Published by :
http://www.ijert.org

International Journal of Engineering Research & Technology (IJERT)
ISSN: 2278-0181
Vol. 6 Issue 11, November - 2017

Aerodynamic Drag Reduction on Locally Built Bus Body using Computational Fluid Dynamics (CFD): A Case Study at Bishoftu Automotive Industry

H. Yesfalgn Damissie
Lecturer,
Wolaita Sodo University,
B Sc (Agricultural Engineering & Mechanization),
Hawassa University,
M Sc (Automotive Engineering),
Adama Science and Technology University, Ethiopia

N. Ramesh Babu
Associate Professor,
Adama Science and Technology University,
Adama, Ethiopia

***Abstract*----** **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. Drastic reduction in the pressure at front and wake region was obtained. Redesigning the overall body shape also improves vehicle stability and handling. This project modified the outer surface of the bus aerodynamically in order to reduce the effect of drag force of the bus which in turn results in reduction of fuel consumption of the bus. Three Models are used and the models were entirely designed using the software CATIA V5R19 and SOLIDWORK 2014. The models are namely model 1, model 2 and model 3. Model 1 is existing Yutong cross-country bus model and model 2 and model 3 are modifications of existing Yutong bus. A simulation of a bus model dimensions 10771 x 2446 x 2727 mm was conducted using CFD software FLUENT 14.5, in order to investigate the flow around a bus and to reduce the drag force. The study was conducted for three wind speeds. Numerical tests have been conducted in CFD to prove the effectiveness of the new concept design. It is evident from the test result, that there has been a considerable reduction in drag force of about 39.4% from the existing bus to the new concepts and 22.8%(5 to 6 liters) of fuel can be saved for the every 100 Km.**

Keywords: Yutong Bus, Drag Force, Pressure coefficient, Fuel consumption, Wind Speeds.

I. INTRODUCTION

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 [3]. 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 raise the fuel consumption rate.

The external flow analysis of a bus is important as the aerodynamics drag dominate at speed above about 65-80 kmph. Generally speaking, an incremental decrease in drag can translate into significant fuel savings. Furthermore, buses are used mostly for long-distance travel and the fuel consumption is a big deal for the bus operators. As shown in Table 1, the current typical value of drag coefficient for the buses is smaller than Pre-1970 vintage, and it is projected to be smaller in future by more radical or thorough approaches.

Conventional aerodynamic development of bus is carried out by wind tunnel testing of a miniature model with the working floor representing the road. Recognizing the limitations of the wind tunnel boundary conditions, considerable efforts were made to study aerodynamics computationally. In this study, a computational Fluid Dynamics (CFD) tool, called ANSY Workbench 14.5 and Fluent 14.5 were used to predict the physical parameters on the external surface of a bus, drag and lift force analysis, pressure and velocity distribution of the bus. The flow distribution was calculated by solving the Reynolds-averaged Navier-Stokes equations with a realizable K-Eturbulencemodel.

II. OBJECTIVE

This research aims to modify the outer shape of the existing bus (Yutong bus) aerodynamically in order to reduce the effect of drag force which in turn results in reduction of fuel consumption of the bus using CFD tool fluent software.

- To model an existing Yutong bus (baseline model) and the two new models using CATIA and SOLIDWORK (CAD modeling software).
- To mesh the models using ANSYS workbench**.**
- To perform the flow analysis on the baseline model and the two new design models using CFD tool fluent.
- To compare the pressure force, viscous force, total drag force, lift force, drag coefficient, lift coefficient and pressure coefficient of the three models in three different high speeds.
- Drawing out the outcomes of rolling resistance, power required and fuel consumption comparing the three models in three different high speeds.
- Analyze the results and draw the conclusion between the cases on the aerodynamics performance and drag.

III. METHODOLOGY

In this study numerical simulation of different buses configurations are performed .Starting with evaluating the results of the baseline simulation. For each configuration the CFD process described below is performed. The CFD-process can be divided into three steps; pre-processing, solving and post-processing.

3.1 Bench Marking the Baseline Model

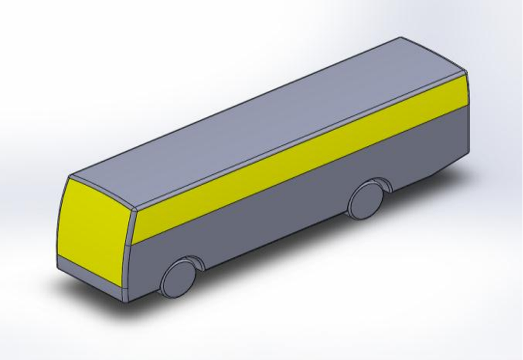
Yutong bus was bench marked for the study. It is the latest model in the China market and is popular in Ethiopia nowadays. Engineering parameters of this model was kept as same for the new bus design and was selected as the baseline for studying the aerodynamic performances. This bus model was used to carry out CFD and aerodynamic parameters such as, flow separation and stagnation areas, pressure and velocity distribution around the body, C_d, and contribution of various exterior parts on the overall drag coefficient of the vehicle.

Fig.1 Existing Yutong Bus (Baseline Model)
Criteria for selecting Yutong bus (baseline model)

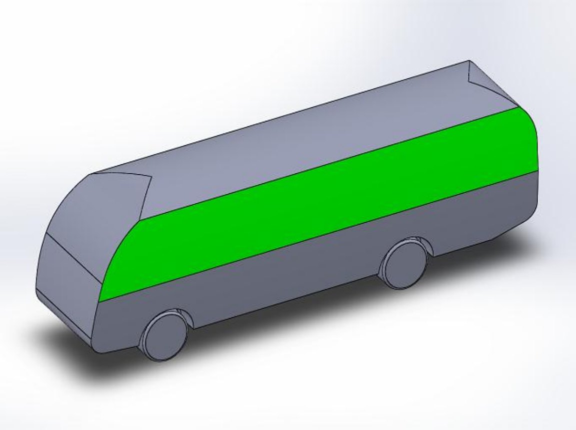
- ✓ The bus is currently popular in Ethiopia. Many government and private organizations are using the bus as a service and public transport.
- ✓ Even if the design and the model of the bus are imported from China, it is assembled here in Ethiopia in Bishoftu Automotive Industry.
- ✓ It is the only long and cross-country bus assembled in Bishoftu Automotive Industry (in Ethiopia). It has 60 seats.
- ✓ The bus has no good aerodynamic shape.
- ✓ It consumes 23 liters per 100 km. This needs to be improved in total resistance of the bus. Due to these, I have selected the aerodynamic resistance. This is serious in cross-country bus while the speed is above 60 kmph and has a significant role in increasing fuel consumption.

3.2 Geometry Generation

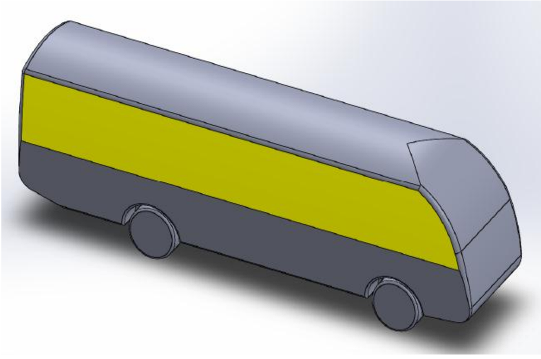
Three dimensional model of the baseline model(10771 mm, 2446 mm, 2727 mm), model 2, and model 3 were created using CATIA V5R19 and SOLIDWORK 2014 as shown in (Figure 30, 31, 32 & 33) below. The original (existing) dimension (10771 mm, 2446 mm, 2727 mm) of the baseline model was used for all the models. Fluid domain of 50 m x 30 m x 30 m was created around the bus model which was 3 to 5 times the length of vehicle to X, Y and Z-direction. Bus model was placed inside this domain in such a way that 1/3 length was kept in front of the vehicle. The larger domain was kept at the rear to capture the essential flow features (Figure 5). A smaller domain was created inside this domain to generate fine mesh in and around the bus body. ANSYS WORKBENCH 14.5 preprocessor and FLUENT 14.5 post processor were the software tools used to generate the mesh and to solve fluid dynamics problems respectively. Outer volume was meshed with coarse elements. Unstructured tetrahedral hybrid elements of 1,342,263 for model 1, 1,391,681 for model 2, and 1,380,386 for model 3 were used to mesh the entire fluid domain.



Model 1



Model 2



Model 3

Fig. 2: Solid Models of Baseline and Two Modified Models (Asymmetric view)

3.3 Boundary Conditions

The analysis was carried out in moving road and rotating wheel condition. In the simulation only straight wind condition was considered at 3 different vehicle speed of 85, 100, 115 kmph. Constant velocity inlet condition was applied at the inlet to replicate the constant wind velocity conditions. Zero gauge pressure was applied at the outlet with operating pressure as atmospheric pressure. All the boundary conditions used in the analysis are listed in table 1.

Table 1: Boundary Conditions

Boundary	Boundary Conditions	Values
Inlet	Constant velocity Turbulent Intensity Length scale	V=23.1 m/s V=27.78 m/s V=31.94m/s
Outlet	Pressure outlet	Constant pressure= 0 N/m^2
Road	Moving wall No slip	V=23.1 m/s V=27.78 m/s V=31.94m/s
Bus body	No slip – stationary wall	-
Domain top and side	Stationary wall Specified shear	Shear stress = 0

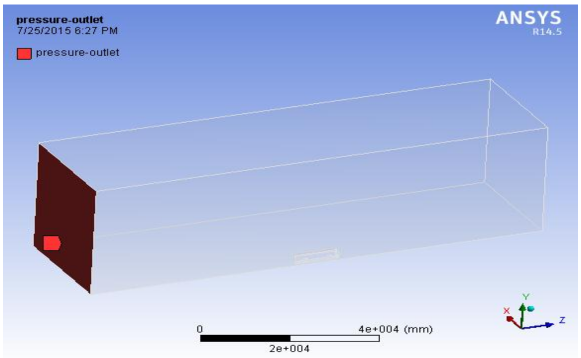


Fig. 3: Boundary Conditions

3.4 Turbulent Model

The solver used for the analysis was FLUENT 14.5 and it uses a control-volume-method to solve the governing equations that can be solved numerically. The solver selected was the pressure based implicit solver. In this type the equations of continuity and momentum are solved sequentially. The flow is considered to be incompressible and steady in nature and the equations are solved using second order upwind method.

3.5 Model Design Specifications and Considerations

I. Model Design Specifications

It scaled 3-D model of bus with its actual dimensions in table 2, by using the CAD software. We know the outer dimensions of the bus then the model is modeled. It found out the external shape of the bus. After, for the analysis the 2D format is a compatible one to view the result. After the bus model is converted in to IGES (igs) format, this file can be used in the CFD software to analyze the external shape of the vehicle for the analysis. The specifications of aerodynamic bus are in table 3.

Table 2: Bus Specifications

Overall Dimensions	
Length (mm)	10771
Width (mm)	2446
Height (mm)	2727
Interior height (mm)	1925
Wheel base (mm)	5800
Fuel consumption (L/100km)	23
Front/Rear Axle (kg)	6500/9500
Max. Total Mass(kg)	16000
Max. Speed (km/h)	110
Model ZK6116D	

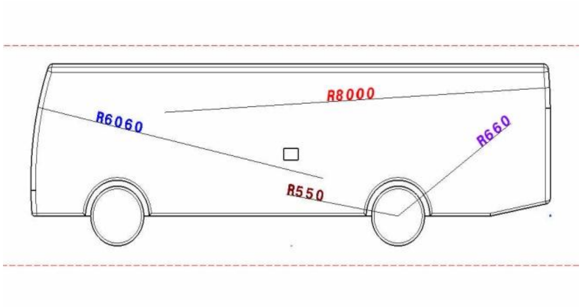
Table 3: Aerodynamic Specifications

S.NO	AERODYNAMIC SPECIFICATIONS
1	Minimum front corner radius of 100 mm
2	Smooth and covered under body from 11.5 ° to 15 °
3	Minimum trailing edge radius of 100 mm
4	Side panel tapering of 100 mm
5	Rear roof tapering of 1000 mm
6	Curved front end
7	Roof end lowering
8	Increasing Rake Angle from 30º to 45º

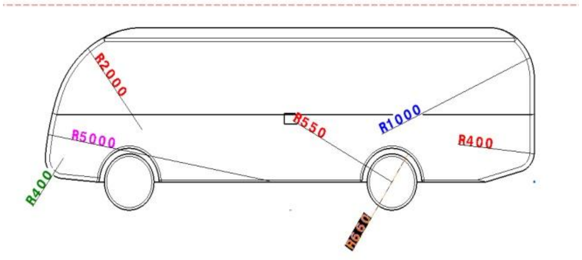
II. MODEL DESIGN CONSIDERATIONS

The exterior of the bus was designed with the concept of robust appearance of a killer whale and other modified models displayed below (Fig. 4).

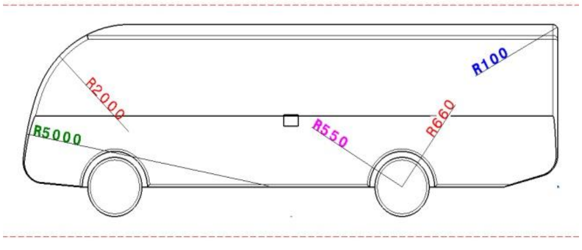
Fig.4: New Bus Concept Design



Model 1



Model 2



Model 3

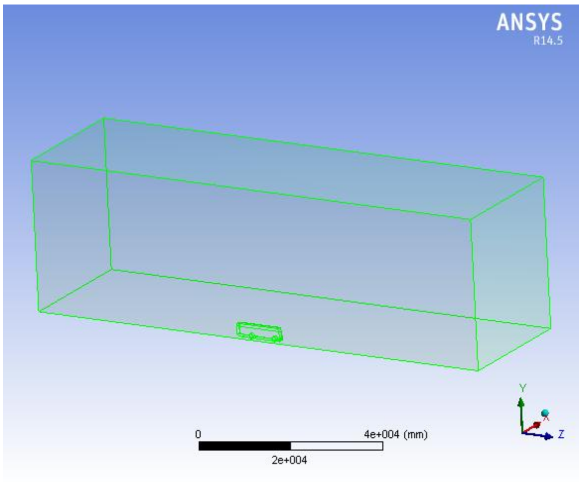
Fig.5: Side view of model 1, model 2 and model 3 with dimensions

3.6 Mesh Generation

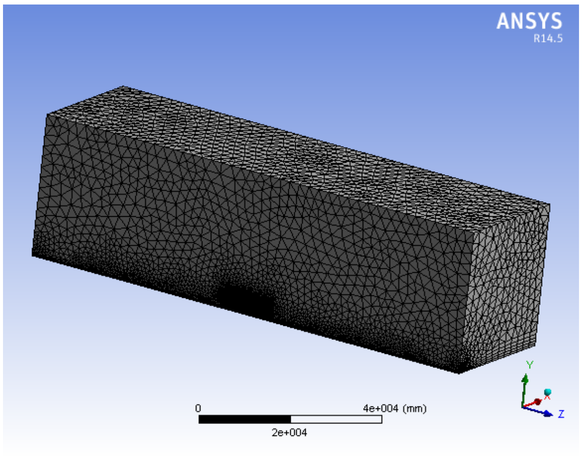
The computational domain is designed to lead to a free with neglectable blockage, which essentially means a box that consists of an inlet, an outlet, two sides, a roof and a ground Surface(road). The size of the domain is taken based on the dimension of the bus such that the real-time road conditions were satisfied.

The surface mesh was created on the geometry of the bus as well as on the surface of the domain. Between the surface of the bus and the domain the computational grid was generated. To capture certain areas of interest (where separation might occur and where the degree if turbulence is high) the cells have to small enough to solve all irregularities and achieve a robust solution. The grid has been redefined around the bus, at the rear and especially at front the bus since this study is focusing on the overbodies influence on the flow field.

I. Model 1



Model 1 inside Enclosure



Meshed Model 1

Fig.6: Model 1 inside Enclosure and Meshed Model 1

Element Details:

1342263 mixed cells, zone 2, binary.

2780796 mixed interior faces, zone 1, binary.

41075 triangular wall faces, zone 5, binary.

552 mixed velocity-inlet faces, zone 6, binary.

606 mixed pressure-outlet faces, zone 7, binary.

13754 mixed symmetry faces, zone 8, binary.

1140 triangular symmetry faces, zone 9, binary.

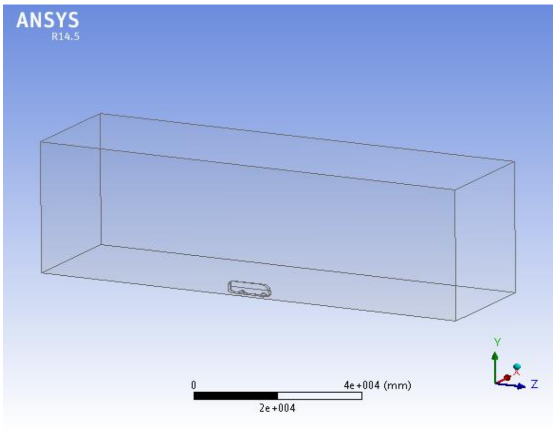
1304 mixed symmetry faces, zone 10, binary.

11399 triangular wall faces, zone 11, binary.

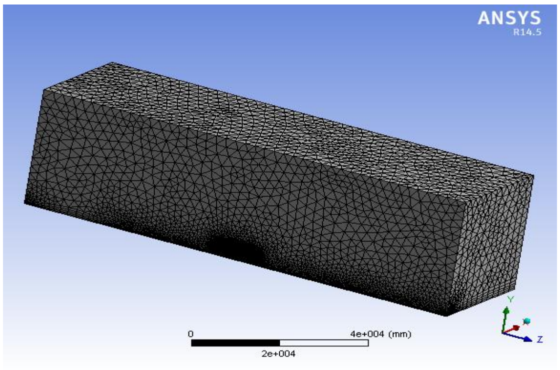
330556 nodes, binary

330556 node flags, binary

II. MODEL 2



Model 2 inside Enclosure

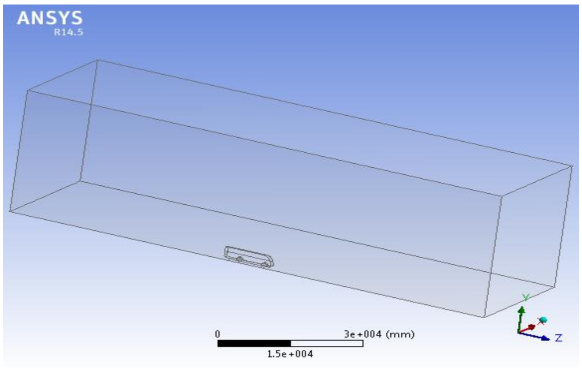


Meshed Model 2

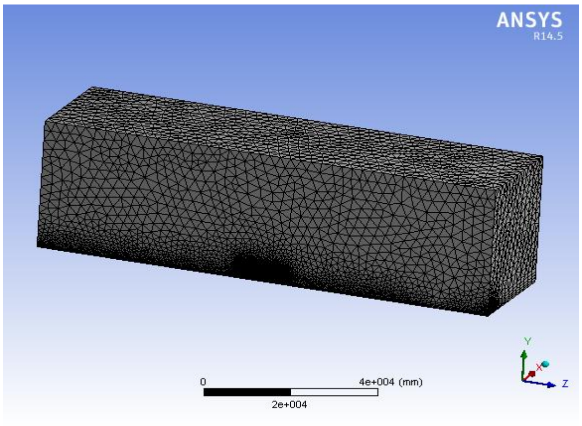
Fig.7: Model 2 inside Enclosure and Meshed Model 2

Element Details:
1391681 mixed cells, zone 3, binary.
10381 triangular wall faces, zone 12, binary.
1342 mixed symmetry faces, zone 11, binary.
3864 triangular wall faces, zone 1, binary.
2878769 mixed interior faces, zone 2, binary.
37997 triangular wall faces, zone 6, binary.
568 mixed velocity-inlet faces, zone 7, binary.
586 mixed pressure-outlet faces, zone 8, binary.
14204 mixed symmetry faces, zone 9, binary.
1158 triangular symmetry faces, zone 10, binary.
301236 nodes, binary
37078 nodes, binary

III. MODEL 3



Model 3 inside Enclosure



Meshed model 3

Fig.8: Model 3 inside Enclosure and Meshed Model 3

Element Details:
1380386 mixed cells, zone 3, binary.
10448 triangular wall faces, zone 12, binary.
1348 mixed symmetry faces, zone 11, binary.
3768 triangular wall faces, zone 1, binary.
2855644 mixed interior faces, zone 2, binary.
37690 triangular wall faces, zone 6, binary.
554 mixed velocity-inlet faces, zone 7, binary.
580 mixed pressure-outlet faces, zone 8, binary.
14258 mixed symmetry faces, zone 9, binary.
1140 triangular symmetry faces, zone 10, binary.
298950 nodes, binary
36931 nodes, binary

IV. RESULTS AND DISCUSSIONS

4.1 Aerodynamic Pressure and Velocity Distribution of the Models at Various Speeds

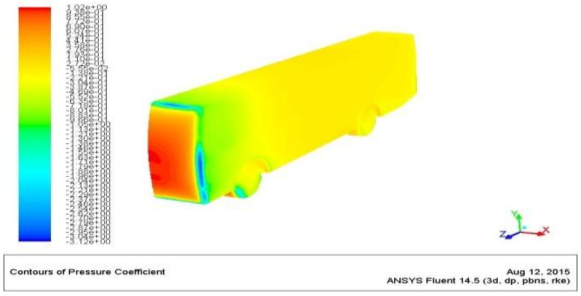


Fig.13 (a): Pressure distribution of Model 1 at 85 Kmph

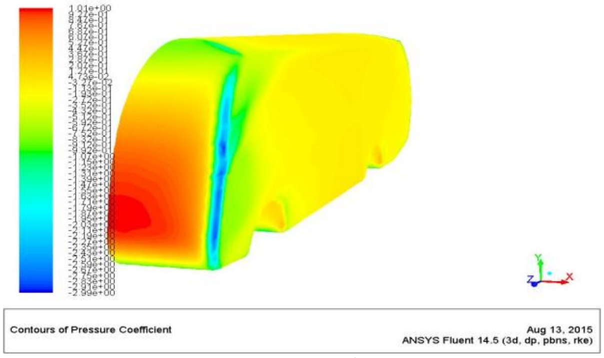


Fig .13(b): Pressure distribution of Model 2 at 85 Kmph

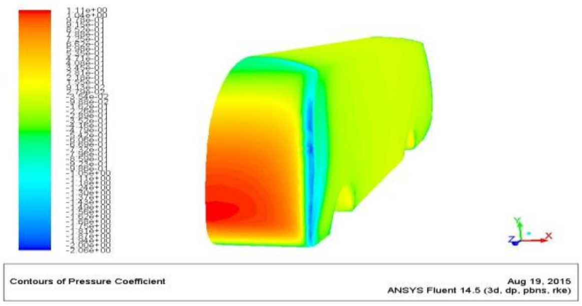


Fig.13(c): Pressure distribution of Model 3 at 85 Kmph

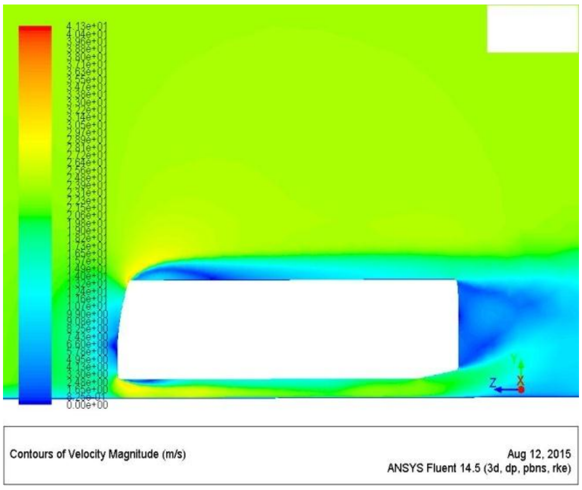


Fig.14 (a): velocity distribution of Model 1 at 85 Kmph

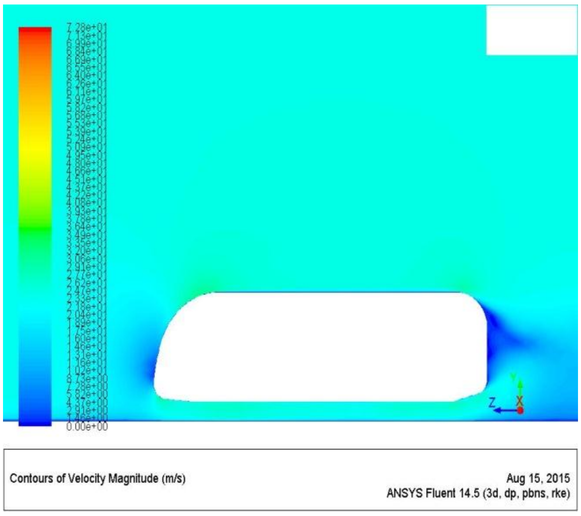


Fig.14 (b): Velocity distribution of Model 2 at 85 Kmph

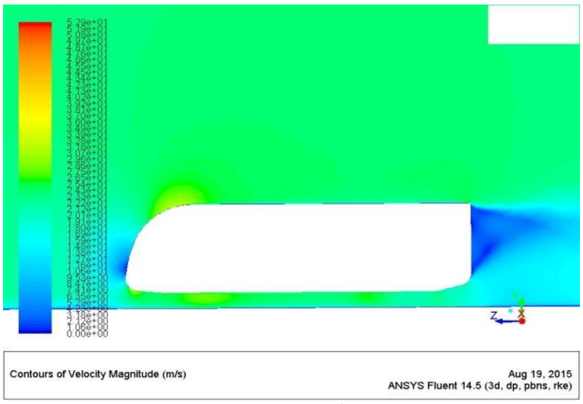


Fig.14(c): Velocity distribution of Model 3 at 85 Kmph

4.2 Drag Coefficient

Table 4: Drag coefficient (C_d) at various speeds

Kmph	Speed in m/s (v)	Model1	Model2	Model3
85	23.61	0.525	0.328	0.418
100	27.78	0.533	0.338	0.452
115	31.94	0.557	0.377	0.476

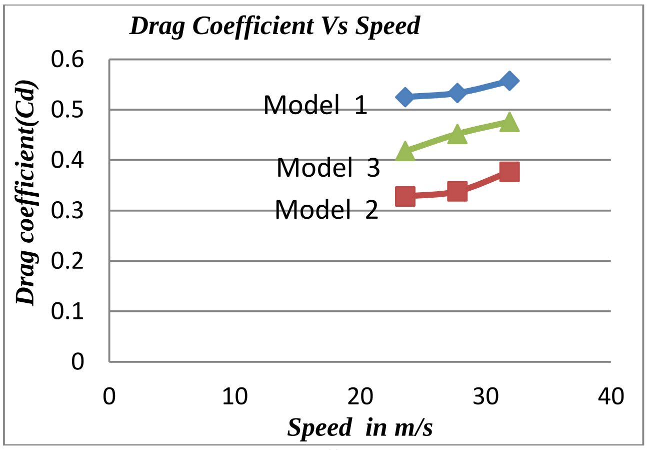


Fig.15: Drag Coefficient Vs Speed

Table 4 and figure 15 show the comparison of drag coefficient acting on the base line model and the new designs at different vehicle speeds. Clear decrease in the drag coefficient is visible in the analysis.

From the above results the percentage of drag coefficient reduction from model1 to model 2 is approximately 38 %, 37%, and 32% at 85, 100 and 115 kmph respectively and on average 35.7%. From model1 to model3 is approximately 20%, 15%, and 15% at 85, 100 and 115 kmph respectively and on average 16.7%. From model 3 to model 2 is approximately 22%, 25%, and 21% at 85, 100 and 115 kmph respectively and on average 22.7%. These reductions are mainly a result of the shape and size of the frontal area of the buses

Thus, the Drag Coefficients are not constant, but depends on a number of factors, including; Shape of object, the orientation relative to the flow and the fluid's viscosity, mass, density, flow speed and object size.

4.3 Drag Forces

A. Total Drag Force (N) [58]

$$F_d = 1/2\ \rho\ A\ C_d\ V^2$$

Total Drag Force = Pressure Force +Viscous Force

Table 5: Total Drag Force (N)

Kmph	Speed in m/s	Model1	Model2	Model3
85	23.61	656.37	397.79	527.30
100	27.78	917.85	533.28	673.36
115	31.94	1265.70	812.42	1027.87

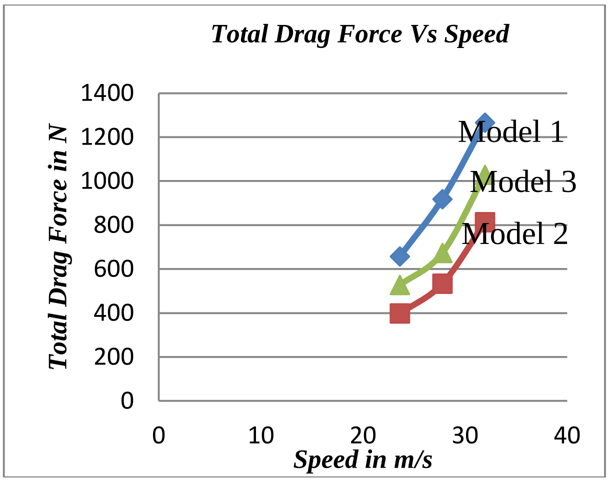


Fig.16: Total Drag Force Vs Speed

Table 5 and figure 16 show the comparison of drag force acting on the base line model and the new designs at different vehicle speeds. Clear decrease in the drag force is visible in the analysis and the total drag force is reduced from 656.37 N to 397.79 N at 85 kmph with model 2 and 656.37 N to 527.30 N at 85 kmph with Model 3. This is an improvement of 39.4% and 19.7% respectively. The total drag force for model1 decreases quickly for model 2 and it slightly decrease for model 3.

4.4 Calculation of Fuel Consumption [24]

Percentage fuel reduction = 3/5[percentage total drag reduction]

Table 6: Fuel saving in (%)

Models	Speed(kmph)	Drag coefficient(C_d)	C_d reduction(%) from baseline	Fuel saving (%)
Model1 1	85	0.525	---	---
	100	0.533	---	---
	115	0.557	---	---
Model 2	85	0.328	38	22.8
	100	0.338	37	22.2
	115	0.377	32	19.2
Model 3	85	0.418	20	12
	100	0.452	15	9
	115	0.476	14.5	8.7

Table 6 shows a maximum of 22.8% fuel can be saved in Model 2 at 85 kmph from Model 1 at the same speed. Which means 22.8% of the total fuel consumed by Model 1 can be saved. The average fuel consumed by Model 1 per 100 km is 23L (Table 14). From this the bus Model 2 can save 22.8% of 23 L, which is 5.24 L. Therefore, Bus Model 2 consumes 17.8 L per 100 km.

4.5 Lift Coefficient

Table 7: Lift Coefficient at various Speed

Kmph	Speed in m/s (v)	Model1	Model2	Model3
85	23.61	-0.143	-0.020	-0.202
100	27.78	-0.147	-0.029	-0.347
115	31.94	-0.177	-0.109	-0.767

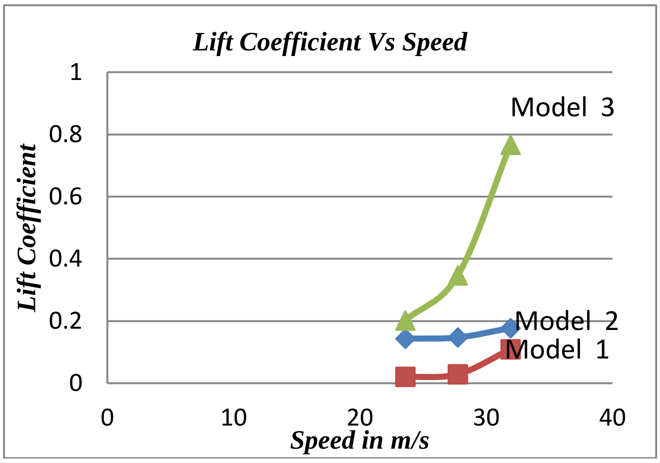


Fig.17: Lift Coefficient Vs Speed

4.6 Lift Force

Table 8: Lift Force at Various Speed

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

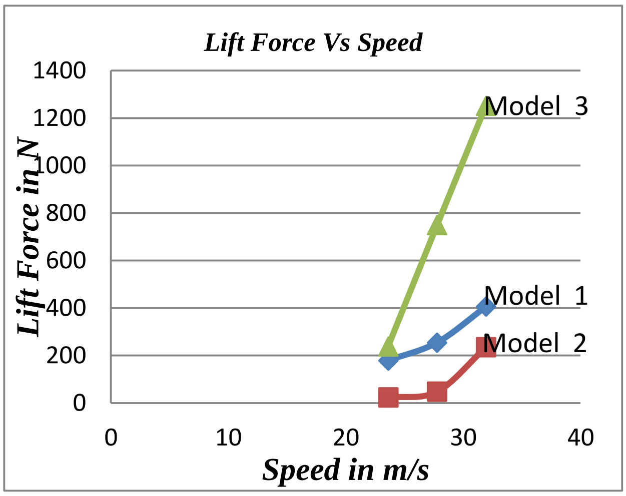


Fig.18: Lift Force Vs Speed

Table 8 shows the values of lift forces are negative, this indicates the lift forces are downward lift and which is very useful to improve the vehicle's road holding capacity and reduce steering instability of the bus. But, even though there are adequate reductions of drag coefficient, drag forces and power while using Model 2, its downward lift force is very less compared to Model 1 and Model 3. This needs to modify the under body profile (shape) and add additional downward force improvement methods like, under body dam, negative rear spoiler, etc. Figure 18 shows the comparison of lift force acting on the base line model and the new designs at different vehicle speeds. Clear increase in the lift force is visible in Model 3. Therefore, an additional underbody designs (mentioned above) are required for Model 1 and especially in Model 2. The lift force is reduced from -178.976 N to-23.282 N at 85 kmph in Model 2(Table 14). Which is 87% reduction of lift force and the lift force is increased from -178.976 N to -238.234 N in Model 3. This is an improvement of 33%.

V. CONCLUSION

In the process of redesigning, exterior styling with improved aerodynamics of existing cross country bus (Yutong bus) plying on Ethiopian roads, a detailed computational analysis has been done. This computational analysis shows that there is possibility of improving the aerodynamic performance of bus by modifications in exterior design of bus body. These modifications are helpful in reducing the coefficient of drag which affects the fuel consumption. A three dimensional flow analysis has been performed on the models at various speeds to predict the airflow characteristics around bus. The results provide the flow pattern and associated drag of the bus body.

The three buses body has been modeled for performing numerical analysis using CFD software. The Bus No.1 is the existing model, Bus No.2 is the existing model with the front and rear end modified, and Bus No.3 is the existing bus with modification at front end and partially rear end. The three models have been separately tested for optimizing how each modification is contributing towards the drag force.

Velocities given to the fluent analysis are 85 kmph, 100 kmph, and 115 kmph. It was found that the least drag force was acting on Bus No.2. Bus No.3 gave an intermediate result as expected. The percentage reduction in drag force of Bus No.2 from Bus No.1 is found to be 39.4%. The average drag coefficient of the baseline model is 0.54, model 2 is 0.35 and model 3 I got is 0.45. By these modifications the coefficient of drag is reduced by model 2 and model 3 are approximately 35.7% and 16.7% respectively. Fuel consumption of about 22.8% can be reduced from 85 kmph to 115 kmph. This improvement in fuel efficiency will have a high impact on the reduction of annual fuel consumption. Hence the aim has been achieved.

REFERENCES

[1] Ahmad Kamal Matamin, "Aerodynamic Study of a Coach Using Computational Fluid Dynamic (CFD) Technique", August 2006
[2] Ahmad Kamal Matamin, "Aerodynamic Study of a Coach Using Computational Fluid Dynamic (CFD) Technique", August 2006
[3] A. Huminic and G Huminic. Numerical Flow Simulation for a Generic Vehicle Body on Wheels with Variable Underbody Diffuser. SAE Technical Paper 2012-01-0172,2012, 2012.
[4] AIS – 052 - (2008), "Automotive industry standards code of practicefor bus body design and approval (First Revision)", The automotive research association of India on behalf of automotive industry standards committee.
[5] D Bitchtas. Aerodynamic evaluation of hgv trailer upper surface curvature. Master's thesis, Cranfield University, 2012.