Arsobre a Carroceria de um Ônibus," USP São Carlos, 2006.
2. Videira, L.C.P., "Estudo Experimental de um Modelo deÔnibus com ênfase no escoamento da Esteira," ITA – Instituto Tecnológico de Aeronáutica, 2001.
3. Hucho, W.-H., *Aerodynamics of Road Vehicles* Fourth Edition (Warrandale: SAE International, 1998).
4. Rodrigues, A.F.A., "Aerodynamics Analysis of Vehicle Minibus," SAE Technical Paper 2014-36-0327, doi:10.4271/2014-36-0327.
5. Shekhar, V.T., "A Holistic Approach to Aerodynamics of Intercity and Interurban Buses," SAE Technical Paper 2014-01-0581., 2014, doi:10.4271/2014-01-0581.
6. Hirz, M. and Stadler, S., "A New Approach for the Reduction of Aerodynamic Drag of Long Distance Transportation Vehicles," SAE Technical Paper 2013-01-2414, doi:10.4271/2013-01-2414.
7. Callen, M.C., Salari, K., Ortega, J., Browand, F. et al., "Effort to Reduce Truck Aerodynamic Drag – Joint Experiments and Computations Lead to Smart Design," *AIAA Fluid Dynamics Conference*, 2004.
8. Patil, C.N., "Aerodynamic Study and Drag Coefficient Optimization of Passenger Vehicle," *International Journal of Engineering Research & Technology*, 2012.
9. Ahmed, S.R., "Wake Structure of Typical Automobile Shapes," *Journal of Fluids Engineering* 130, 1981.
10. Ahmed, S.R., "Influence of Base Slant on the Wake Structure and Drag of Road Vehicles," *Transactions of the ASME, Journal of Fluids Engineering* 105:429-434, 1983.

Contact Information

Vivek Waman Nagose,
Student – Mtech Program (Automotive Technology), College of Engineering, Pune.
nagosevw17.mech@coep.ac.in/nagosevivek19@gmail.com
Address: COEP Hostel, Shivaji Nagar, Pune.
Contact no: 9403327151

Acknowledgments

I express sincere gratitude towards the Automotive Research Association of India, Pune and College of Engineering, Pune that has provided me valuable assistance and requirements for the development of my topic.

Definitions/Abbreviations

3-D - 3-Dimensional

ADAS - Advanced Driver Assisted Steering

CFD - Computational Fluid Dynamics

Dolphin - Aquatic Mammal having streamline body which uses 20% of brain.

ISSN 0148-7191