ICMIEE20-286

Drag Reduction in Bus through Exterior Shape Modification

Pranto Karua[*], Pronob Debnath Partho
Department of Mechanical Engineering, Khulna University of Engineering & Technology, Khulna-9203, BANGLADESH

ABSTRACT
Analysis of drag is a major concern in automotive sector based on the issues of fuel economy as well as CO_2 discharge. Drag rate, which is mainly expressed by drag coefficient depends on several parameters. The parameters involved with diminishing drag are critical to designers of moving vehicles, aircrafts, ships and other structures. As the aerodynamic properties of a vehicle rely chiefly upon its shape, so the exterior shape of vehicle gets huge attention in planning a vehicle. So, the variation of drag with the adjustment in exterior shape of bus has got the priority in this work. Buses with their diverse mathematical measurements give distinctive stream qualities over the front and rear end of models. A total of six models have been designed to perform the numerical analysis. The width and height of the models are kept same, but the exterior shape in the rear and front end of the bus has been varied in each of the models. For different highway speed regulations, computational investigations have been carried out on the models using ANSYS Fluent 19.2 (Student Version). The results of the simulation are acquired in terms of coefficient of drag for different speeds of the bus. Then, the value of drag coefficient for different shaped models has been analyzed, which gives the outcome of the investigation as per improvement in shape.

Keywords: Bus, Drag Coefficient, Shape Advancement, Fuel Consumption, CFD.

1. Introduction

The improvement of eco-friendliness is a significant issue in automotive industry. Producers are acquainting more eco-friendly vehicles onto the market with the expanding request. The streamlined properties of a vehicle altogether influence the performance and fuel utilization attributes of the vehicle. A lot of air is uprooted and streamed around the vehicle when a bus moves. Aerodynamic drag is one of the fundamental hindrance in the acceleration of a solid body when it moves through air. Around 50 to 60% of total fuel is lost uniquely to defeat this troublesome streamlined power [1].

The drag coefficient assesses the obstruction of an item in a fluid environment. It changes with speed and bearing of stream, object shape and size [2]. The lower is the drag coefficient, the less streamlined drag happens on an item. Moreover, the structural changes are done so as to improve street strength of the vehicle and to diminish the drag through coordinating the wind current in various manners. In this manner, vehicle fashioners are keen on planning vehicles with expanded mileage through advancing the exterior formation of vehicle.

Though friction between the wheels and the road is also an important consideration in the analysis of fuel economy, but the main emphasis of the paper has been put only between the relation of drag reduction with the change in exterior shape. Frontal shape of a vehicle has a critical commitment in the outward composition of a vehicle. Buses with their diverse exterior measurements give distinctive stream attributes over the front appearance. Hence the resultant net streamlined power, for example, drag and lift differs. Thus, the enhancement of vehicle front shape has more prominent effect on the fuel utilization quantities.

In addition to this, consumption of fuel is straightforwardly connected with the CO_2 emanation from the vehicle. The temperature of the world is expanding rapidly. The excessive increment of CO_2 has the main accountability in this regard. About 20% of all CO_2 emanations is accounted from the transport division [3]. The decrease of CO_2 is the most noteworthy issue to oppose an earth-wide temperature boost. To decrease the CO_2 discharge rate from vehicles, the improvement of efficiency is the primary inference. This paper will show an effective adjustment of bus model by advancing its exterior shape to improve mileage and consequently to lessen CO_2 outflow from vehicles. Moreover, the use of public vehicle is expanding quickly and individuals are contemplating on decreasing the expense of fuel consumption. In this motivation, vehicle producers are worried in planning the vehicle to lessen the drag through improving various boundaries [4]. In this work, computational investigation has been completed for different speed regulations depending on traffic criteria and change in exterior shape of the bus models for those velocities give a sign of the reduction in drag, thus a fuel efficiency improvement.

2. Method

A sum of six models have been prepared to perform the numerical analysis. The technique is begun with a solitary initial base model. After that the model is imported in the Design Modeler of ANSYS 19.2. Subsequent process of creating the mesh, it is simulated in Fluent characterizing the boundary conditions. After completing the whole run, the model is upgraded and a similar method is rehashed. The whole process has been shown in the schematic diagram in Fig.1.

* Corresponding author. Tel.: +88-0191135959
E-mail addresses: prantokarua@me.kuet.ac.bd[1*], pronobdebnathpartho@gmail.com[2]

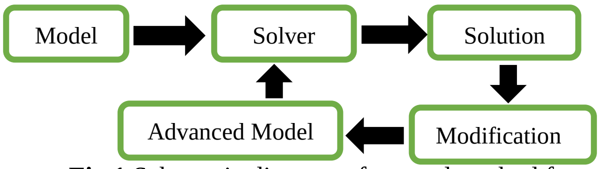

Fig.1 Schematic diagram of general method for shape advancement

The models to be analyzed are shown below:

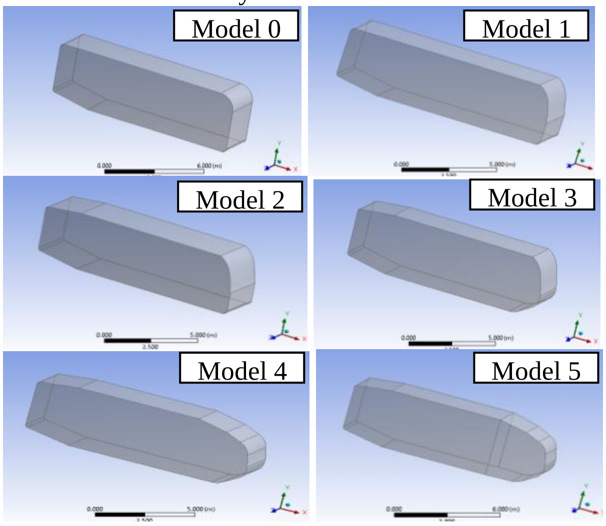

Fig.2 Geometrical models of this investigation

A length of 11.9m and width of 2.55m is utilized while planning the model. The length and width of the vehicles are saved same for all the models. Model 0 is taken as the base model of the investigation. Changes have been made in the remainder of models. Model 1 is exposed to a semi curvature front shape, Model 2 is subjected to a roof rear end chamfer, Model 3 is exposed to a fillet edge at the lower front portion of the bus, Model 4 is subjected to a flat curvature at the upper front portion of the bus. And lastly Model 5 is designed through chamfering from the roof to bottom on both side walls at about 3m from the front. All the models are shown in Fig.2.

A fluid volume has been made to simulate the airflow around the vehicle. For this reason, an enclosure has been created that encompasses the vehicle and boolean operation has been executed to deduct the vehicle body in the Design Modeler module of Fluent. This surrounding pretends as the air area, which is shown in Fig.3. The essences of the boundary in area has been named as inlet, outlet, topwall, bottomwall, sidewall and symmetry. The front of the busbody is kept 11m from the inlet of the enclosure. And the distance from the rear end to the outlet is 23m because of having probability of backflow formation at the rearend of the bus. The height of topwall from the bus roof has been kept 6m because of rapid pressure variation on the roof. The distance from the bus bottom to bottomwall is 1m as it proceeds onward the street. To decrease the computational cost, a symmetry plane is presented at the half of the model.

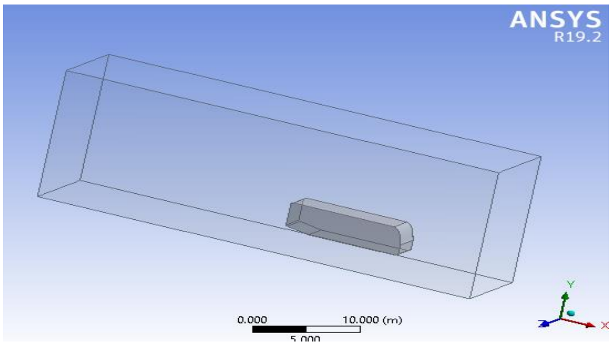

Fig.3 Computational domain in Design Modeler of Fluent Solver

3. Governing Equations

The continuity and momentum equations with a turbulence model are used to solve the airflow. The equations are:

$$\frac{\partial u}{\partial v}+\frac{\partial v}{\partial y}+\frac{\partial w}{\partial z}=0 \qquad (1)$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial x}+\frac{1}{\rho}\left(\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{xz}}{\partial x}\right)+B_x \qquad (2)$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial y}+\frac{1}{\rho}\left(\frac{\partial \tau_{xy}}{\partial y}+\frac{\partial \tau_{yz}}{\partial x}\right)+B_y \qquad (3)$$

$$u\frac{\partial u}{\partial x}+v\frac{\partial v}{\partial y}+w\frac{\partial w}{\partial z}=-\frac{1}{\rho}\frac{\partial p}{\partial z}+\frac{1}{\rho}\left(\frac{\partial \tau_{xz}}{\partial y}+\frac{\partial \tau_{yz}}{\partial x}\right)+B_z \qquad (4)$$

Where u is x-component of velocity vector, v is y-component of velocity vector and w is z - component of velocity vector. ρ is density of air, p is static pressure, τ is shear stress and B_x , B_y , B_z are the body forces [5].

For turbulent kinetic energy, k :

$$\frac{\partial k}{\partial t}+U_j\frac{\partial k}{\partial x_j}=P_k-\beta^* k\omega+\frac{\partial}{\partial x_j}[(\nu+\sigma_k\nu_T)\frac{\partial k}{\partial x_j} \qquad (5)$$

For specific dissipation rate, ε :

$$\frac{\partial \omega}{\partial t}+U_j\frac{\partial \omega}{\partial x_j}=\alpha S^2-\beta\omega^2+\frac{\partial}{\partial x_j}[(\nu+\sigma_\omega\nu_T)\frac{\partial \omega}{\partial x_j}]+2(1-F_1)\sigma_{w2}\frac{1}{\omega}\frac{\partial k}{\partial x_i}\frac{\partial \omega}{\partial x_i} \qquad (6)$$

SST k - ω turbulence model is used with second order discretization scheme for pressure and momentum. The solver is first run 10000 iterations with convergence criteria 0.001.

4. Mesh Generation

A volume mesh is generated using sizing functions of 0.5m per element. A face size of 0.2m is utilized on the busbody to refine the mesh. Before importing in fluent, the mesh shaping has been converted into polyhedral. The final mesh is shown in Fig.4 and the extended view of the inflation layers in mesh is shown in Fig.5.

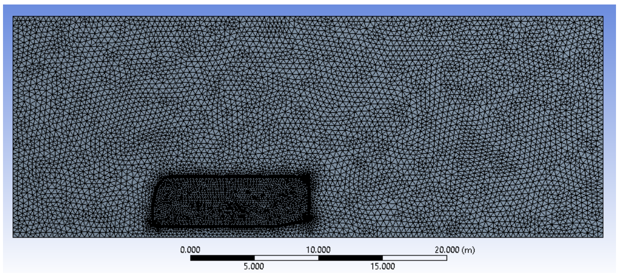

Fig.4 Final Mesh of Bus Model 1

Also, ten layers of inflation has been added around the busbody to adapt to the detachment of boundary layer.

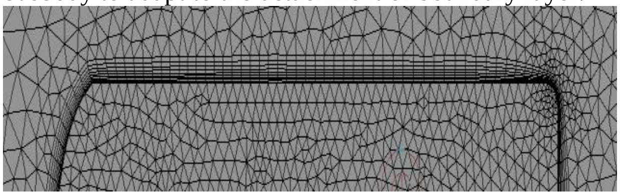

Fig.5 Ten layers of inflation

The complete number of nodes and elements have been presented in Table 1.

Table 1 Number of nodes and elements in each model

Model Name	Nodes	Elements
Bus Model 0	95460	371277
Bus Model 1	89657	351131
Bus Model 2	92757	367765
Bus Model 3	91323	366279
Bus Model 4	91520	365628
Bus Model 5	98421	419726

After generating the initial mesh, the boundary conditions has been set in the solver to run the simulation. And the result got from each run is compared to perform the mesh dependency test through changing the element size on the busbody.

4.1 Accuracy of Analysis

This simulation work is verified by the journal named "Aerodynamic Exterior Body Design of Bus" published in International Journal of Scientific & Engineering Research, Volume 4, Issue 7, July, 2013 [6].

4.2 Verification of Bus Model 0

The verification has been made in terms of coefficient of drag and the percentage of deviation from the paper study has also been shown in Table 2.

Table 2 Verification of Present Work to Paper Work

Velocity (km/hr)	Paper work, C_D	Present work, C_D	Percentage of error, %
80	0.466	0.471	1.06
100	0.702	0.709	0.98
115	0.925	0.941	1.71

4.3 Plots of Verification

The line represented through blue color is showing reference paper work and the line represented through orange color is showing present work. Almost similar results have been got in the verification, which is shown in Fig.6 through plotting a graph.

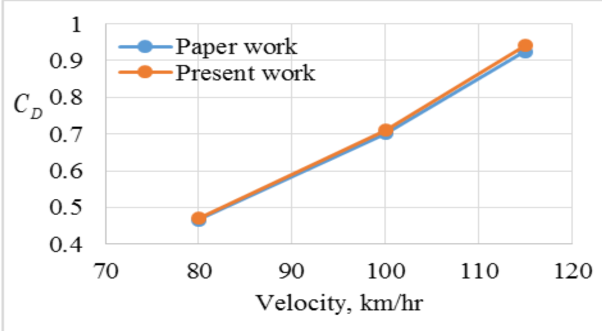

Fig.6 Verification of Present work to Paper work

4.4 Mesh Sensitivity

The mesh independence test has been shown in Table 3. Almost similar result has been found for the meshes mentioned, thus the least number of element (371277) is selected to reduce the computational cost.

Table 3 Mesh dependency test for Base Model 0

Number of Mesh	Coefficient of drag, C_D
371277	0.7380
386121	0.7362
401532	0.7401
425292	0.7399
466318	0.7411
490445	0.7353

5. Post Processing

The contours of static pressure and velocity streamlines at the front and rear end of all the geometries are illustrated on the following sections.

5.1 Static Pressure Contours of the Models

The static pressure variation along the models has been shown from Fig.7 to Fig.12. Pressure is maximum at the point where the flow strikes the bus. The pressure along the whole roof is almost constant except at the contact points of the front and rear end roof.

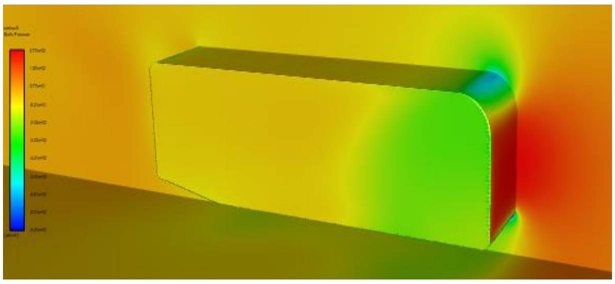

Fig.7 Static Pressure contour of Model 0

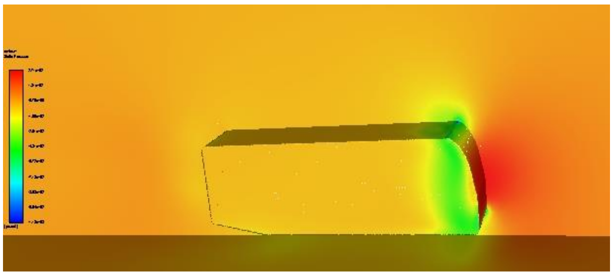

Fig.8 Static Pressure contour of Model 1

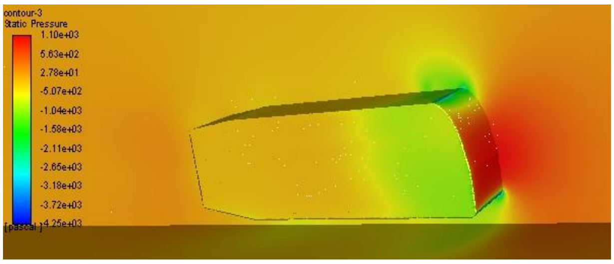

Fig.9 Static Pressure contour of Model 2

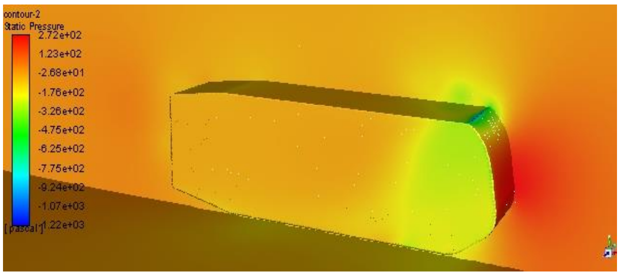

Fig.10 Static Pressure contour of Model 3

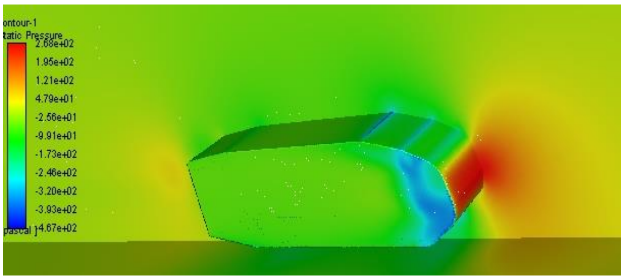

Fig.11 Static Pressure contour of Model 4

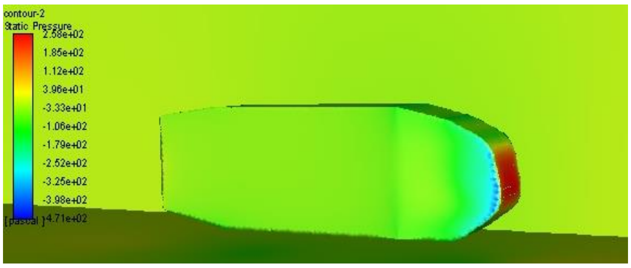

Fig.12 Static Pressure contour of Model 5

5.2 Velocity Streamlines through the Models:

The velocity streamline for all the models has been represented in the contours from Fig.13 to Fig.18. The pathlines are deflected significantly when the flow is passing over the varying frontal shape on different models. In all of models, the vortex generation takes place at the rear end of the models because of having back pressure.

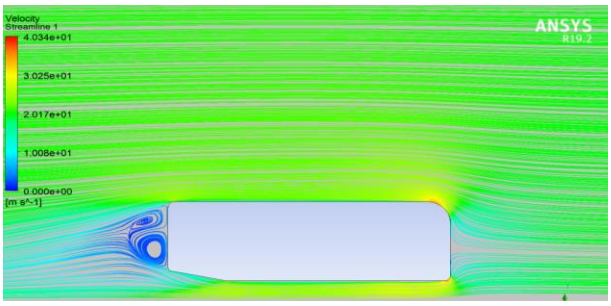

Fig.13 Velocity Streamlines of Bus Model 0

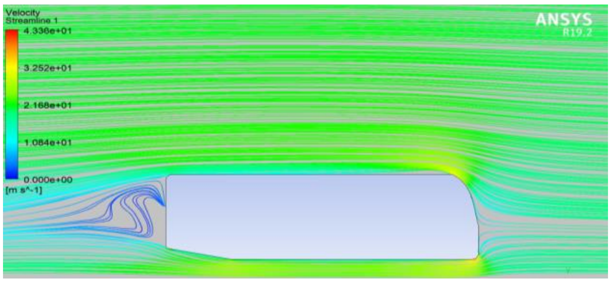

Fig.14 Velocity Streamlines of Bus Model 1

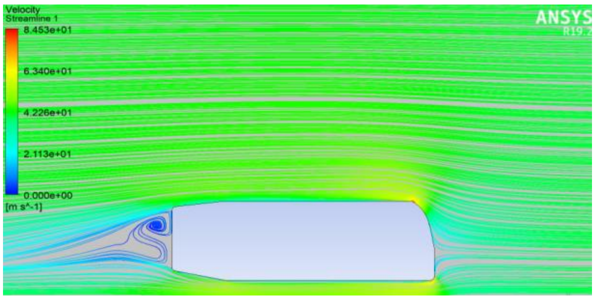

Figure.15 Velocity Streamlines of Bus Model 2

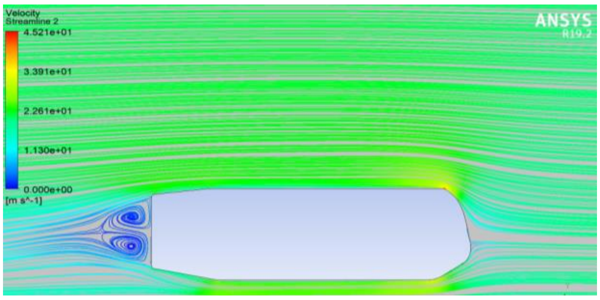

Figure.16 Velocity Streamlines of Bus Model 3

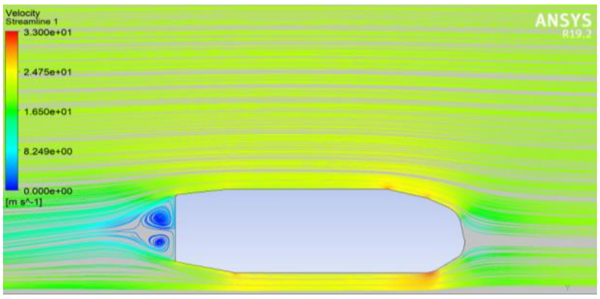

Figure.17 Velocity Streamlines of Bus Model 4

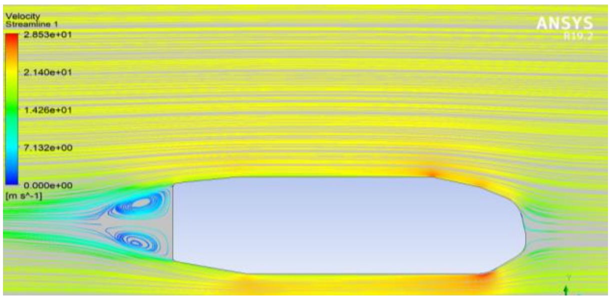

Fig.18 Velocity Streamlines of Bus Model 5

6. Result and Discussions

6.1 Result

In Table 4, the result has been represented in terms of drag and lift when the models are subjected to 108 km/hr velocity. Normally in Bangladesh, the speed of high performance road vehicle ranges from 80 km/hr to 120 km/hr. That's why 108 km/hr has been taken as a standard to show the post processing results in the whole simulation.

Table 4 The value of C_D and C_L for the Models

Model Name	Coefficient of drag, C_D	Coefficient of lift, C_L	Iterations to converge
Bus Model 0	0.738	0.799	635
Bus Model 1	0.592	0.731	511
Bus Model 2	0.588	0.609	689
Bus Model 3	0.481	0.502	453
Bus Model 4	0.406	0.451	707
Bus Model 5	0.375	0.249	326

6.2 Velocity vs Coefficient of Drag Plot for Model 0

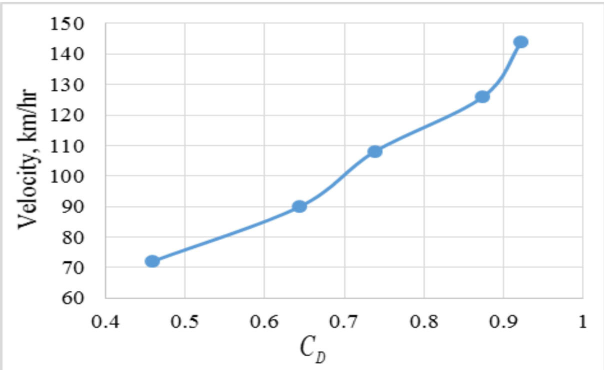

Fig.19 Velocity vs Coefficient of Drag Plot for Model 0

In Fig.19, the above plot shows the variation of coefficient of drag with increasing the velocity. Drag increases proportionally with the velocity for the base model.

6.2 Velocity vs Coefficient of Drag Plot for Model 1

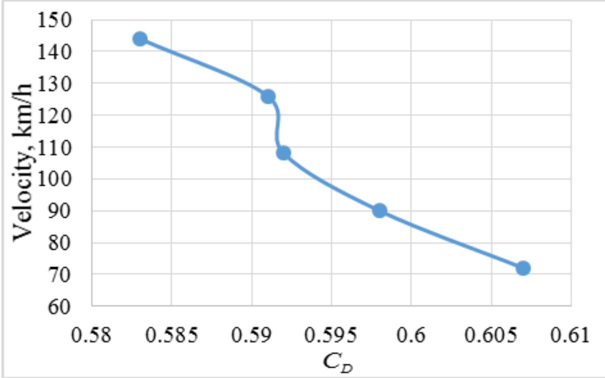

Fig.20 Velocity vs Coefficient of Drag Plot for Model 1

The plot in Fig.20 indicates the relationship between drag and flow velocity. Drag changes inversely with the velocity for the model having semi curvature frontal shape.

6.3 Velocity vs Coefficient of Drag Plot for Model 2

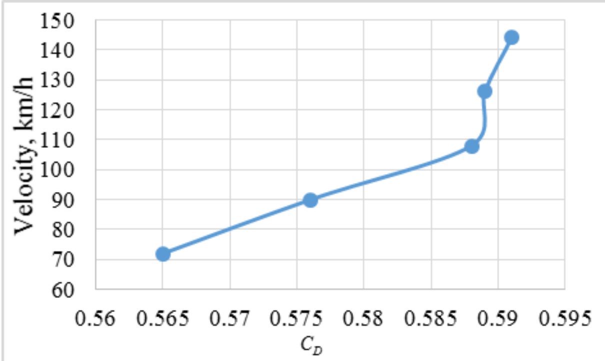

Fig.21 Velocity vs Coefficient of Drag Plot for Model 2

This plot relates drag generation with flow velocity. Drag increases rapidly with the velocity for a bus model having roof rear end chamfer as compared to Model 1, which is shown in Fig.21.

6.4 Velocity vs Coefficient of Drag Plot for Model 3

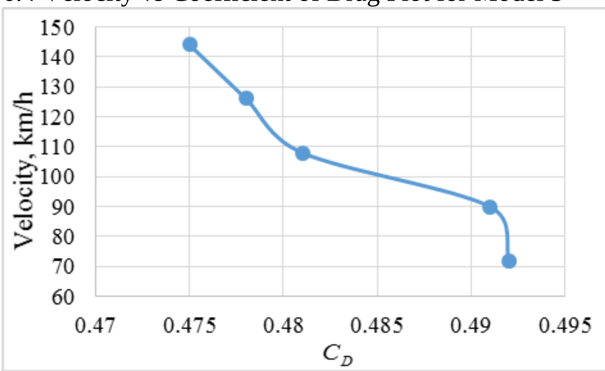

Fig.22 Velocity vs Coefficient of Drag Plot for Model 3

In this plot in Fig.22, drag changes inversely with the velocity for the model having fillet edge at the lower front portion of the bus.

6.5 Velocity vs Coefficient of Drag Plot for Model 4

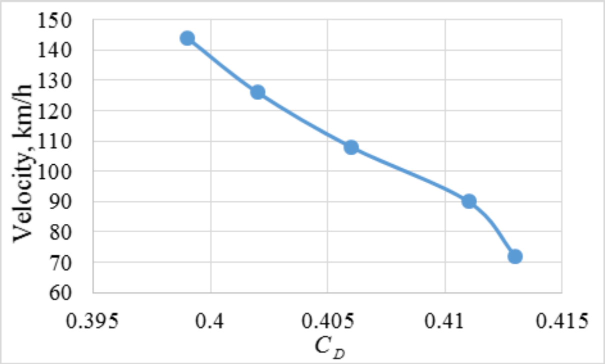

Fig.23 Velocity vs Coefficient of Drag Plot for Model 4

From the Fig.23 shown above, it is seen that drag changes inversely with the velocity for the model having flat curvature at the upper front portion of the bus.

6.6 Velocity vs Coefficient of Drag Plot for Model 5

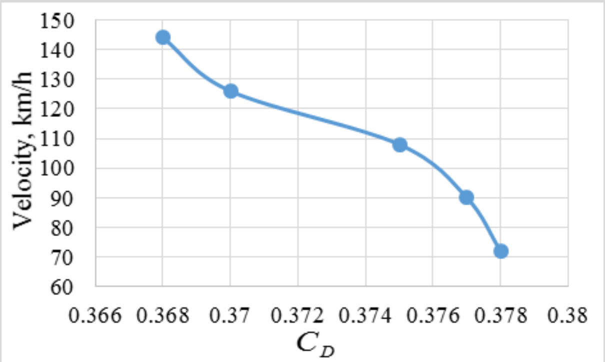

Fig.24 Velocity vs Coefficient of Drag Plot for Model 5

This plot in Fig.24 informs that drag has an inverse relationship with the velocity for the final model which is designed through chamfering from the roof to bottom on both side walls at about 3m from the front.

6.7 Coefficient of Drag vs Velocity Plot for All Models

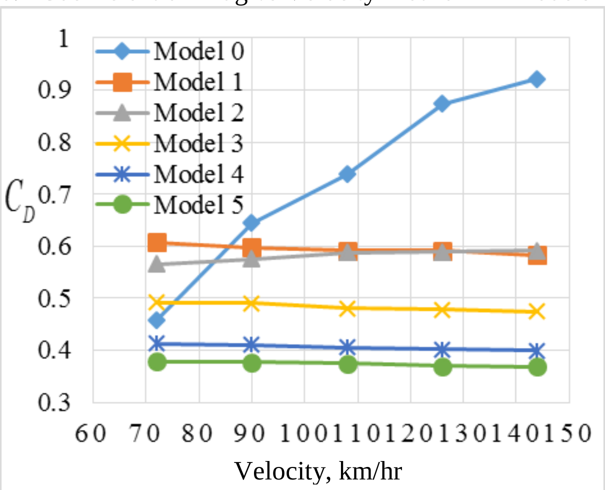

Fig.25 Coefficient of Drag vs Velocity Plot for All Models

The difference in coefficient of drag between all the models subjected to same velocity has been shown in Fig.25. The modified bus Model 3, 4, and 5 provide a significant decrease in drag under high velocities as compared to Car Model 0 and 1.

7. Conclusions

The resulting drag coefficient provides an indication of the fuel economy improvement for those models discussed. The value of drag coefficient in the initial model was 0.738 while finishing the run at a velocity of 108 km/hr. After that, five modifications had been made in the geometry. On the first modification, the value of drag coefficient was 0.592 and the further modification in geometry gave the value of drag coefficient lastly 0.375, which can be considered as the most updated model. Thus, the modification of shape in exterior body to reduce drag can be concluded as the fulfillment of the investigation. The work can further be extended by comparing the accuracy of the simulation with the experimental wind tunnel testing.

8. References

[1] VL Srinivas ‘Shape Optimization of a Car Body for Drag Reduction and to Increase Downforce” Seminar Report, Indian Institute of Technology Bombay, March, 2016.

[2] Al-Saadi, Ahmed Ali Shakir “Analysis of Novel Techniques of Drag Reduction and Stability Increase for Sport Utility Vehicles using Computational Fluid Dynamics” PhD thesis, University of Leeds, 2019.

[3] A. K., Lilley, G., Syred, B. A., Swirling flows, Abacus press, Ed. 1, England, 1984.

[4] FeiqiLiu, FuquanZhao, ZongweiLiu, HanHao “Can autonomous vehicle reduce greenhouse gas emissions? A country level evaluation.” Energy Policy, Volume 132, Pages 462-473, September 2019.

[5] ANSYS Fluent, Release 16.2: Help Topics

[6] A. Muthuvel, M.K. Murthi, Sachin N.P, Vinay. M. Koshy, S. Sakthis, E. Selvakuma, “Aerodynamic Exterior Body Design of Bus” published in International Journal of Scientific & Engineering Research” Volume 4, Issue 7, July, 2013.

NOMENCLATURE

C_D : Coefficient of drag
C_L : Coefficient of lift
V : Velocity, ms^{-1}
ρ : Density of air, Kgm^{-3}
k : Turbulence kinetic energy, J/Kg
ε : Specific dissipation, m^2/s^3