American Journal of Mechanical and Industrial Engineering
2023; 8(4): 99-109
http://www.sciencepublishinggroup.com/j/ajmie
doi: 10.11648/j.ajmie.20230804.12
ISSN: 2575-6079 (Print); ISSN: 2575-6060 (Online)

Numerical Analysis for Reducing Fuel Consumption and CO_2 Emissions in a Passenger Bus

Md. Rezaul Karim Sikder*, Mohammad Zoynal Abedin*

Department of Mechanical Engineering, Dhaka University of Engineering and Technology, Gazipur, Bangladesh

Email address:

20103001@student.duet.ac.bd (Md. Rezaul Karim Sikder), abedin.mzoynal@duet.ac.bd (Mohammad Zoynal Abedin)

*Corresponding author

To cite this article:

Md. Rezaul Karim Sikder, Mohammad Zoynal Abedin. Numerical Analysis for Reducing Fuel Consumption and CO_2 Emissions in a Passenger Bus. *American Journal of Mechanical and Industrial Engineering*. Vol. 8, No. 4, 2023, pp. 99-109.
doi: 10.11648/j.ajmie.20230804.12

Received: July 23, 2023; **Accepted**: August 23, 2023; **Published**: October 31, 2023

Abstract: The transportation system is one of the most important ways for the public, commercial goods and materials to transit from one place to another place. Some of the vehicles are used such as cars, passenger buses, trucks and heavy vehicles. The drag reduction is directly effect on the fuel saving as well as the efficiency of the vehicle. Hence, for the analysis, a Computational Fluid Dynamic (CFD) analysis has been done using ANSYS^{R}19.0 workbench and an investigation has been done. Three demo base models are considered for the analysis namely model-1 made of plastic body, model-2 made of metal body and model-3 of BD bus that run in the Bangladeshi roads. Some extensive modifications are done in the bus body such as front and rear side area which helps to reduce the aerodynamic drag of the bus. The numerical analysis for the drag forces, drag coefficient, fuel consumption and CO_2 reductions have been done for model-1,2 & 3 as well as for their respective modified models. For model-3 validated with the Sunglong China bus model in the analysis. For base model-1 &2, the fuel savings 19.50% and 22.20% respectively. On the other hand, the CFD values at a velocity of 110 km/h for base model-3 the drag forces are reduced for base model modified-1, base model modified-2 as 1987.20 N, and 2499.80 N, respectively. In addition, the fuel savings for base model modified-1, base model modified-2 is 2.53 liters/hour, 4.11 liters/hour, respectively. In addition, the CO_2 reductions for base model modified-1, base model modified-2 is 7.072 liters/hour, 11.488 liters/hour, respectively.

Keywords: Aerodynamics Drag Reduction, Drag Force, Bus, Fuel Saving, CO_2 Reduction, CFD, Ansys

1. Introduction

Aerodynamic is a part of fluid dynamics which expresses the air velocity during through the solid object. It's generally consists between the vehicle and fluid flow during run on the road. The fluid characteristics as like viscosity, density, compressible or non-compressible, and the speed of the fluid are directly effect on the aerodynamic drag. The drag forces are divided into two components as such pressure drag perpendicular to the surface and friction drag along the surface. A dimensionless number is used the measure the aerodynamic drag of the vehicle which called as the drag coefficient or the co-efficient of drag (C_d) [1].

The automotive industries focused to reducing the fuel consumption by reduction drag of their products due to the economically and climate change of the world. Besides that the legislation and governments promote the companies to reduce the emissions of their vehicles for friendly environment. The emission can be reduced by different approaches such as fuel quality, powertrain, and electric or hybrid system of the vehicle. The most effective way is to a reduction in driving resistance of the vehicle. The following expression can determine the required force to propel or move the vehicle on the road:

$$Freq = FD + FR + FA + FG$$

Where,
Freq = Required force
FD = Force due to aerodynamic drag
FR = Force due to rolling resistance
FA = force needed to accelerate the vehicle
FG = Force due to gravity

Above the expression revealed that the rolling resistance, the mass, and the aerodynamics drag are the main parameters of the vehicle. When a passenger bus run in 100 km/hour Aerodynamic resistance and rolling resistance are assumed equal [2].

Figure1shows the required force or thrust force F, overcome the drag force F_{darg} and to propel the vehicle on the road. Reactive force and weight of the vehicle are directly applied force to the perpendicular of the road. Lift force act to the opposite of the vehicle weight.

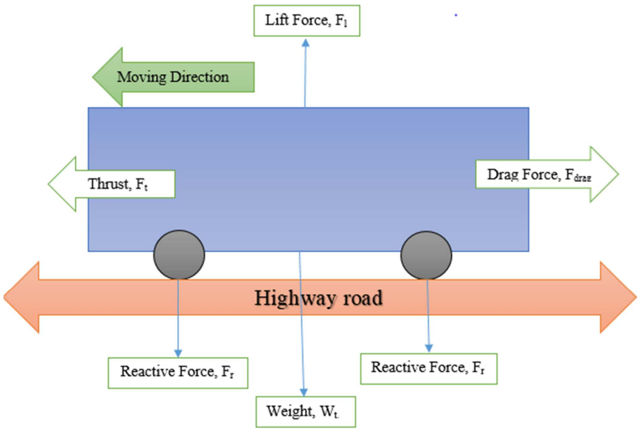

Figure 1. *Required force to propel the bus on road [2].*

There are many literature review published in the field of aerodynamic drag of the vehicles. The additional purposes of the review papers are comparing among the papers and to investigate the outcomes of the results.

Newland [3] the aimed of the passenger bus to develop and obtain with fuel consumption. The function of fuel consumption based on the relationship which executed in the literature review. To compare of the fuel consumption between the reference bus model and various passenger bus operating characteristics. Arun Raveendran et al. [4] carried out extensive product study, market study, also aspirations and frustrations of travelers were recorded. CFD Simulation was done using ANSYS FLUENT commercial package from various studies and simulation a new design of bus with low floor height and appealing exterior with extemporized aerodynamics was developed. The modification in exterior design resulted in reduction in Coefficient of drag from 0.53 to 0.29 and over all drag reduction by 60%.

Roy and Srinivasan [5] described the aerodynamics bus and various high-speed buses have the target and significant to reduce the road accident and improve the fuel consumption of the bus by aerodynamic drag reduction. To study the aerodynamic drag and provide the exterior rear view mirrors also equipment setup with body. To modifying bus geometry can reduce aerodynamic drag and to obtain the fuel economy.

Carr [6] illustrated the effects of fluid flow of the front and rear side of the rectangular vehicle is investigated in ground proximity. In the experimental results the coefficient of drag value is calculated with low leading edge is 0.21. W H Hucho [7] stated the one way to develop fuel efficient vehicles is the reduction of aerodynamic drag as it accounts for around 80% of the total drag at vehicle cruising speeds over 80 km/h. The reduced drag will not only lower the fuel consumption but also the CO_2 emissions. The major drag reductions have been achieved by optimizing vehicle exterior body shapes over four decades. At low speeds the main source of drag is the rolling resistance. Typically, the aerodynamic drag of a medium-sized car accounts for 75-80 percent of the total resistance to motion at 100 km/h, the rest being mainly rolling resistance.

Abdel Gawad and Abdel Aziz [8] illustrated the effect of front shape of buses is investigated numerically and experimentally of the characteristic of fluid flow. Also the rear side of the bus considered the heat transfer in driving tunnels. Three bus models considered with modification of the front side as flat, inclined and curved. It's observed that the modification of inclined and curved of front side of vehicle is better than the aerodynamic drag of modification flat by about 20%.

A. Muthuvel et al. [9] explained the name of the experiment is Aerodynamic exterior body design of bus executed a wind tunnel with numerical test and to obtain the effectiveness result of the new design model. It has been proved that from experimental result and the aerodynamic drag force is considered about 30-40% from the existing value of the bus. Also fuel consumed is reduced about 6-7 liters for the every 100 km.

P. Gopal et al. [10] illustrated the aerodynamic drag force, lift force, drag coefficient, pressure coefficient with and without vortex generator. A wind tunnel considered 1: 15 scaled as well as the different yaw angle as 10°, 15°, and 20° considered. The different velocities of the vehicles are 2.42, 3.7, 5.42 and 7.1 m/s. The aerodynamic drag reduction with vortex generators to obtain in the experiments.

(a) L=180 mm, W = 58 mm, H = 60 mm.

(b) L = 200 mm, W = 80 mm, H = 85 mm.

(c) L -11275 mm, W -2950 mm, H -3090 mm

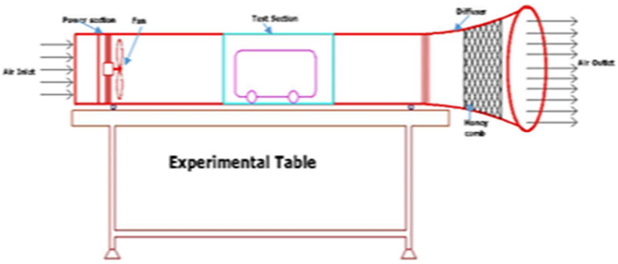

Figure 2. Setup in experimental wind tunnel for (a) Base Model 1 (b) Base Model 2 (c) & Base Model 3 (d) Experimental Table.

Alam et al. [11] described to overcome the aerodynamic resistance almost 80% (out of total power) vehicle powers is required and remaining power is used for rolling resistance. The total fuel cost of the system depends on the extra drag due to add-ons causes. The preliminary purpose of this thesis was the aerodynamic drag measure in experimentally under range of vehicle speeds 25 m/sec.

2. Computational Analyses

CFD is the analysis of fluid flows using numerical solution methods. To analysis complex problems consists fluid-fluid, fluid-solid or fluid-gas interaction. It is based on the Navier-Stokes equation. It is divided in different way such as; Viscous-laminar K-epsilon (2-eqn) equation, Reynolds-Averaged Navier-Stokes (RANS) equations, Euler equation and Direct Numerical Simulation (DNS). The turbulence model equations are divided into two equations, one is k–omega (k–ω) and another is k-omega (k–ω). For two variables k and ω, the primary variables k is indicated that turbulence kinetic energy (K. E) [12].

2.1. Numerical Model

Three passenger bus models are planned to be simulated by ANSYS R 19.0 CFD software. The following conditions for the parameters of aerodynamic drag is simplified and advanced of the bus. The experimental data is to be used to validate with the simulations result.

2.2. Models Preparation

The models are prepared by ANSYS R19.0 software before simulation. The dimensions of the models are measured physically during experimental conditions. For the preliminary step the dimension are measured before the simulation of the models as per measured physically. The simulation systems are more accuracy than the standard physical models for surface finishing. So that, the simulation process provides a precious parameters of the models.

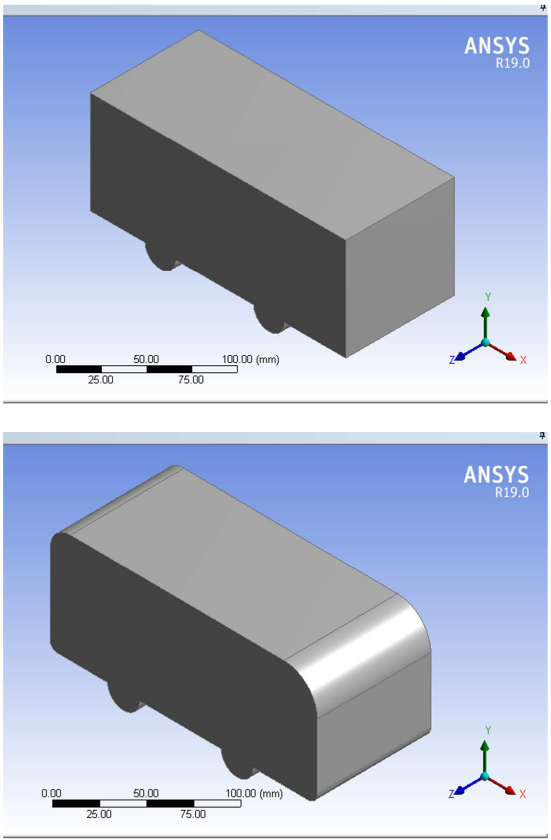

Figure 3. Model base model and base model modified.

Table 1. Computational domain conditions of the base model-1, 2 & 3.

Items	Computational domain
Software	ANSYS ^{R}19.0 CFD software
Mesh	Hexahedral mesh with prism layer
Geometry	Simplified box created by itself
Yaw angle	Zero degree angles and only one angle is considered.
Aerodynamic bus designs	Base Treatment
Processor	Intel (R) CORE (TM) i5-8250 CPU @ 1.80GHz, RAM 4.00GB

2.3. Geometry of the Models

The geometric modelling is a part of computational geometry that studied the process and mathematical explanation of the modelling shapes. The model can be prepared by many standard software such as solid works, CATIA, AutoCAD or ANSYS^{R}19.0 software etc. The major components dimension of the vehicle are counted and measured. The minor components are ignored due to obtain simplicity of the design. All linear dimensions as like length, width, height and radius are measured in millimetre (mm) [13].

***Table 2.** Specification in mm of base model 3.*

S. N	Specification	Details	S. N	Specification	Details
1	Overall length(L)	11275 mm	5	Front over hang	2875 mm
2	Overall width(W)	2950 mm	6	Rear over hang	2850 mm
3	Overall height(H)	3090 mm	7	Wheel radius(r)	350 mm
4	Wheel base	6000 mm	8	Min. ground clearance	265 mm

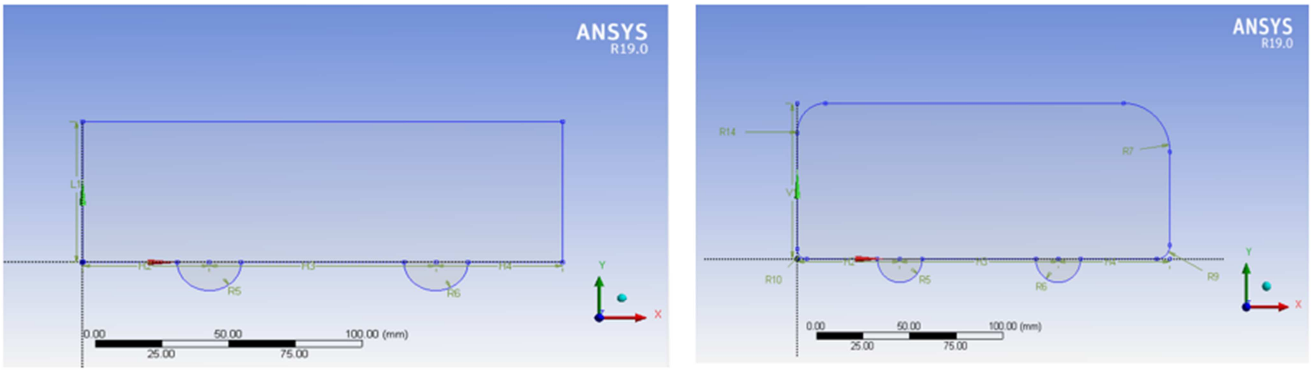

***Figure 4.** Geometry of the base model 3 and base model modified 1.*

2.4. Computational Grid (Mesh)

A rectangular box is selected which consists of an inlet, an outlet, a roof, a ground surface and two sides of the body. The mesh was designed on the geometry of the bus model and surface domain. The inlet and outlet of the tunnel distance from the bud model centroid were keep11250 mm and 16250 mm respectively. The mesh was created by hexahedral which discretized the computational domain.

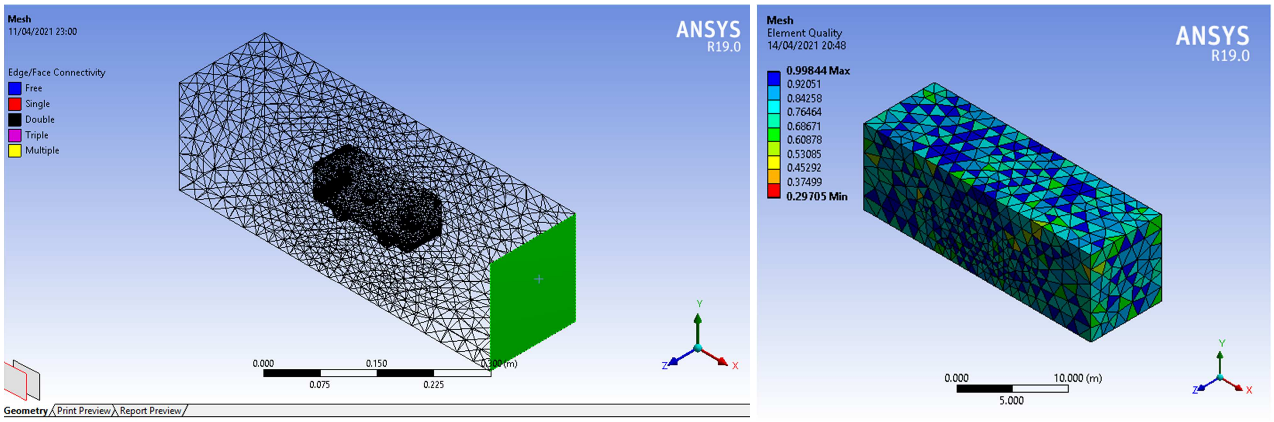

***Figure 5.** Computational grid mesh of the base model 3 & base model modified 1 & 2.*

***Table 3.** Mesh Condition of the base model 3 & base model modified 1 & 2.*

S. N	Type	Base Model 1	Base Model 2	Base Model 3
1	Length X	0.65 m	0.65 m	30.225 m
2	Length Y	0.232 m	0.232 m	10.44 m
3	Length Z	0.225 m	0.225 m	9.95 m
4	Volume	3.2557e-002 m^3	3.2557e-002 m^3	3032.7 m^3
5	Nodes	8610	8610	9829
6	Elements	45610	45610	52885
7	Smoothing	high	high	high

2.5. Boundary Conditions

The bus is subjected or faced by the different cross wind during run the vehicle on the road. The boundary conditions is the vital point for aerodynamic drag force calculation of the cross wind. In generally it's defined as the surrounding of the vehicle body to contact with air during run on the road. There are many parameters were consists as inlet and outlet of the air flow.

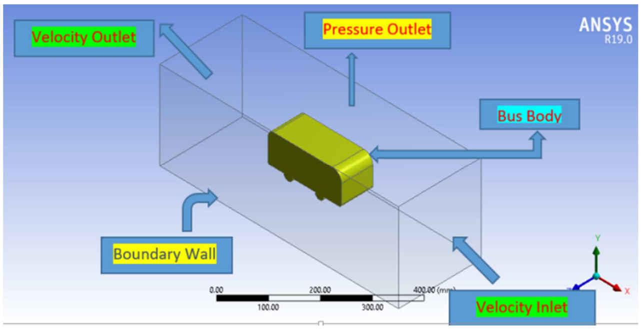

Figure 6. *Boundary condition of the base model-1, 2 & 3.*

2.6. Simulation

In these models, the simulations are performed by ANSYS[R]19.0 CFD software. After completed the geometry, mesh analysis, setup, solutions and results of the model performed the simulation of the bus. Every simulation developed from minimum 200 iterations number to 2500 iterations number. The residuals and drag plot are performed in the solutions and checked. In the results, velocity streamline, velocity vector, pressure contour found in different three dimensional views. The deepest area of the figures of any parameters indicated that more pressure drop or velocity drop of the fluid flow.

3. Aerodynamics Drag Calculations

The results are performed by ANSYS[R]19.0 CFD software which is calculated by the following equation. The equation defined that, the relation among the drag force *(F_d)*, drag coefficient *(C_d)*, frontal area of the bus *(A_f)*, Velocity of the bus *(V_b)*, and density of the fluid which through the bus. The following equation 6 is given bellow.

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

4. Results and Discussion

Figures 7 and 8 show the drag force (N) vs. speed (km/h) for base models 1 and 2 along with the y and x axes, respectively. When the speed is increased, the drag force also increases and vice versa for the base model and the modified base model. Base model modified 1 and 2 have lower drag forces than base models 1 and 2.

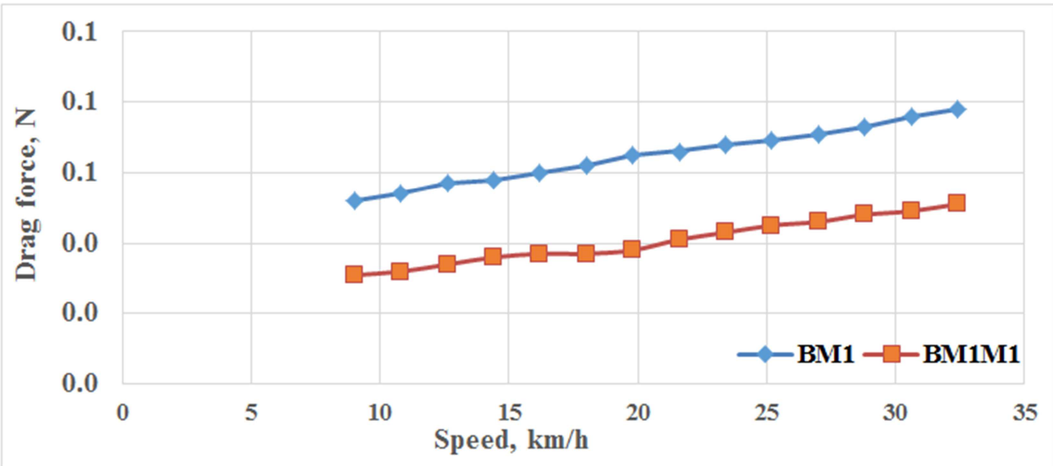

Figure 7. *Drag force versus speed of base model 1 & base model modified 1.*

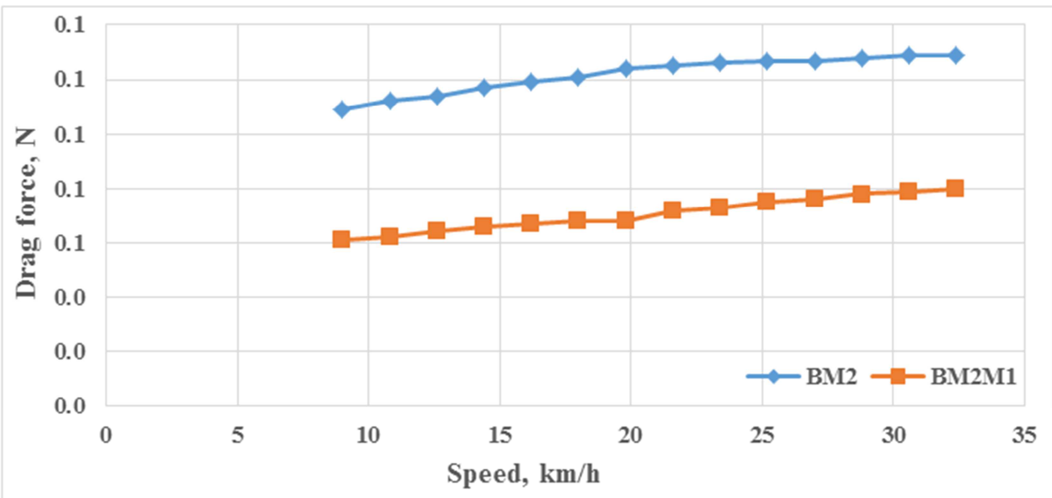

Figure 8. *Drag force versus speed of base model 2 & base model modified 1.*

***Table 4.** Drag force for base model modified 1 & 2 with Sunlong China bus 1 & 2.*

Velocity, Km/h	Drag Force, Fd0 (BM-3)	Drag Force, BM3M1	Drag Force, BM3M2	Sunlong China Bus-1
80	2218.60	1600	1436.50	1366.06
90	2806.60	2016	1820.50	2264.12
100	3462.20	2466.1	2180.70	2913.11
110	4977.00	2989.8	2477.20	3978.14

Figure 9 illustrates the relationship between drag force (N) and speed (km/h) for the Sunlong China bus 1 and 2 and base model 3. The y axis represents drag force (N), while the x axis represents speed (km/h). For the base model 3 and Sunlong China bus, the drag force increases as the speed increases and vice versa. In comparison to Sunlong China buses 1 and 2, the base model modified has lower drag forces.

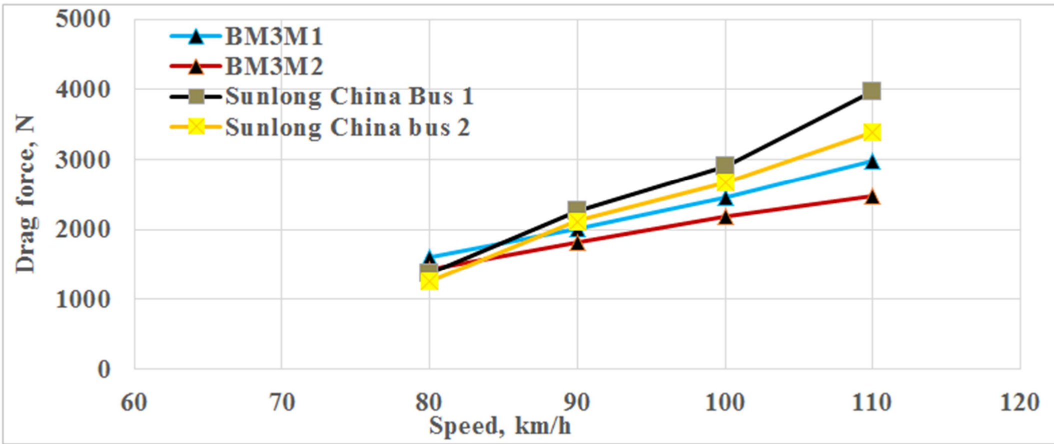

***Figure 9.** Drag force for base model modified 1 & 2 with Sunlong China Bus 1 & 2.*

4.1. Fuel Saving of the Models

The fuel system is the vital points which directly effect on efficiency and costing of the vehicles. There are many systems are developed to reduce the fuel consumption of the vehicles. The following equation are derived that the fuel savings calculated in percentage (%). This equation executed the relation between the fuel savings and the coefficient of drag in specific case and reference case of vehicle.

$$\text{F. S } (\%) = \frac{\Delta Cd.\ 16}{30} \quad (2)$$

Above the formula is used in the experiments for a long distance vehicle of bus weight almost 40 ton and multiplying factor considered 16. The denominator 30 is executed the fuel consumption of 30Liters/ 100 km during experiments [14].

$$\text{F. S } (\%) = \Delta Cd/1.88 \quad (3)$$

Figure 10 represents the drag reduction in percentage (%) and Fuel saving in percentage (%) with velocity for base model-1 & 2. Drag reduction and fuel saving indicated by Y axes and speed indicated by X axis. For base model 1 & 2, drag reduction and fuel saving slightly decreased with respect to speed.

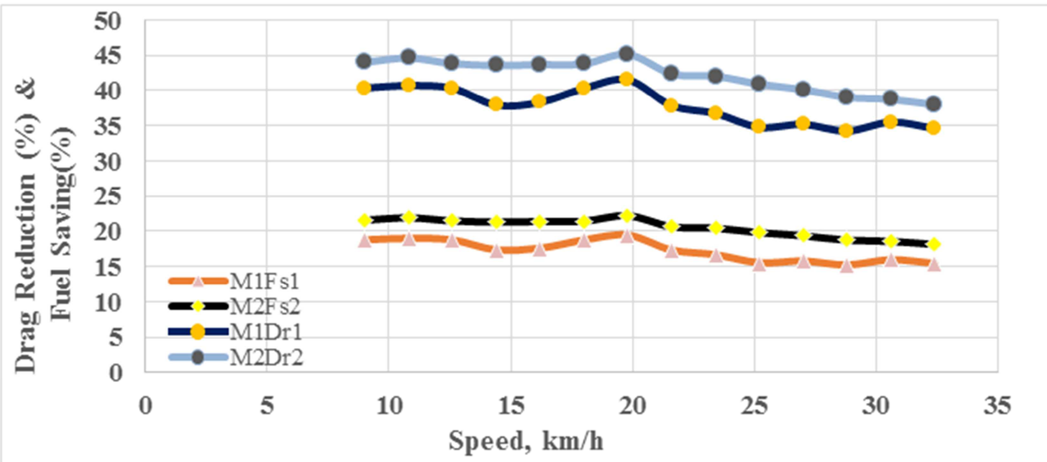

***Figure 10.** The results of drag reduction and fuel saving with velocity for model 1 & 2.*

Table 5. *The Results of Drag Reduction with Fuel Saving of Base Model 3 and Base Model Modified 1 & 2.*

Velocity, Km/h	Drag Reduction, (BM3M1)	Drag Reduction, (BM3M2)	Fuel Saving (%) (BM3M1)	Fuel Saving (%) (BM3M2)
80	35.3	27.9	11.1	15.4
90	35.1	28.2	11.3	15.4
100	37.0	28.8	11.7	16.5
110	50.2	39.9	11.5	18.7

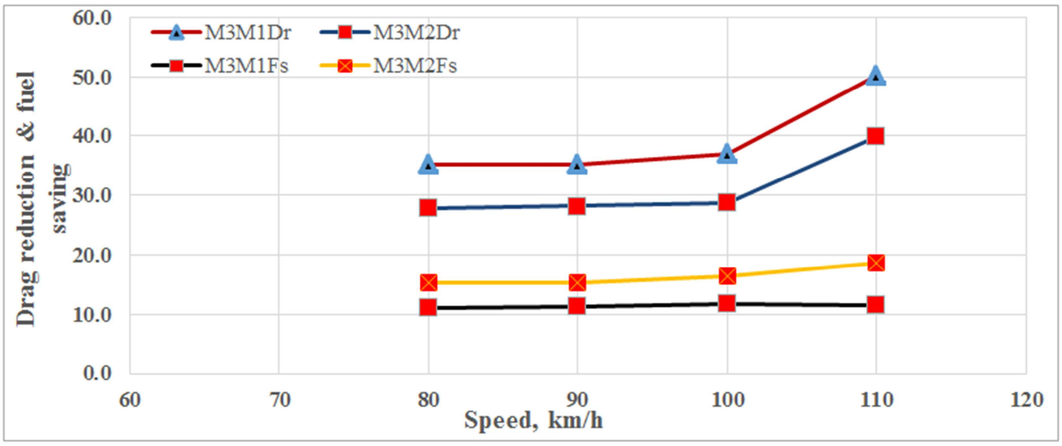

Figure 11. *The results of Drag Reduction and Fuel Saving with velocity of base model 3 and base model modified 1 & 2.*

Table 6. *Fuel saving at various Speeds for Base Model 3 Modified 1 & 2 with Sunlong China Bus1.*

Velocity (km/h)	Fuel Saving (%) Base Model Modified-1	Fuel Saving (%) Base Model Modified-2	Fuel Saving (%) Sunlong China Bus-1
80	2.44	3.38	0.654
90	2.49	3.39	1.107
100	2.57	3.63	4.659
110	2.53	4.11	5.059

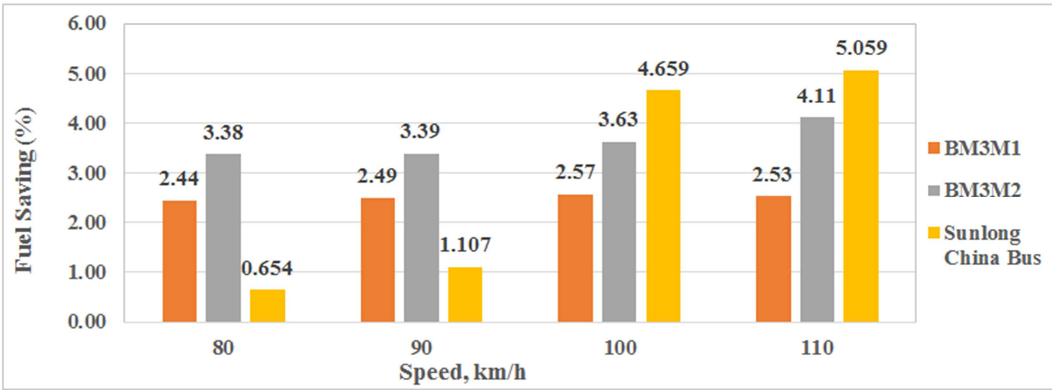

Figure 12. *Fuel Saving Versus Velocity of Base Model Modified- 1 & 2 with Sunlong China Bus.*

4.2. CO_2 Reduction for Base Model 3

The carbon dioxide (CO_2) emissions are produced by two main sources such as natural source and human source. To calculated the CO_2 emissions from fuel combustions of fossil fuels by the analytical equations. The oxidations of fossil fuels are depending on the quality of fuels and equipment where it's used. It's lowest for solid fuels and highest for natural gas. The following analytical equation is given:

CO_2 emission = AL*CL*OF*44/12, Where, AL = amount of combustion fossil fuel, CL = carbon content of fossil fuel, 0.77ton/kl, OF = oxidation factor of fossil fuel, 0.99 for gas/ diesel.

Table 7. *Annual CO_2 Reduction (ton/year) with velocity of Base Model-3 Base Model Modified- 1 & 2 with Sunlong China Bus [15].*

Velocity (Km/h)	CO_2 Reduction (ltrs/hrs)- BMM 1	CO_2 Reduction(ltrs/hrs)- BMM 2	CO_2 Reduction (ltrs/hrs)- Sunlong China Bus
80	6.820	9.447	1.828
90	6.960	9.475	2.843
100	7.183	10.146	13.022
110	7.072	11.488	14.14

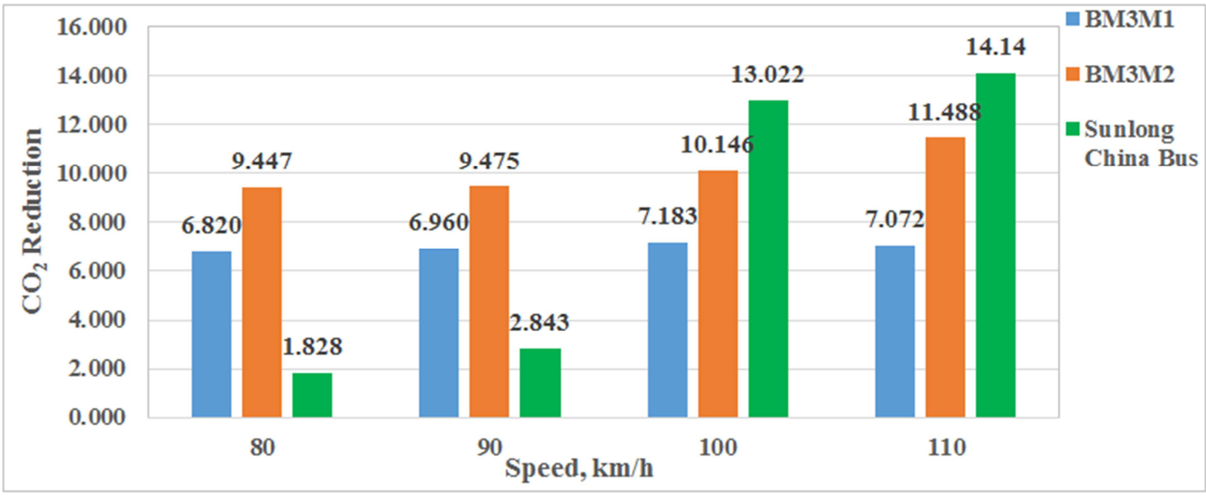

Figure 13. *CO_2 reduction for Base Model Modified 1 & 2 with Sunlong China Bus.*

The numerical for the drag forces, drag coefficient, fuel consumption and CO_2 reductions have been done for model-1, 2 and model-3 for the base models as well as for their respective modified models. The simulation results of the drag forces drag coefficient, fuel consumption and CO_2 reductions for the Base Model-3 and Base model Modified-1 and Base model Modified-2 are validated with the Sunglong China bus model in the analysis.

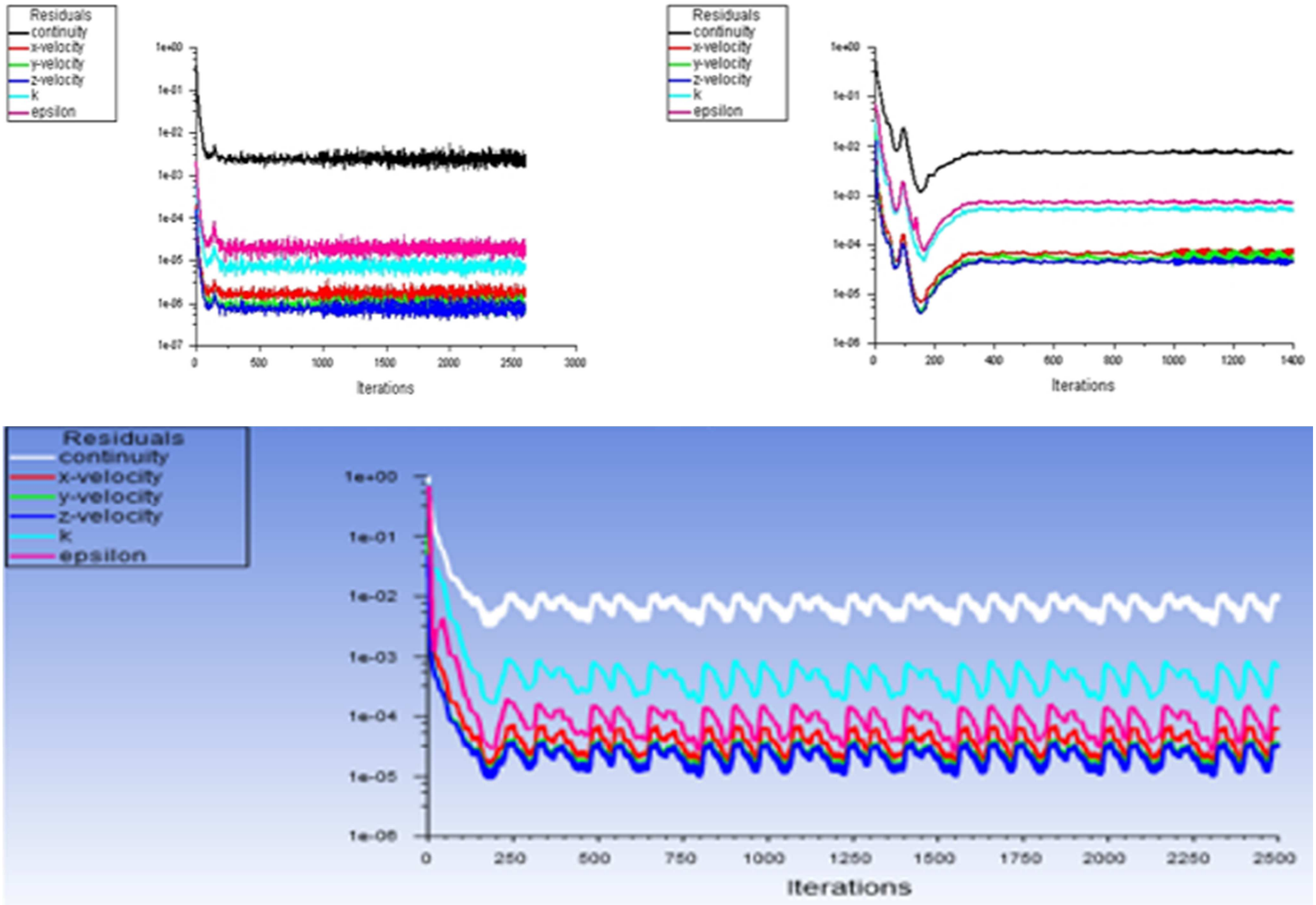

Figure 14. *Graph of the Base Model 1, 2 & 3 with Base Model Modified at 0^0 yaw angle.*

The following figure of Pressure contour, velocity streamline and velocity vector with velocity for base model and modified model. The highest density of red color indicated that highest air resist by vehicle body. Fluid streamline indicated that the direction of air movement which flow the opposite vehicle movement.

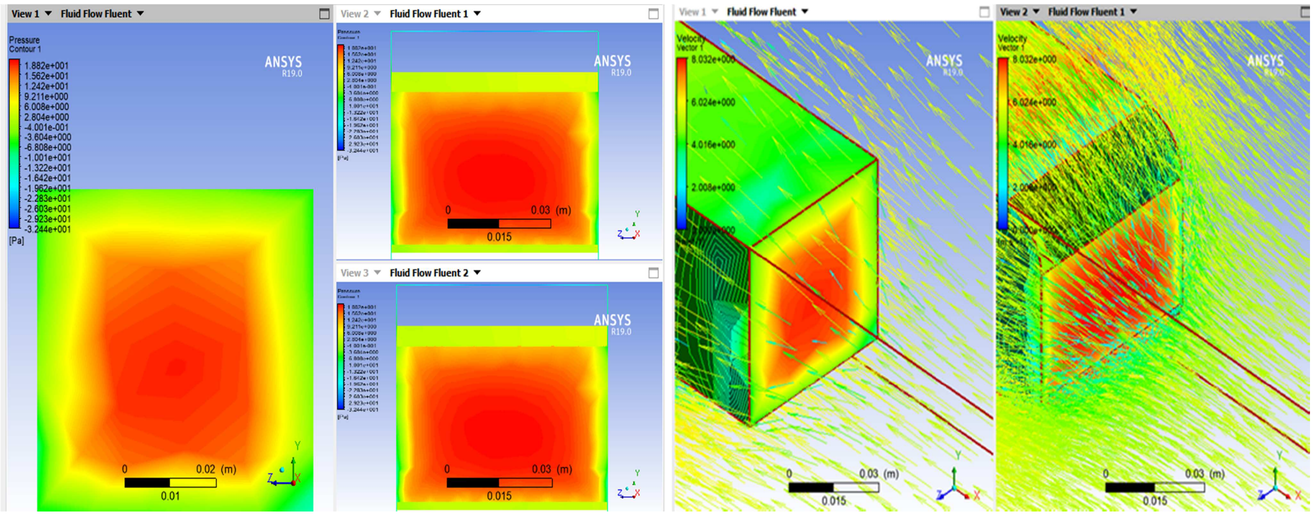

Figure 15. Pressure contour, velocity vector and velocity streamline of the Base Model 1 and Base Model Modified 1.

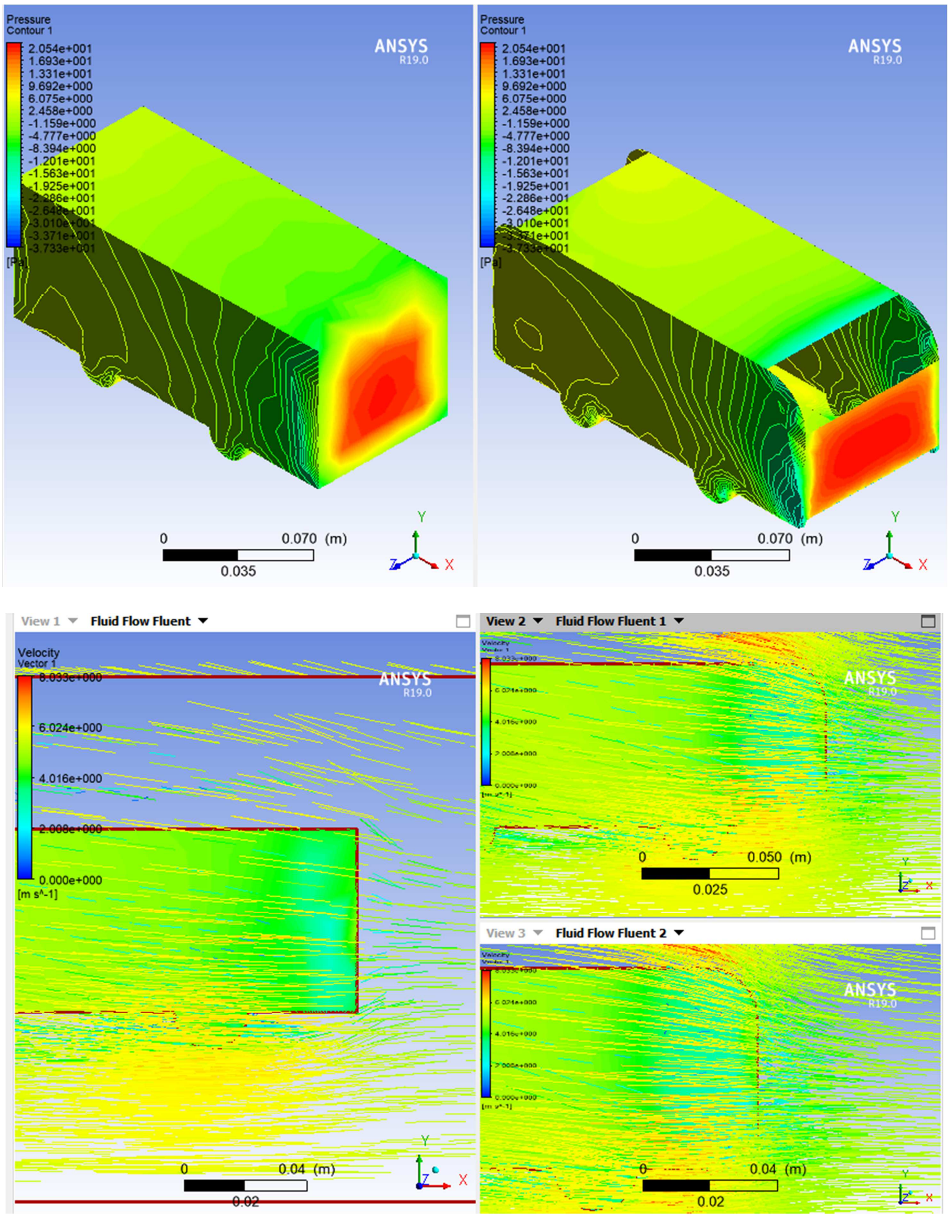

Figure 16. Pressure contour, velocity vector and velocity streamline of the Base Model 2 and Base Model Modified 1.

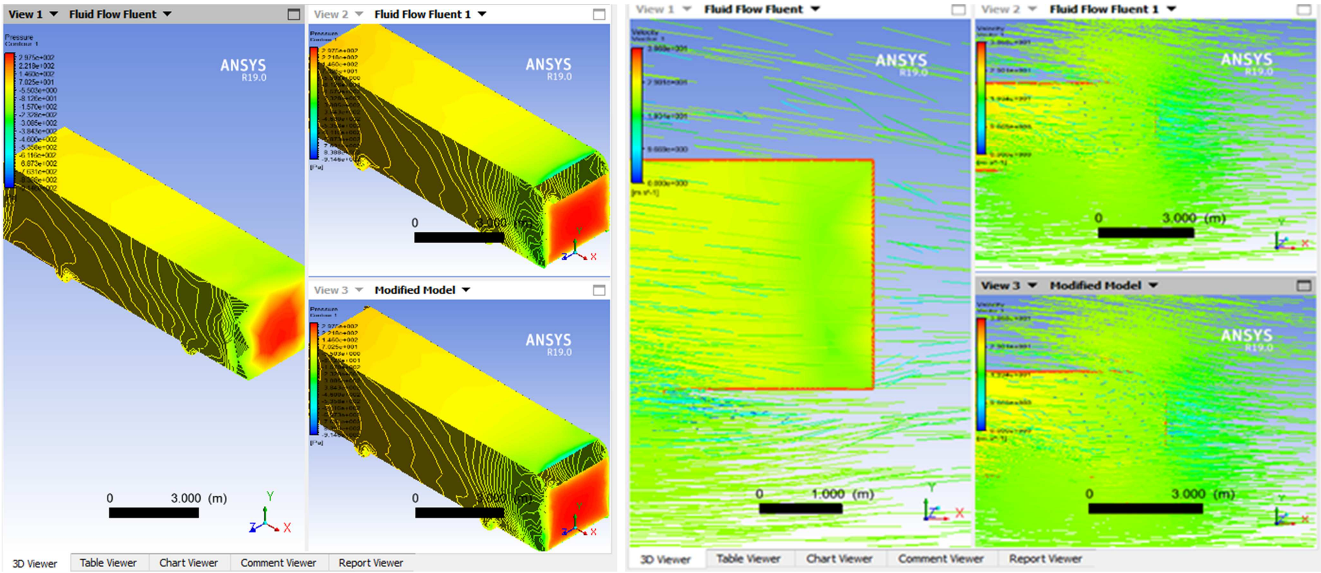

Figure 17. *Pressure contour, velocity vector and velocity streamline of the Base Model 1 and Base Model Modified 1.*

5. Conclusions

It has been seen that the aerodynamic drag on a vehicle directly affects not only is fuel consumption but also the generation of carbon dioxide (CO_2) from the combusted fuel which directly affected on the environment. Drag force has been reduced and performance of bus gets increased due to the fuel consumption decreased and hence the efficiency of the bus increases. Other findings are provided in the following way.

Model-1: The drag force at a velocity of 20 km/h for base model-1 the drag force is reduced as 0.027 N or 42.19% In addition, the fuel consumption for base model modified-1 is 19.30 liters.

Model-2: The drag force at a velocity of 20 km/h for base model-2 the drag force is reduced as 0.062 N or 49.21%. In addition, the fuel consumption for base model modified-1 is 22.20 liters.

Base Model-3: The CFD values of the drag forces at a velocity of 110 km/h, the drag forces are reduced for base model modified-1, base model modified-2 and Sunlong China Bus as 1987.20 N or 39.93%, 2499.80 N or 50.53% and 3803.8 N or 46.8%, respectively. In addition, the fuel savings for base model modified-1, base model modified-2 and Sunlong China Bus are 2.53 liters/hour or 9108.0 liters/year, 4.11 liters/hour or 14796.0 liters/year and 5.059 liters/hour or 18212.4 liters/year, respectively. The annual savings for base model modified-1, base model modified-2 and Sunlong China Bus are Tk. 592020.00, Tk. 961740.00 and Tk.1183806.00. The Bus is assumed to be running for 10 hours every day and 30 days in a month and also assumed that per liters diesel price is Tk.65.00 in Bangladesh.

CO_2 Reduction for Base Model-3: The CO_2 reductions at 110 km/h velocity for base model modified-1, base model modified-2 and Sunlong China Bus are 7.072 liters/hour, 11.488 liters/hour and 14.14 liters/hour, respectively.

References

[1] N. Govindha Rasu, A. M. Renil, S. J. Sachin, and J. kevin, CFD analysis of commercial bus models for improvement of aerodynamic performance, JCHPS, Issue 3. PP.106-110, 2016.

[2] Mr. G. Singh, Assistant Prof. Mr. Vinod Kumar, CFD Analysis for Drag Force Reduction in Buses, IJAERD, Vol. 5, Issue 01, 2018.

[3] L. E. Newland, "A Fuel Consumption Function for Bus Transit Operations and Energy Contingency Planning", Technical Report: UM-HSRI-80-53, Highway Safety Research Institute, The University of Michigan, 1980.

[4] Arun Raveendran, D. Rakesh, S. N. Sridhara. “Exterior Styling of an Intercity Transport Bus for Improved Aerodynamic Performance”, SAE Technical Paper 2009-28-0060 09/29/2015.

[5] S. Roy, and P. Srinivasan, External Flow Analysis of a Bus for Drag Reduction, International Bus and Bus Meeting & Exposition, Paper #: 2000-01-3500, 2000.

[6] Carr. G. W., The aerodynamics of basic shapes of road vehicles, part 1, Simple rectangular bodies”, MIRA report No. 1982.

[7] W H Hucho, “Aerodynamics of Road Vehicles”. Fourth ed., 1998, SAE International, ISBN 0-7680-0029-7, p. 1-918.

[8] A. A. Abdel Aziz, and A. F. Abdel Gawad, “Aerodynamic and Heat Transfer Characteristics around Vehicles with Different Front Shapes in Driving Tunnels,” Proceedings of Eighth International Congress of Fluid Dynamics & Propulsion, ICFDP 8, December 14-17, 2006.

[9] A. Muthuvel, M. K. Murthi, Sachin. N. P, Vinay. M. Koshy, S. Sakthi, E. Selvakumar, Aerodynamic Exterior Body Design of Bus, IJSER, Volume 4, Issue 7, 2013.

[10] P. Gopal, and T. Senthilkumar, "Aerodynamic Drag Reduction in a Passenger Vehicle Using Vortex Generator with Varying Yaw Angles", ARPN Journal of Engineering and Applied Sciences, ISSN 1819-6608, Vol. 7, No. 9, September 2012.

[11] Alam, H. Chowdhury, H. Moria, and S. Watkins, “Effects of Vehicle Add-ons on Aerodynamic Performance”, Proceedings of the 13th Asian Congress of Fluid Mechanics, Dhaka, Bangladesh, 17-21 December 2010.

[12] H. reza Shiri. Aerodynamic Analysis of Drag Reduction Devices on the Simplified Body for Tractor and Trailer by Using CFD, Eastern Mediterranean University, December 2016.

[13] M. Govardhana, B. V. Reddy, Estimation of Drag and Lift on Ahmed Body Using CFD Analysis, Issue 8, Vol. 4, 7 July 2017.

[14] F. Alam and S. Watkins, Implication of vehicle aerodynamics on fuel savings and the environment, International Conference on Mechanical, Industrial and Materials Engineering 2013 (ICMIME2013), Paper ID: KL-07, 2013.

[15] Optimum Aerodynamic Design of a High-speed China Sunlong Bus For fuel saved and carbon Dioxide reduction. Journal Paper for Automotive Engineering Vehicle Body Design and Aerodynamics. Department Of Mechanical And Automotive Engineering Adama Science And Technology University.2016.