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

ICMERE2015-PI-261

CFD ANALYSIS OF SPOILER'S CREATED DRAG AND DOWNFORCE WITHCHANGING ANGLE OF ATTACK

Rubel Chandra Das[1,*], Md. Rubaiat Islam[2], Shahedul Islam[3] and Mohammad Mashud[4]

[1 – 4]Department of Mechanical Engineering,
Khulna University of Engineering & Technology, Khulna-9203, Bangladesh
[1,*]rubeldas1105062@gmail.com,[2]rubaiat1116024@gmail.com, [3]shahed522630@gmail.com,
[4]mdmashud@me.kuet.ac.bd

Abstract- *In the fastest world, automotive vehicles have become so much faster experiencing uplift force which creates unexpected accidents. Spoilers have a great effect in case of improving traction with ground of such moving vehicles. Providing a spoiler at the rear end portion of the vehicle forms the air slice longer, gentler slope from the roof to the spoiler, which is helpful for reducing the flow separation. Reduced flow separation has effect on drag & lift forces of automotive vehicles.With the rapid developments of digital computers, CFD is used as an important tool for modern fluid dynamics research. This paper focuses on the relationship between* C_D *,*C_L*& angle of attack(α) based on the result found using AUTODESK SIMULATION CFD 2015. For the simulation work, vehicle generic model and required wind tunnel are designed in SOLIDWORKS 2015 and relevant boundary conditions are applied in AUTODESK SIMULATION CFD 2015. And models have been simulated for the evaluation of* C_D*&* C_L *with the change of α. The simulated results were analyzed with graphical representation. This helps for making proper decision of which angle of attack is preferable for maintaining optimized criteria for automotive vehicles with spoiler.*

Keywords: Drag, Down-force, Spoiler, Angle of attack, Autodesk Simulation CFD

1. INTRODUCTION

At present, automotive vehicles runs so much fast which creates unexpected accidents for such excessive amount of speed. This shows the necessity for inventing an aerodynamic wing, spoiler which creates a carefully controlled stall over the wing portion behind the spoiler, basically by reducing the lift of that wing section. Spoilers are designed to reduce lift also making considerable increase in drag [1].

The invention of the spoiler was inevitable, because of having capability for reducing aerodynamic lift; in the view of safety aspect is more important than the aerodynamic drag, which influences the car performance as well as its economical side. It is observed that an automobile experiences an uplift force while traveling at higher speeds. Therefore, the automobile losses traction with the road and drivers fall into problem for maintaining proper control.

Aerodynamic phenomena i.e. spoiler created drag & down-force is important aspect for the analysis of the phenomena of car. It is estimated that the aerodynamic drag is the governing form of resistance when vehicles run at speeds of 80 km/h or greater, especially considering the fact that 65% of the power required at 110 km/h is consumed due to overcoming aerodynamic drag [2, 3].

Aerodynamic characteristics such as flow separation, force & pressure distribution phenomena of car with rear end spoiler or having no spoiler are a lot of dissimilarities. Besides, angle of attack (α) is also important concern for the change of aerodynamic properties of moving vehicles. Drag & down-force created from the rear end spoiler is greatly influenced by the change of attack angle. For observing these effects clearly, proper analysis is needed. Use of Computational Fluid Dynamics (CFD) software greatly reduces time-to-market by reducing the need for costly physical testing and prototyping. Computational fluid dynamics can be considered as a practical tool for the analysis of vehicle aerodynamics. Numerical studies involved for the evaluation & the use of computational fluid dynamics for the prediction of the overall drag coefficient for vehicles have been carried out by a number of researchers [4-6].

Numerical investigation is so much helpful for declaration of spoiler's capability of better traction, faster turning, proper controlling i.e. acceleration & brake as well as increasing the vehicle safety.

This paper analysis is basically related to the help for suitable angle of attack optimization maintaining best performance.

2. VEHICLE GENERIC MODEL & DIMENSIONS

A generic model of a private car is shown in Figure 1 with its relevant dimensions. The length of this model is 4.36 m, width 1.885m & the height is 1.424 m. For finding our required analysis results, a sort of modifications are performed. Spoiler was mounted on the edge of the rear side of the vehicle without leaving a gap between spoiler & the surface of the vehicle. This model is prepared with the help of SOLIDWORKS 2015 software.

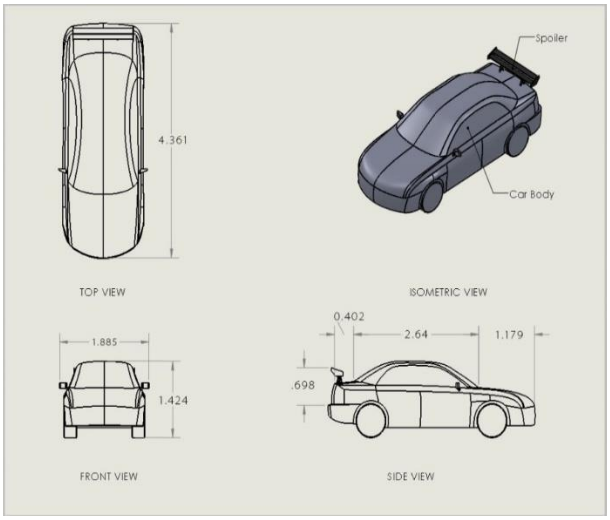

Fig.1: 3D car model with relevant dimensions (meter)

3. THE COMPUTATIONAL DOMAIN

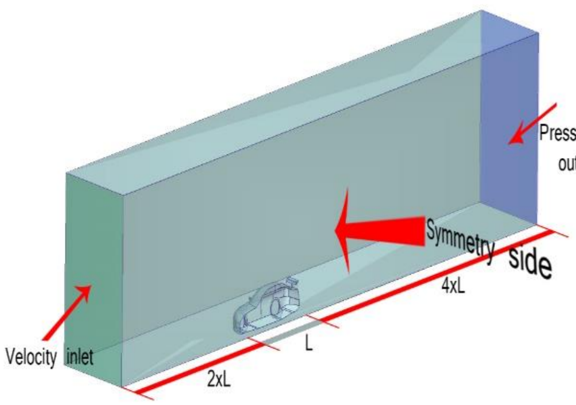

Fig.2: A virtual wind tunnel

Virtual wind tunnel with the chosen car & spoiler is prepared as shown in the above figure. A large air domain is used that avoids artificial acceleration due to squeezing air around the side and top of the car. The velocity inlet surface, in front of the car is situated at the two times the vehicle length. But, outlet is four times the vehicle length behind the car. There is a symmetry side shown in the mentioned figure. The material used for this domain is air.

4. MESH GENERATION

For finding accurate simulation results, appropriate mesh sizing is one of the basic criteria. Since, we are more interested in the rear side of vehicle, which is where the "wake of vehicle" phenomenon occurs, enough space has been kept in the rear portion of the vehicle model to capture the flow behavior mostly behind the vehicle. First of all, automatic meshing is created. Then, specific regions where rare spoiler are attached with car i.e. basic flow separation phenomena is generated, are meshed properly. The advantage of limiting this mesh sizing within the described region helps us for improving the meshing quality only within the area where we need high resolution mesh & basically this leads to get decent results of simulation. Zoom in view of region mesh from total mesh is shown in Figure 3.

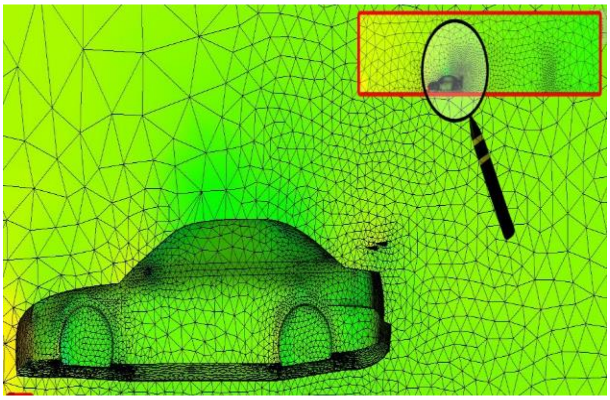

Fig.3: Region mesh with modified sizing setting

5. BOUNDARY CONDITIONS

Boundary conditions for faces & volumes were configured in AUTODESK SIMULATION CFD 2015. The velocity of the air at the inlet is set in the range of 60 km/h to 120 km/h. The outlet boundary surface is set at pressure 0 Pa. A slip/symmetry boundary condition was given at the symmetry side of the wind tunnel shown in the figure 2. The value of density of air is 1.2041 kg/m^3 at STP. The angle of attack is adjusted for four different values at 5 degree increments. These are 4.5, 9.5, 14.5 & 19.5 degrees.

6. RESULTS AND DISCUSSIONS

All the simulation works for the variation of attack angle were performed with same region meshing & same relevant criteria for same case. For obtaining accurate results, the model was run to convergence (higher number of iterations). Simulations were run with varying velocities in the range of 60 km/h to 120 km/h. Flow separation phenomena, velocity distribution & drag as well as lift coefficients were investigated with great care. From simulation, it was found that there is a higher pressure concentration on the front part of the car. Particularly, when air proceeds to the front side of the car, it slows down gradually because of accumulating more air molecules into a smaller space. Flow separation phenomena shown in figure 4, 5, 6 & 7 for four types of attack angles are of great variety with their relevant vortex region.

After that, C_D& C_L values were plotted keeping air velocity as an independent factor. Coefficient of drag is a unit less number, which indicates a body's ability to generate fluid resistance [7]. The magnitude of the coefficient of lift depends on the shape of the body and its angle of attack [8].

It has been appeared that, for higher angle of attack the drag coefficients are higher. The basic cause behind this is due to the fact that with smaller angle of wind collision, the spoiler would create smaller recirculation zone behind the rear end of the racing car.

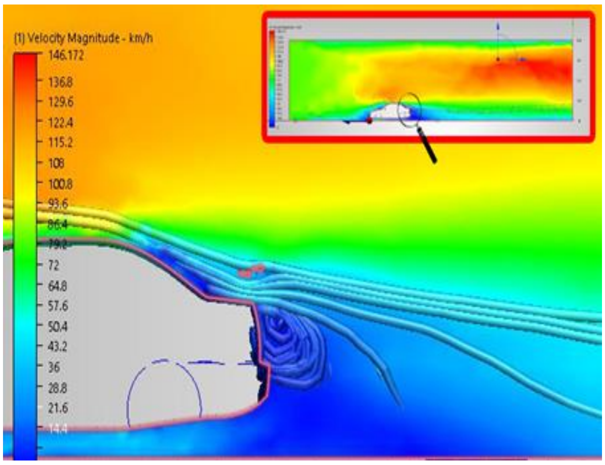

Fig. 4: Flow separation shown with traces at V=80 km/h & α=4.5 degree

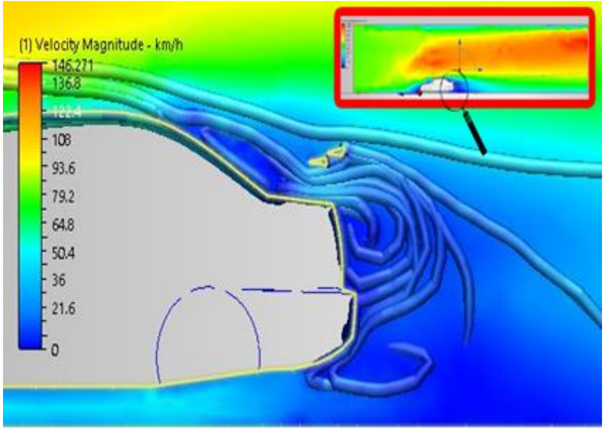

Fig.5: Flow separation shown with traces at V=80 km/h & α=9.5 degree

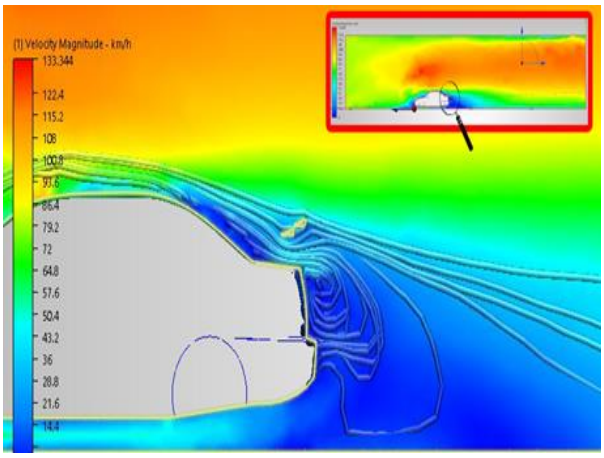

Fig.6: Flow separation shown with traces at V=80 Km/h & α=14.5 degree

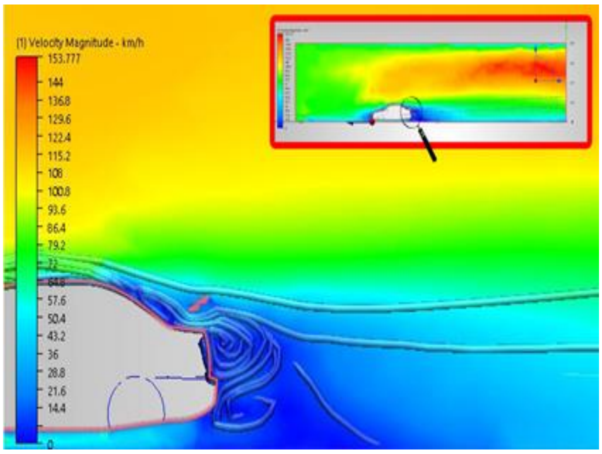

Fig.7: Flow separation shown with traces at V=80 km/h & α=19.5 degree

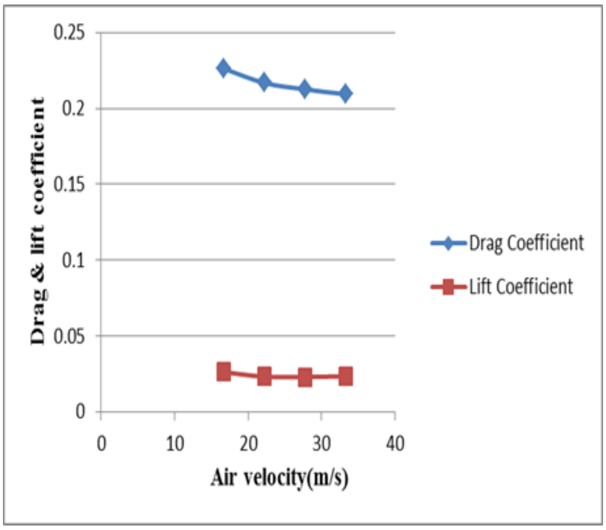

Fig.8: Variation of C_D& C_L over the range of Air Velocity when α is 4.5 degree

For angle of attack 4.5 degree, C_D decreases gradually, but C_L is decreasing with negligible amount.

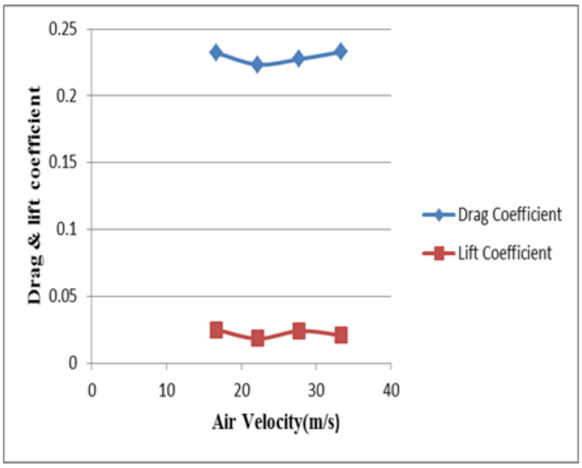

Fig.9: Variation of C_D& C_L over the range of air velocity when α is 9.5 degree

For 9.5 degree attack angle, graph phenomena is wavy type with fluctuation of the amount of C_D& C_L .So, this is totally unfavorable for usual car.

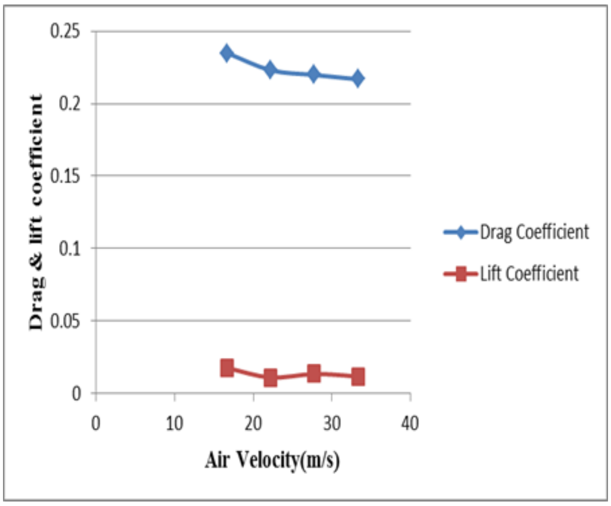

Fig.10: Variation of C_D& C_L over the range of air velocity when α is 14.5 degree

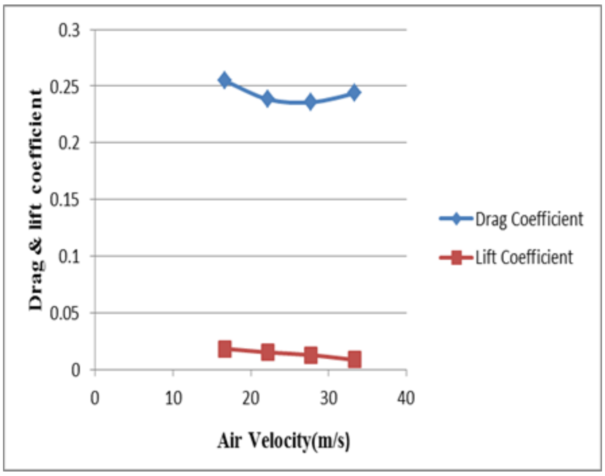

Fig.11: Variation of C_D& C_L over the range of air velocity when α is 19.5 degree

For 14.5 degree attack angle, C_D is decreasing with increased velocity, but there is some fluctuation of C_L characteristics. For variation attack angle, pressure distribution over the rear end of the car has influenced down force. For 19.5 degree, C_D values are of variety, but down-force has been spoiled efficiently.

Four graphs of Drag & Lift coefficient vs angle are plotted for the specified velocity range from 60 km/h to 120 km/h. With the increase of attack angle, C_D is increasing for all cases expect for the case when angle close to 14.5 degree. But, there is a significant loss of C_L .After analyzing the complete simulation result; we can make a decision that 14.5 degree is more preferable than any of these angle of attack taking drag & lift coefficient on consideration. Though, C_D is increasing, the lift force is more important for controlling the car stability, thus the drag coefficient can be sacrificed in order to achieve the desired lift coefficient. So, attack angle plays an important role in changing phenomena of drag & lift coefficient for a car having rear end spoiler.

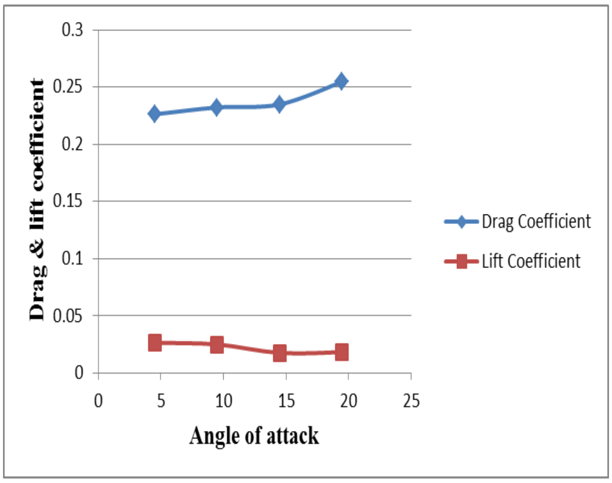

Fig.12: C_D& C_L VS α when V is 60 km/h

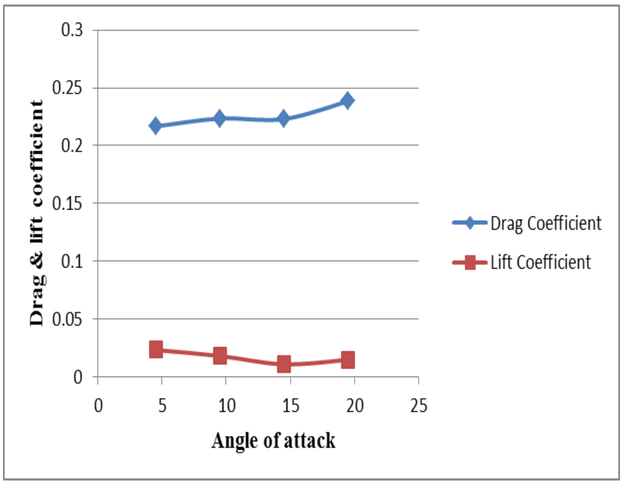

Fig.13: C_D& C_L VS α when V is 80 km/h

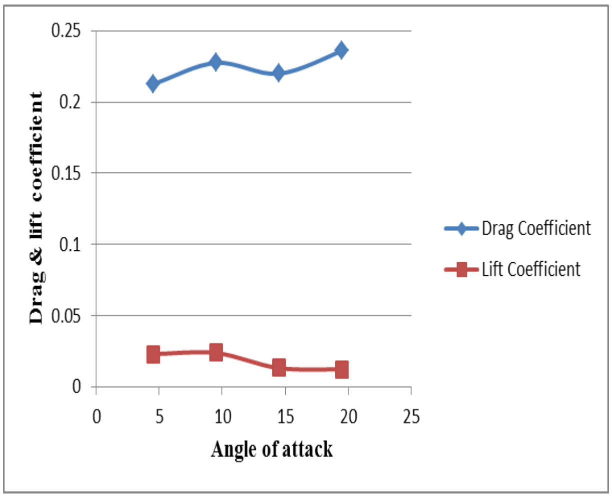

Fig.14: C_D& C_L VS α when V is 100 km/h

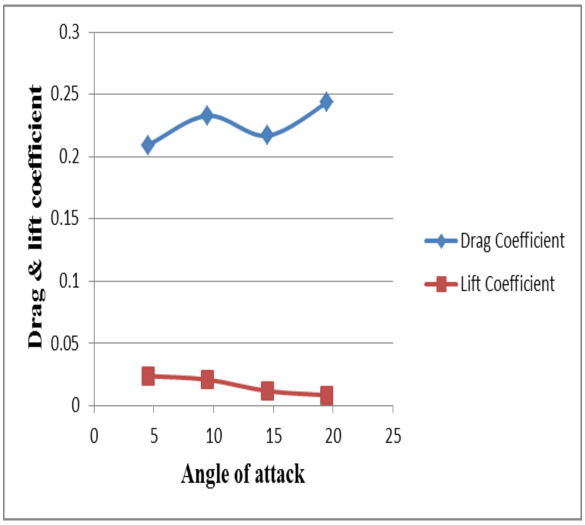

Fig.15: C_D& C_L VS α when V is 120 km/h

7. CONCLUSION

On the basis of car model, several computational fluid dynamics simulations are performed using AUTODESK SIMULATION CFD 2015 to visualize the airflow, velocity distribution & force characteristics around the car with changing attack angle. Simulation results are graphically represented which indicate that angle of attack is so crucial factor for drag & lift coefficient of a car. It is noticed that the angle which causes highest down-force was not the same for another velocities of the airflow. The mentioned changes of attack angle result in the variation of flow separation of rear end portion, velocity distribution as well as drag & down-force phenomena. So, angle of attack implies a great relationship with drag & down-force for better stability of car. But, spoilers are very efficient vehicle attachment, because it creates down-force for proper traction and thereby with small effect to increasing drag.

8. ACKOWLEDGEMENT

The authors gratefully acknowledge the Department of Mechanical Engineering, KUET due to allowing touse Computational Fluid Dynamics (CFD) lab. Also, the authors express their sincere gratitude to Dr. Mohammad Mashud for extending his expertise in the progress of this work.

9. REFERENCES

[1]Spoiler - CFD-Wiki, the free CFD reference.
[2] Leduc G., 2009, "Longer and heavier vehicles, an overview of technical aspects", JRC Scientific and Technical Reports, European Communities.
[3] Diamond S., 2004, "Heavy Vehicle Systems optimization", Annual Progress Report for Heavy Vehicle Systems Optimization, Washington, D.C, U.S.A
[4]Salari K, J. Ortega and P. Castellucci. "Computational prediction of aerodynamic forces for a simplified integrated tractor–trailer geometry". AIAA Paper 2004 - 2253. 2004.
[5] Pointer D. "Evaluation of commercial CFD code capabilities for prediction of heavy vehicle drag coefficients". AIAA Paper 2004-2254. 2004.
[6] Maddox S., K. Squires, K.E. Wurtzler, J. Forsythe. "Detached- eddy simulation of the Ground transportation system. The Aerodynamics of Heavy Vehicles: Trucks Buses and Trains", 2004, Vol. 1: p. 89–104.
[7] Halliday, David, Resnich, Robert - "How do wings work".
[8] Santiago Giraldo, Manuel J. Garcia, Pierre Boulanger - "CFD Based Wing Shape Optimization Through Gradient-Based Method " EAFIT University, No 7 Sur 50, Medellin, Colombia University Of Alberta, 2-21 Athabasca Hall, T6G 2E8, Canada.

10. NOMENCLATURE

Symbol	Meaning	Unit
C_D	Co-efficient of drag	Dimensionless
C_L	Lift co-efficient	Dimensionless
V	Air Velocity	Km/h
α	Angle of attack	Degree