Computational Fluid Dynamic Analysis on Bus Shell to Improve Efficiency by Reduction in Drag Force

Sanjeev Tiwari[1], Dr.Satyendra Sharma[2]

[1]RGPV Bhopal
[2]Professor, SAGE University, Indore

Abstract - India launched second Automotive Mission Plan (2016-2026) [1][2] in 2016 and following is the vision statement,

AMP 2026: "Vision 3/12/65": "By 2026, the Indian automotive industry will be among the top three of the world in engineering, manufacture and export of vehicles and auto components, and will encompass safe, efficient and environment friendly conditions for affordable mobility of people and transportation of goods in India comparable with global standards, growing in value to over 12% of India's GDP, and generating an additional 65 million jobs".

As can be seen from the above, automobiles play a very important role in the county [3], be it contribution to GDP or number of people employed. Currently only 22 people per 1000 own cars in India. If compare this to the developed economies USA 890, UK 850 and even China is as at 164, we realize that it is bound to go up and above goals set by the GOI are actually attainable.

Also, automobiles play a very important role in increasing the reach of people in rural areas, thereby improving their livelihood, reducing influx to cities, and delivery of goods and services. Thus, contributing to much more than the targeted 12% of GDP. The importance of the sector in development of any country cannot be overemphasized.

Key Words: CFD (Computational Fluid Dynamics), Automotive Sector, Fuel Efficiency etc. Aerodynamics, Drag

1. INTRODUCTION

Fuel efficiency of a vehicle is very important in deciding whether the vehicle will be successful in the marketplace or not. The cost of fuel in India is very high as compared to several countries in dollar term. Secondly our currency is weaker and thus as an Indian we pay much higher price in purchasing power parity. This makes our markets more and more sensitive to the cost per kilometer. We all remember Maruti Suzuki's campaign that focused on "Kitna Deti Hai". It was a hit campaign and helped in boosting their sale. That is one of the reasons why they hold close to 50% market share in a market that has more than 12 players.

Another aspect of fuel consumption is related to environment. The more fuel we consume as a whole, the more we pump Carbon Dioxide (Co2) which is a Green House Gas which causes global warming. Vehicle's tailpipes also emanate following hazardous gases; like

1. Carbon Monoxide,
2. Oxides of Nitrogen,
3. Sulphur Dioxide,
4. Particulate matter
5. Hydro Carbons [4]

Our country unfortunately has the distinction of having 12 of the world's 20 most polluted cities [5]. Automobiles are responsible for substantial part of the same. Despite the fact that our country has only 22 car per thousand people, the pollution levels are very high, it means substantial part of this should be coming from Commercial Vehicles. That is the reason why the Supreme Court ordered for conversion of entire fleet to CNG from diesel as early as 1998 [6]

Thus, this paper deals with improvement of Coefficient of drag in commercial vehicles, especially buses.

So, any which way we look at it, be it commercial, environmental or foreign exchange point of view, the importance of fuel efficiency improvement, more so, in commercial vehicles cannot be overstated.

Fuel efficiency depends of many parameters of vehicles including engine design, aerodynamic drag, weight, fuel type and driving skills etc. Indian automotive sector is the fourth largest petroleum consumer after china and USA also increased 5.3% petroleum products in this year. Indian government fixed a target to reduce 10% reduction on import of petroleum products [7].

Our country did not have good roads till a few years ago or had automobiles capable of running at higher speeds. This resulted in a very modest average kilometers per hour even on highways. But the scene is fast changing on both counts.

Recently Mr. Nitin Gadkari, the Minister of Road Transport and Highways of India (MoRTH) announced that the country is developing the highways that will have maximum permissible speed of 140 Km/Hour [8]. This means that the vehicles will be travelling at much higher speeds. Thus, aerodynamics becomes a major factor in improvement in fuel efficiency, as can be seen in the table below.

Table-1

Vehicle Speed	Aerodynamic Drag	Rolling & Accessory Drag
20 mph (32 kph)	28%	72%
33 mph (53 kph)	33%	66%
40 mph (64 kph)	36%	64%
50 mph (80 kph)	50%	50%
60 mph (96 kph)	62%	38%
65 mph (105 kph)	67%	33%
70 mph (113 kph)	70%	30%

Relative magnitude of drag components. [9] CSTT, 2012

With better roads and more powerful automobiles aerodynamics can go a long way in improving fuel efficiency. While this will result in savings to the operator of the bus, it will reduce pollution and help in conserving precious foreign expense because we import 82.5% of our needs of crude oil close to 100 Billion USD and is the single largest item comprising of 25.2% of the total import [10].

Aerodynamics is as old as cars themselves, but it did not acquire great significance in early evolution of automobiles because the fuel was cheap and the speeds were low. However, lot of research was done in aeronautical science as the countries needed faster and faster planes not only to transport people and goods but also to win wars. The lessons learnt from aeronautics were later applied to racing cars and then to regular automobiles so that the drag could be minimized resulting in increased speeds and reduced fuel consumption.

Here density of air remains constant (@ sea level 1.225 kg/m3 at 15 Degree Celsius [11]), Cross sectional area is dictated by usage and styling of the vehicle. We would want to drive at higher speeds [8] to save time, and then we are left with only Cd to play around with to reduce the drag in order to improve the fuel efficiency.

There are only 2 ways to find out the value of drag. One is through wind tunnel experimentations [12]. Firstly, with scaled models and then with full scale models. And the second one is through Computational Fluid Dynamics. Many softwares like Ansys Fluent, Solid Works etc. are available that can simulate the wind tunnel testing and give fairly accurate results. As can be see testing in wind tunnel can be expensive and time consuming, more so when one considers large number of iterations before the design is finalized.

There are some enhancement techniques to improve airflow around the vehicle thus improving the Coefficient of drag Cd.

- Creating a round edges at front of the vehicle
- Experiment with front fascia grill shape
- Design of rear spoiler to reduce lift value.
- Design a side skirts for airflow around the vehicle.
- Aerodynamic wheel shape

2. OBJECTIVE

An improvement in fuel efficiency is good for individuals, countries and the environment and every effort should be made to achieve this.

In this study we are going to attempt to reduce fuel consumption of a bus by altering its external profile so that we can improve its coefficient of drag (Cd) and thus reduce aerodynamic drag.

3. METHDOLOGY

The primary objective of this research work is to improve fuel efficiency of a bus by reducing the aerodynamic drag. This will be performed through Computational Fluid Dynamics (CFD) analysis of a bus body shell and reduction of the drag force value by changing its shape. Which may reduce the fuel consumption and improve the efficiency of the vehicle? In this research we started our work with previous research on the same field. Firstly, we performed a CFD analysis of simple vehicle design and then modified the same to improve drag coefficient value.

To perform a computational fluid dynamic analysis, we used Ansys 18.1 as a pre-processor and fluent as a solver. So, we need to import CAD data into Ansys Design modeler and develop enclosure as specified dimensions then export this data in to mesh modeler where we mesh it. Start with simple tetra meshing using .5mm size, import patch conforming method then import body sizing and body sizing to re-mesh it. At last import inflation on bus surface with first layer first layer thickness .3mm. assign boundary condition names like inlet, outlet bus surface and wall. Import this data into fluent launcher with double precision option and consider 4 processors

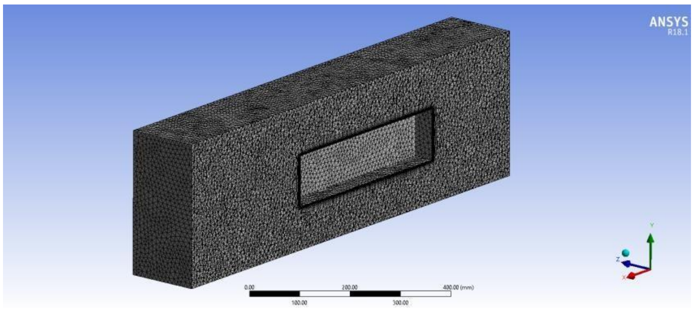


Fig -1: Mesh Model

In fluent solver setup the desk as per analysis.

Setup → Models → Energy → On

Setup → Models → K-Epsilon → Realizable→ Standard wall function

Setup → Material→ Air →Density (1.225 Kg/m3) →Viscosity (1.7E-05 Kg/m-s)

Setup → Boundary Condition→ Inlet →Velocity (33.34 m/s or 120 KM/Hr)

Solution initialization → Hybrid initialization

4. Results

In this research we analyzed various designs, starting with simple base design. A basic body shape of rectangular profile was made for initial calculations.

Design 1 (Base Design)

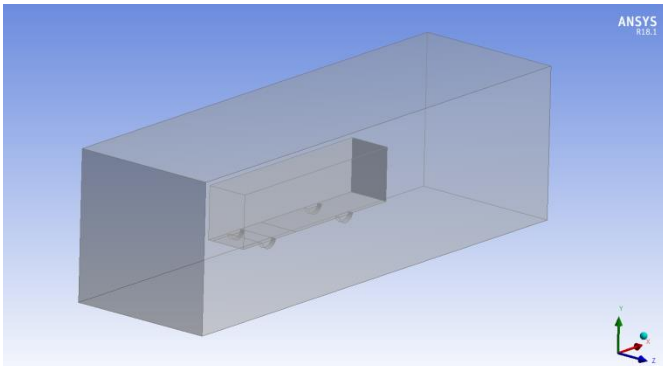


Fig -2: Base Design

Result- Drag Coefficient – 1.31 and Drag Force – 8.73165

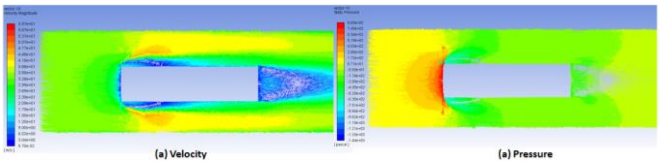


Fig -3: Results

Design 2 – As shown in figure we replaced the windshield with a curved one.

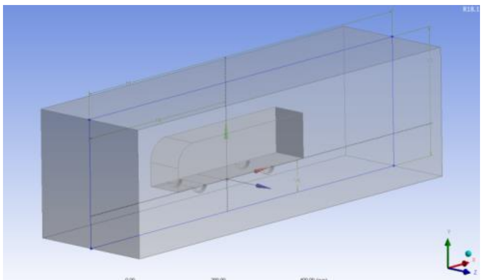


Fig -4: Design2

Results –Drag Coefficient – 1.22 and Drag Force – 7.45

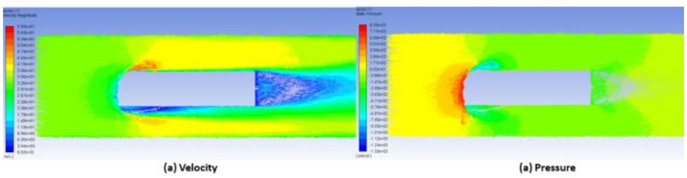


Fig -5: Results

Design 3 – As shown in figure along with front curved windshield, we create around edges at all front face and roof edges.

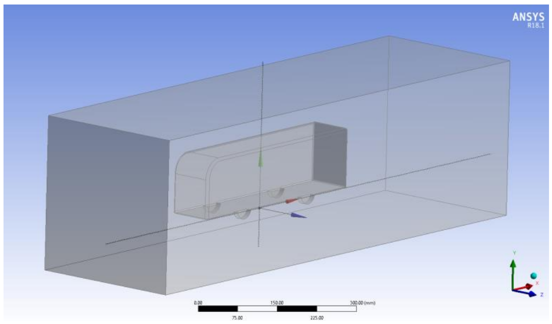


Fig -6: Design3

Results –Drag Coefficient – 0.801 and Drag Force – 5.320

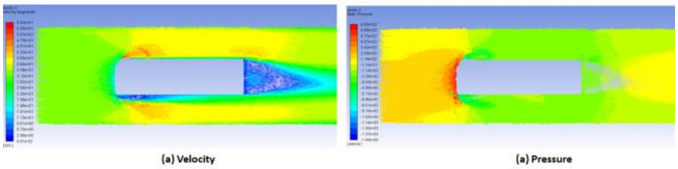


Fig -7: Results

Design 4 – In this design windshield is curved, side roof and back too is curved.

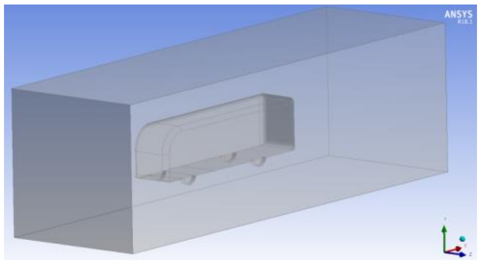


Fig -8: Design4

Results –Drag Coefficient – 0.432 and Drag Force – 2.872

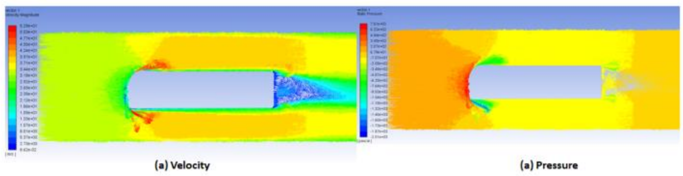


Fig -9: Results

Design 5 – After so many iterations we finally took inspiration from an existing Volvo bus and made design very similar to that.

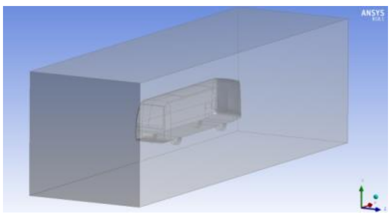


Fig -10: Design5

International Research Journal of Engineering and Technology (IRJET) e-ISSN: 2395-0056
Volume: 08 Issue: 12 | Dec 2021 www.irjet.net p-ISSN: 2395-0072

Results –Drag Coefficient – 0.3590 and Drag Force – 2.010

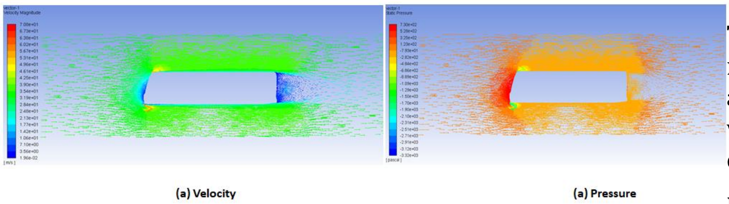


Fig -11: Results

5. Analytical Results

The Drag force is a force exerted on an object by a fluid to resist its motion. If fluid is air is named as aerodynamic drag force and the fluid is water than it is hydrodynamic drag. The drag force acts in opposite direction to the motion of the object. The drag force directly depends on density of fluid, velocity, drag coefficient and cross section area. It can be expressed using Bernoulli theorem, is a relation between pressure, kinetic energy and gravitational potential energy of the fluid, It state that in increase in the speed of fluid occurs simultaneously decrees in static pressure or potential energy.

$$\text{Constant} = P + \frac{1}{2}(\rho V^2) + \rho g h$$

Where,

p	is the pressure exerted by the fluid
V	is the velocity of the fluid
ρ	is the density of the fluid
h	is the height of the container

Pressure as force per area - F = PA and substituting the dynamic contribution of pressure.

$$F = \left(\frac{1}{2}\rho V^2\right)A$$

Now for calculation of drag force we can add 'Cd 'as coefficient of drag force, which depends on the shape of the object subjected to flowing fluid around it. The Cd also depends on texture, viscosity, compressibility and lift etc.

$$F_d = \frac{1}{2}(C_d \, \rho \, V^2 A)$$

F_d	Drag Force
C_d	Drag coefficient
ρ	Air Density
V	Velocity
A	Area of the object

6. CONCLUSIONS

The aim of this research was to improve vehicle efficiency by reducing the drag force. In computational fluid dynamics analysis, we can easily predict drag coefficient and lift value which helps engineering team to design aerodynamic shape of the vehicle.

Design1

Most of the buses in our country are made by local body builders who convert a drive-away chassis in to a bus by making the body and fitting seats in them. They do not have wherewithal to consider aerodynamics. Thus, this design is very similar to the boxy shapes that are made by them. However there is a Bus Body design code approved by ARAI (AIS-052) (which they are expected to follow) but it doesn't delve in to aerodynamics [13]

Design2

In this design the windscreen of the bus was given a curvature so that the pressure built up in front is reduced and the air flows over it with lesser restriction.

Design3

In this design additional apart from the curved wind screen sides of the bus too were given curvature to further ease the travel of air over the body.

Design4

In the addition of design 3 we added proper curvature in the rear face of the bus.

Design5

After several iterations we developed our final designed similar to an actual bus made by Volvo. It has the lowest coefficient of drag.

Table-2 Results Comparison

Concept	Drag Coefficient	Drag Force
D1	1.31	8.73
D2	1.12	7.45
D3	0.801	5.32
D4	0.432	2.87
D5	0.359	2.01

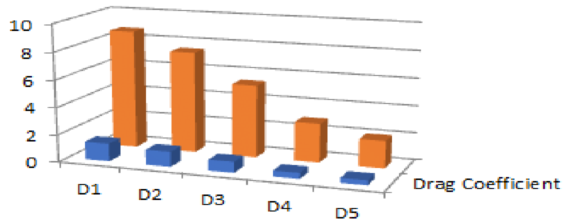


Chart- 1 Results Comparison

Computational Fluid Dynamics is a great tool and can be used to predict Drag, Co-efficient of Drag and lift values. We have used the same in the current study to predict Drag and

Cd values for various shapes of a bus. Lift values have not been captured because they are of little consequence in case of heavy vehicles. After trying out various iterations the optimum Cd value of 0.3590 was found. Which has resulted in almost 77% reduction in drag from the first design. This will surely improve the fuel efficiency of the vehicle which was the objective of this study.

REFERENCES

[1] https://www.siam.in/uploads/filemanager/47AUTOMOTIVEMISSIONPLAN.pdf

[2] https://www.business-standard.com/article/automobile/govt-aims-to-raise-auto-sector-contribution-to-gdp-job-creation-gadkari-121082501375_1.html

[3] https://auto.economictimes.indiatimes.com/news/passenger-vehicle/cars/india-has-22-cars-per-1000-individuals-amitabh-kant/67059021

[4] 14/RN/Ref./June/2018 – Lok Sabha Secretariat New Delhi

[5] https://smartairfilters.com/en/blog/25-most-polluted-cities-world-2021-rankings/

[6] https://indiankanoon.org/doc/1351218/

[7] Bureau of energy efficiency , government of India, ministry of power,

[8] https://beeindia.gov.in/content/fuel-efficiency

[9] https://www.youtube.com/watch?v=wpl6wCiy-3M&t=72

[10] https://lynceans.org/all-posts/supertrucks-revolutionizing-the-heavy-tractor-trailer-freight-industry-with-science/

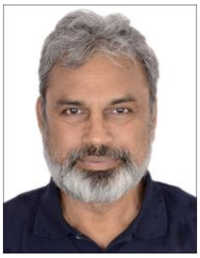
[11] https://www.india-briefing.com/news/indias-import-export-trends-in-2020-21-trade-diversification-fta-ftp-plans-23305.html/

[12] https://en.wikipedia.org/wiki/Density_of_air

[13] A.Muthuvel, M.K.Murthi, Sachin.N.P, Vinay.M.Koshy, S.Sakthi, E.Selvakumar "Aerodynamic exterior body design of bus" International Journal of Scientific & Engineering Research, Volume 4, Issue 7, July-2013

[14] Bus Design type approval As per Indian standard AIS052,2008

BIOGRAPHIES



Mr Sanjeev Tiwari did his B Tech in 1980 in Mechanical Engineering SGSITS Indore. Has been in automobile industry since then and has worked in Design, Manufacturing, Sales and Service. He is currently pursuing M Tech in Automobile Engineering from RGPV Bhopal, Through Sagar Institute of Research and Technology Indore.