ISSN: 0971-1260
Vol-22-Issue-16-August-2019

Simulation of Air Flow Through Bus For Different Position of Two Top Openings By Using Open foam Package

[1*]Mr.Keval Nikam,Ravinder Kumar[2]

[1]Faculty,Department of Mechanical Engineering, Dr.D.Y.Patil Institute of Engineering, Management and Research, Akurdi, Pune, Maharashtra,India 411044

Email: nikamkeval26@gmail.com

[2]Faculty, Department of Mechanical Engineering, Lovely Professional University, India,

Email: ravinder.22218@lpu.co.in

Corresponding author: nikamkeval26@gmail.com

ABSTRACT

Bus is considered as common mode of transportation usually in cities and suitable up to travel distance of 500Km as per comfort condition of passenger is concerned. Passenger comfort conditions are needed to study with respect to bus geometrical features. The previous investigations have been limited to explore vehicles with air flow through closed windows vehicle body. The top opening of bus is considered at different location and analysis is done. CATIA V5 software is used to build model and Open source software OpenFOAM is used to do analysis. This analysis indicates the suitable arrangement of top opening provided on the roof of bus. Top opening directed forwarded found to better option as per coefficient of drag is concern and one forward and other one backward top opening found to be better option as per air distribution in confined space is concerned.

Keywords: OpenFOAM; Drag force; Top opening

1.0 Introduction

Most of the population in developing country are uses public transportation like buses, trains etc. From past 10 to 15 years government are heavily investing money on road in order to reduce the

ISSN: 0971-1260
Vol-22-Issue-16-August-2019

travel time of passenger. Therefore, it has become important to analyze buses with improved fuel efficiency. Fuel cost is increasing exponentially in past few years depend on its availability from other country. The bus body in developing country like India is considered to be bluff body with applying knowledge of aerodynamics. These public transportation bus are designed as per the customer convince. It thus becomes imperative to improve comfort levels while relying on motion-generated circulation. In a study[1], top opening placed at two different position are studied and analyzed by using OpenFOAM Software. It was also discussed various methods to reduce drag force for case of bluff body with circular and rectangular shape[2]. It was observed that significant changes occur when there is change in frontal area of the bluff body. In 1993, Cooper considered the vehicle to be approximated as Bluff body and test in wind tunnel test[3]. Author also suggested different kinds of vehicles which can be approximated as bluff body like road automobiles, railways, marine's vehicles etc. Researchers also proposed new model of bus rear part considering drag reduction phenomenon[4]. It was found that by changing optimal roof junction and including vortex trapping cavity in the tail. An overview of turbulence modeling and how the relation is formed for solving complex problem in any computational software was also suggested[5]. Author also discussed various cases of turbulent flow and its use in various computational software like STAR-CCM+. Davidson discussed the influence of the moving floor instead of scattered spectra in steady floor and concluded that by incorporating moving floor affect the result of experimental setup and numerical simulations[6]. Results obtained from experimentation and a numerical simulation tends to comes nearer to theoretical results.

1.1 OpenFOAM

The OpenFOAM (Open Field Operation and Manipulation) is open source CFD toolbox is used visualize flow of air as surrounding fluid through the bus cases. OpenFOAM hasvariety of plugin to solve complex fluid flow problems related to chemical reactions, fluid mechanics and rate of heat flow to solid dynamics and electromagnetic [7].

Bus model is constructed in CATIA V5modeling software. It is then converted into thestereolithogrphyfile (STL). STL file is then imported to meshing background using snappyHexMesh. Command potentialFoam was used to startfirst phase flow. Command simpleFoamuse to allot initial boundary conditions and performing calculations.Command

ISSN: 0971-1260
Vol-22-Issue-16-August-2019

ParaFoam is use to open paraview ads in with OpenFOAM for visual purpose and to perform post processing. The velocity and pressure distribution over the geometries are visualized and discussed at constant speed of 70Km/Hrs. Through the visualization a contour of pressure and velocity, bus body with two top opening in front side and back side combination. Finally coefficients of drag, drag force are estimated. General Assumption of bus without wheel is made to perform this analysis[1].

1.2 Basics of Aerodynamics

Drag Force (F_d)is the force responsible to resist the forward motion of any vehicle. For the ground vehicle drag force contributes to 65% of the net forces acting on the whole body.The drag force is represented as follows:

$$Fd = \frac{1}{2}\rho V^2 C_D A \qquad (1)$$

Where
F_d = Drag Force
C_D =Coefficient of Drag
A = Effective Frontal Area of Bus
V =Velocity of air
ρ = Density of air

2.0 Case details

The cases discussed are as follow

a. Bus with close windows front positioned Top Opening.
b. Bus with close windows front and back positioned Top Opening.

3.0 Computational Set-up

The simulation setup is identical for both bus cases.Modeling is performed in CATIA V5 and meshing is done in the OpenFOAM Environment. It is saved into the stereolithogrphy file

ISSN: 0971-1260
Vol-22-Issue-16-August-2019

(STL)[8].The preprocessing is partially done in OpemFOAM environment while Post processing is totally done in paraview featured built in OpenFOAM.

3.1 Specification of Domain

Domain is designed in such a way that bus model get adjusted into it. It consist of fixed boundaries such as front(inlet),Back(outlet),side wall, roof top and base surface(ground).Names such as front and back surface is been allotted as per the length of domain varying from 0 to L indicated in Fig 1.Wall is specified name give in other than X-direction shown in Fig 2[8].

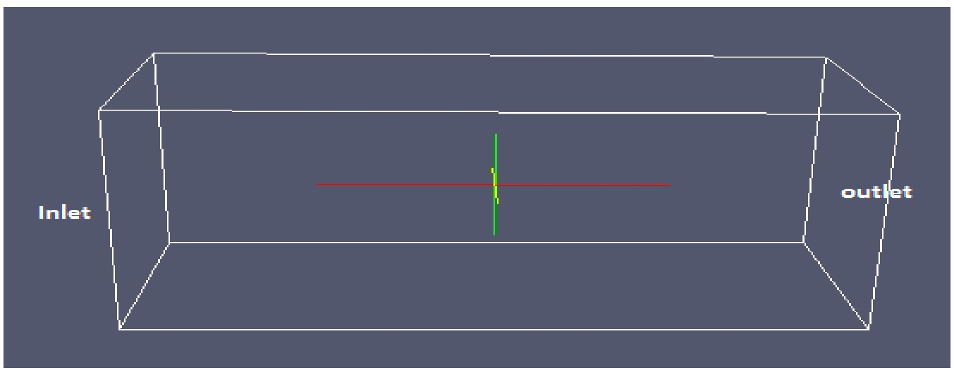


Fig 1. Front and back surface domain

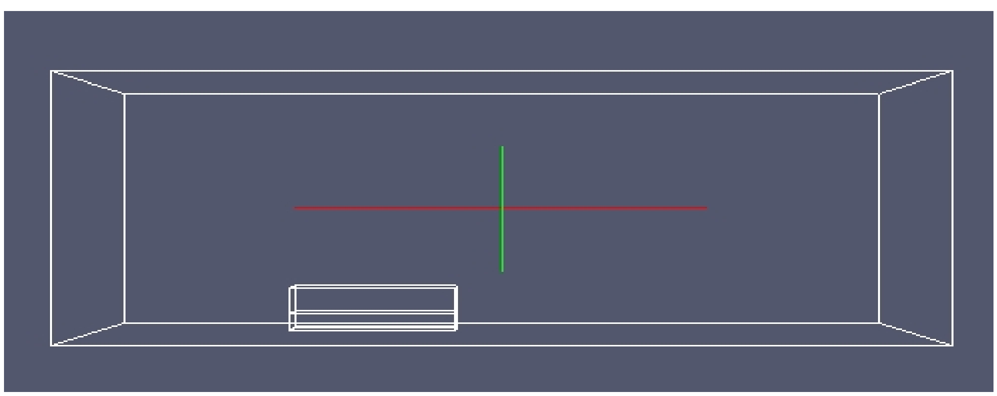


Fig 2. Domain with geometry

3.2 Mesh formation in OpenFOAM software:

Utility snappyHexMesh is used to create high quality hex dominant meshes based on arbitrary geometry as shown in Fig 3. Parallel computing is performed by utilizing the both core of RAM. Parameter of mesh is controlled by command snappyHexMeshDict[9].

ISSN: 0971-1260
Vol-22-Issue-16-August-2019

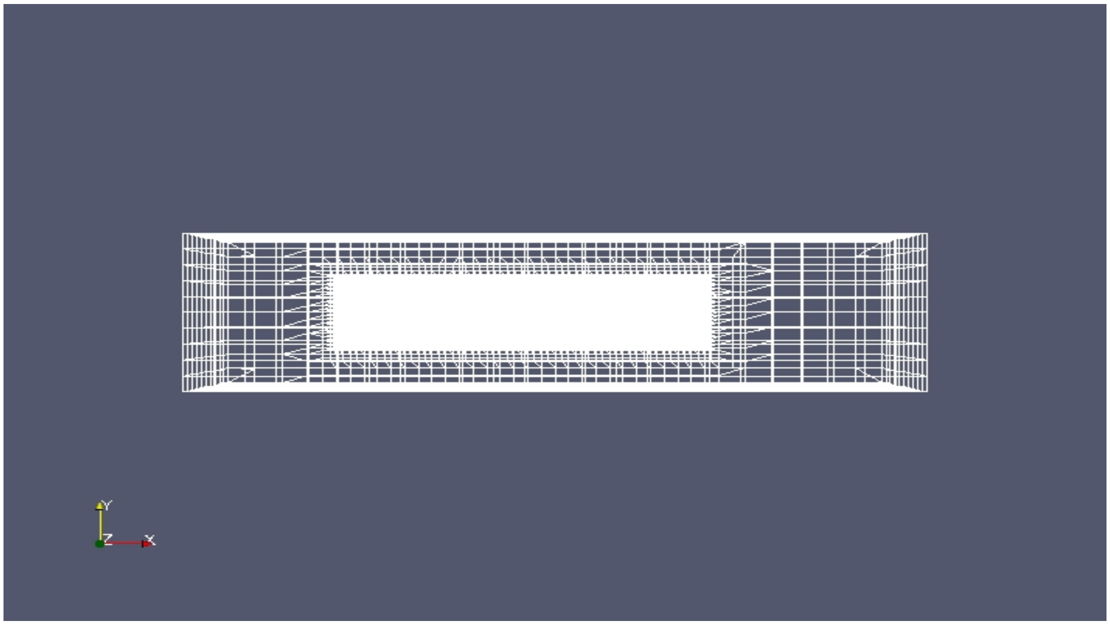


Fig 3. Top view after meshing

3.3 Boundary conditions

In this present work, the tests carried out at operating speed varying from 40km/hrs, 50km/hrs,60km/hrs,70km/hrs,80km/hrs. Each case is be analyzed in different opening speed as mentioned. Sample case is explained as case 2encounter 40km/hrs of velocity of the inlet.The initial boundary condition is considered as standard temperature of 300K. The outlet boundary condition is considered to pressure of zero Pascal. Theair density and viscosity of 1125g/$m^3$and of 1.7894×10^{-5} kg/msec) is considered[10].

3.4 Turbulence Modeling

The kω-SST turbulence model is considered for carrying out the simulation in OpenFOAM tool. ω has ability to be proven reliability in different zones and feasible to apply on the free stream model[5].

3.4.1 Intensity of Turbulence(I)

The intensity of turbulence is the ratio of the root-mean-square of the fluctuating velocity, V', to the mean free stream velocity, V.

ISSN: 0971-1260
Vol-22-Issue-16-August-2019

$$I = \frac{V'}{V}. \qquad (2)$$

Intensity of turbulence is generally in range of 1% to 10% in case of internal flow. The relation of turbulence intensity for fully developed region inside a duct is represented as follow

$$I = 0.16(Re)^{-0.125} \qquad (3)$$

Based on the properties of flow, intensity of turbulence for external flow is in the range from 0.01% to 0.05%. Intensityof turbulencein present case is considered as 0.02%.

3.4.2 Turbulence Length (*l*)

The turbulence lengthsignify the size of the large eddies formation in turbulent flows (Davidson 2018).The turbulence length is estimated by using following empirical relation

$$l = 0.07L \qquad (4)$$

Where 'L' is the characteristic length, *l*is turbulence length.

3.4.3 Turbulent Kinetic Energy(K) and Specified Dissipation Rate (ω)

Kand ω are estimated from following equations. Table 1 indicates the values considered for computation calculation purpose

$$K = \frac{3}{2}(VI)^2 \qquad (5)$$

$$\omega = \frac{K^{0.5}}{C\mu^{0.25} \times l} \qquad (6)$$

Table 1: Values of K and ω

Cases No.	Velocity in km/hrs	Velocity in m/sec	*K*(calculated)	ω(calculated)
1	40	11.11	0.074	0.258
2	50	13.89	0.1157	0.3231
3	60	16.67	0.1667	0.3878

ISSN: 0971-1260
Vol-22-Issue-16-August-2019

4	70	19.44	0.4523	0.4523
5	80	22.22	0.2962	0.5169

4. Simulation Post Processing

Simulation result of bus with top opening with both flap opened in forward direction. Figure 4, shows pressure profile from different views and for same velocity.

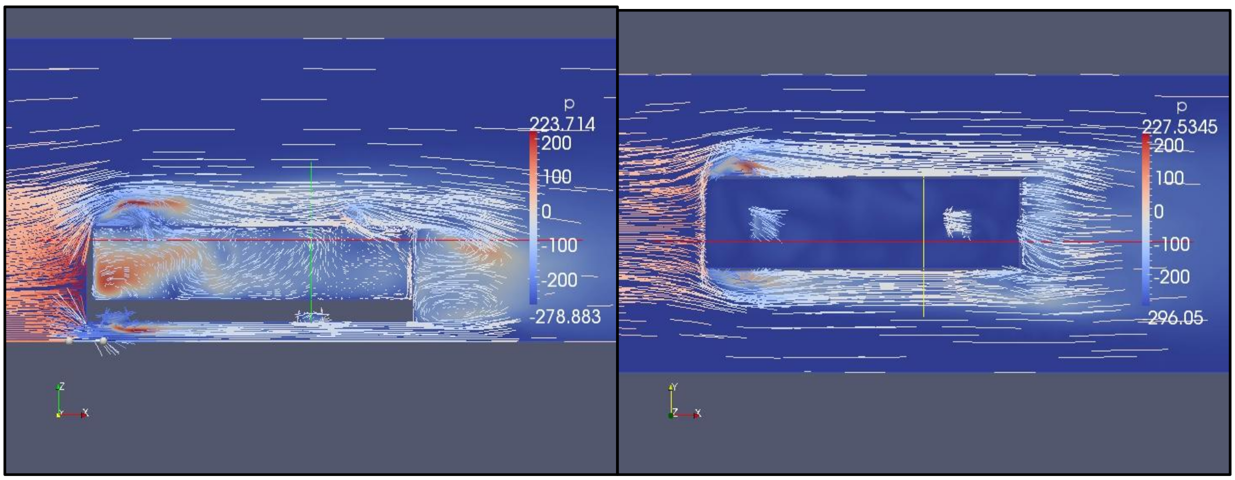


Fig 4: Pressure profile side and top view after slice operation(Case a)

Simulation result of bus with top opening with one flap opened in forward direction and other flap opened in reverse direction. Figure 5, shows pressure profile from different views and Velocity.

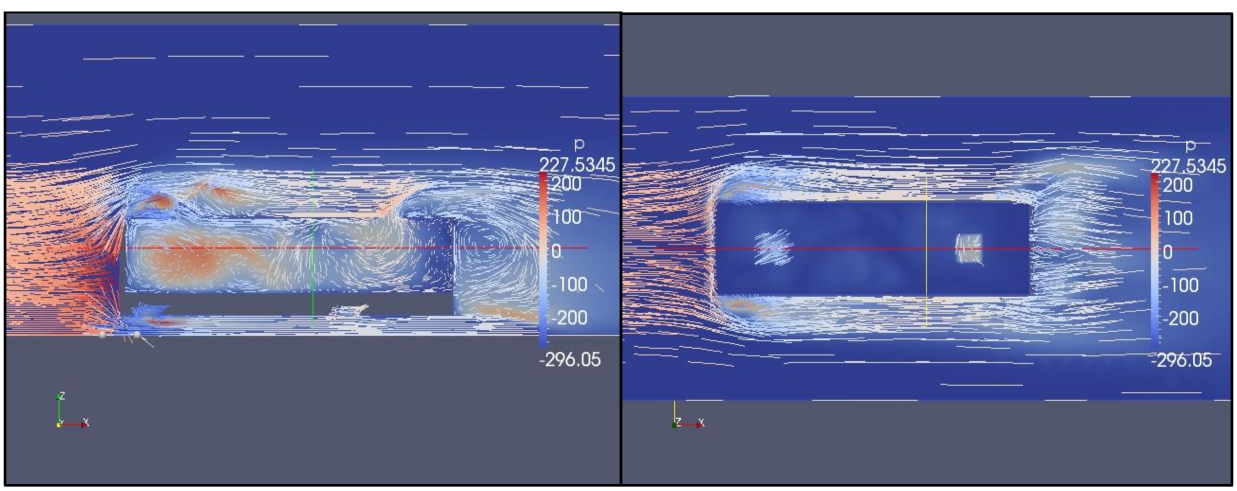


Fig 5: Pressure profile side and top view after slice operation(Case b)

5. Results and discussion

Air will try to move inside the bus from the top opening. It is been observed that Air distribution in (case a) is not much effective as shown in Figure. This may lead to the suffocation condition for the passengers comfort. In view of this (case b) is considered where the orientation of second top opening is change to backward and results are analyzed which tell us that air get distributed in all the portion of the bus which can lead to better comfort of the passengers. This can be viewed in Fig 6 and Fig 7.

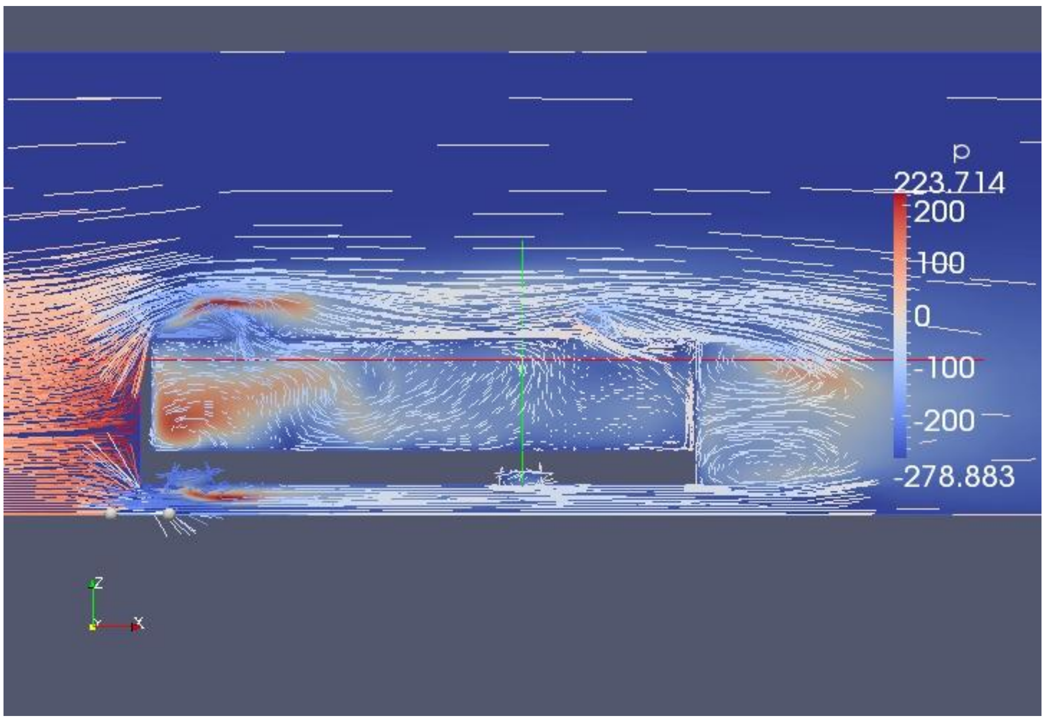


Fig 6: Visualization of fluid flow in Case a

ISSN: 0971-1260
Vol-22-Issue-16-August-2019

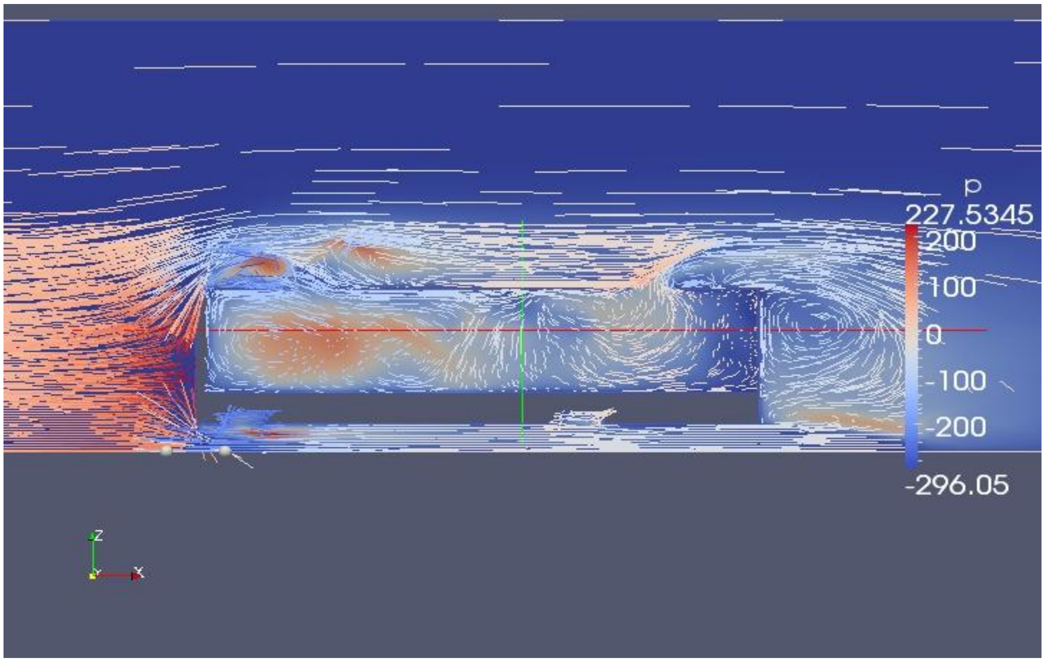
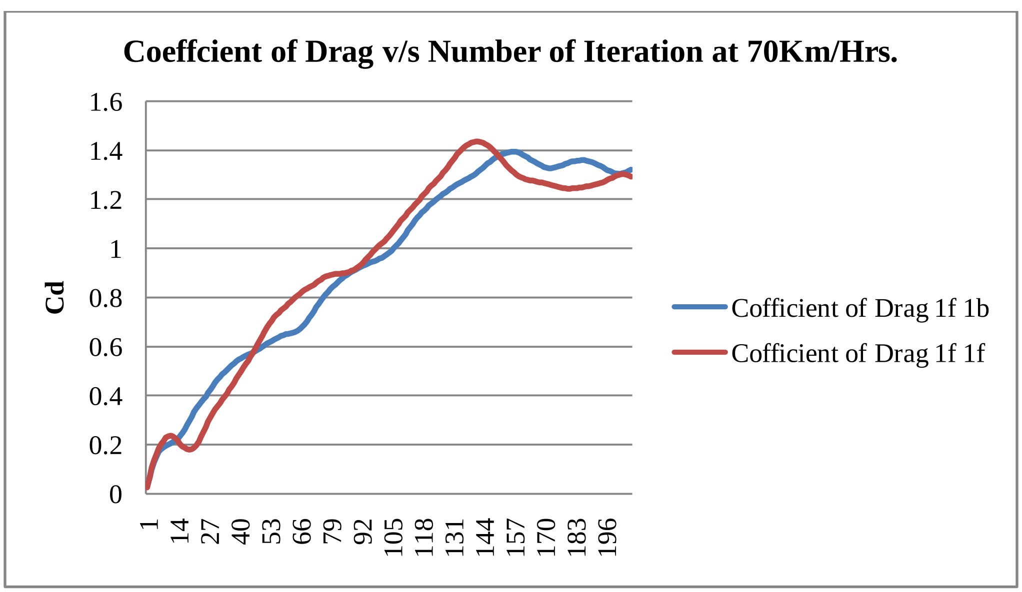


Fig 7: Visualization of fluid flow in Case b

Particular 70Km/Hrs speed of the air is considered for the analysis. Coefficient of Drag and Drag Force is plotted opposite to number of iteration as shown in Figure8-9. It is observed that the Coefficient of Drag and Drag force for both the cases are different at different instance of iteration. As per drag force is concern (Case a) is better than (Case b) and if other side is considered i.e Comfort of passenger (Case b) fulfill the comfort condition of passenger.



ISSN: 0971-1260
Vol-22-Issue-16-August-2019

Fig 8:Coefficient of Drag

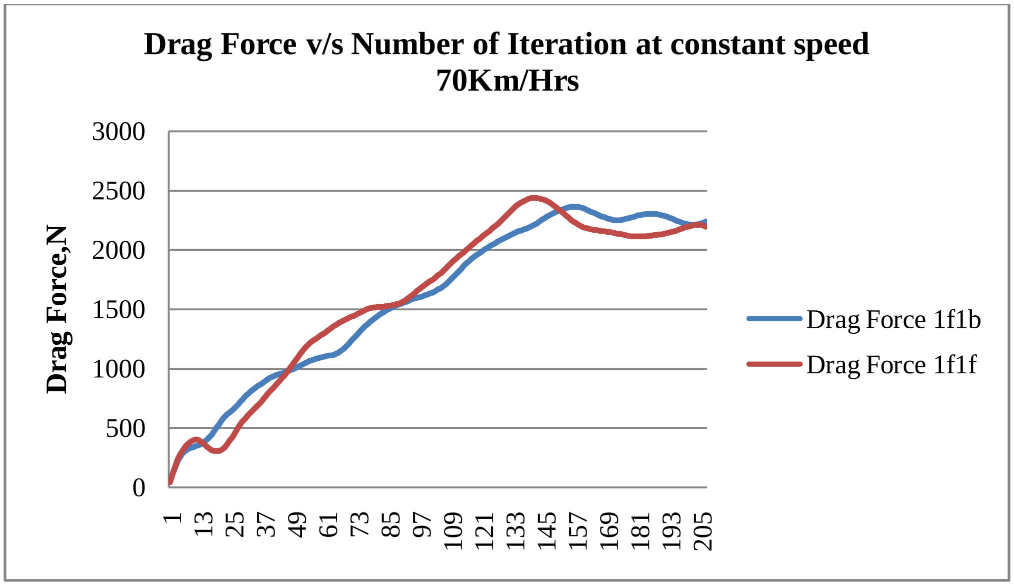


Fig 9:Coefficient of Drag Force

6. Conclusion

This paper presents the application of computational technique in analysis of positioning the top opening provided to the City bus. Open Source OpenFOAM package was used for computational fluid dynamics of the Bus model. Post processing result gives the idea of fluid flow around the bus body with two different cases. Comparing two different considered cases it seems to go for two opening one to front and other one back side of the roof of the bus rather than other case. Drag force required for case 'a' is significantly lower as compared with case 'b'. One forward and other one backward top opening found to be better option as per air distribution in confined space is concerned. Researcher working on opening in bluff vehicles and its effect on drag forces will get insight idea for from this paper about positioning of opening in vehicles for passenger comfortless.

References

[1] K. C. Nikam and S. L. Borse, "Study of air flow through a open windows bus," *International Journal of Fluids Engineering*, vol. 6, no. 1, pp. 53–64, 2014.

ISSN: 0971-1260
Vol-22-Issue-16-August-2019

[2] I. S. G. F. Lesage, "A method of reducing drag and fluctuating side force," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 25, pp. 229–245, 1987.

[3] K. R. Cooper, "Bluff-Body aerodynamics as applied to vehicles," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 49, 1993.

[4] G. D. H. S. C.A.J. Fletcher, "Bus drag reduction by the trapped vortex concept for a single bus and two buses in tandem," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 24, pp. 143–168, 2006.

[5] L. Davidson, "Fluid mechanics , Turbulent flow and turbulence modeling," 2018.

[6] L. Davidson, "Influence of floor motions in wind tunnels on the aerodynamics of road vehicles," *Journal of Wind Engineering and Industrial Aerodynamics*, vol. 93, pp. 677–696, 2005.

[7] H. K. Melbø, *A graphical user interface for the computational fluid dynamics software OpenFOAM*, no. June. Norwegian University of Science and Technology, 2014.

[8] L. D. Jathar and K. C. N. S. L. Borse, "Study of flow over sedan car body by changing different parameters using OpenFOAM," *International Journal of Fluids Engineering*, vol. 2, no. 2, pp. 342–348, 2015.

[9] H. Rahimi, E. Daniele, B. Stoevesandt, and J. Peinke, "Development and application of a grid generation tool for aerodynamic simulations of wind turbines," *Wind Engineering*, vol. 40, no. 2, pp. 148–172, 2016.

[10] S. M. Murman, "Lift and drag behavior of unconstrained bluff bodies," *NASA*, pp. 1–13, 2010.