Aerodynamic study of state transport bus using computational fluid dynamics

To cite this article: Siddhesh Kanekar *et al* 2017 *IOP Conf. Ser.: Mater. Sci. Eng.* **263** 062052

IOP Conf. Series: Materials Science and Engineering **263** (2017) 062052 doi:10.1088/1757-899X/263/6/062052

Aerodynamic study of state transport bus using computational fluid dynamics

Siddhesh Kanekar, Prashant Thakre and E Rajkumar
School of Mechanical Engineering, VIT University, Vellore - 632014, Tamil Nadu, India.

Email: rajkumar.e@vit.ac.in

Abstract: The main purpose of this study was to develop the aerodynamic study of a Maharashtra state road transport bus. The rising fuel price and strict government regulations makes the road transport uneconomical now days. With the objective of increasing fuel efficiency and reducing the emission of harmful exhaust gases. It has been proven experimentally that vehicle consumes almost 40% of the available useful engine power to overcome the drag resistance. This provides us a huge scope to study the influence of aerodynamic drag. The initial of the project was to identify the drag coefficient of the existing ordinary type model called “Parivartan” from ANSYS fluent. After preliminary analysis of the existing model corresponding changes are made in such a way that their implementation should be possible at workshop level. The simulation of the air flow over the bus was performed in two steps: design on SolidWorks CAD and ANSYS (FLUENT) is used as a virtual analysis tool to estimate the drag coefficient of the bus. We have used the turbulence models k-ε Realizable having a better approximation of the actual result. Around 28% improvement in the drag coefficient is achieved by CFD driven changes in the bus design. Coefficient of drag is improved by 28 % and fuel efficiency increased by 20% by CFD driven changes.

1. Introduction

Richard A. Drollinger [1] explained the need truck aerodynamics due to increasing fuel cost in 1987. He gave the test data for wind tunnel and proposed improved shape of truck aerodynamics. He explored the computational fluid dynamics method for calculating the drag force on vehicle body. He also studied the limitations of CFD. Due to enhancement of technology, the computational fluid dynamics based simulation techniques have been developed for exploring the aerodynamics of vehicles. Earlier aerodynamics is mainly for design racing cars and aeroplanes. However, Subrata Roy and Pradeep Srinivasan [2] solved compressible Navior-Stokes equation with k-ε turbulent model. They had taken two different types of trucks for computing coefficient of drag. Sachin Thorat and G Amba Prasad Rao [3] performed an experiment on CFD Analysis of Intercity bus on Indian roads and reduced the drag force by 30%. Rodrigues et al[4] found that small alteration in geometry makes huge impact on drag force . Further Ashok Patidar et al [5] performed aerodynamic analysis based on numerical study on effect drag resistance on fuel efficiency of bus. They used ANSYS fluent for solving and got 30 % improvement in drag coefficient. Aerodynamic study for commercial vehicle was started by Sherwood [6] in 1953. He conducted wind tunnel experiments on heavy vehicles like trucks and buses. This paper focuses on buses of Maharashtra state transport. Maharashtra State road transport corporation (MSRTC) began their operation from 1948 as BSRTC. It is state bus service

IOP Conf. Series: Materials Science and Engineering **263** (2017) 062052 doi:10.1088/1757-899X/263/6/062052

with around 18000 buses in fleet which carry around 7 million passengers daily on various intrastate and interstate route. The Parivartan, Hirkani and City Buses are built at MSRTC's own in-house workshops at Dapodi, Nagpur and Aurangabad on Ashok Leyland and TATA chassis. Theses workshops don't have modern techniques to build aerodynamic bus coaches. It uses conventional way for building. that is why we have made changes in design in such way that their implementation should be possible in conventional workshops. In this paper, Central workshop Aurangabad(CWA) built Ashok Leyland's "Parivartan" model have been considered for analysis. This work is very beneficial in terms of increasing fuel efficiency and maximize profit by reducing drag force**.**

2. CFD Methodology

The methodology used in this analysis consists of the design geometry of the bus in SolidWorks and mesh with CFD analysis using ANSYS FLUENT. The study includes the analysis of 2 bodies from an initial model and modified model to reduce the aerodynamic drag through small changes in the design. Imagining the air flow field around the bus is required to find the drag force on the bus. It can be taken either by using CFD techniques or by wind tunnel testing. From the flow pattern, flow separation and wake strength behind the body can be recognized and fixed by changing the CAD geometry. A CFD methodology followed in this study and each step of the procedure is explained below

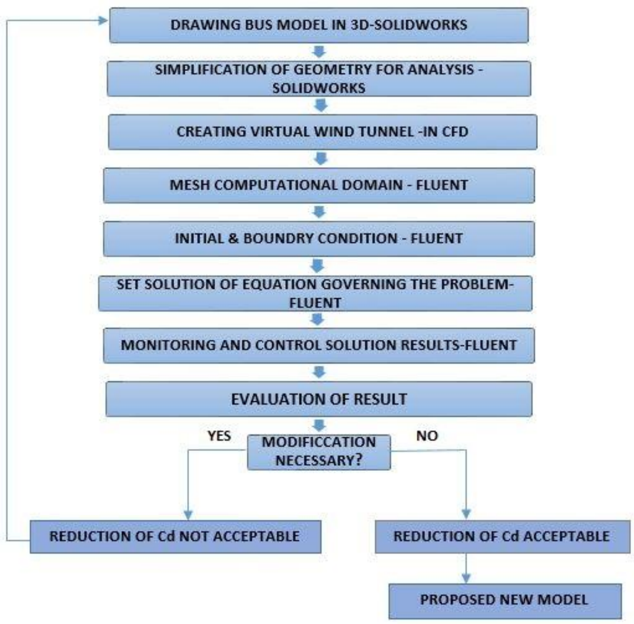


Figure 1. Methodology

2.1. Geometric Model

The vehicle taken in this project is known as "Parivartan" which is built on Ashok Leyland cheetah chassis (Figure 2). It is Maharashtra state transport vehicle with capacity of about 45 passengers, designed to have the safe and affordable interstate and intrastate journey

Figure2. CWA Built Ashok Leyland Parivartan

*2.1.1. Original body.*The original model of bus without simplification is shown in fig. for performing aerodynamic analysis, it has been simplified to figure 4. The main simplifications are eliminating small parts of gaps, wheel frames, taillights and other small details which are insignificant for this study.

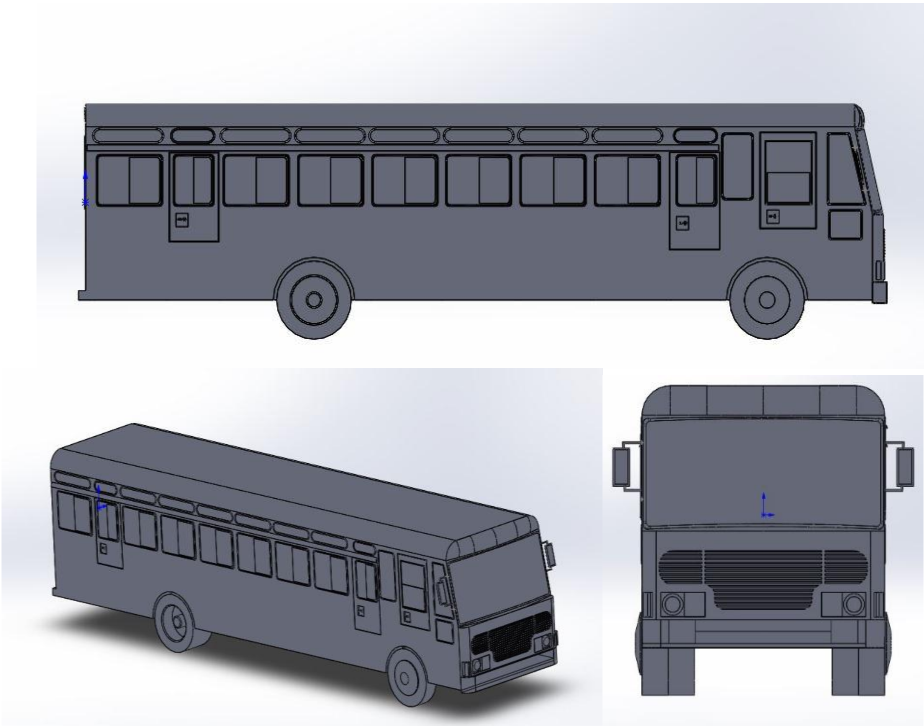


Figure 3. Original model without simplification

The original body is now present with the simplification they are mention in above

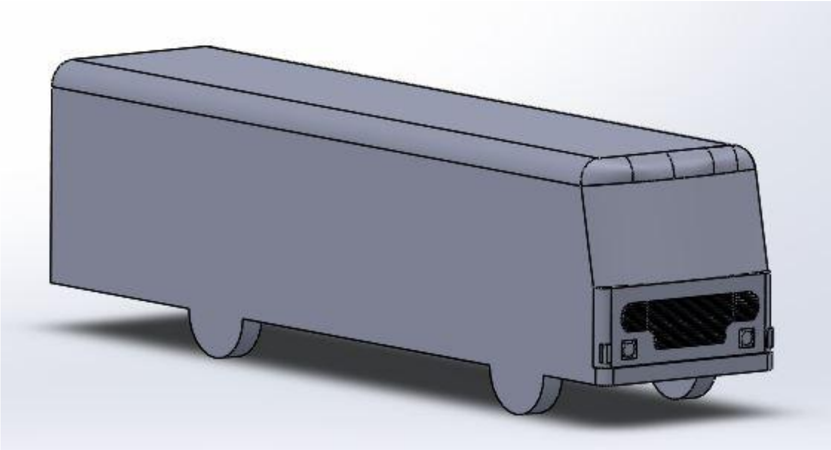


Figure 4. Original model with simplification

2.2 Design of Virtual Wind Tunnel

Based on the design of the bus was made virtual wind tunnel, which is the domain of the flow, using in ANSYS FLUENT as shown in figure 5. The measurements of the tunnel are relational to the bus. At work, the full body was inserted into the tunnel for computational analysis. The dimension of wind tunnel is where 10H, 20W and 13L clear height, width and length of bus respectively. Virtual wind tunnel that contains of an inlet, an outlet, wall, and a ground surface.

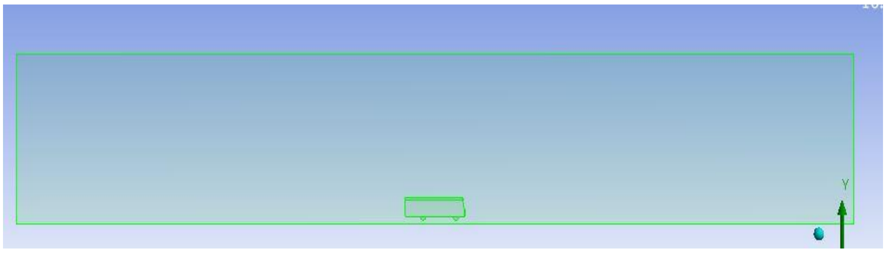


Figure 5. Wind tunnel

2.3 Mesh Generation

The surface mesh was created using ANSYS fluent. The mesh is produced on the design of the bus and on the surface of the domain. The type of mesh used to discretize the computational domain was tetrahedrons. The mesh established for the surface bodies had as base format the triangular mesh. fine mesh is done in sizing. Everything else is kept default for quickening the process.

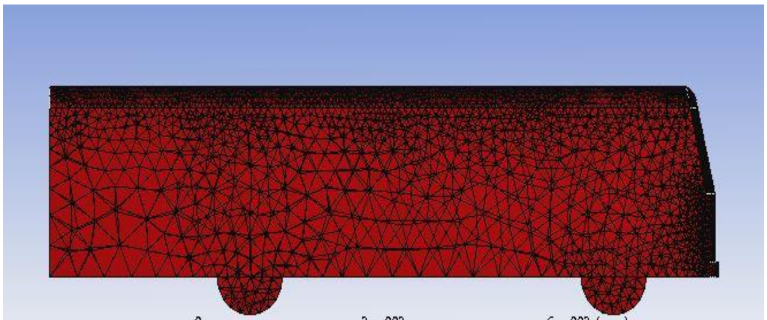


Figure 6. Mesh generation

2.4 Solver Setup

IOP Conf. Series: Materials Science and Engineering **263** (2017) 062052 doi:10.1088/1757-899X/263/6/062052

ANSYS fluent solver based on pressure is used for this study. It is steady analysis with absolute velocity formulation. Reynolds-Averaged Naiver-Stokes equations are solved for simulating an incompressible turbulent flow. The selection of a viscous model is tough in external flow analysis. A standard k-ε model is generally used due to its strength and easy convergence. Since it is not recommended for external geometry with flow separation, realizable k-ε model with non-equilibrium wall functions is used for this analysis. Coupled Scheme is taken for the pressure-velocity coupling with Least squares Cell based gradient. Bus body is considered stationary for study. In the simulation, vehicle speed of 80 Kmph with only straight wind condition is measured. Constant velocity inlet condition is applied at the inlet and at the outlet zero-gauge pressure is applied with operating pressure as atmospheric pressure.

3. Results and Discussion

The total pressure distribution on the vehicle, velocity vectors and air flow path lines results are analysed & discussed in this section. A modest estimation of change in fuel consumption due to change in drag coefficient is also explained in this section.

3.1 Original Body

The coefficient of drag (C_d) of Parivartan is found to be 0.66. This value can be considered normal that can be seen in reference Patidar et al [6]. Drag force is 1373.5 N.The pressure contour figure obtained is showing the great influence of the windshield and cowl. Plotted path lines visualize the air flow separation and wake region behind the bus body as shown in figure- Due to flow separation strong vortices are produced behind the bus which create low pressure zone. This low-pressure zone jerks the bus back and straight away this is shape dependent.

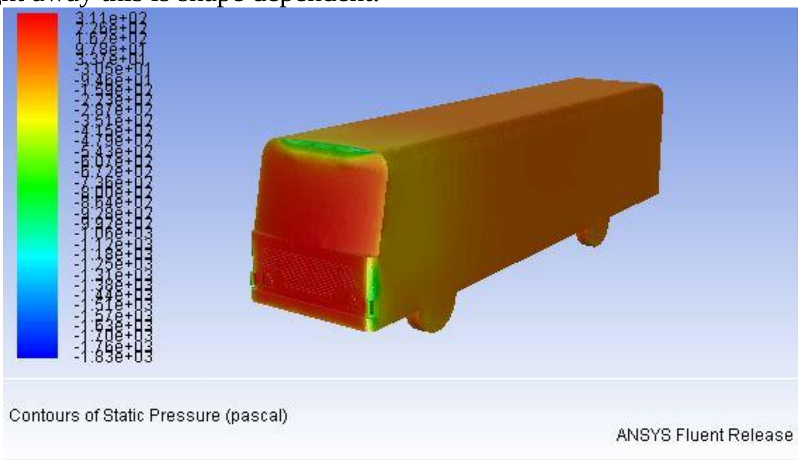
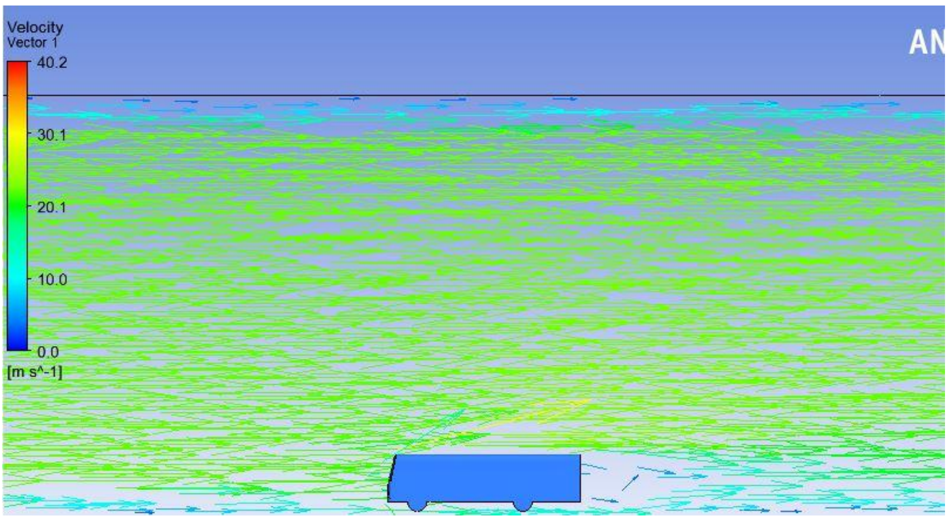


Figure 7. Pressure contour of original design



IOP Conf. Series: Materials Science and Engineering **263** (2017) 062052 doi:10.1088/1757-899X/263/6/062052

Figure 8. Velocity vector of original design

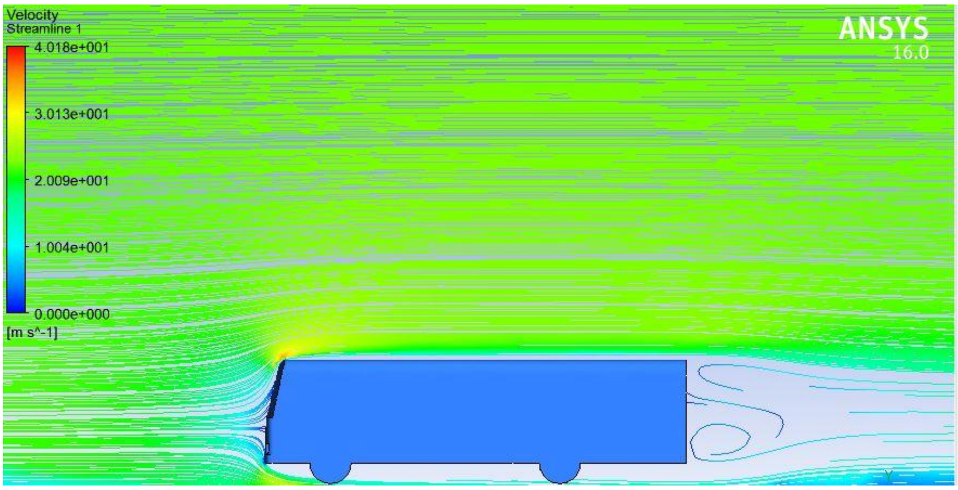


Figure 9. Velocity streamline of original design

After evaluating the aerodynamic performance of original parivartan body. It is found that there is very much scope for modification so as to improve coefficient of drag and fuel efficiency. After trial and error method it is found that design shown in fig. giving optimum performance during aerodynamic test.

3.2 *Design Proposal*


Figure 10. Modified proposed designed

In the proposed model, angle of windshield and cowl has been changed. also, cowl is made curvy as compared to previous one. In this model windshield and cowl are attached to each other in smooth manner. Proposed model has 0.46 coefficient of drag. and new drag force is found to be 1040.5N.

Pressure contour shown in fig emphasizing on pressure zone at critical frontal area of bus body. It is found that new model has less static pressure as compared to original one. The overall impact of this reduction can be measured in terms of reduction of drag.

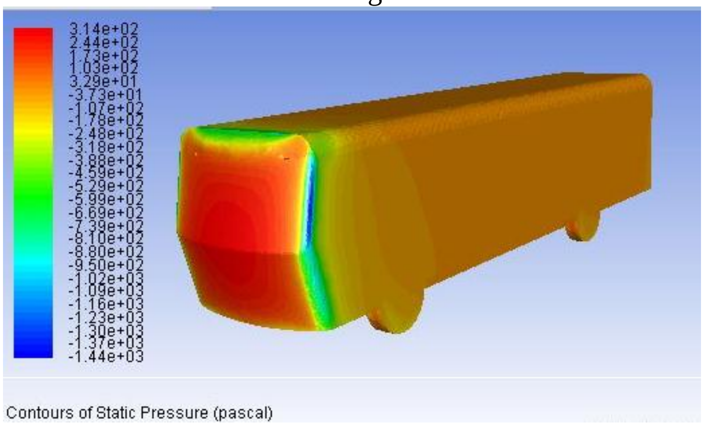


Figure 11. Pressure contour of modified design

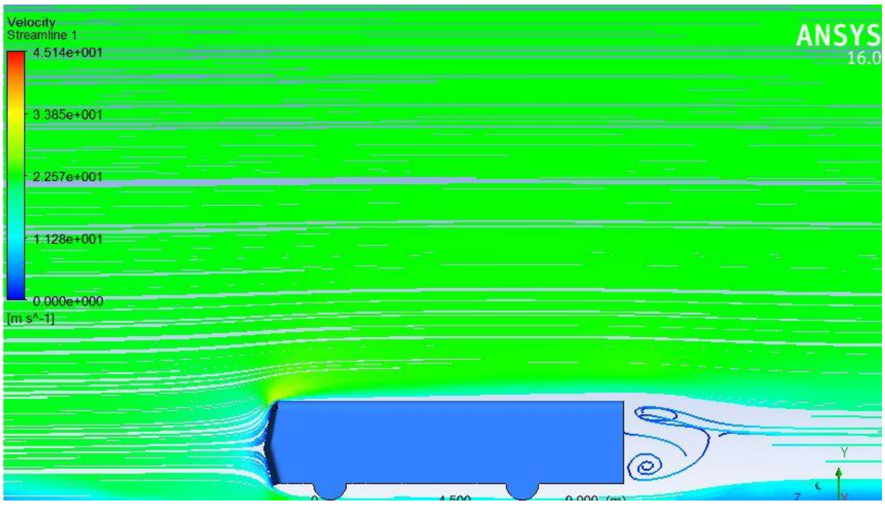


Figure 12. Velocity streamline of modified design

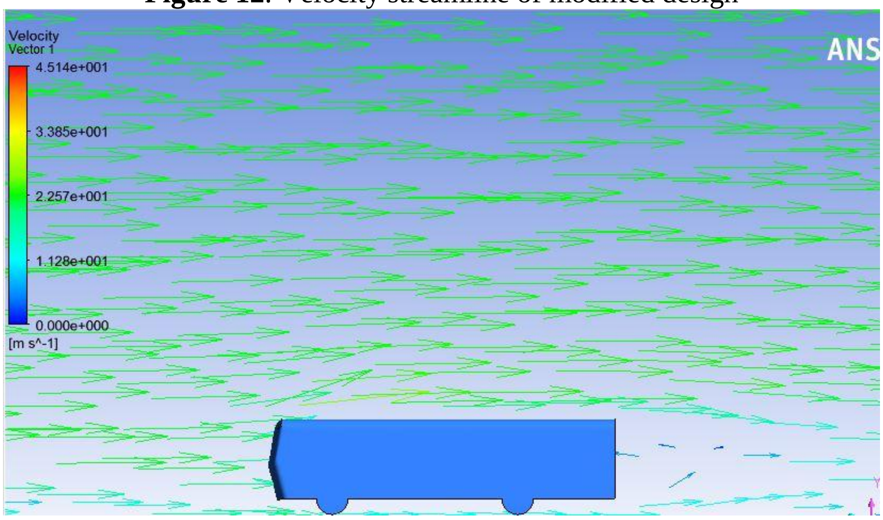


Figure 13. Velocity vector of modified design

4. Estimation of Fuel Consumption

4.1 Aerodynamic resistance

Aerodynamic resistance is the force of wind resistance pushing the body in the opposite direction to the motion of the body. Aerodynamic resistance is given by following equation,

$$F_A = \frac{C_d \rho A V^2}{2}. \quad (1)$$

Where,
drag coefficient= C_d
the frontal area =A (m^2)
the density of the air =ρ(kg/m^3)
the relative velocity of air or bus speed relative to the road =V (m/s).

4.2 Rolling Resistance

Rolling resistance is the force that resist the motion of vehicle when vehicle rolls on a surface. While tire rolling, resistance depend on the temperature, pressure of air, tread depth and side force.

The experimental relation the unit of F_R is pounds (lbf).

$$F_R = (0.0041 + 0.000041.V_T).GVW \quad (2)$$

Where,
GVW =the gross vehicle weight in ponds.
V_T = the relative bus speed in miles per hour for this equation.

4.3 Requirement Of Power

The Aerodynamic resistance is one of the part total power requirement on the engine. The additional key sources of power requirement are rolling resistance (F_R), road grad (lifting the vehicle) (F_G), and other equipment power losses (P_E) .The total power requirement (P_T) for the bus engine can be estimated by

$$P_T = \frac{(F_A + F_R + F_G).V}{\eta_m} + P_E \quad (3)$$

Following Assumption are made for calculating power requirements.
Mechanical efficiency (n_m)=90%
road grad (F_G)=0
Power demand P_E=6KW

4.4 Consumption of Fuel

An exact calculation of the fuel economy is difficult. A good and easy calculation of fuel economy or fuel efficiency in kilometre per litter (KMPL) can be estimated as

$$KMPL = \frac{V.\eta_t.E_F}{P_T.10^6} \quad (4)$$

Where n_t =the maximum thermal efficiency=35%
E_F =energy content of the fuel=39.7 × 106 Joules per Litre for diesel fuel.
V = the vehicle speed in m/s=22.22

For original body KMPL is found to be 6.2 while for proposed model it is 7.5 that is around 20% improvement. It is shown in table 1.

Table 1.Comparison between Original parivartan and Modified parivartan

Type	C_d	Drag	KMPL
Original parivartan	0.66	1373.5N	6.2
Modified parivartan	0.46	1040.5N	7.4

5. Conclusion

External aerodynamics performance of the original design of Maharashtra state transport bus is studied. Critical areas where improvement can be done are identified in design and made three basic level changes: 1. Rounded cowl, 2. Attachment of cowl and windshield and 3. Fit of cowl to the body. These changes can be easily implemented at regular workshop. The drag coefficient of modified design is reduced by 28% and at 80 kmph speed fuel economy is increased by 20%. It is noted that the use CFD analysis as design instrument at early stage of vehicle body design, can improve the vehicle performance by large margin.

References

[1] Drollinger, , *Heavy Duty Truck Aerodynamics*, SAE Technical Paper 870001, doi:10.4271/870001, (1987).

[2] S.Roy, and P. Srinivasan, *External Flow Analysis of a Truck for Drag Reduction*, SAE Technical Paper, doi:10.4271/2000-01-3500, (2000).

[3] Thorat Sachin, and Rao G Amba Prasad, *Computational Analysis of Intercity Bus with Improved Aesthetics and aerodynamic Performance on Indian Roads*, International Journal of Advances in Engineering and Technology, 2,No **3**, p.p. 103-109, (2011).

[4] Rodrigues, Antonio Flavio Aires, *Aerodynamic Analysis of a Vehicle Minibus*, SAE Technical Paper, 36, pp.3-27, (2014) .

[5] A. Patidar, U. Gupta, and Bansal, A., *Fuel Efficiency Improvement of Commercial Vehicle by investigating Drag Resistance*, SAE Technical Paper, doi:10.4271/2015-01-2893, (2015).

[6] A. Sherwood., *Wind Tunnel Tests of Traler-mobile Trailers*, University of Maryland, No. 85, (1953).