Aerodynamic Drag Reduction of Heavy Vehicle Using Computational Fluid Dynamics (CFD)

Rutuja Ravindra Kapote

rutujark2017@gmail.com
Sinhgad College of Engineering, Pune Maharashtra
Department of Mechanical Engineering

DOI: 10.29322/IJSRP.14.02.2024.p14634
https://dx.doi.org/10.29322/IJSRP.14.02.2024.p14634

Paper Received Date: 13th February 2024
Paper Acceptance Date: 16th February 2024
Paper Publication Date: 21st February 2024

Abstract

The aerodynamic study of State Transport Bus minimizes road accidents and fuel consumption. The increasing fuel prices has made private transportation uneconomical. The Vehicle utilizes around 40% of available engine power to overcome the drag resistance. Drag coefficient can be reduced by streamlining the surface, modifying the outer structure of bus, roof optimization, etc. considering ergonomics and aesthetics. Drag resistance can be reduced by reducing drag coefficient. Exterior design of city bus is poor considering aerodynamic aspect. The objective of this paper is to optimize state bus and use Computational Fluid Dynamics to calculate drag coefficient which will eventually reduce the fuel consumption of vehicle. For this 3D CAD (SOLIDWORKS) model of Maharashtra State Transport bus is prepared and optimized considering standard parameters and drag coefficient is calculated using CFD software ANSYS 2021 R1 (FLUENT). The drag coefficient has reduced from 0.9 to 0.6 Around 33 % reduction in drag coefficient is achieved. Due to which the fuel economy is reduced by 26%. The turbulent model standard k-eplison is used for better approximation of results.

Keywords

Drag Reduction, Fuel Economy, Computational Fluid Dynamics, InterCity Bus, Wind Tunnel

1. Introduction

Buses are the easy and cheapest mode of mass transportation, which can go large distance at high speed by carrying almost about 30-45 passengers in single stretch. The design of city buses is very poor considering aerodynamic aspect. The flow around a heavy vehicle, including buses, trucks is three dimensional with turbulent boundary layers, separation and reattachment on the vehicle surface, huge separation at the rear trailing edge, shear layer evaluation, and a large wake behind a vehicle. The high Reynolds number develop turbulent boundary layer above the most vehicles surface, except very near to the front surfaces. The exterior styling and aerodynamically efficient design of buses help to reduce drag and attract customers. This help to reduce load on engine and increase the fuel efficiency. The power generated by engine is mainly use to overcome the rolling resistance, aerodynamic drag and climbing resistance and aerodynamic drag assume importance as it increases with respect to vehicle speed. Reduction is driving resistance is however one of the most effective approaches

The force required to propel a vehicle is :

$$F_{req} = F_d + F_r + F_a + F_g \quad F_d = \tfrac{1}{2} C_d \rho A V^2 \qquad [10]$$

At the speed of 100km/hr. the aerodynamic drag force is responsible for driving resistance

Previous study as per S. Thorat et al [1] showed the reduction in Cd from 0.581 to 0.41 at the speed of 100 km/hrand obtained overall drag reduction of 30% using CFD. S. Kanekar, Prasad Thakre [2] used turbulence model k-ε realizable and achieved 28% improvement in drag coefficient. The simulation of flow of air was performed over the bus in two steps: modelling using SOLIDWORKS and analysis from ANSYS fluent. prof. Mr.S.Sudhakar [3]did CFD analysis of two prototype bus models one is the baseline model and another is the optimized model to increase the performance and reduce the fuel requirement occurred as the result of reduction in drag coefficient. This paper discuss about the cross over speed beyond which aerodynamic losses are dominant. A loaded intercitybus of 24 tone has a cross over speed of 70km/hr. Whereas the empty 17 tone bus has a crossover speed of 60km/hrand tractor-trailer with gross weight of 36 tone has cross over speed of 90km/hr. Lower the drag density (ratio ofrolling resistance to aerodynamic resistance) the aerodynamic loss of intercity bus is greater than the mechanical loss at lower vehicle speed than tractor trailer. At high speed the significant fraction of energy expanded is dissipated into aerodynamic loss . J Abinesh [4] in his study got the overall reduction in aerodynamic drag force of 10% which result in increase in performance and reduced the fuel requirement. A review of Aerodynamic DragReduction Devices of heavy vehicle [5] focus on reduction of aerodynamic drag, and hence Green House Gas (GHG) emission without negatively affecting usefulness and profitability of vehicle by using various devices in tractor trailer gap. This review even include the management of air flow around a heavy vehicle. Various Techniques like base bleeding is used and design changes like change in geometry of side mirrors, boat tailing, gap fillers side skirts done for modifications in design of buses, truck to reduce the aerodynamic drag.

This paper focus on drag reduction and design optimization of Maharashtra state transport bus through CFD analysis using ANSYS 2021 R1 (FLUENT). One model is a current intercity bus model and another is the optimized model use for long distance travelling which is aerodynamically efficient and the reduction in drag coefficient is found which decreases drag force acting on vehicles. This reduction in drag help to reduce fuel consumption to overcome the drag resistance.

2. Physics of Air Flow Around a Vehicle

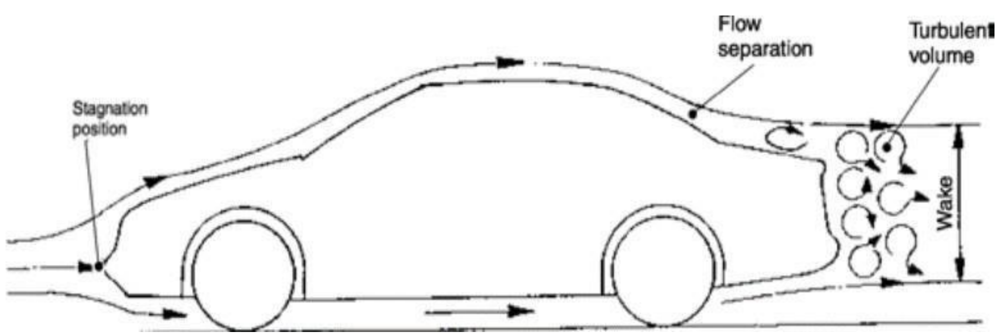


Fig 2.1 flow of air around a vehicle surface

The flow of air around a vehicle is governed by velocity and pressure relation expressed in Bernoulli's Equation assuming flow as incompressible.

P static + P dynamic = P totalPS
+ ½ ρV2 = Pt

In the absence of friction, the flow passes over the surface of vehicle exchanging the pressure for velocity. The pressure on front side balances the pressure on back side of vehicle and there will be no drag produced. But experimentally drag is produced. The drag is formed by friction of air on surface of vehicle as it flows, over the vehicle. Boundary layer begins at the point of stagnation i.e at the bumper of vehicle where velocity reduces due to friction. The pressure at stagnation point is total pressure and decreases as flow travels. Unfortunately, as the flow turns again to follow the body, the pressure again increases, reducing the velocity. At separation point the main stream is no longer attached to body but is break and continue in approximate straight line. The pressure in this region drop below the ambient pressure. The wake is highly turbulent and consist of large-scale eddies and start at the separation point [13]

The pressure is identified as negative or positive with respect to ambient pressure measured some distance from the vehicle. Negative pressure is developed at the front of hood as the flow over the front of vehicle and follow horizontally along the hood. Near the base of windshield and cowl the flow is upward and thus high pressure is experienced. Over the roof line the pressure again goes negative as the air flows tries to follow the roof. The pressure remains low down over the Bakelite and on the truck because of continuing curvature. It is in this area where separation of flow is most likely.

The three forces and three moments acting about vehicle are :

Direction	Force	Moment
Longitudinal(x-axis positive, rearward)	Drag	Rolling
Lateral (y-axis, positive towards right)	Side force	Pitching
Vertical (z- axis, positive upward)	Lift	yawing

Fig 2.2 Aerodynamic Forces

Aerodynamic drag (Fd) = ½ ρ V2 Cd AWhere,
½ ρ A V2 is dynamic pressure
Cd varies over broad range with different shape

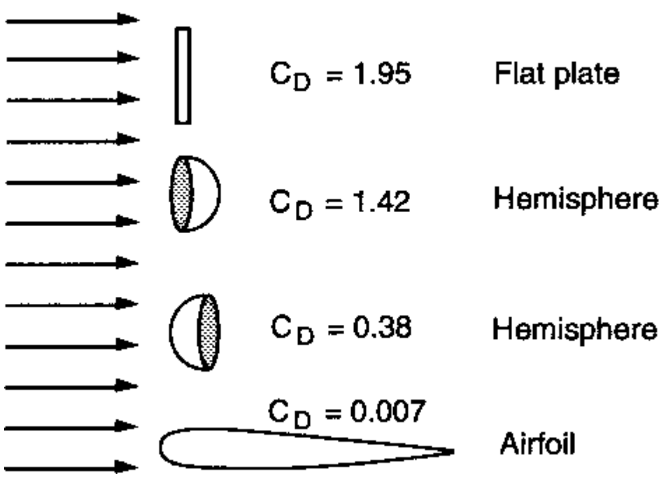


Fig 2.3. Cd value for various shapes

The aerodynamic drag consist of two drag forces. One which is due to pressure difference from front of car reaching the rear over the surface is called as "pressure drag". Whereas the drag caused due to friction between air and surface of vehicle know as skin fiction is called as "friction drag". About 90% of drag is due to the pressuredifference created at different regions of the vehicle.

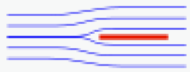
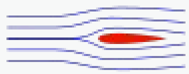
Shape and flow	**Form Drag**	**Skin friction**
	0%	100%
	~10%	~90%
~90%		~10%
100%		0%

Fig 2.4 drag and skin friction effect

For heavy vehicles such as tractor trailer and buses, the drag coefficient increase with yaw angle. Axial drag coefficient (Cd) analyzed for yaw angles in the range −6° to +20° and various ground lengths [8]. Cross winds are called as the winds flowing across the direction of travel of vehicle. The cross winds have component perpendicular to the direction of travel. Cross winds move the path of vehicle sideways and can be hazardous. Crosswinds are strongly related to yaw angle. "Crosswinds sensitivity" generally refers to lateral and yawing response of vehicle in the presence of transverse wind disturbances which affect the driver's ability to hold the vehicle in position.

Aerodynamic Drag directly affects the fuel economy. The power loss due to braking and acceleration is dominant in urban area whereas the power loss due to aerodynamic drag is dominant on highways. The reduction of aerodynamic losses is a significant area in which fuel consumption improvements can be made [7]. The exact improvement in fuel economy that may be expected from improvement in road load are difficult to predict becauseof uncertainty about the ways vehicles are used and driven.

Source	Urban	Highway
Drivetrain	10-15%	5-10%
Inertia/ braking	35-50%	0-5%
Rolling Resistance	20-30%	30-40%
Auxiliary Load	15-20%	2-10%
Aerodynamic Losses	10-25%	35-55%

Fig 2.5 Effect of vehicle properties on fuel efficiency

3. Computational Fluid Dynamics Analysis and modelling of Intercity Bus

The CFD process is divided into three steps; Pre-processing, solving and post processing. These processes are applied for two bodies one is a baseline model which is a current intercity bus design and another is the aerodynamically modified design. Virtual wind tunnel is created to analysis the nature of flow around a bus.This helps to find the drag force around a bus from the flow pattern, flow separation and wake strength behind the body.

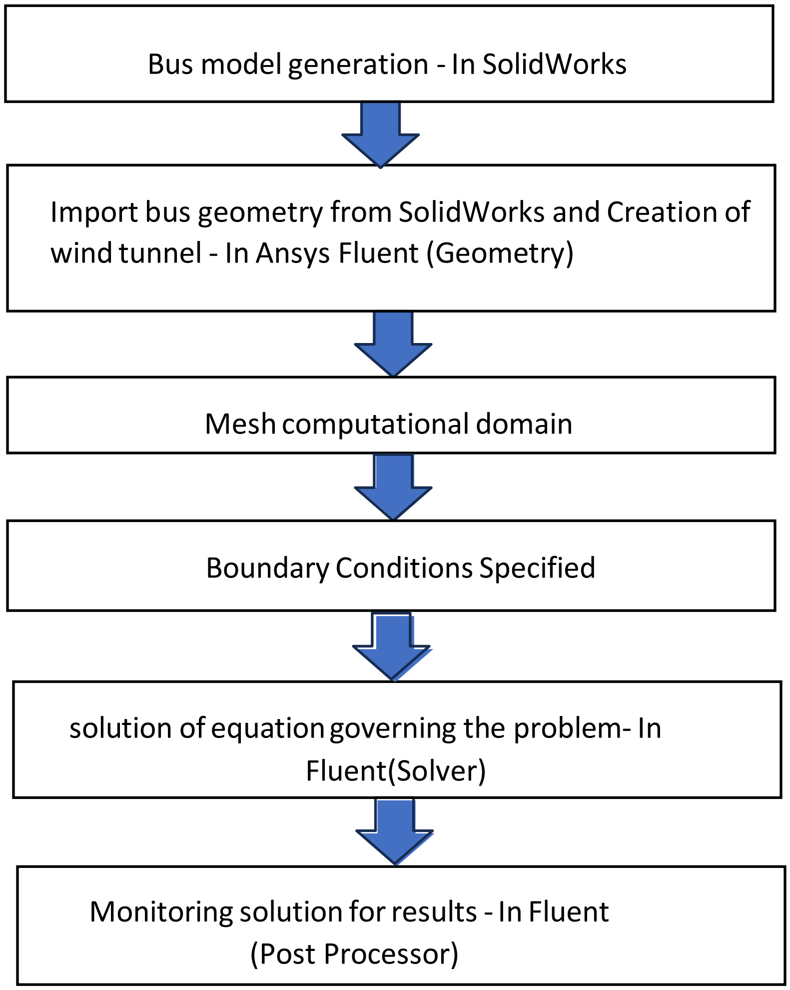
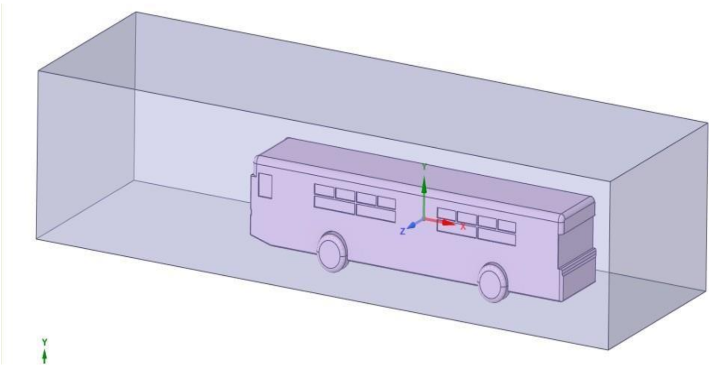
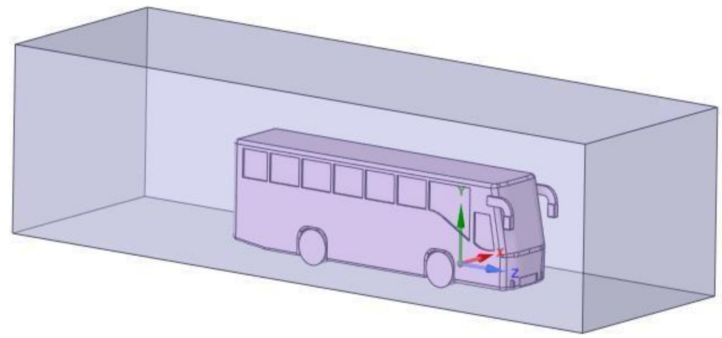


Fig 3.1 Fluent Methodology Flow Chart

The 3D geometry of intercity bus created in SolidWorks is imported into ANSYS 2021 R1 (Fluent). The dimension of wind tunnel are 5H, 10W and 7L. The bus model is placed inside this domain in such a way that 1/3length was kept in front of the model. The large domain is kept at the rear to capture essential flow features. The virtual wind tunnel consist of an inlet, outlet, wall and ground surface. To capture certain area of interest the mesh should be fine to solve all irregularities and achieve a robust solution. Boundary Conditions are applied on meshedmodel. In order to replicate the constant wind velocity like wind tunnel test, constant velocity inlet condition is applied at the inlet. With the atmospheric pressure as operating pressure, zero-gauge pressure is applied at theoutlet.



3.2 Geometry of Maharashtra State Bus



3.3 Geometry of Coach (Traveler)Bus

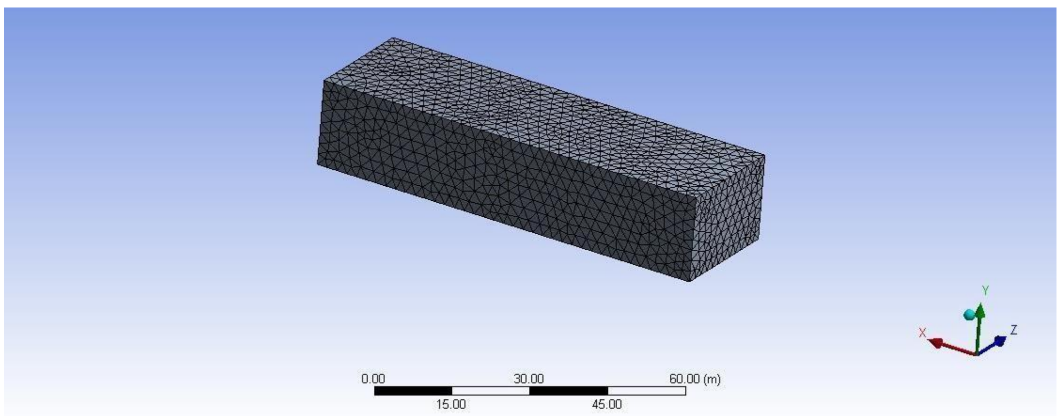


Fig 3.4 Mesh model

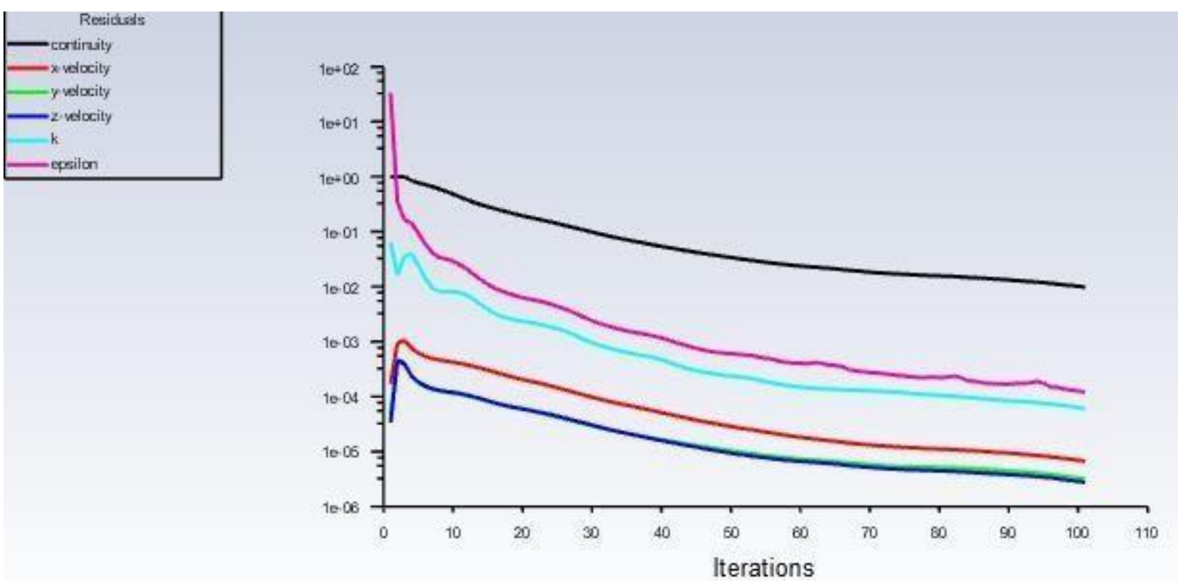


Fig 3.5 Convergence

4. Observation and Discussion

From the analysis in ANSYS (FLUENT) the following value of drag force and drag coefficient are found.

Model 1

State Bus

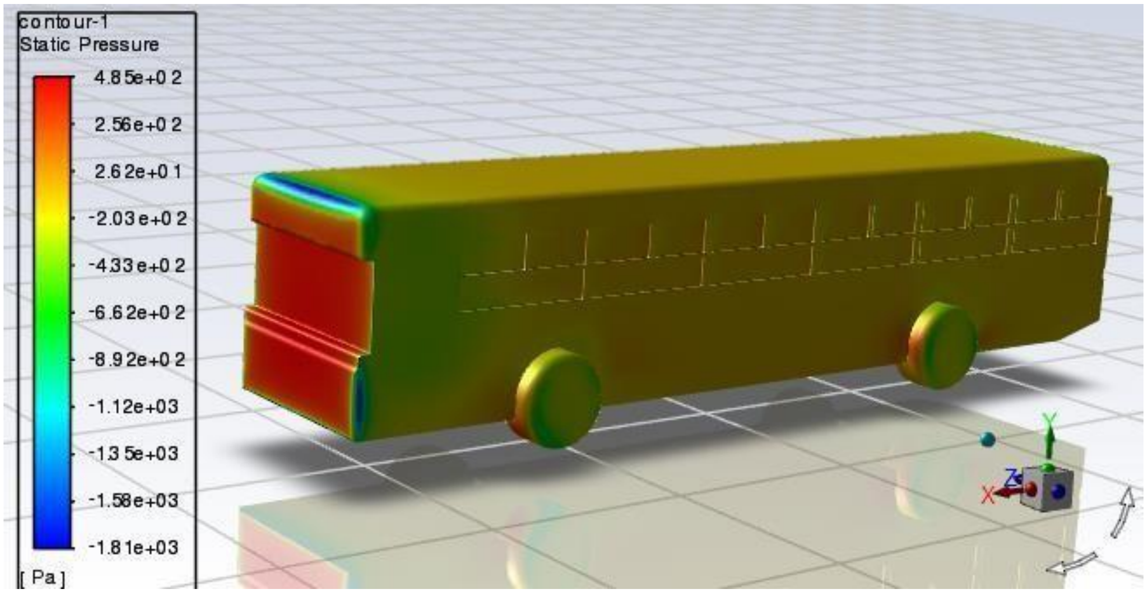


Fig 4.1 Pressure Contour (Intercity Bus)

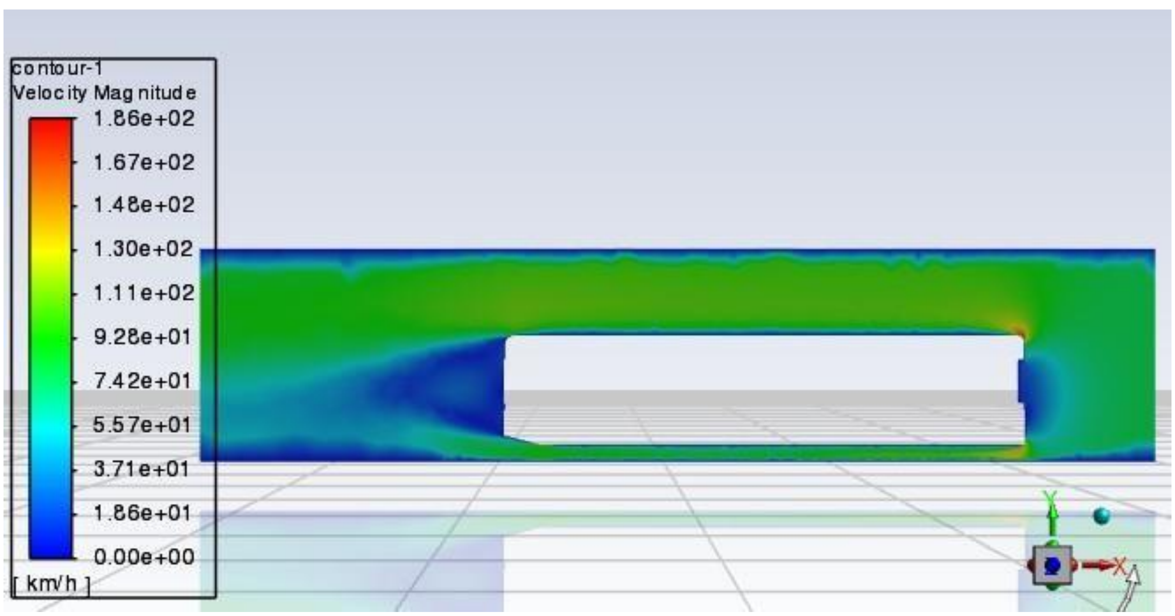


Fig 4.2 Velocity Contour (Intercity Bus)

Coefficient of friction (calculated from fluent)

Cd due to Pressure = 0.88Cd due to friction = 0.02 Total Cd = 0.90

Drag Force (calculated from fluent)

Fd due to pressure = 2787 N Fd due to friction = 48 N Total drag force Fd = 2833 N
Rolling resistance (Fr) = 583.34 N

Fuel Economy

the power required to overcome the drag is Power (PT) = total resisting force × velocity

= (2833 +583.34) × 27.78×1/ η

= 105.477 KW

Fuel consumed in KMPL = (27.78 × 0.35 × 39.7×106) / (105.477×106)

= 3.7

Model 2 (Modified)

This model has modified at its outer design. This model is design under the consideration of aerodynamicaspects and standard parameters

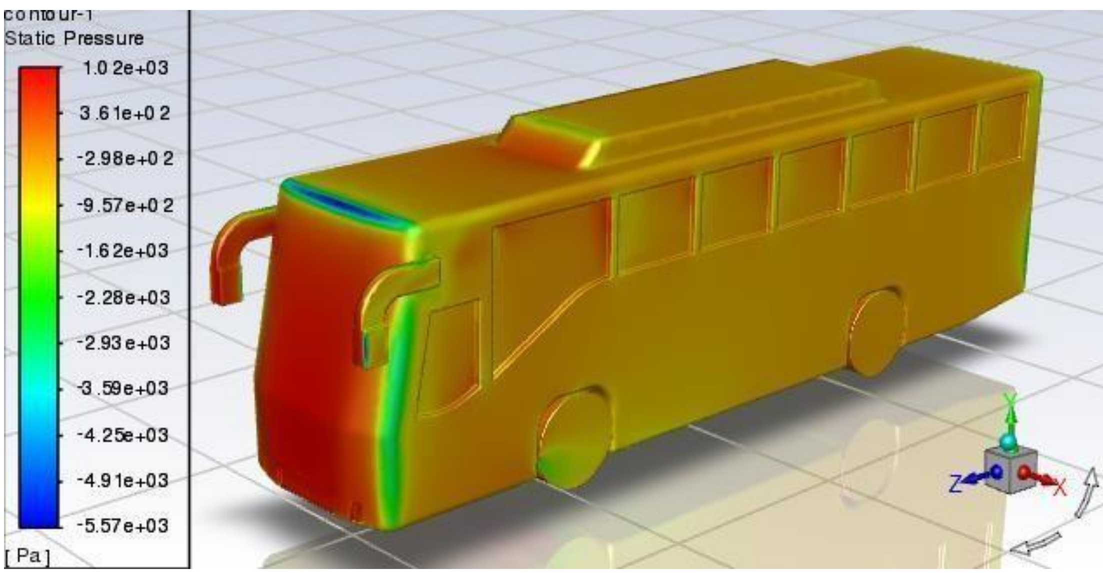


Fig 4.3 Pressure Contour (Modified Bus Design)

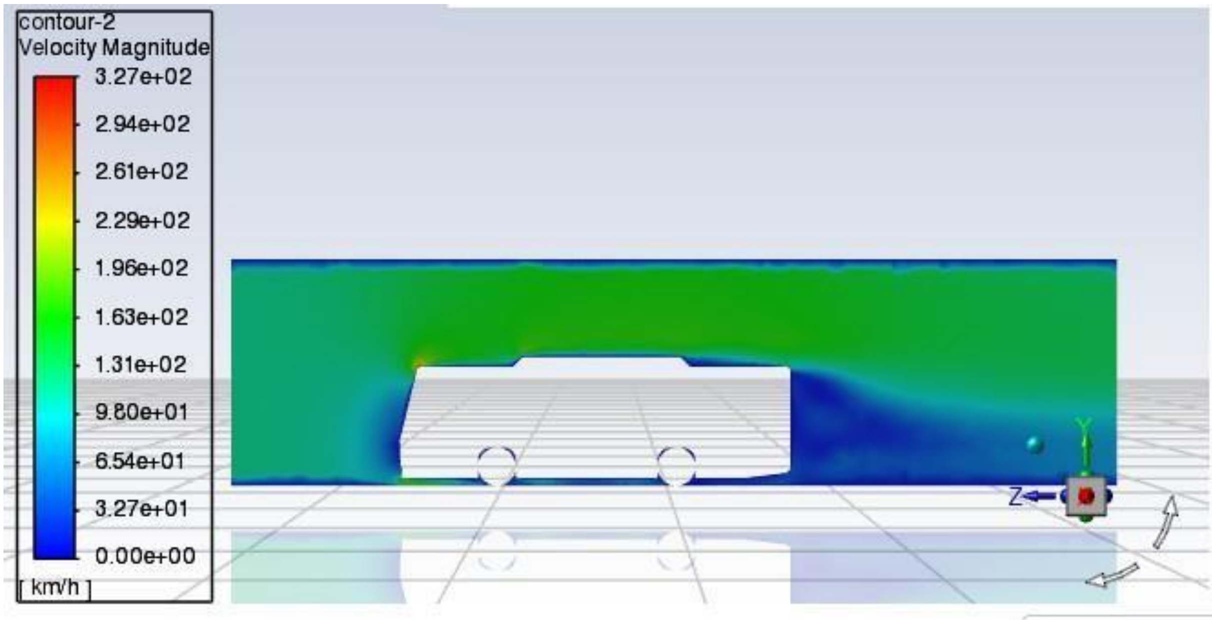


Fig 4.3 Velocity Contour (Modified Bus Design)

Coefficient of friction (calculated from fluent)

Cd due to Pressure = 0.58 Cd
due to friction = 0.034
Total Cd =0.614

Drag Force (calculated from fluent)

Fd due to pressure = 1839 N Fd
due to friction = 92 N Total
drag force Fd = 1930 N
Rolling resistance (Fr) = 583.34 N

Fuel Economy

the power required to overcome the drag is Power (PT) = total resisting force × velocity

= (1930 + 583.34) × 27.78 ×1/ η
= 77.608KW

Fuel consumed in KMPL = (27.78 × 0.35 × 39.7×106) / (77.608×106)

= 4.97

Type	Cd	Drag Force (N)	KMPL
State Bus model	0.90	2833	3.7
Coach Bus model	0.63	1930	4.97

Fig 4.4 Comparison Table

5. Conclusion

From above calculations and Simulation done using Computational Fluid Dynamics the drag coefficient for StateBus is 0.9 and that for modified bus is 0.6. This reduction in drag is obtained because of modification in exterior aerodynamic design. Eventually the fuel economy has decreased by 26%. Thus we get the amount of drag we were able to reduce by designing aerodynamically efficient vehicle having a good aesthetics attracting customers.The one which is comfortable for passengers and follow 95 % male and 5% female rule with good ergonomics. The modified bus attains very high speed for less fuel consumption.

6. Reference

1. Fred Browand, "Reducing Aerodynamic Drag and Fuel Consumption" Aerospace and Mechanical Engineering Viterbi School of Engineering University of Southern California, Global Climate and Energy Project Workshop on Advanced Transportation October 10-11, Stanford University
2. T. kumar, Dayananth.D, R.S.Manibharathi, S.Arun kumar Asst prof, "Design and Analysis of Aerodynamic Drag reduction in Truck prototype", International Journal of Pure and Applied Mathematics, Volume 118 No. 11 2018, 535-539 ISSN: 1311-8080 (printed version), ISSN: 1314-3395 (on-line version)
3. J Abinesh1 and J Arunkumar "CFD Analysis of Aerodynamic Drag reduction and Improved Fuel Economy", International Journal of Mechanical Engineering and Robotics Research, Vol. 3, No. 4, October, 2014,ISSN 2278 – 0149
4. S. Kanekar, P. Thakre and E Rajkumar "Aerodynamic study of state transport bus using computational fluid dynamics", IOP Conference Series: Materials Science and Engineering, Materials Science and Engineering 263 (2017) 062052
5. H. Yesfalgn Damissie, N. Ramesh Babu Associate Professor, "Aerodynamic Drag Reduction on Locally Built Bus Body using Computational Fluid Dynamics (CFD): A Case Study at Bishoftu Automotive Industry"
6. 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, ISSN 2229-5518
7. Jeff Patten, P. Eng. Brian McAuliffe, Ph.D. William Mayda, P. Eng. Bernard Tanguay, Ph.D, "Review of Aerodynamic Drag Reduction Devices for Heavy Trucks and Buses", Project 54-A3578, May 11, 2012, CSTT- HVC-TR-205
8. Sachin Thorat, G.Amba Prasad Rao "Computational Analysis of InterCity bus with improved aesthetics and Aerodynamic Performance on Indian Road", International Journal of Advanced Engineering Technology, Vol.II/ Issue III/July-September, 2011/103-109, EISSN 0976-3945
9. Eyad Amen Mohamed , Muhammad Naeem Radhwi, Ahmed Farouk Abdel Gawad "Computational investigation of aerodynamic characteristics and drag reduction of a bus model", American Journal of Aerospace Engineering, 2015; 2(1): 64-73 Published online October 20, 2014
10. Thomas D Gillespie"Fundamental of Vehicle Dynamics", Society of Automotive Engineering

11. Mr. Devesh yadav , Mr. Sumit chauhan , Mr. Shashank karki , Mr. Satya ranjan nayak , Mr. Nitin kumar, Associate Prof. Mr.S.Sudhakar babu "CFD Analysis of Drag Force Reduction in Intercity Bus", International Research Journal of Engineering and Technology (IRJET), Volume: 04 Issue: 05 | May -2017

12. Ma. Janice J. Gumasing, and Charm Hanalae C. Dela Cruz "Ergonomic Design of Public Bus in The Philippines with Provision for Senior Citizens and Persons with Disability", MATEC Web of Conferences 150, 05006 (2018)

13. Zulfaa Mohamed-Kassim, Antonio Filippone, "Fuel savings on a heavy vehicle via aerodynamic drag reduction", Transportation Research Part D 15 (2010) 275–284