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

ICMERE2019-PI-173

INFLUENCE OF THE ROOF OF CAR TO REDUCE FUEL CONSUMPTION

Pranto Karua[1], Dr.Md. Shahidul Islam[1]

[1]Department of Mechanical Engineering, Khulna University of Engineering & Technology,Khulna-9203, Bangladesh
karuapranto@gmail.com[1] shahidulbitk@gmail.com[2]

Abstract-*Fuel consumption characteristics plays an important role in the choice of an automobile. Every manufacturer wants to design a car with the lowest fuel consumption. Aerodynamic forces like drag, lift, weight, side forces affect the fuel consumption characteristics. The reduction of drag is the main concern of this analysis to improve the fuel consumption and thus* CO_2 *emission can be reduced. The shape of car is one of the significant factor in designing a vehicle as the aerodynamic properties of a car depend mainly on its shape. Aerodynamic drag varies according to the shape variation which can be estimated by performing experiment on wind tunnel using model. As the experimental procedure is time consuming, the designers have to go for the computational analysis to calculate the drag. Besides, it is difficult to implement a physical prototype through changing the design parameters each time in the experimental method. Automobiles with their different geometric dimensions provide different flow characteristics over the roof. Thus the resultant net aerodynamic force such as drag and lift varies. In this reason, the variation of drag with the change in roof geometry has been prioritized in the work. A total of three model have been prepared and the simulation is carried out on the models. The result of simulation is obtained in terms of coefficient of drag. And the result is investigated in accordance with the optimization. In addition to this, the analysis gives a vast information about the flow characteristics over car and possible aerodynamic shape optimization using ANSYS Fluent 16.2.*

Keywords: Drag, Fuel Consumption, Optimization, Aerodynamic Shape.

1. INTRODUCTION

The improvement of fuel efficiency is a major issue in automotive industry. Manufacturers are introducing more fuel efficient cars to the market day by day with the increasing demand. The aerodynamic properties of a vehicle significantly affect the performance and fuel consumption characteristics of the vehicle. A large amount of air is displaced and flowed around the vehicle when an automobile is in motion. Aerodynamic drag is one of the main impediment to accelerate a solid body when it moves in the air. About 50 to 60% of total fuel energy is lost only to overcome this unfavorable aerodynamic force [1].

The drag coefficient estimates the resistance of an object in a fluid environment. It is not an absolute constant, rather it varies with speed and direction of flow, object shape and size [2]. The lower is the drag coefficient, the less aerodynamic drag occurs on an object. Thus the performance of vehicles depends upon the reduction of drag as it ensures the efficient fuel consumption. And the geometrical changes are done in order to improve road stability of the vehicle and to reduce the drag through directing the air flow in different ways. Thus, vehicle designers are interested in designing cars with increased fuel economy through optimizing the shape of vehicle.

Roof of an automobile has a significant contribution in the design of an automobile. Automobiles with their different geometric dimensions provide different flow characteristics over the roof. Thus the resultant net aerodynamic force such as drag and lift varies. So, the optimization of car roof has greater influence on the fuel consumption characteristics.

Moreover, fuel consumption is directly related to the CO_2 emission from the car. The temperature of the world is increasing day by day. The rapid increase of CO_2 is responsible for this. About 20% of the total CO_2 emissions is accounted from the transportation sector [3]. The reduction of CO_2 is the most significant issue to resist global warming. To reduce CO_2 emissions from cars, the improvement of fuel economy is the main criteria. This paper will show an efficient modification of vehicle by optimizing its roof to improve fuel economy and thus to reduce CO_2 emission from cars. Besides, the utilization of public transport is increasing rapidly and people are thinking about reducing the cost of fuel. In this reason, vehicle manufacturers are concerned in designing the vehicle to reduce the drag through optimizing different parameters [4]. In this work, computational analysis have been carried out on the car model and modification of the models provide an indication of the fuel economy improvement.

2. METHOD

A total of three models have been analyzed. The method is started with a single initial model. Then the model is imported in the Design Modellerof ANSYS 16.2. After generating the mesh, it is solved in Fluent defining the boundary conditions. Then the model is optimized and the same procedure is repeated.

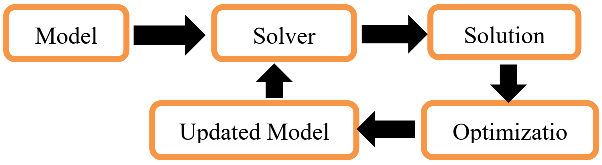

Fig 2.1: Schematic diagram of general method for shape optimization

The models of my investigation are shown below:

Fig 2.2: Model 0Fig 2.3: Model 1Fig 2.4: Model 2

A length of 1170mm and width of 400mm is used while designing the geometry. The length and width of the cars are kept same for all the models. But the height of the roof from the underbody has been changed. Model 0 is taken as the base model of the analysis having trapezoid roof. Modification has been made in the rest of models. Model 1 is subjected to a curvature roof, whereas Model 2 is subjected to a flatter roof.
A fluid volume has been created to simulate the airflow around the vehicle. For this purpose, an enclosure has been generated that surrounds the vehicle and Boolean operation has been implemented to subtract the vehicle body in the Design Modeller module of ANSYS. This enclosure acts as the air domain. The faces of the enclosure has been named as inlet, outlet, topwall, bottomwall, sidewall and symmetry. The front of the carbody is kept 2000mm from the inlet of the enclosure. And the distance from the rear end to the outlet is 6500mm because of the probability of formation of backflow at the rearend of the car. The height of topwall from the car roof has been kept 5000mm because of rapid pressure variation on the roof. The distance from the car bottom to bottomwall is 350mm as it moves on the road. To reduce the computational cost, a symmetry plane is introduced at the half of the model.

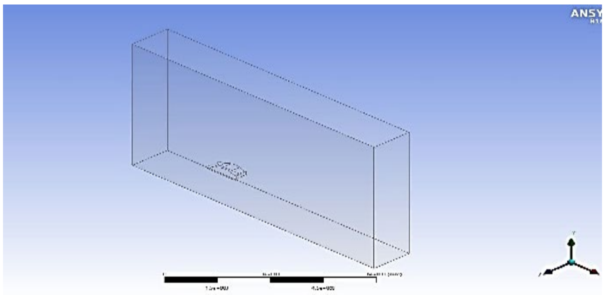

Fig 2.5: Computational domain in Design Modeler of Fluent Solver

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:

$$\frac{\partial}{\partial t}(\rho k)+\frac{\partial}{\partial t}(\rho k u_i)=\frac{\partial p}{\partial x_j}\left[\left(\mu\frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right]+G_k+G_b-\rho_\varepsilon-Y_M-S_k$$

(5)

For dissipation, ε:

$$\frac{\partial}{\partial t}(\rho\varepsilon)+\frac{\partial}{\partial t}(\rho\varepsilon u_i)=$$

$$\frac{\partial p}{\partial x_j}\left[\left(\mu\frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right]+C_{1e}\frac{\varepsilon}{k}(G_k+C_{3e}G_b)+C_{2e}\rho\frac{\varepsilon^2}{k}+S_\varepsilon$$

(6)

In these equations, the generation of turbulence kinetic energy due to the mean velocity gradients is represented by G_k and the generation of turbulence kinetic energy due to buoyancy is represented by G_b.

k - εturbulence 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 280mm per element.A face size of 5mm is used on the carbody to refine the mesh. And five layers of inflation has been added around the carbody to cope with the separation of boundary layer. The total number of nodes and elements obtained are shown in Table 4.1.

Table 4.1: Total number of nodes and elements in each model

Model Name	Nodes	Elements
Car Model 0	583327	2132493
Car Model 1	560161	2059217
Car Model 2	575260	2106127

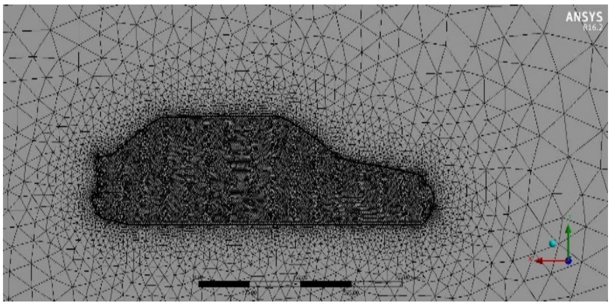

Fig 4.2: Final mesh of Car Model 0

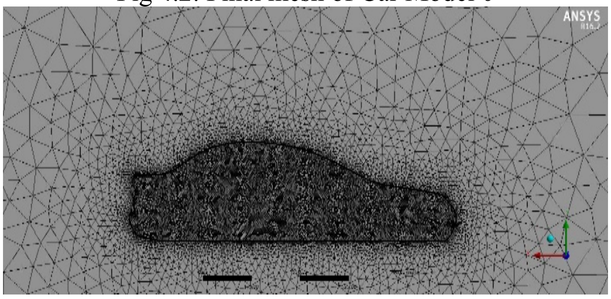

Fig 4.3: Final mesh of Car Model 1

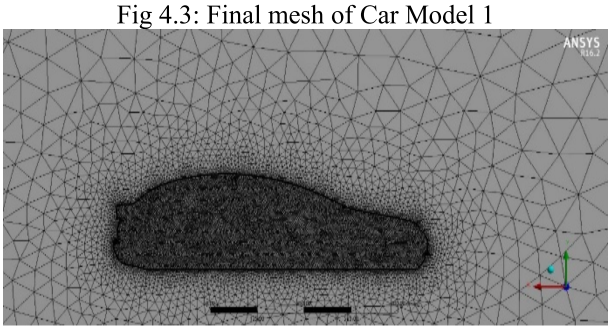

Fig 4.4: Final mesh of Car Model 2

The final mesh of the car models has been shown in the above figures. Air coming through the inlet has been given a velocity of 20 m/s (72 km/hr).

4.1 Accuracy of Analysis:
This thesis work is verified by the journal named "Investigation of Drag and Lift Forces over the Profile of Car with Rearspoiler Using CFD" published in International Journal of Science and Research (IJSR) Volume 4, Issue 9, September, 2015.

4.2 Verification of Car Model 0:

Table 4.2.1: Verification of Present Work to Paper Work

Velocity (km/hr)	Paper work, C_D	Present work, C_D	Percentage of error, %
72	0.329	0.3295	0.16
90	0.330	0.3298	0.06
108	0.331	0.3301	0.27
126	0.333	0.3319	0.33
144	0.334	0.3321	0.57

4.3 Plots of Verification:

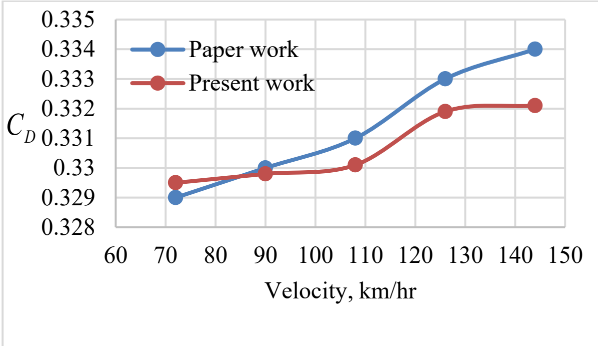

Fig 4.3.1: Verification of Present work to Paper work

4.4 Mesh Sensitivity:

4.4.1 Car Model 0:

Table 4.4.1: Mesh Sensitivity Test of Car Model 0

Number of Mesh	Coefficient of Drag, C_D
2132493	0.3295
1980691	0.3301
1785036	0.3289
1597066	0.3306

4.4.2 Car Model 1:

Table 4.4.2: Mesh Sensitivity Test of Car Model 1

Number of Mesh	Coefficient of Drag, C_D
2246639	0.3037
2059217	0.3061
1954124	0.3013
1744091	0.3003

4.4.3 Car Model 2:

Table 4.4.3: Mesh Sensitivity Test of Car Model 2

Number of Mesh	Coefficient of Drag, C_D
2106127	0.2927
1932912	0.2933
1734122	0.2935
1594862	0.2947

5. POST PROCESSING

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

5.1 Static Pressure Contours of the Models:

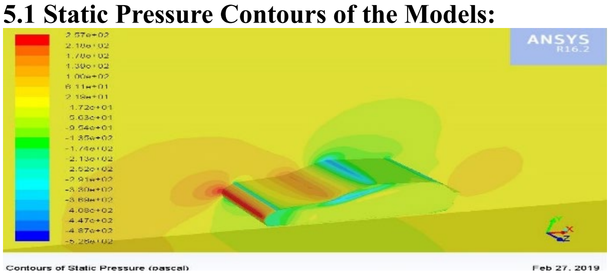

Fig 5.1.1: Static Pressure contour of Model 0

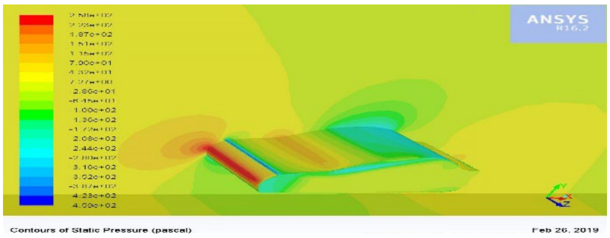

Fig 5.1.2: Static Pressure contour of Model 1

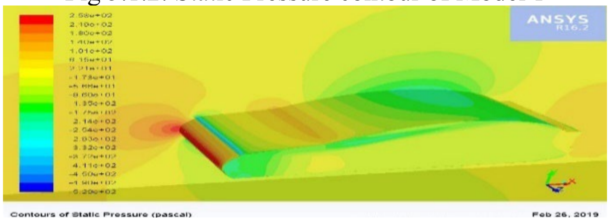

Fig 5.1.3: Static Pressure contour of Model 2

The above contours show the static pressure variation along the model. Pressure is maximum at the point where the flow strikes the car. The pressure along the whole roof is almost constant except at the contact points of the windshield and car roof.

5.2 Velocity Streamlines through the Models:

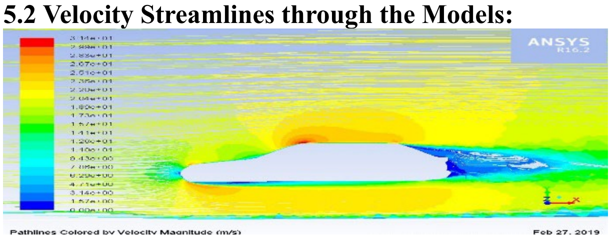

Fig 5.2.1: Velocity Streamlines of Car Model 0

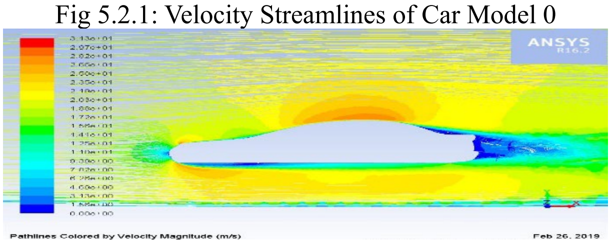

Fig 5.2.2: Velocity Streamlines of Car Model 1

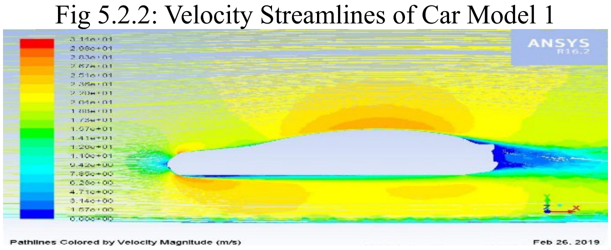

Figure 5.2.3: Velocity Streamlines of Car Model 2

The above contour shows the velocity streamlines of air. The streamlines are deflected significantly when the flow is passing over the windshield and car roof. It becomes almost straight at the rear portion.

5.3 Results:

Table 5.3.1: 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
Car Model 0	0.32953503	-0.21881488	435
Car Model 1	0.30918703	0.05635878	453
Car Model 2	0.29271003	0.16588451	501

6. PLOTS

6.1 Velocity vs Coefficient of Drag Plot for Car Model 0:

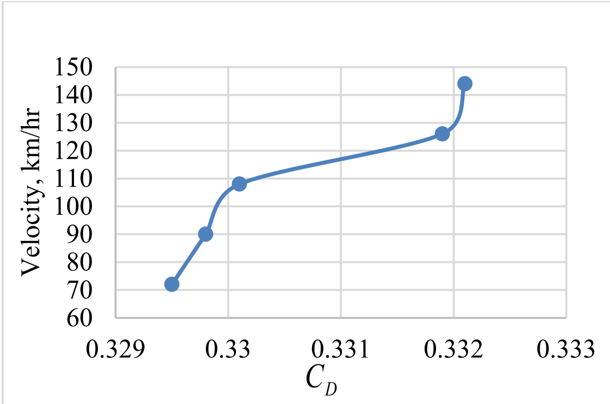

Fig 6.1: Velocity vs Coefficient of Drag Plot for Car Model 0

The above plot shows the variation of coefficient of drag with increasing the velocity. Drag increases proportionally with the velocity for a car model having trapezoid roof.

6.2 Velocity vs Coefficient of Drag Plot for Car Model 1:

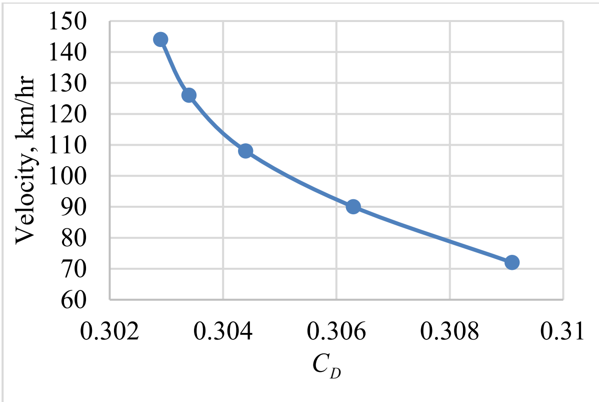

Fig 6.2: Velocity vs Coefficient of Drag Plot for Car Model 1

The above plot shows the variation of coefficient of drag with increasing the velocity. The plot indicates the relationship between drag and flow velocity. Drag changes inversely with the velocity for a car model having curvature roof.

6.3 Velocity vs Coefficient of Drag Plot for Car Model 2:

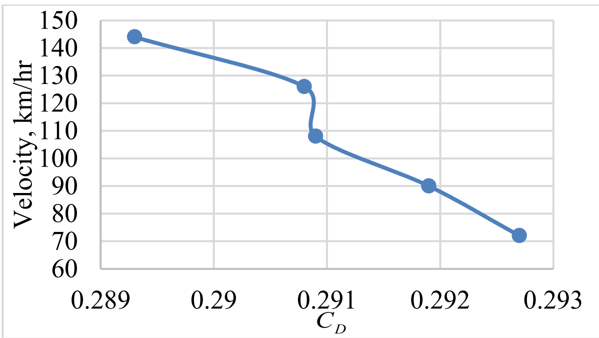

Fig 6.3: Velocity vs Coefficient of Drag Plot for Car Model 2

The above plot shows the variation of coefficient of drag with increasing the velocity. The plot indicates the relationship between drag and flow velocity. Drag decreases rapidly with the velocity for a car model having more flatter roof as compared to Car Model 1.

6.4Coefficient of Drag vs Velocity Plot for All Models:

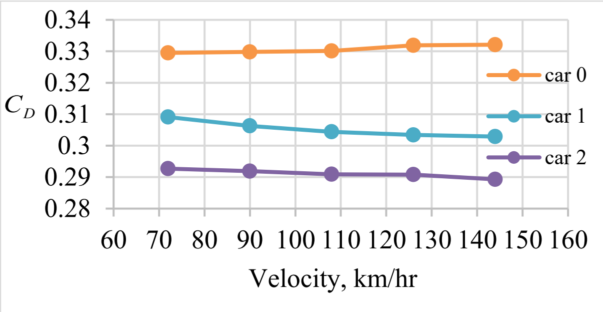

Fig 6.4: Coefficient of Drag vs Velocity Plot for All Models

The difference in coefficient of drag between all the models subjected to same velocity has been shown in the plot. The modified Car Model 1 and 2 provide a significant decrease in coefficient of drag as compared to Car Model 0.

6.5 Pressure vs Velocity Plot for All Models:

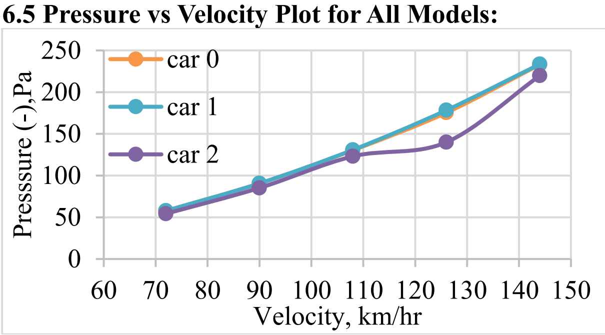

Fig 6.5: Pressure vs Velocity Plot for All Models

The difference in pressure between all the models subjected to same velocity has been shown in the plot. The change in pressure is quite similar in all the models. The lowest pressure occurs in the model having flatter roof.

6.6 Turbulence Kinetic Energy vs Velocity Plot for All Models:

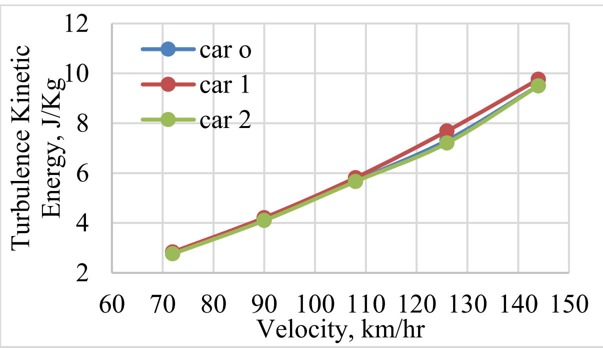

Fig 6.6: Turbulence Kinetic Energy vs Velocity Plot for All Models

The difference in turbulence kinetic energy between all the models subjected to same velocity has been shown in the plot. The change in kinetic energy is quite similar in all the models.

6.7 Relation of Drag with the Fuel Consumption:

Aerodynamic drag is proportional to the velocity square and is therefore proportional to the velocity cube to the energy required to overcome drag. This implies that a very powerful connection exists between the velocity travelled by a car and the percentage of the fuel used to overcome drag. Aerodynamics is liable for a much greater percentage of the fuel used on the highway for passenger vehicles. This implies that 10% decrease in aerodynamic drag will enhance the fuel economy by around 5% [6].

7. CONCLUSION AND RECOMMENDATIONS

7.1 Conclusion:

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 0.3295, which was verified with the paper. After that, two modifications had been made in the geometry, firstly the value of drag coefficient was 0.3091 and the further modification in geometry gave the value of drag coefficient 0.2927.

Thus, the optimization of the shape of roof can be concluded as the fulfillment of the work.

7.2 Recommendations:

a) In this present work, the roof geometry has been modified and all other parameters has been kept same. In future, other parameters that affect the vehicle aerodynamics can be taken into account to perform the analysis.

b) The work can also be modified by comparing the accuracy of the simulation with the experimental wind tunnel testing.

c) Idea of the analysis can be implemented to optimize the shape of truck to improve its fuel economy.

8. REFERENCES

[1]https://ec.europa.eu/clima/policies/transport/vehicle/car (Reducing CO_2 emissions)

[2]https://www.comsol.com/blogs/studying-the-airflow-over-a-car-using-an-ahmed-body (Airflow over an Ahmed Body)

[3] K. Takama *"CO_2 Reduction from Vehicles"* Journal of Society of Automotive Engineers of Japan, March, 2004.

[4] Abdul Razzaque Ansari *"CFD Analysis of Aerodynamic Design of Tata Indica Car"*, International Journal of Mechanical Engineering and Technology (IJMET) Volume 8, Issue 3, pp. 344–355, March, 2017.

[5] ANSYS Fluent, Release 16.2: Help Topics.

[6]http://www.arcindy.com/effect-of-aerodynamic-drag-on-fuel-economy.html (Drag and fuel economy)

9. NOMENCLATURE

Symbol	Meaning	Unit
T	Temperature	(K)
P	Pressure	(Pa)
A	State transition matrix	Dimensionless
C_D	Co-efficient of Drag	Dimensionless
C_L	Co-efficient of Lift	Dimensionless
D	Drag	N
A	Frontal Area of Car	m^2
k	Turbulence Kinetic Energy	J/Kg
ε	Turbulence Dissipation	m^2/s^3
σ_k	Turbulence Prandtl Number for k	Dimensionless
σ_ε	Turbulence Prandtl Number for ε	Dimensionless
γ	Ratio of Specific Heat of Gas	Dimensionless