Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2015
(ICMERE2015) 26 – 29 November, 2015, Chittagong, Bangladesh

ICMERE2015-PI-282

AERODYNAMIC ANALYSIS OF A CONCEPT CAR MODEL

S. Pal[1], S. M. H. Kabir[2] and M. M. M. Talukder[3,*]

[1-3]Department of Mechanical Engineering, Chittagong University of Engineering & Technology, Chittagong-4349, Bangladesh
[1]shubhankar09me@gmail.com, [2]dalimuou@yahoo.com, [3,*]mehdi.cuet@gmail.com

Abstract-*Aerodynamic drag is the force opposite to the direction of motion that acts on a body moving through air [1]. Car is a high speed vehicle and when it runs on the road it faces huge amount of drag force opposite to its moving direction. The main purpose of this paper is to propose a concept car model which will face very small drag force and also to reduce the corresponding drag coefficient. The purpose is served by analyzing different shapes of the car and incorporating necessary changes to it to achieve the desired outcome. Both computational and experimental approaches are applied to identify the areas where we can modify to produce a new aerodynamic shape which will face a very small drag force as well as provide small drag coefficient. Solidworks flow simulation is used to perform the computational analysis which is later verified by a subsonic open circuit suction type wind tunnel testing of the concept car model. A good agreement between the computational and experimental results suggest the reliability of this investigation and scope for further analysis of the car model. However, the variation in computational and experimental drag force suggests a higher degree of dependency on the grid quality and elements selection.*

Keywords: Aerodynamic Drag, Coefficient of Drag, CFD, Concept Car, Solidworks Flow Simulation.

1. INTRODUCTION

When a car runs on the highway, it loses 45% of its total energy to overcome the drag forces that oppose the car's motion .In the mid of nineteenth century the car were boxy and has large frontal areas because of limited applications of aerodynamic analysis. The growth of computing technology in the 1970's and 80's allowed car manufacturers to analyze air flow around cars and recognize the importance of aerodynamics in vehicle development [2]. This progress allowed further research to be focused on reducing the drag coefficient. It has been shown that 40% of the drag coefficient is dependent on the external shape of the car and is concentrated at the rear end of the geometry [3].The goal of this investigation is to modify the aerodynamic shape of the car by using effective aerodynamic analysis both computational and experimental.

2. THEORY

(a) Drag Force: The force a flowing fluid exerts on a body in the flow direction is called drag. The drag force is due to the combined effects of pressure and wall shear forces in the flow direction[4].Drag force due to the combine effects of the wall shear stress and pressure forces is define by Eq. (1)

$$F_d = C_d \, {}^1\!/_2 \, \rho V^2 A \qquad (1)$$

Where, C_d is the drag coefficient, ρ is the density of air, V is the velocity, and A is the cross-sectional area. Aerodynamic drag consists of two main components: skin friction drag and pressure drag. Pressure drag accounts for more than 80% of the total drag and it is highly dependent on vehicle geometry due to boundary layer separation from rear window surface and formation of wake region behind the vehicle [5].

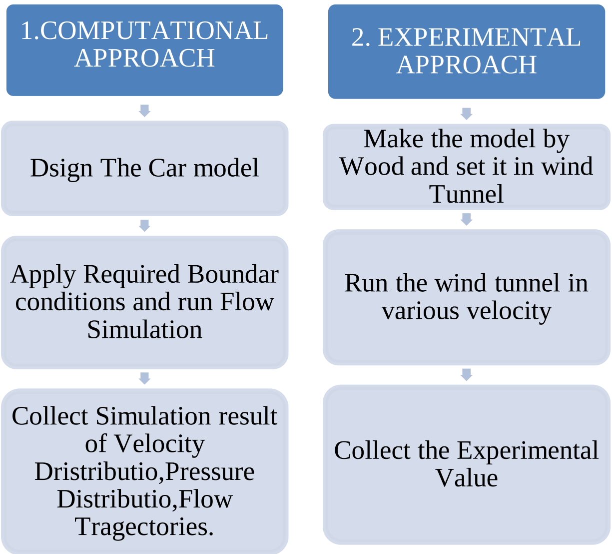
(b) Stream Line: A stream line is an imaginary line drawn in a flow field such that a tangent drawn at any point on this line represents the direction of the velocity vector [6].

(c) Vorticity: Vorticity is the measurement of rotation of a fluid particle. Specifically Vorticity is equal to twice the angular velocity of the fluid particle [6]. For the racing car and other high speed aerodynamic vehicles, vorticity has larger effect. Vorticity is created by leakage of flow from high-pressure side to low-pressure side.

(d) Computational Fluid Dynamics: Computational fluid dynamics (CFD) is one of the branches of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. CFD provides numerical solutions to the governing equations of fluid dynamics throughout the desired flow region. It allows for complex problems to be solved without losing the integrity of the problem due to over-simplification [7].

3. METHODOLOGY

Both computational and experimental approaches are used to analyze the concept car model. The methodology of the analysis is shown below in a block diagram form.



4. COMPUTATIONAL APPROACH

In computational approach, the car models are designed by the Solidworks 2013 and Key shot 4 is used for the rendering purpose to make the model realistic. Then Solidworks Flow Simulation is applied for simulation purpose. Different result viewing options related to Flow Simulation is applied to analyze the models and also to find the best one among them. The result viewing options are

- Cut plot (Shows pressure distribution, velocity distribution etc.)
- Surface plot (Shows pressure distribution.)
- Flow Trajectories

In the figure of the velocity distribution, Pressure distribution and Flow trajectories, the mesh and isolinesare concealed to visualize the result bar and value change region clearly.

4.1Design of Simple Car Model

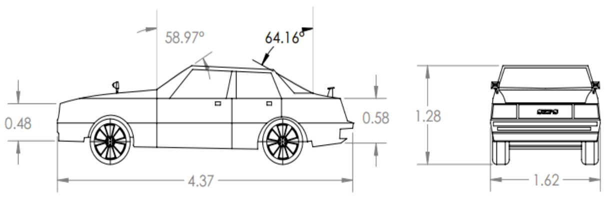


Fig.1: Schematic representation of the simple car (Linear distances are in meter and angles are in degree)

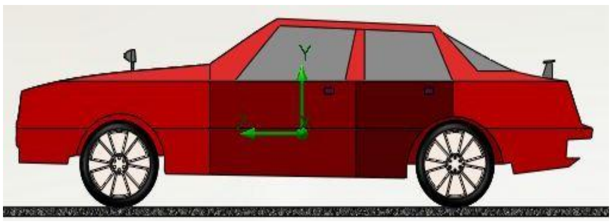


Fig. 2: Three dimensional solid model of simple car

4.1.1 Analysis of the Simple Car

4.1.1(a)Velocity Distribution

Inside the cut plot contour, mesh and isolines are used to visualize the velocity distribution. To visualize the result bar only the contour is used.

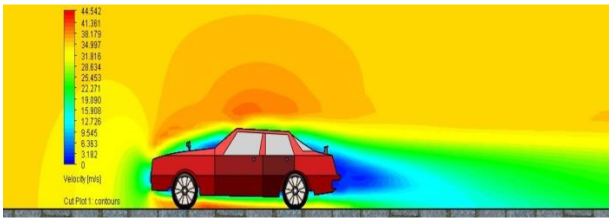


Fig. 3: Velocity distribution of Simple car model.

Figure 3, shows the velocity distribution over the car body. The red region shows high velocity region and blue region shows low velocity and deep blue region has nearly zero velocity. The Figure 3 demonstrates that the velocity of the air reduces as it reaches the front of the car. This velocity then increases due to a positive slope above the front end of the car. The airflow velocity over the roof reduces significantly and remains constant as it flows towards the rear of the car [8].The car frontal area has low velocity distribution containing velocity 15.908m/s and the back position is nearly zero because it is opposite to the direction of the air flow. In the frontal area there are some regions where the airflow is resisted. For this reason the velocity of the car reduces.

4.1.1(b) Pressure Distribution

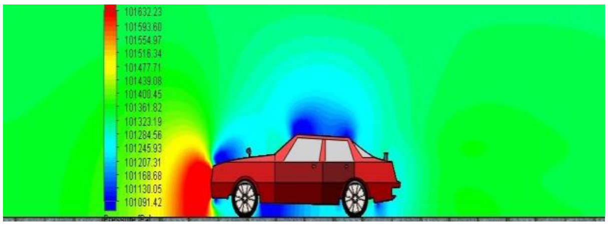


Fig.4: Pressure distribution of the simple car model

Pressure distribution is also shown by cut plot using contour with mesh. In Figure 4, the light blue region indicates atmospheric pressure and red region is the high pressure region. It is visible that the car frontal area has a larger pressure effect than other regions, so by improving in the design of the car frontal area the pressure effect over the car can be reduced. The targeted areas are too much vertically inclined, for this the air flow strikes the targeted areas which create pressure on car. When the car is in running condition, it has to overcome that extra pressure effect. For this extra pressure a portion of velocity spends to overcome the pressure. For this reason the velocity of the car reduces.

4.1.1(c)Flow Trajectories

Figure 5, shows velocity flow over the car body in different ways. In the flow trajectory figure it is observed that the stream line flows are disorganized for the shape of the car which should have to be smooth to maintain continuous uniform flow. If the stream lines pass perfectly without obstacles, the velocity of the car will increase.

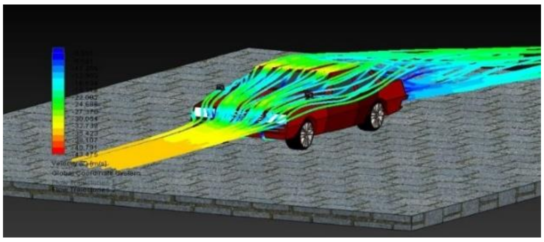


Fig.5: Flow Trajectories of the simple car.

So the regions where the stream lines get obstacles, is being modified. Specifically, the sharp edges of the car is modified to control the smooth stream line flow.

4.2Targeted Areas for Modification

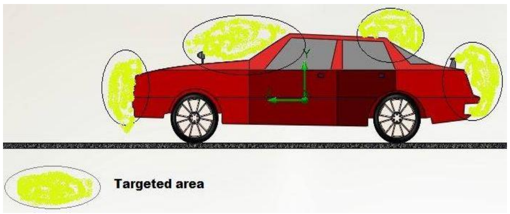


Fig.6: Targeted Areas for Modification

From the analysis of design of the simple car, particular region are detected where the flow of air gets the maximum resistance. For this reason those areas are picked out as the targeted areas for modification.

4.3 Design of Modified Car Model

In case of simple car and modified car, there is minor change in dimension but we changed the appearance both in front and back side between them. Such as in modified car we took change in the front glass and in the edges of the rear end geometry.

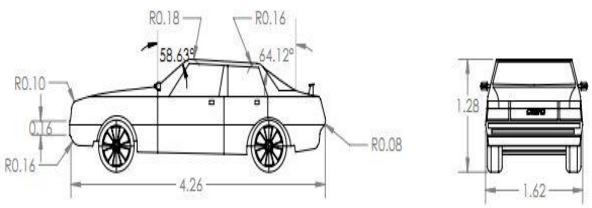


Fig.7: Schematic representation of the modified car (Linear distances are in meter and anglesare in degree)

Fig.8: Three dimensional solid model of modified car.

In the modified version of the simple car model, the sharp edge of the car is modified into fillet with concern in style and fashion to reduce the resistance of air. The targeted regions of the front is transformed by making smooth inclination so that the velocity can flow smoothly over the car. A remarkable modification is also accomplished in the sight glasses in the front of the car.

4.3.1 Analysis of The Modified Car Model

4.3.1(a) Velocity Distribution

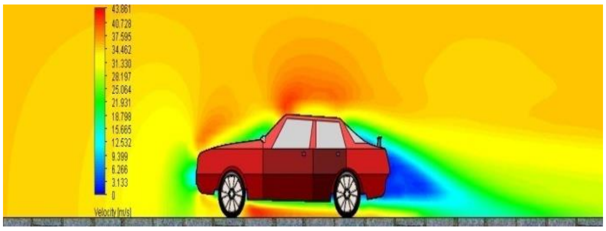


Fig. 9: Velocity distribution over modified car model

Figure 9, shows the velocity distribution over a car model which have been modified by using smooth areodynamic property of car.Here, in the front of the car in the light blue region the velocity is 18.798m/s and in the back side of the deep blue region the velocity is 3.133m/s which is better than the simple car model. It represents that using of these aerodynamic edges and under low resistance of velocity treatment reduce the velocity drop over the simple car model.

4.3.1(b)Pressure Distribution

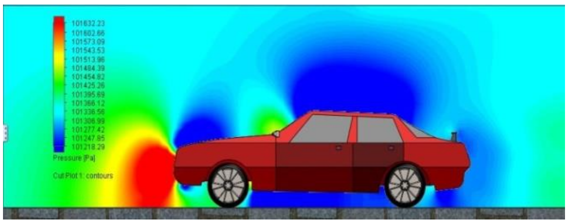


Fig.10: Pressure distribution over modified car model

Figure 10, shows the pressure distribution over a car model which have been modified by using smooth aerodynamic property over the car and the smooth curved shape at back side of the car. It represents that using of these aerodynamic edges and under modified shape treatment reduces the pressure effect over the modified car model.In the frontal red region of the car,the pressure is 101632.23N and in the back the light blue region,the pressure is 101277.42N.The Flow trajectory figure represent the pressure effect over the surface of the body clearly.

4.3.1(c) Flow Trajectories Showing Velocity of Flow

Flow trajectories shows how stream lines are flowing over the modified car model. It is observed that the air in the path of its flow gets less resistance than the simple car model. The velocity of flow over the car body showing yellow region is 35m/s. In the hood of the car showing slightly red region where the velocity raises and it is 40.72m/s. In the back side showing light green region represent velocity 21.93m/s.

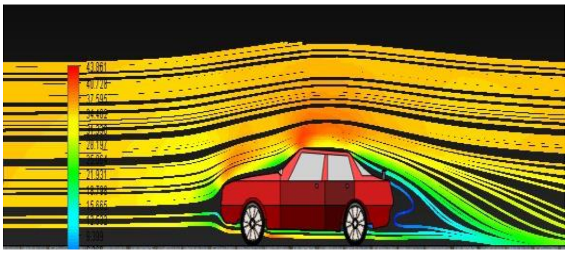


Fig.11: Flow trajectories showing velocity of flow over modified car model.

4.4 Design of Concept Car Model

In case of concept car we brought significant changes as comparative to previous two models.

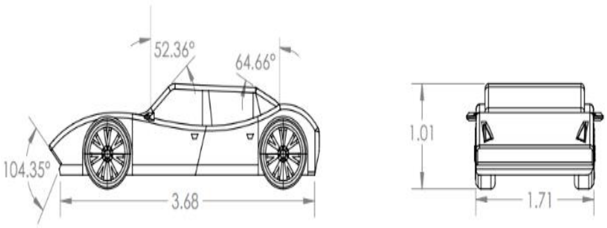


Fig.12: Schematic representation of concept car (Linear distances are in meter and angles are in degree)

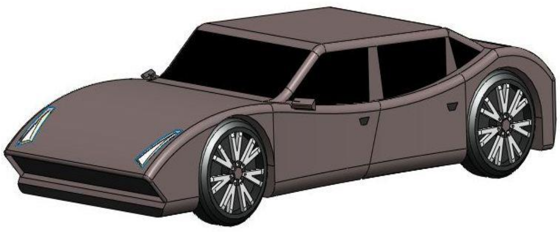


Fig.13: Three dimensional solid model of concept car

In the concept car model further adjustment in the frontal shape of the car is accomplished by incorporating aerodynamic shapes. This geometric features give minimum drag than the previous two models.

4.4.1 Analysis of The Concept Car Model

4.4.1(a)Velocity Distribution

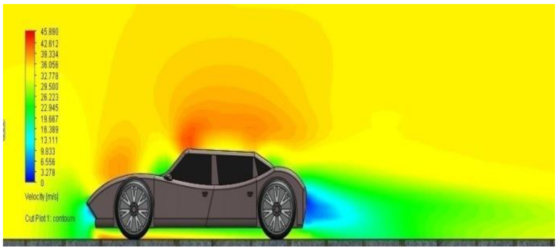


Fig.14: Velocity distribution over concept car model.

In the above figure it is observed that the velocity drop is very little as comperative to both the simple model and modified model.In the frontal light green region the velocity is 22.945m/s and in the rear deep blue region the velocity is6.556m/s.For this reson the speed of the concept car will be more than the previoustwomodels.

4.4.1(b)Pressure Distribution

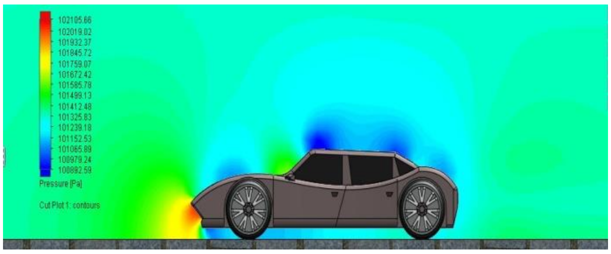


Fig.15: Pressure distribution over concept car model.

The above figure shows the pressure distribution over a concept car model.Here high pressur region is less than the previous two models and also a smooth distribution of pressure occurs .For this reason the resistances of air is very little.In front of the car under the front glass there is a minor pressure drop for the shape of the car. However it is very negligible and considered to make the car more stylish.

4.4.1(c) Flow Trajectories Showing Velocity of Flow

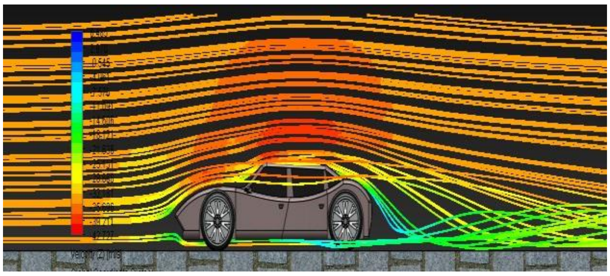


Fig.16: Flow trajectories showing velocity of flow over concept car model

In the flow trajectory analysis, it is found that the flow can pass over the car body almost smoothly occurring a trifling disturbance. For the trivial disturbance in flow velocity, the velocity drop is comparatively less than the previous two models. For this reason the velocity of the concept car will be high.

5. EXPERIMENTAL APPROACH

In the experimental approach, first a wooden model is prepared maintaining the dimensional constraints and the shape of the concept car. Twenty holes are pierced in the wooden car model to create pressure taps. Pressure data is collected by inserting plastic tubes in the holes and using multi tube manometer. The experimental investigations were performed on an open circuit suction type subsonic wind tunnel having a 0.355 m x 0.330 m x 0.380 m test section.

5.1 Wooden Model of Concept Car

In the wooden model there are 20 holes incorporating 20pressuretubes. The Length, width and height are 0.197m, 0.086mand 0.06mrespectively and the frontal area is $0.086 \times 0.06 = 0.0052\ m^2$

Fig.17: Wooden model of the concept car with pressure tapings.

5.2 Specification of Conventional Wind Tunnel

Table 1: Wind tunnel specifications

Type	**Open type wind tunnel (S HUNT)**
Test section	380 × 355 × 330mm
Blower	Compatible capacity
Motor Type	DC Motor in 220v,10A,2800rpm
Speed controller:	For variable seed
Air velocity	Maximum 5.3 m/s (in test section)
Number of Tube in Manometer	20
Fluid inside the Manometer	Kerosene.

Among the 20 pressure tubes, one remains open to the air which gives atmospheric pressure. The pressure head is obtained from the difference between actual head to the atmospheric head.

Fig.18: Wooden model inside conventional wind tunnel

5.3 Governing Equations

Specific gravity of air=0.0013
Specific gravity of kerosene=0.787
H_k=Pressure Head of Kerosene (m)
H_a= Pressure Head of Air (m)
Equations used to calculate the drag force:
Pressure head of Air,

$$Ha = \left(H_k \times \left(\frac{Sp.of kerosene}{Sp.of air} - 1\right)\right) / (1000 \times 1.5) \quad (m) \qquad (2)$$

$$\text{Pressure, } P = H_a \times \text{Density of Air} \times g \quad (N/m^2) \qquad (3)$$

$$\text{Drag Force, } F = P \times A \quad (N) \qquad (4)$$

$$\text{Coefficient of Drag, } C_d = \frac{2 \times F}{Density \times Frontal area \times velocity^2} \qquad (5)$$

6. RESULTS AND DISCUSSIONS

6.1 Computational Data

Table 2: Drag force with respect to velocity

Velocity (m/s)	Drag Force(N)		
	Simple Car	**Modified Car**	**Concept Car**
30	467.644	385.746	376.8
35	635.992	525.356	509.21
40	830.487	685.982	664.133
45	1051.416	869.255	841.042
50	1298.187	1074.355	1039.22

6.2 Curve of Computational Drag Force with Respect to Velocity

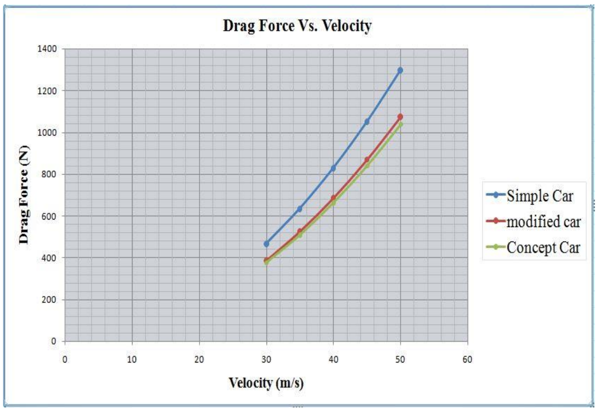


Fig.19: Variation of drag force with air velocity

By flow simulation the drag force for five distinct velocities over the simple, modified and concept car model is measured. The distinct five velocities are 30m/s, 35m/s,40m/s, 45m/s, 50m/s. It is observed that with the increase of velocity drag force also increases. It proves the convergence of the investigation. Among the three models the concept car faces low drag force than the others. For this reason the concept car is selected for the experimental investigation.

6.3 Computational Drag Co-Efficient of Three Models

From the above figure it can be observed that the coefficient of drag is 0.611 for simple car, 0.5048 for modified car and 0.438 for concept car model.

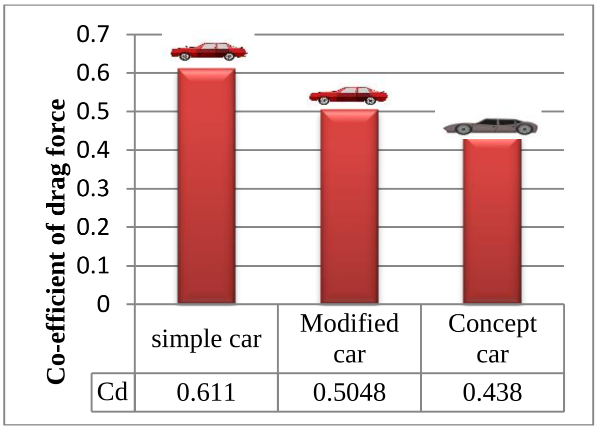


Fig.20: Computational of drag coefficients

Here the coefficient of the concept car model is the lowest between the three models. As the coefficient of concept car model is the lowest, it faces minimum resistance of air. For this we suggest concept car as the more efficient and high speed car.

6.4 Experimental and Computational Drag Force of Concept Car at Wind Tunnel Velocity

Table 3: Velocity vs. Experimental Drag Force

Velocity (m/s)	**Drag Force(N)**	
	Experimental	Computational
3.3	0.086	0.012
4.3	0.129	0.020
5.3	0.151	0.030

6.6 Experimental and Computational Drag Force Curve of Concept Car

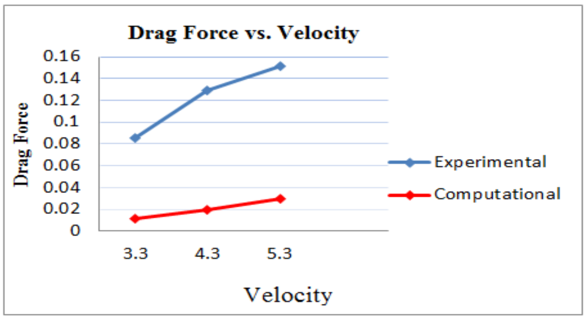


Fig.21: Experimental and computational drag force of concept car at different wind velocities.

In figure 21, a comparative study between the experimental and computational drag force of concept car is accomplished by plotting the experimental and computational drag force at different wind tunnel velocities. From the trend of the two curves it is evident that in both experimental and computational approach, the drag force increases with the increase of the velocity of flow which substantiated the validity of this investigation. However, there is a difference between the experimental and computational value of the drag force at a certain wind velocity which suggests a higher degree of dependency on the grid quality and elements selection. Since in Solidworks Flow Simulation module there are very limited options to control mesh size, mesh number and grid size, the built in mesh, number of mesh and grid sizes are used. If the mesh size and mesh number can be controlled more precisely it is possible to get more accurate results that will commensurate closely with the experimental values. However, the wind tunnel used in this investigation also needs some modification and calibration for acquisition of more reliable test data.

7. CONCLUSION

Flow simulation over simple and modified car model facilitated the finding out of the target regions where the aerodynamic modifications are applied to develop the final concept car model. Several result viewing options are applied like cut plots, surface plot, goal plot, equation goal plots etc.to visualize the effect of air flow over the car models. Velocity distribution, pressure distribution and flow trajectories of three models are studied comprehensively. Velocity distribution of the three models confirmed that the velocity drop of the concept car model was negligible with respect to simple and modified car model. The high pressure region is also minutest than the simple and modified car model. In the analysis of flow trajectories, the stream lines pass over the body very smoothly and get the least resistance from the concept car. Finally drag force is calculated from the computational values with respect to different air velocities for all the car models. From the plot of the drag force with respect to velocity it is evident that for all the models as the velocity increases drag force also increases which is convergent with the previous prediction and also substantiated the applicability of the investigation. Among the three models the drag force and drag co efficient of the concept car is minimum and that's why the experimental validation of this model is accomplished. To do this a geometrically similar reduced scale wooden model of the concept car is tested in a subsonic suction type wind tunnel. There is a good agreement between the trend of the experimental and computational result which suggests the reliability of this investigation and scope for further application. However the difference in experimental and simulation drag force at a particular wind velocity appears to suggest a higher degree of dependency on the details included in the geometric modeling, grid quality and elements selection. Also the experimental setups may need some rigorous modification to get precise data. Although there are some variation, both the experimental and computational analysis falling well within the allowable range, endorsing acceptance and promising a fuel efficient performance of the proposed profile of the concept car.

8. ACKOWLEDGEMENT

This work was partially funded by the ChittagongUniversity of Engineering & Technology (CUET) as anundergraduate project.

9. REFERENCES

[1] Toukir Islam, S.M. Rakibul Hassan and Dr.M.Ali,"Aerodynamic Drag of Racing Cars", Proceedings of the Global Engineering, Science and Technology Conference, Dhaka, Bangladesh, 28-29 December 2012.

[2] Wolf-Heinrich Honcho, "Aerodynamics of Road Vehicles". Cambridge:Butterworth's, 1987, ch.1-4.

[3] W. Kieffer, S. Moujaes, N. Armbya "CFD study of sectioncharacteristics of Formula Mazda race car wings" March 2005

[4] Yunus A. Cengel, John M. Cimbala, Fluid Mechanics Fundamentals and Applications; Second Edition, Tata McGraw Hill Education Private Limited, Six Reprint 2012.

[5] Vishal Shukla, Gaurav Saxena,"Computational Drag Analysis of Passenger Car Using Splines and Spoiler"International Journal of Engineering Trends and Technology (IJETT) – Volume 21 Number 1 – March 2015.

[6] Dr. A.K. Jain, Fluid Mechanics Including Hydraulic Mechanics, Khanna Publishers, Tenth Edition 2009.

[7] SatyanChandra, Allison Lee, Steven Gorrel and C. Greg Jensen,"CFD Analysis of PACE Formula-1 Car", Project and Thesis, Brigham Young University.

[8] Chainani. A, Perera. N,"CFD Investigation of Airflow on a Model Radio Control Race Car", Proceedings of the World Congress on Engineering 2008,Vol II, WCE 2008, July 2 - 4, 2008, London, U.K.

10. NOMENCLATURE

Symbol	Meaning	Unit
F_d	Drag Force	(N)
C_d	Drag Coefficient	[-]
ρ	Density of Air	kg/m^3
V	Air Velocity	m/s
A	Car Frontal Area	m^2
H_a	Pressure Head (Air)	m
H_k	Pressure Head (Kerosene)	m
g	Gravitational Acceleration	m/s^2
P	Pressure	N/m^2