Proceedings of the
International Conference on Mechanical Engineeringand Renewable Energy 2019
(ICMERE2019) 11 – 13 December, 2019, Chittagong, Bangladesh

ICMERE2019-PI-181

A COMPUTATIONAL STUDY IN FUEL ECONOMY IMPROVEMENTS FOR HIGHWAY BUS IN BANGLADESH

M. H. Pranta*, **M. S. Rabbi, and Md. Anamul Haque**

[1]Department of Mechanical Engineering, Chittagong University of Engineering and Technology, Chattogram-4349, Bangladesh
*mhp432@gmail.com, rabbi@cuet.ac.bd

Abstract*- Continuous rising price and strict regulations of fuel makes the transportation system uneconomical in Bangladesh in the recent years. This research aims to modify the shape of the highway bus frame to improve the fuel economy. Two prototypes of the modified shape named after M_1 and M_2 has been modelled considering the base model of Hino AK1J series highway bus which has been popular in Bangladesh for years. Solidworks was used to create the frame geometry and the flow analysis was done in FLUENT. It was observed that the drag coefficient has been significantly reduced for the modified models. The investigation concludes that the modified aerodynamic shape of the bus frame improved the fuel economy remarkably.*

Keywords: Aerodynamic, Hinobus AK1J, Drag coefficient, Fuel economy.

1. INTRODUCTION

In our country, bus has been considered the major transport system for years. Petroleum is the significant energy source for such highway vehicle. It has been stated that petroleum covered 66% to the total fuel consumption for highway vehicles. Continuous increasing fuel price made the petroleum operated vehicle driving uneconomical day by day. Driving with better fuel economy could overcome this problem. The exterior design of bus plays a vital role in the variation of drag force. Manufacturers in this region always follow the conventional exterior thus giving the least importance in this sector. A better aerodynamic exterior design of highway bus can reduce the drag force and consequently can improve the fuel efficiency.

A new aerodynamic exterior body has been proposed by E Selvakumar et. al. and conducted computational and experimental work [1]. The result shows that 30% to 40% drag force has been considerably reduced from the existing bus to the new concept and for every 100 km 6 to 7 liters of fuel consumption also contracted. Md. Jahangir Alamand Mohammad Mamun [2] used a rectangular box of 36 m x 4 m x 10 m as fluid domain. They placed half bus-length in front of the bus to the inlet of the wind tunnel and the outlet was 1.5 times bus-lengths behind the bus.A research [3] conducted by Siddhesh Kanekar et. al. on Aerodynamic study of state transport bus using computational fluid dynamicsthrough solvingReynolds-Averaged Naiver-Stokesequation for incompressible turbulent flow and also a model of standard k-εis used. They had taken an Ashok Leyland's "Parivartan" model for the computation of drag coefficient. J Abineshand et. al. performed[4] on two Volvo intercity buses a CFD test. The original bus model is model one and the updated version is modified one. This they did so as to reduce the streamlined drag and fuel utilization. After the CFD investigation they discovered drag decrease of about 10%.The Original model's coefficient of drag was seen as 0.8 and for the changed model was seen as 0.7.Ashok Patidar et. al. [5] conducted aerodynamic analysis on the basis of numerical study on effect of drag resistance on fuel efficiency of bus. They used ANSYS fluentand got 30 % improvement in drag coefficient.H. YesfalgnDamissie and N. Ramesh Babu [6]performed analysis on drag reduction and fuel consumption and they proved from 85 kmph to 115 kmph about 22.8% fuel consumption can be reduced. The frame of the bus varies region to region. G. Sivaraj et al. carried out [7] an investigation for understanding the impact of stream condition on streamlined powers on the vehicle to look at between the base vehicle design and connection of base bleed. They found the streamlined drag coefficient of vehicle model with the connection of base bleed is decreased from 0.3329 to 0.3123. This happened in view of the end of the wake locale at the back side of the vehicle. Additionally, streamlined drag coefficient decrease in hatchback cars was determined by experiment with a 1:12 scale-down model.L. Anantha Raman et. al. directed [8] a near investigation of various techniques for streamlined drag decrease to diminish fuel utilization in vehicles. They led passive tests on a SUV model by expanding its backside (back fairing), including a back plate (back screen) what's more, by including a vortex generator (Delta wing and bump shaped).A 6.5% and 26% decrease of drag was found by establishment of back screens and back fairing individually. Among the vortex generators the delta wing type were found increasingly compelling drag reducers. Yiping Wang et. al. [9] done numerical simulation on a nonexclusive vehicle to upgrade the streamlined drag by utilizing a dimpled non smooth surface. Dimpled surface aides in turbulent wind stream around the vehicle consequently deferring the partition point and subsequently getting a littler wake and lesser structure drag. They had utilized Kriging surrogate model to plan the dimpled non smooth surface. The product utilized for CFD was ANSYS fluent. It was discovered that decrease in drag coefficient was gotten when velocity was underneath 10 m/s and a consistent drag coefficient was found in the speed run over 20 m/s. G. Shankar et al. done [10] both numerical and experimental studies on sedan car vehicle model utilizing dynamic wind stream alteration procedure for reducing streamlined drag coefficient and lift coefficient. The investigation was performed with three delta formed vortex generators mounted at the rooftop end of vehicle model where the stream partition starts from the outside surface of the vehicle. The test results show that 4.53% and 2.55% decrease of drag coefficient and lift coefficient individually. In our country, most of the highway non-ac Hino bus assembled by Aftab automobiles and the drag force of that frame yet to be examined. The purpose of this study is to find out the passenger bus aerodynamics and hence proposed better design which able to perform with less drag force results lower energy consumption.

In this study numerical simulation of three bus models (one is base model and other two are proposed models) with different configurations are performed. Pre-processing, solving and post-processing these are the three steps for CFD analysis. Section 2 describes the design consideration of all the bus models. Finite element modeling of all models is explained in section 3. Observed result from the conducted experiment is discussed in section 4. Finally, conclusions are drawn in section 5.

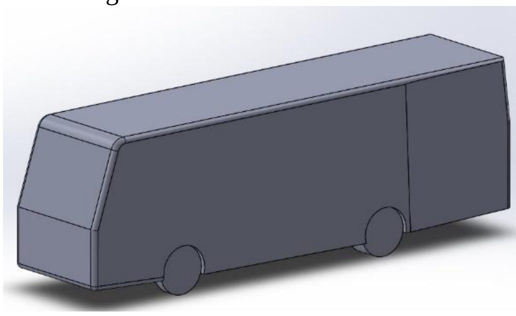
2. DESIGN

2.1 Base Model

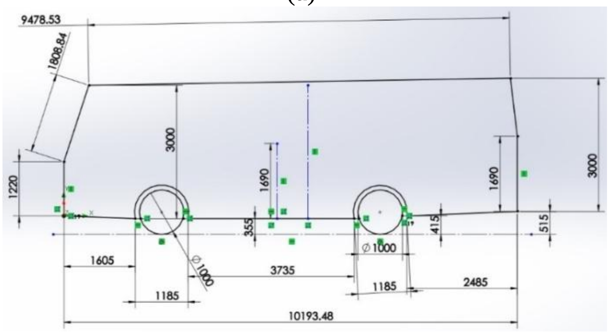
Design of the highway bus frame is the elementary step to study the aerodynamics. Figure 1 shows the typical popular bus assembled locally on Hino Ak1J chassis.

Fig. 1. A typical highway bus built on Hino AK1J chassis

Such kind of bus mainly designed for achieving a comfortable journey and about 45 persons can travel on it. Corresponding Solidworks design in a simplified way is depicted in Figure 2.



(a)

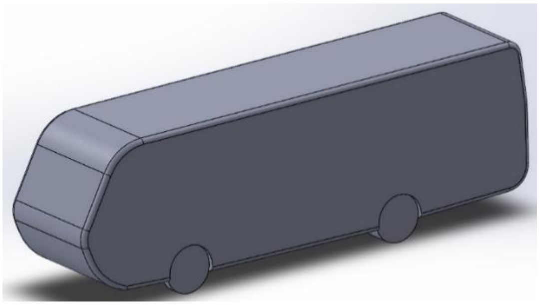


(b)

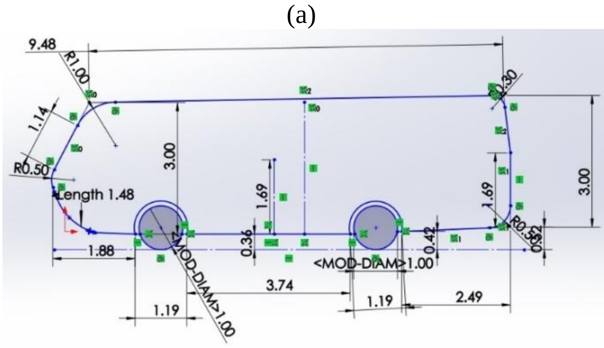
Fig. 2.(a)Isometric view &(b)Dimensions (mm) of base model

2.2 Modified Model 1 (M_1)

In this model, only angle of cowl has been changed shown in Figure3(a). The cowl and windshield are connected with each other very smoothly and cowl is kept curvy in angled manner. Roof and rear module also attached smoothly with each other. These modifications can easily be done in Bangladeshi workshops. All the dimensions are shown in Figure3(b). For this model the frontal area and rear module has been modified.



(a)

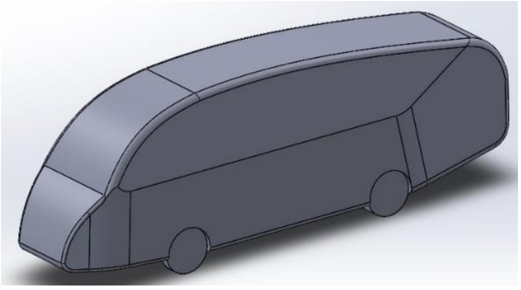


(b)

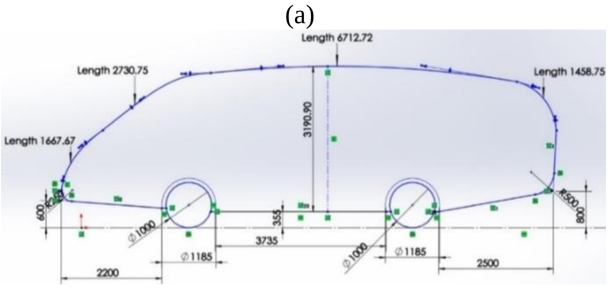
Fig. 3. (a) Isometric view, (b) Dimensions (m) of M_1

2.2 Modified Model 2 (M_2)

The outer design of this model has modified shown in Figure 4(a). While designing the aerodynamic aspects and parameters have been taken into consideration. Cowl and windshield are attached more smoothly with each other. Angle of cowl and windshield are also changed. All the dimensions are shown in Figure 4(b). For this model the frontal area, rear module, roof and side mirror shape has been modified.



(a)



(b)

Fig. 4. (a)Isometric view, (b) Dimensions (mm) of M_2

3. FINITE ELEMENT MODELLING

3.1 Meshing

A rectangular box of 36 m x 4 m x 10 m was used as fluid domain. The bus model was placed in that rectangular block. The block simulated in a wind tunnel and allowed for the outer surface of the bus to be studied. Distance from the inlet of the tunnel to the bus model centroid kept 12 m and to the outlet of the tunnel kept 24 m shown in Figure 5.By using ANSYS fluent the surface mesh was generated. The surface mesh was generated both on the surface of the domain and on the bus geometry. To discretize the computational domain the type of mesh used was tetrahedrons. For the surface bodies the mesh established had as the basic format of the triangular mesh. Mesh sizing done in the analysis is fine. The process is quickened by keeping all other options default.

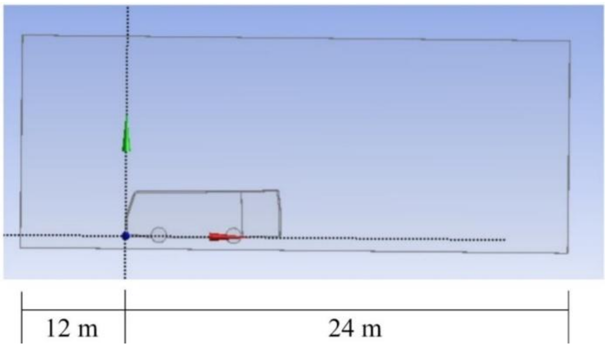


Fig. 5. Bus model placed in the rectangular fluid domain

Reynolds-Averaged Naiver-Stokes equations are analyzed for the simulation of an incompressible turbulent flow. A standard **k-ε** model is strong and convergence is easy. For this reason, it is commonly applied. In this analysis attainable k-ε model is applied with non-equilibrium wall functions. Also, it is considered that bus body is stationary. Ambient operating conditions are considered. Figure 6, 7, and 8 shows accordingly all the meshing elements and details of base model, modified model 1 and modified model 2.

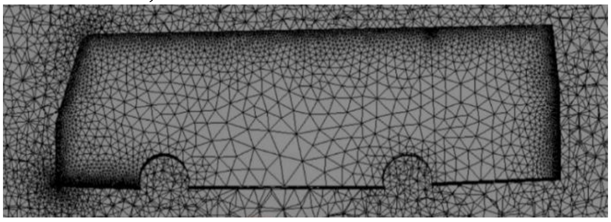


Fig. 6. Base model meshing

- 127664 nodes
- 24412 nodes
- 36870 triangular wall faces
- 1646012 triangular interior faces
- 598 triangular velocity-inlet faces
- 592 triangular pressure-outlet faces
- 10756 triangular wall faces
- 835210 tetrahedral cells

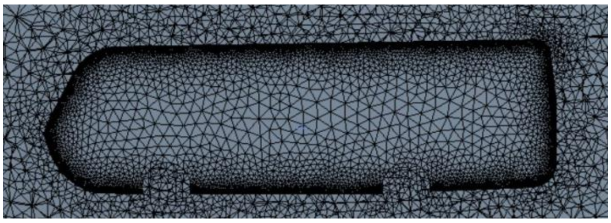


Fig. 7. Meshing of model M_1

- 154309 nodes, binary
- 29231 nodes, binary
- 45862 triangular wall faces, zone 1, binary
- 1988403 triangular interior faces, zone 2, binary
- 598 triangular velocity-inlet faces, zone 6, binary
- 574 triangular pressure-outlet faces, zone 7, binary
- 11420 triangular wall faces, zone 8, binary
- 1008815 tetrahedral cells, zone 3, binary

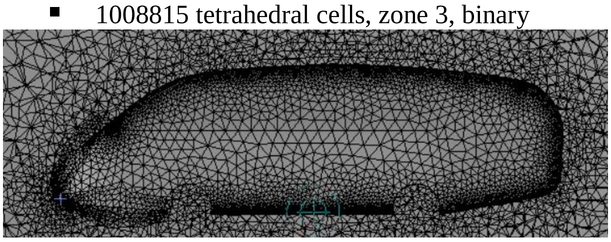


Fig. 8. Meshing of model M_2

- 113886 nodes, binary
- 22356 nodes, binary
- 32386 triangular wall faces, zone 1, binary
- 1470512 triangular interior faces, zone 2, binary
- 592 triangular velocity-inlet faces, zone 6, binary
- 586 triangular pressure-outlet faces, zone 7, binary
- 11140 triangular wall faces, zone 8, binary
- 746432 tetrahedral cells, zone 3, binary

3.2 Boundary Conditions

In this analysis it is considered that the wheel is in rotating condition and also the road is moving. Air is the fluid that was used in the analysis with 1.225 kg/m3 density. Here atmospheric pressure is considered as the operating pressure and at the outlet zero-gauge pressure is applied. Table 1 shows all the boundary conditions that have been used in this analysis.

Table 1**.** Boundary conditions for computational study

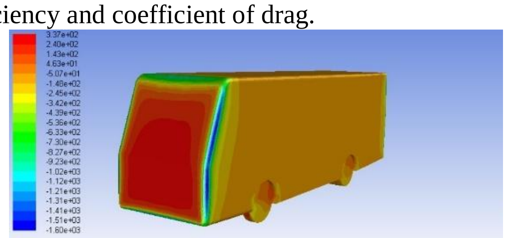
Boundary	Boundary Conditions	Values
Inlet	Velocity	V=22 m/s (79.2 km/h) (Constant)
	Turbulent Intensity	5 %
	Turbulent Viscosity Ratio	10
	Supersonic/Initial Gauge Pressure	0 pa (Constant)
Outlet	Gauge Pressure	0 pa (Constant)
	Backflow Turbulent Intensity	5 %
	Backflow Turbulent Viscosity Ratio	10
Bus body	No Slip Stationary Wall	-
Domain	Stationary Wall Specified Shear	Shear stress = 0

4. DATA ANALYSIS AND RESULT

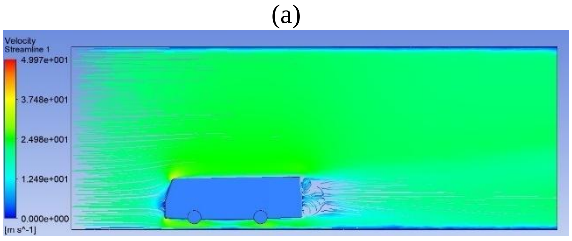
4.1 Pressure Distribution and Air-flow Path

4.1.1 Base model

The drag coefficient (C_d) is found for base model is 0.52 and drag force is 1272.73 N. Great influence of cowl and windshield can clearly be seen from the pressure contour in Figure 9(a). The air flow path lines in Figure 9(b) shows that for strong vortices low pressure zone is created behind the bus as a result of flow separation. This creates jerking situation at the rear end of the bus. After the evaluation of aerodynamic analysis of base model, it is clear that detailed modification can improve fuel efficiency and coefficient of drag.



(a)

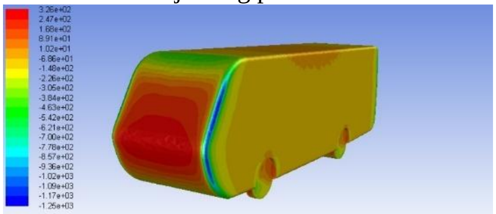


(b)

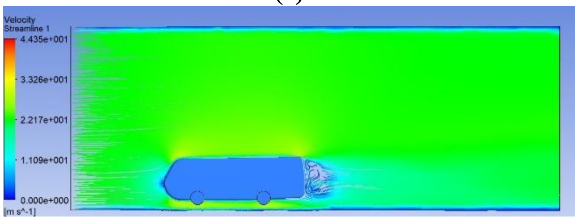
Fig. 9. (a)Pressure contour, (b)Air flow path lines of simplified base model

4.1.2 M_1

The drag coefficient (C_d) is found for this model is 0.402 and drag force is 984.75 N. Here amount of drag force is clearly smaller than the base model due to smoothing the windshield and cowl joint and also making the cowl curvy. From the pressure contour in Figure 10(a) it is seen that static pressure is less in this modified model with the comparison to the base model. Due to smoothing the roof with rear module the air flow path lines in Figure 10(b) are also smoother than the base model. So somewhat jerking problem will be reduced.



(a)

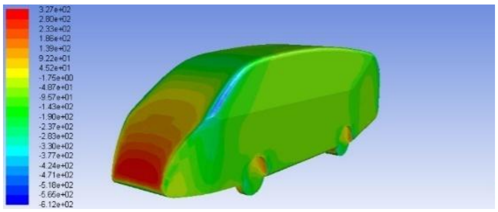


(b)

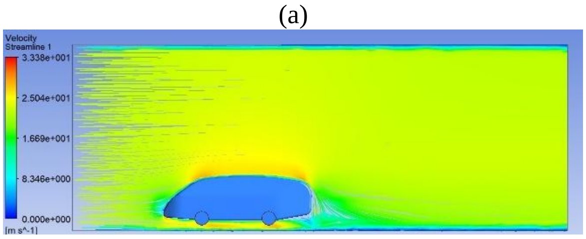
Fig. 10. (a)Pressure contour,(b)Air flow path lines of simplified M_1

4.1.3 M_2

The drag coefficient (C_d) is found for this model is 0.44 and drag force is 1088.22 N. Figure 11(a) shows the pressure contour and static pressure is very less in this modified model with the comparison to the base model and as well as to the M_2. The air flow path lines in Figure 11(b) are also smooth and less vorticity produces than the two models analyzed earlier. So jerking problem will also less than the two others.



(a)



(b)

Fig. 11. (a)Pressure contour, (b)Air flow path lines of simplified M_ 2

4.2 Fuel Consumption

Total drag force and fuel consumption [7] calculated according to the following equation:

$$F_D = (1/2) \times \rho \times A \times C_d \times v^2 \qquad (1)$$

Where,ρ = Density of air,

v = Speed of the bus,

C_d = Drag co-efficient, and

A= Frontal area.

$$Percentage\ fuel\ reduction = 3/5(percentage\ total\ drag\ reduction) \qquad (2)$$

Front area and total drag force data found in this study are listed in Table 2.

Table 2. Frontal area and total drag force

Parameters	Base model	Modified Model 1	Modified Model 2
Frontal area, A	8.2762 m^2	8.265 m^2	8.351 m^2
Total drag force, F_D	1272.73 N	984.75 N	1088.22 N

Figure 12 shows the variation in drag force acting on these three models at different frontal areas.From the analysis it can be said that drag force has been decreased and the total drag force is reduction from 1272.73 N to 984.75 N and 1088.22 N at 79.2 kmph for M_1 and M_2 respectively. This is an improvement of 22.63% and 14.50% respectively as compared to base model.

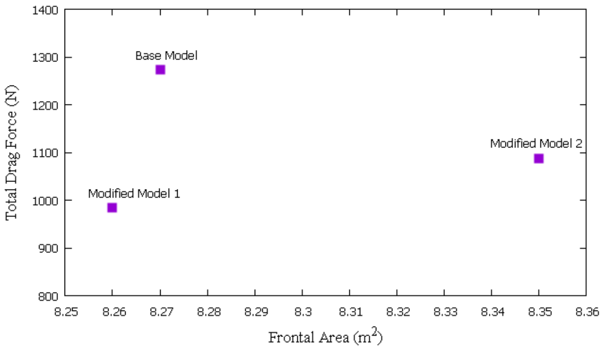


Fig. 12. Variation of drag force with frontal area

At the identical speed (79.2 kmph), by measuring drag coefficient(C_d), percentage reduction of $C_d$has been calculated. Percentage of fuel reduction is found by multiplied the percentage total drag reduction with 0.6 [7] and listed in Table 3.

Table 3. Data of drag coefficient

Model	Drag co-efficient, C_d	%C_d reduction	Fuel saving (%)
Base model	0.520	-	-
M_1	0.402	22.69%	13.61%
M_2	0.440	9.23%	9.23%

It is observed that maximum of 13.61% fuel withM_1 and 9.23% fuel can be saved with M_2.

5. CONCLUSION

The main purpose of this study is to reduce the drag force focusing improvement the fuel economy. Several numerical tests have been done on three models to observe the variation in drag force. Less drag force means less energy consumed at high speed which is good for our environment and economy as well. Changing the model of front and rear module of the M_1 bus doesn't cost any passenger but for M_2 bus may cost few passengers but it should not yield any profit or loss in the long run. These variations in design reduce the drag coefficient for M_ 1 and M_ 2 are on an average of 22.69% and 15.38%. As a consequence, fuel consumption will be reduced. In conclusion, it can be said that this reduction in fuel consumption by the proposed design of highway bus would put an impact in national economy.

6. REFERENCES

[1] E.Selvakumar, M. K.Murthi, and A.Muthuvel, "Conducted Research on Aerodynamic Exterior Body Design of Bus",Reproduced with permission of the copyright owner. Further reproduction prohibited without permission,2013.

[2] M. J.Alam, and M. Mamun, "CFD Analysis of Newly Designed Aerodynamic Shape for a Highway Passenger Bus to Reduce Drag Force", International Conference on Mechanical Engineering, 2015.

[3] SiddheshKanekar, P.Thakre, and E. Rajkumar, "Aerodynamic study of state transport bus using computational fluid dynamics",Materials Science and Engineering Conference Series. Vol. 263, No. 6, 2017.

[4] J.Abinesh, and J.Arunkumar, "CFD Analysis of Aerodynamic Drag Reduction and Improve Fuel Economy", International Journal of Mechanical Engineering and Robotics Research, Vol. 3, No. 4, 2014.

[5] A.Patidar, U. Gupta, and Bansal, "Fuel Efficiency Improvement of Commercial Vehiclebyinvestigating Drag Resistance", SAE Technical Paper, 2015.

[6]H. Yesfal, and N. R.Babu, "Aerodynamic Drag Reduction on Locally Built Bus Body using Computational Fluid Dynamics (CFD)",International Journal of Engineering Research & Technology (IJERT), Vol. 6, No. 11,2017.

[7] G. Sivaraj, K. M. Parammasivam, and G. Suganya, "Reduction of Aerodynamic Drag Force for Reducing Fuel Consumptionin Road Vehicle using Basebleed," vol. 11, no. 6, pp. 1489–1495, 2018.

[8] L. Anantha Raman and Rahul Hari H., "Methods for reducing aerodynamic drag in vehicles and thus acquiring fuel economy," Journal of advanced engineering research, ISSN: 2393-8447, Vol. 3, Issue 1, 2016, pp 26 32, 2016.

[9] Yiping Wang, Cheng Wu, Gangfen Tan and Yadong Deng, "Reduction in the aerodynamic drag around a generic vehicle by using a non-smooth surface," Journal of Automobile Engineering. Vol: 231(1), 130-144, 2016.

[10]G. Shankar and G. Devaradjane, "Experimental and Computational Analysis on Aerodynamic Behavior of a Car Modelwith Vortex Generators at Different Yaw Angles," vol. 11, no. 1, pp. 285–295, 2018.