

**CFD ANALYSIS FOR DRAG FORCE REDUCTION IN BUSES**Mr. Ganga Singh¹, Assistant Prof. Mr. Vinod Kumar²¹PG Scholar, Department of Mechanical Engineering, JCDM College of Engineering, Sirsa (Hry), India²Assistant Professor, Department of Mechanical Engineering, JCDM College of Engineering, Sirsa (Hry), India

ABSTRACT- The road transportation vehicles are uneconomic nowadays. Because a vehicle after consuming fuel energy in the form of diesel, petrol, CNG etc. Perform an inefficient task of speedy motion on the road. We are calling inefficient because no engine has 100% efficiency, so some of the energy waste in form of heat and other by the vehicle aerodynamics itself. We cannot save this waste energy but one can minimize it by using more efficient aerodynamic design. The fuel price rising and strict government regulations are also a big problem. The bus body building companies' focus on exterior styling only and ignore the aerodynamics. The exterior design of the present intercity buses is very poor; due to this high drag force is produced. Aerodynamically efficient designs can reduce engine load and fuel consumption is also reducing. The aim of this project is using CFD to modify the structure of intercity bus in order to decrease the effect of drag force of the vehicle which in turn leads to reduced fuel consumption of the vehicle. Two prototypes of bus body have been modeled in CATIA and imported into ANSYS Workbench software, which is named as baseline model and modified model. Baseline model is the existing intercity bus model and Modified model is with modification of existing intercity bus. The Modified model is used to reduce the drag force by modifying and analyzing by using CFD. Due to this reduced the fuel requirement and increase the performance occurs as a result of reduced drag force. The overall aerodynamic drag force is reduced.

KEYWORDS- Drag force, aerodynamics, ANSYS, CFD, inter-city buses, CATIA.

1. INTRODUCTION:

The automotive industries concentrate on reducing the fuel consumption of their products due to the climate change. The legislation and governments push the companies to reduce the emissions of their vehicles. The emission can be reduced by different approaches. One of them is to use environmentally friendly fuel and other technologies such as powertrain, electric or hybrid. To make the existing technology more efficient a lot of work has been done; such as, combustion is optimized, and internal friction has been reduced e.g. spark timing, direct Injection, turbo charging etc. The most effective approach is to a reduction in driving resistance of the vehicle. The following expression can determine the required force to propel the vehicle:

$$F_{\text{req}} = F_D + F_R + F_A + F_G$$

Where,

F_D = Force due to aerodynamic drag

F_R = Force due to rolling resistance

F_G = $W \sin$ - climbing resistance due to gravity

F_A = force needed to accelerate the vehicle

Hence the rolling resistance, the mass, and the aerodynamics are the main parameters that can be improved by automotive industry. Aerodynamic resistance and rolling resistance are approximately equal in 100km/H generally for a typical passenger vehicle and the main source of driving resistance at highway velocity is an aerodynamic drag while the rolling resistance stays more or less constant.

1.1 OBJECTIVE:

The aim of this project is designing of Ashok Leyland bus into computer modeling software named as CATIA and imported this model into analysis software named as ANSYS for computational fluid dynamic analysis of aerodynamics and improves the aerodynamic design of current intercity bus for required.

- To use CATIA V5 CAD modeling software to model an existing Ashok Leyland bus, as a baseline model.
- To perform the flow analysis of the baseline model using ANSYS CFD tool fluent.

- To design a new model of a bus with reduced drag force.
- Perform flow analysis on the new models.
- To reduce drag force along with achievement of better fuel efficiency

2. Model Development:

Automobile industry researchers were focused on the sedan, race car aerodynamics rather than heavy vehicles because of market forces at consumer preferences and work. However, there were few types of research like research conducted by G Amba Prasad Rao and Sachin Thorat in 1999 on “Computational Analysis of Intercity Bus with Improved aerodynamic performance on Indian roads and aesthetics, the drag force reduced by 30%. Robert R Meyer and Edwin j Saltzman in 2007 conducted studies on drags reduction of trucks and buses. The final model prepared with rounded, vertical and horizontal corners, smoothed a boat tail and underbody achieved Cd value of 0,242. (Peterbilt Motors Company, 2009) presents a white paper on Heavy vehicle fuel efficiency and aerodynamics. In this paper, the aerodynamic drag losses due to which vehicles utilize large capacity engines are reviewed. In 2007, G Buresti conducted research on methods of their application to heavy road vehicles and to reduce the drag of bluff bodies which declared that to reduce the drag of bluff body, the boat tailing design has been applied and this lead to reduce the drag about 5% to 10%. A device occupies between Trailer-Tractor gap which ensures both easy movements of the trailer around the turns without any clashing and drags reduction. Flow deflection devices and fairings are provided to reduce pressure drags on axle and trailer base. Wheel housings have been provided around wheels of the vehicle because wheels are the main source of aerodynamic drag.

In 2013, E Selvakumar conducted research on “Aerodynamic exterior body design of bus”. In this, the experimental numerical tests were conducted in wind tunnels and effectiveness of new concept design. The drag can reduce by 30% to 40% in new concept than the existing bus and the test result show that for every 100km 6 to 7 liters of fuel is used.

In 2007, Panu Sainio conducted a research from aalto University on aerodynamics possibilities for heavy vehicles aerodynamics solution is boat tail approach. This approach may increase the length of the vehicles but trailing edge blowing is the solution of this problem due to this aerodynamic performance same with same tail length.

In 2004, Mc Callen concludes that by removal of rearview mirror alone vehicle drag will reduce by 4.5%. Flow circulation and flow separation will occur as a result of the presence of any gap in the vehicle body. The front leading edge radii reach 150mm and drag value gets reduced.

2.1. Methodology:

Numerical simulations of two different vehicle configurations are done in this research. Required modifications can be identified by interactive colour graphics of baseline stimulation results. The Computational Fluid Dynamics process described below can be performed for each configuration, which includes three steps; pre-processing, solving and post-processing.

2.2. Boundary Conditions:

Boundary Condition is a very important step. This was applied to the meshed model. The first condition was only straight wind was considered in this simulation at different speeds of the vehicle from 10Km/hr to 100km/hr. The constant wind velocity applied in order to replicate the wind tunnel test, constant velocity inlet condition was applied at the inlet. With the atmospheric pressure as operating pressure, zero gauge pressure was applied to the outlet.

3. Modeling:

Consider the intercity bus model as the baseline model. The dimensions of the bus were considered to the actual design of intercity bus. The outer body of a designed bus was similar to the intercity bus.

The shape of the body plays a vital role in designing for the aerodynamic analysis and is more responsible for the formation of drag force. Therefore in the modified model, the major change has been brought to the shape of the body.

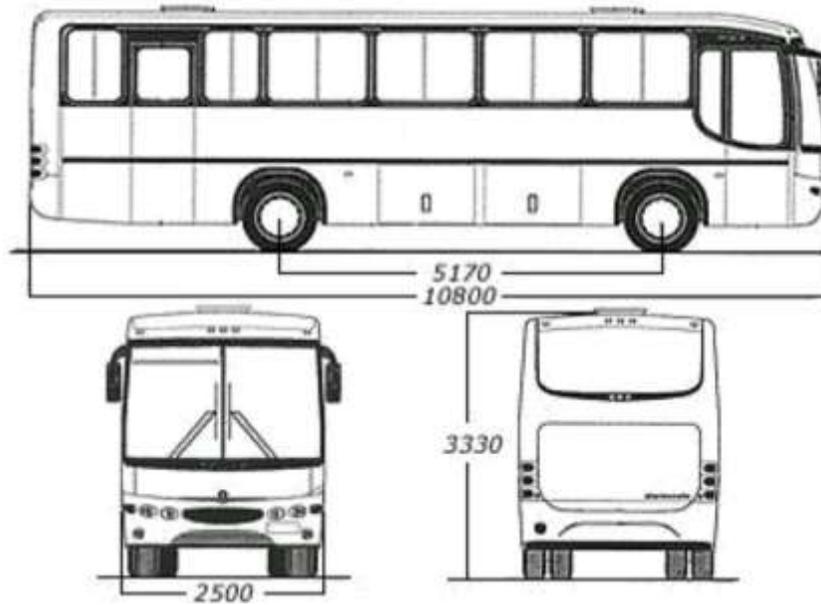


Figure 1: Outer Dimensions

3.1 Baseline Model:

The baseline model is just like the original model of actual intercity bus. In this model only major body shape is design. This is modeled in CATIA designing software. The isometric view of the intercity bus is modeled in the below figure 2.

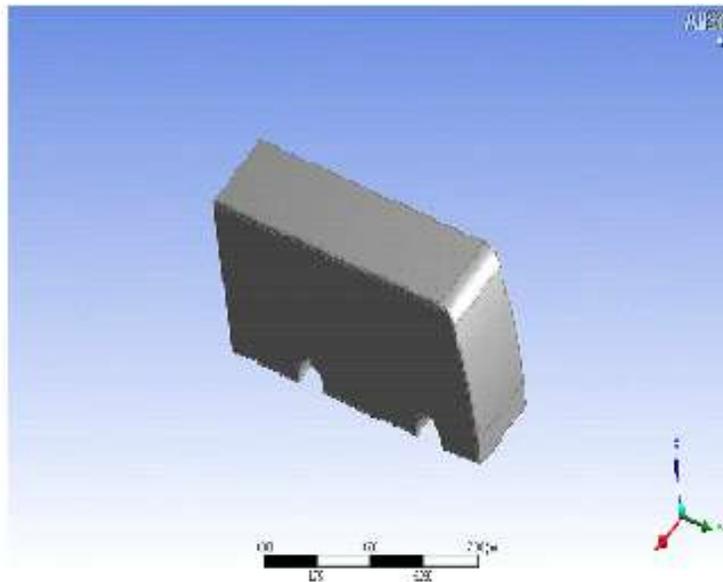


Figure 2: Isometric View (Baseline Model)

3.2 Modified Model:

The above model has been modified in their shape. This vehicle designs under the consideration of aerodynamic aspects and parameters.

Modified Aspects:

Frontal Area: The frontal area there is specially design duct which is going through front side to rear side of the bus which allow the high-pressure air pass through front side to low-pressure rear side of the vehicle by this duct.

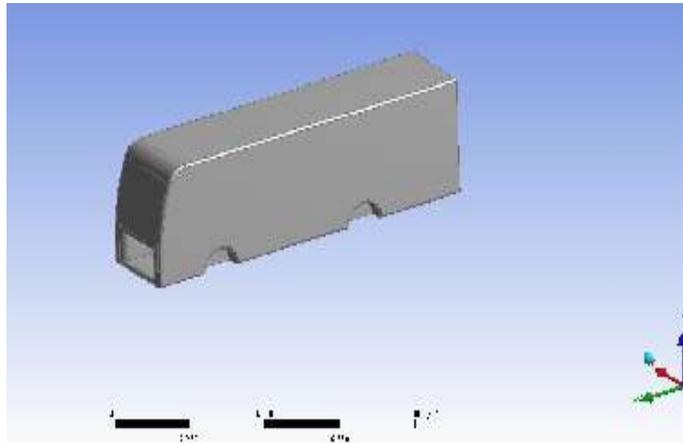


Figure 3: Isometric View (Modified Model)

4. MESHING:-

A domain of computational box is designed in such a way that the blockage is neglectable and free to pilot, which mean a box that involves of inlet and outlet as well as ground and a roof surface this domain is occupied almost such that the real-time road condition fulfilled.

The meshing is very important part of Pre-processing and Finite Element mesh is generated using parabolic tetrahedral elements. An automatic method is used to generate the mesh in the present work. Figure 4 shows the meshed model of intercity Bus in ANSYS workbench. This was generated on the surface of the geometry and the surface of the domain of the vehicle and the computational grid is created between the vehicle and domain. To internment certain area of interest (where the turbulence is high as well as departure might occur) the cell has to solve complete a vigorous solution and all irregularities. The grid has been reassignment Over the bus, and rear of the bus and specially. The grid has been over the bus front and the rear of the body and especially ducting area. The vehicle has been studied and the concentrating on the underbodies of the guidance of the flow filed.

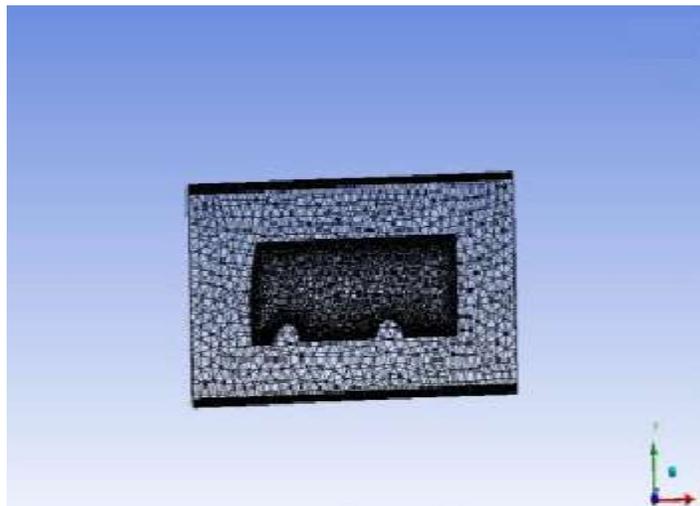


Figure 4: Meshing Model

4.1 Baseline Model

Element Details:

- 1377769 tetrahedral cell, zone 2, binary.
- 2724395 triangular interior face zone 2, binary.
- 724 triangular velocity-inlet faces, zone 6, binary.
- 726 triangular pressure-outlet faces, zone 8, binary.

- 4720 triangular wall faces, zone 7, binary.
- 56116 triangular wall faces, zone 1, binary.
- 245905 nodes, binary.
- 245905 node flags, binary.

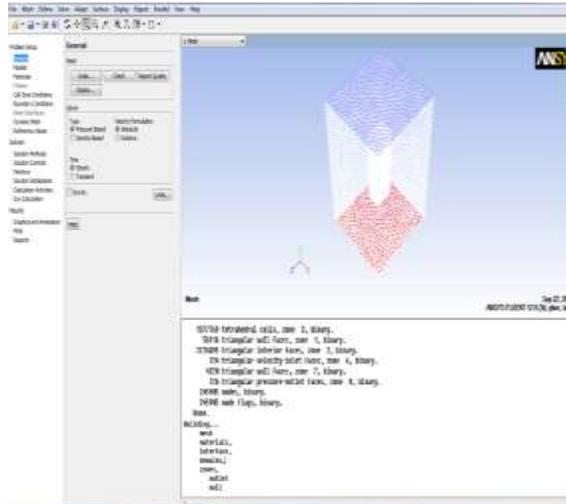


Figure 5: Meshing of Baseline Model

4.2. Modified Model

Element Details

- 1463795 tetrahedral cell, zone 2, binary.
- 2893034 triangular interior face zone 2, binary.
- 714 triangular velocity-inlet faces, zone 6, binary.
- 738 triangular pressure-outlet faces, zone 7, binary.
- 4744 triangular wall faces, zone 8, binary.
- 62916 triangular wall faces, zone 1, binary.
- 34550 nodes, binary.
- 227448 nodes, binary.

5. Results and Discussion:

We analyze and discuss the results in various categories like pressure and velocity Contour distribution on the vehicle in interactive colour graphics or drag force, power resistance, rolling resistance, total resistance and are depicted with the help of a graph and tables.

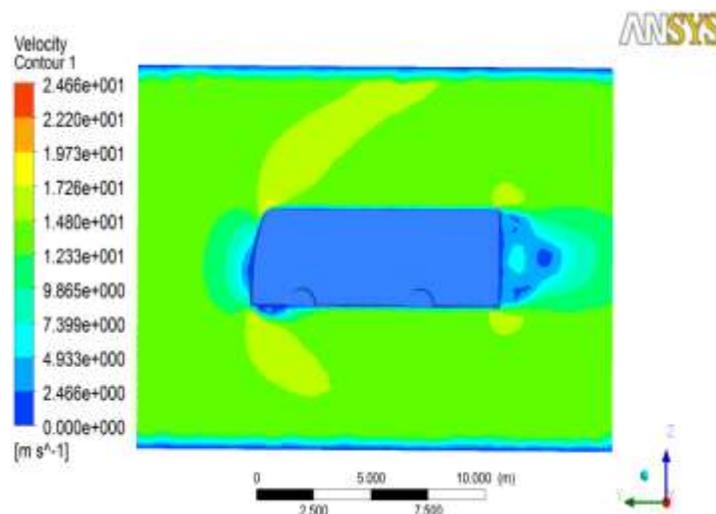


Figure 6: Velocity Contour of Baseline Model of Bus (Side view)

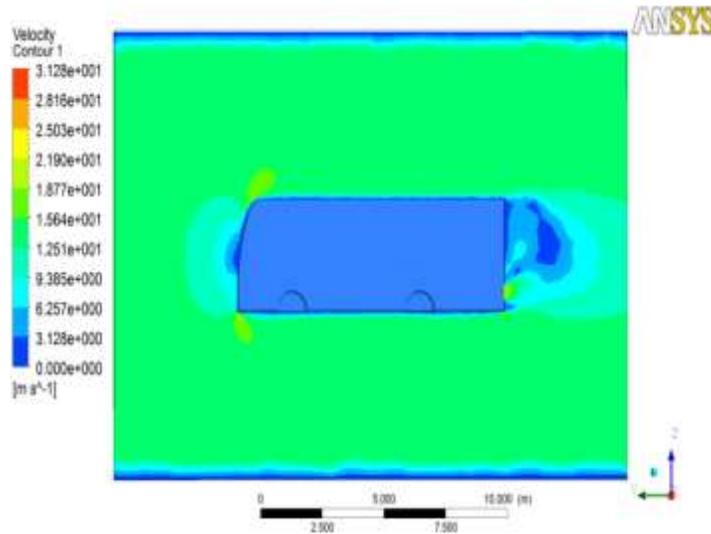


Figure 7: Velocity Contour of Modified Model of Bus (Side view)

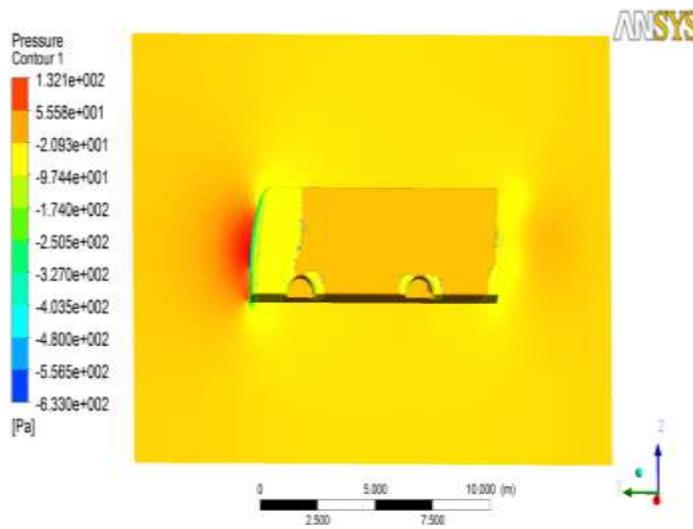


Figure 8: Pressure Contour of Modified Model of Bus (Side view)

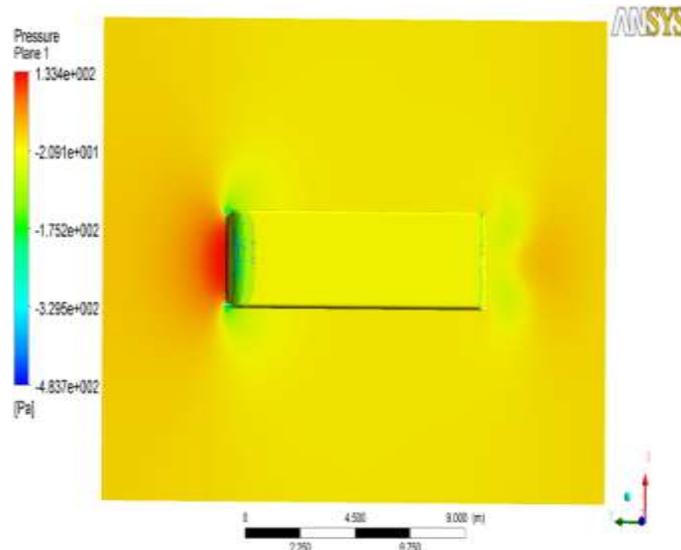


Figure 9: Pressure Contour of Baseline Model of Bus (Top view)

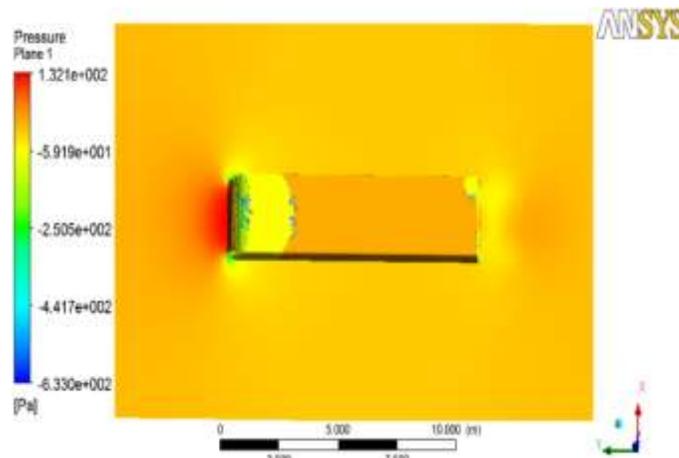


Figure 10: Pressure Contour of Modified Model of Bus (Top view)

5.1 Calculation of Drag Force: $F = \frac{1}{2} \times \rho \times A \times V^2 \times C_D$

ρ = Density of Air, A = Frontal area, V = Velocity, C_D = Drag Coefficient

CALCULATION OF DRAG FORCES IN BASELINE MODEL AND MODIFIED MODEL		
Velocity (Km/h) V	Drag Force of Baseline Model F_D	Drag Force of Modified Model F_{D1}
10	33.96	28.23
20	135.42	109.05
30	304.018	256.59
40	536.68	460.69
50	854.71	599.83
60	1207.33	879.64
70	1627.12	1197.47
80	2121.42	1532.47
90	2686.5	2195.66
100	3355.87	2748.82

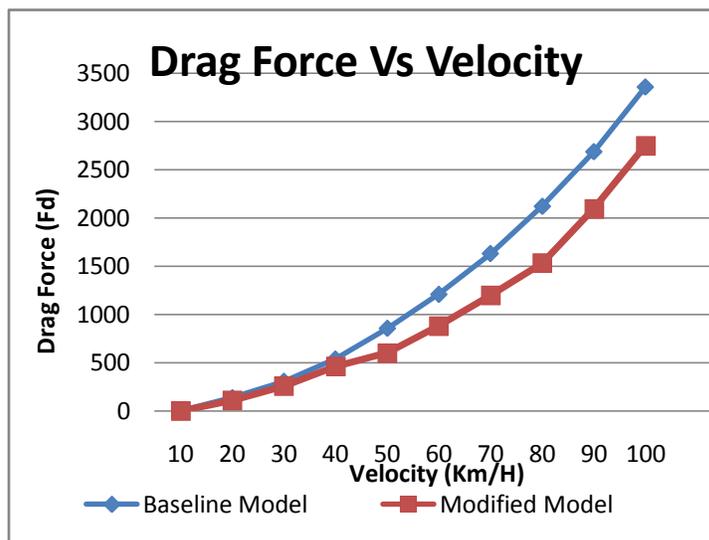


Chart 1: Drag Force

5. ROLLING RESISTANCE (N):

$$F_R = (0.015 + 0.00016V) \times W$$

ROLLING RESISTANCE AT VARIOUS SPEED		
SPEED(KM/Hr)	BASELINE MODEL(F_R)	MODIFIED MODEL(F_{R1})
10	250.207	250.207
20	257.411	257.411
30	264.591	264.591
40	271.797	271.797
50	279.002	279.002
60	286.208	286.208
70	293.388	293.388
80	300.594	300.594
90	307.801	307.801
100	315.005	315.005

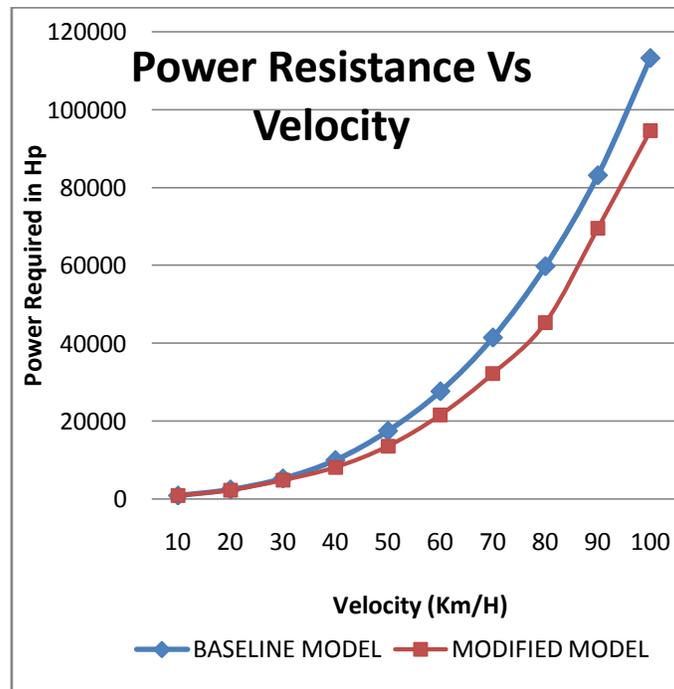
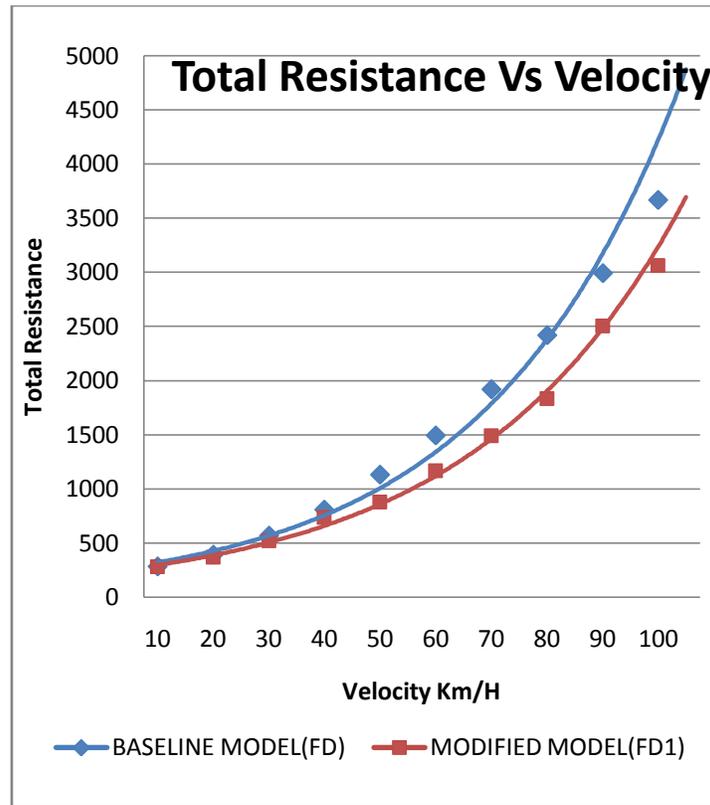
Total resistance (F_t)

$$F_t = F_D + F_R$$

FD: - Drag Force

FR: - Rolling Resistance

TOTAL RESISTANCES AT VARIOUS SPEED		
SPEED(KM/Hr)	BASELINE MODEL(F_T)	MODIFIED MODEL(F_{T1})
10	284.167	278.641
20	392.831	366.461
30	568.609	521.181
40	808.477	732.487
50	1133.712	878.832
60	1493.538	1165.848
70	1920.508	1490.858
80	2422.014	1833.064
90	2994.301	2503.461
100	3670.875	3063.825



P = Power resistance, η_t = Transmission efficiency

$$P = \frac{F_t \times V}{\eta_t}$$

POWER RESISTANCE AT VARIOUS SPEEDS

SPEED(KM/H)	BASELINE MODEL	MODIFIED MODEL
10	877.760	860.691
20	2426.822	2263.914
30	5262.792	4823.819
40	9980.199	8137.930
50	17496.955	13563.307
60	27663.642	21594.095
70	41482.972	32202.532
80	59796.834	45256.313
90	83175.027	69540.583
100	113307.675	94570.065

6. Conclusion:

A Computational Fluid Dynamic analysis has been done using ANSYS workbench on the intercity buses that run in the Indian roads with some modification in their structure which affects the aerodynamics of the bus.

The original structure of Baseline model was modified from frontal area there was specially design duct which went through front side to rear side of the bus which allow the high-pressure air pass through front side to low-pressure rear side of vehicle, this high-pressure air can further used in useful work and mathematical analysis has been done using CFD software. There are two types of the prototype used first is Baseline model which is an Original version of intercity bus and the second model is a Modified version of the intercity bus. Analysis and calculations have been done for both the cases.

The maximum velocity that has been considered is 100 km/Hr and the drag coefficient for the baseline model is 0.85 and it has been decreased in the case of the modified model to 0.71.

By following these criteria the drag has been decreased to approximately 16%.

Drag force has been reduced and performance of bus gets increases due to this the fuel consumption decreases and hence the efficiency of the bus increases.

REFERENCES:

1. G.buresti, G V lungo and Lombardi (2007),”carried out a Research on Method for the drag Reduction of bluff bodies and their application to heavy Road.”
2. MC Callen (2004),” investigated on the reduction in drag value until the front leading edge.”
- 3.E Selvakumar, M K Murthi, and A Murthi (2013),”Conducted Research on the Aerodynamic Exterior body design of bus.”
- 4.Sachin Thorat And G Amba Prasad Rao(1990),” conducted Research on Computational Analysis of intercity Bus with improving Aesthetics and Aerodynamic performance on Indian Roads.”
- 5.Peterbit(2009),”Motors Company Present a White Paper on heavy vehicle Aerodynamics and fuel efficiency”.
- 6.Edwin J Saltzman, Robert, R Meyer, MC Callen and K Salari (2007),”International Papers Carried out studies on Reducing the Drag of Trucks and Buses”.
- 7.Panu Sainio, Kimmo Killstrom and Matti Juhala(2007),”From Aalto University Conducted a Research on Aerodynamics Possibilities for Heavy Vehicles”.