ICMERE2015-PI-262

STUDY ON DRAG ANALYSIS OVER A COVERED VAN

Md. Sourove Akther Momin[1,*], Mohammad Mashud[2] and Mr. Md. Golam Kader[3]

[1-3]Department of Mechanical Engineering,
Khulna University of Engineering & Technology, Khulna-9203, BANGLADESH
[1,*]sur.sor269@gmail.com, [2]mdmashud@me.kuet.ac.bd, [2]mdgolamkader@gmail.com

Abstract: In recent times, CFD analysis, with the advent of computer architectures with superfast processing capabilities are rapidly emerging as an alternative to wind tunnel tests. Numerous aerodynamic designs of vehicles have been made to reduce aerodynamic drag for the consumption of fuel as well as save the global environment and the aerodynamic development of external body shape is also aesthetically attractive, it will much help to attract the customers. In the present work, a design concept was incorporated to optimize the shape of the Covered Van to observe the effects on the reduction of aerodynamic drag. It is found that the total aerodynamic drag on a running vehicle is the pressure drag by stagnation on the frontal side and vortex at the rear side. Computational fluid dynamics (CFD) method was incorporated to analyze the variations of aerodynamic drag's effect on the model van with the change of the configurations. The Autodesk CFD 2015 program was used to obtain a better solution and a better flow around the van. To validate the results, a recognized journal is studied and analyzes the results, only drag force for comparing with the journal.
.

Keywords: CFD, Drag, Velocity Vector, Covered Van

1. INTRODUCTION

Not only to save energy but also to protect the global environment, fuel consumption reduction is main concern of automobile development. In development of vehicle body, it is essential to reduce drag for improving fuel consumption and driving performance. An aerodynamically purified body is not only aesthetically attractive but also it will help much to increase the vehicles appeal to the customer [1].

If a body of arbitrary shape, immerged in a large stationary mass of fluid is moved with constant velocity through the fluid, the body exerts a force to the fluid, which opposes the motion of the body [4]. Also the fluid exerts a force to the body as every action have equal and opposite direction. The force exerts on the body or the fluid results from the relative motion between the emerged body and its surroundings fluid. As such it makes no difference in the magnitude of the force whether the body through the stationary fluid or the fluid flows about the immerged body which is at rest. The force exerted by the fluid on the moving body may in general be inclined to the direction of the motion and hence it has a component in the direction of motion as well as one the perpendicular to the direction of motion. The component of this force in the direction of motion is known as drag force [5].

In general, vehicles are known to be aerodynamically insufficient compare to other ground vehicles due to their large frontal areas and bluff-body shapes. The insufficient aerodynamic shape results in excessive drag which leads to elevated fuel consumption rate [3]. The aerodynamic drag is due to the separation of flow regions in different parts of the vehicle such as by the wake region behind the vehicle as shown in figure 1. Most Probably all road vehicles such as cars, truck, bus, Covered van etc.is bluff bodies. For this reason contributions to the vehicles aerodynamic drag are mainly due to pressure drag also known as form drag by the flow separation at the rear end of the body as shown in the figure 1&2. It is estimated that the pressure drag on heavy vehicles account to more than 80% of the total aerodynamic drag, with frictional drag accounting for the remaining 20%[2].

Computational fluid dynamics provides an important tool for the analysis of heavy vehicles like Covered van, bus etc. aerodynamics .The purpose of the numerical study is to evaluate the use of computational fluid dynamics for the prediction of the overall drag co-efficient for heavy vehicles. In this paper, a three dimensional field flow analysis has been performed to understand the air flow characteristics surrounding a Covered van like bluff body. The commercial software Autodesk CFD is incorporated to converge solution. Results provide boundary layer details and associated drag for the Covered van geometry. Drag can be reduced by modifying the van geometry. The drag is calculated for the vehicles moving at 15m/s, 30m/s, 45m/s, 60m/s.
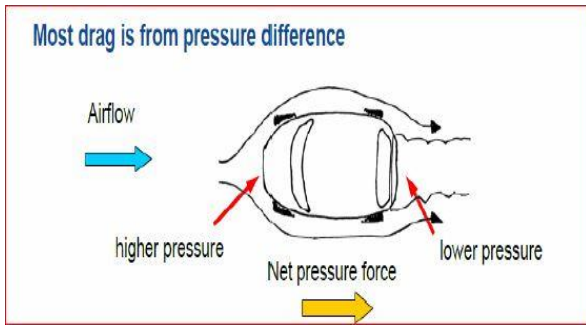
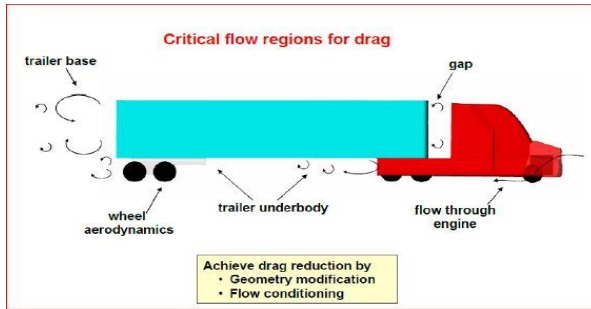Fig.1: Aerodynamic drag due to pressure difference [2]

Fig.2: Drag contributing region of vehicles [2].

2. GEOMETRIC DESIGN

Frontal shape and the configurations of different spoiler are two most important parameters to improve the aerodynamic performance of the model Covered van. As for the analysis the geometric model is prepared with the help of CAD (Computer Aided Designing) software. Solid works 2015 is used as CAD platform for preparing the model. The CAD model is shown in figure 2. The model is about 398.5mm in length, 181.3 mm height and 120mm in width.
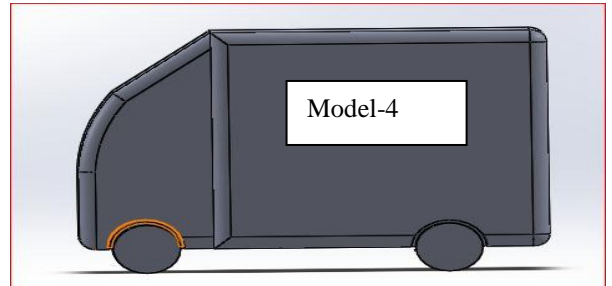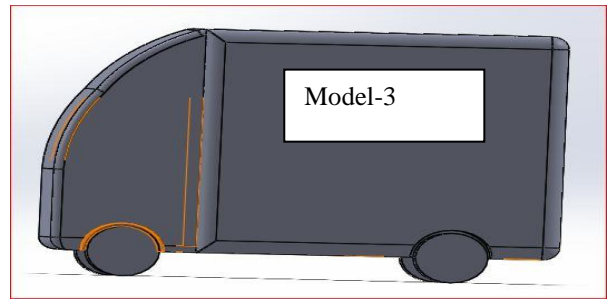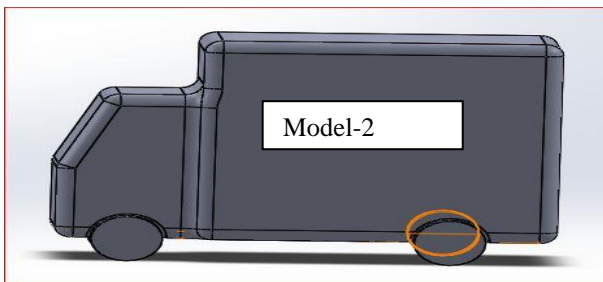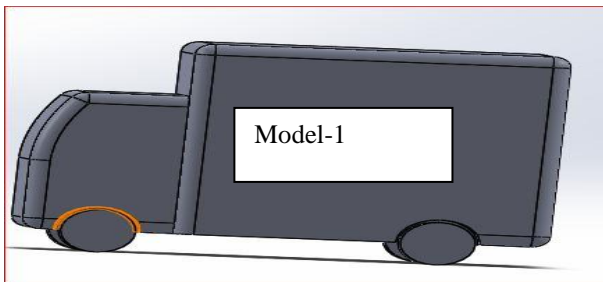

Fig. 3: Different geometric model of Covered van

3. CFD SIMULATION SET UP AND BOUNDARY CONDITIONS

The commercial Autodesk CFD package is incorporated to perform the simulation. In this package FEM (Finite Element Method) was employed to simulate the flow phenomenon around the Covered van. The flow is considered turbulent, incompressible, steady state and k-ε model. The boundary conditions set to the domain (a) Velocity boundary conditions at inlet (b) at exit constant pressure boundary condition (c) no-slip conditions at the domain wall. The steps for a setting up of numerical procedure are shown in figure3. The simulation was performed until the drag co-efficient attains a constant value and is no longer changing with iterations. The drag co-efficient was evaluated for the different model considered in this paper using the flowing equation.

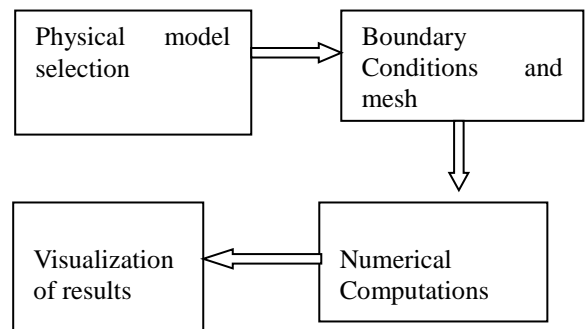$$C_d = \frac{F_x}{\frac{1}{2} * \rho_\infty * V_\infty{}^2 * A} \qquad (1)$$

.

Fig. 3: Steps for setting up of numerical procedure

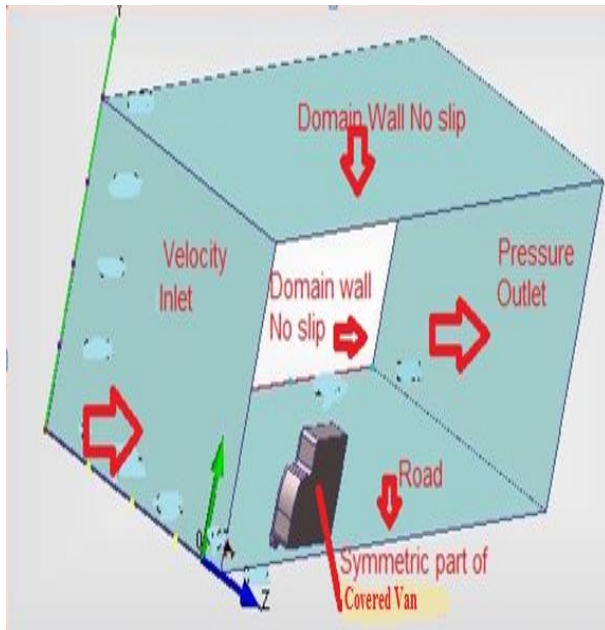Fig. 4: Computational Domain

4. RESULT AND DISCUSSION

Drag co-efficient of three dimensional bodies are always depends on the number of sharp edges which increases the flow separation and drag force. All the three dimensional bodies have drag depending upon the point of separation. The separation phenomenon is not limited to blunt bodies or curved boundary surface. An adverse pressure gradient is obtained in a diffuser. The boundary layer separation does also take place where boundary surface suddenly changes its longitudinal profile providing sharp corner or edges. The in ability of streamlines to bend around the edges leads to the flow separation of flow. The model vans have smooth shape and sharp edges are avoided, since they are responsible for adverse pressure gradient. In figure 5 the velocity vector are shown at different velocity for model-1. In the figure the direction of the flowing air orientations are addressed at different velocity. In every velocity vector figures air slowdown when it reaches the front of the Covered van. For this reason more air molecules are accumulated into a small space. The figure represents that there are two vortex zones created. For model-1 and model-2 there are two vortex zones at the front top of the van called death flow region and another at the rear end of the van. These vortexes are the prime reason for the drag force. But in model-2 and model-4 one vortex zone is reduced by using spoiler at the frontal area. At model-2 curved type and model-4 triangular type spoiler is used. In figure 6,7,8,9 the variations of drag co-efficient is shown with the variations of velocity. In every graph the with variations of velocity small amount of changes is occurred in drag co-efficient for every model.
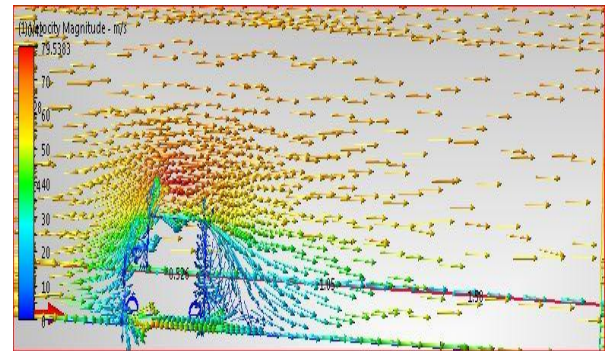

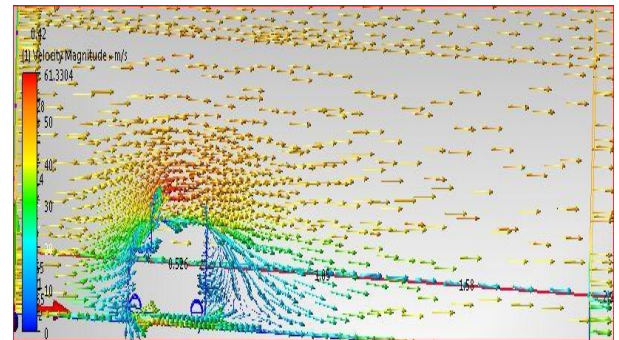Fig. 5.1: Velocity vectors at velocity 15 m/s

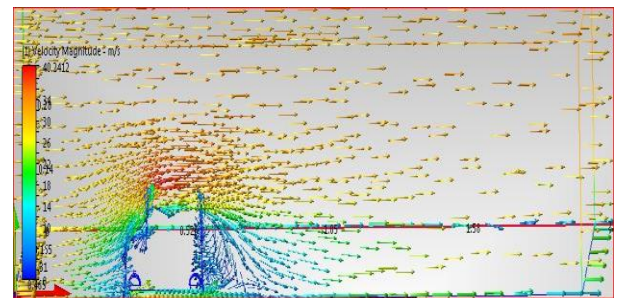Fig. 5.2: Velocity vectors at velocity 30 m/s

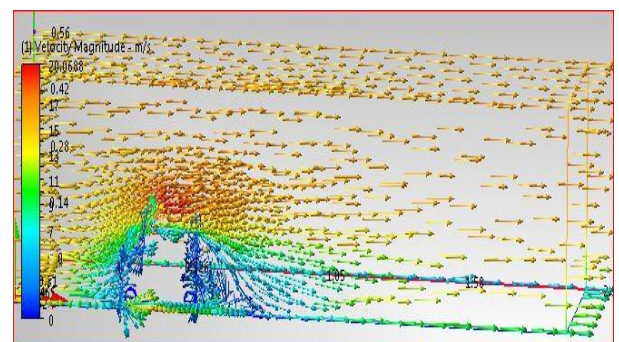Fig. 5.3: Velocity vectors at velocity 45 m/s

Fig. 5.4: Velocity vectors at velocity 60 m/s

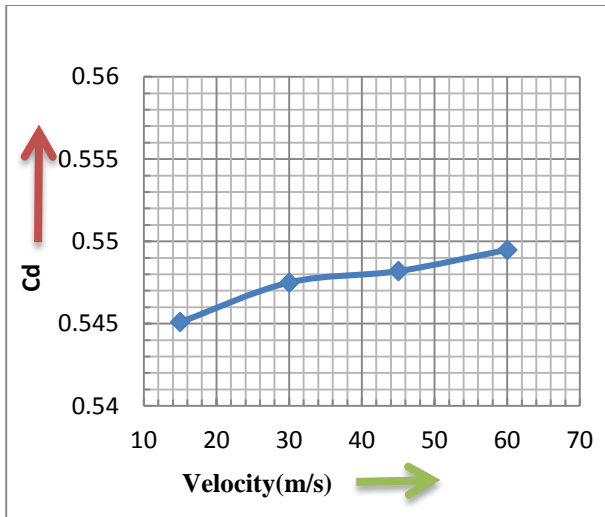Fig. 5: Velocity vectors at mid plane of the model-1 at different velocity.

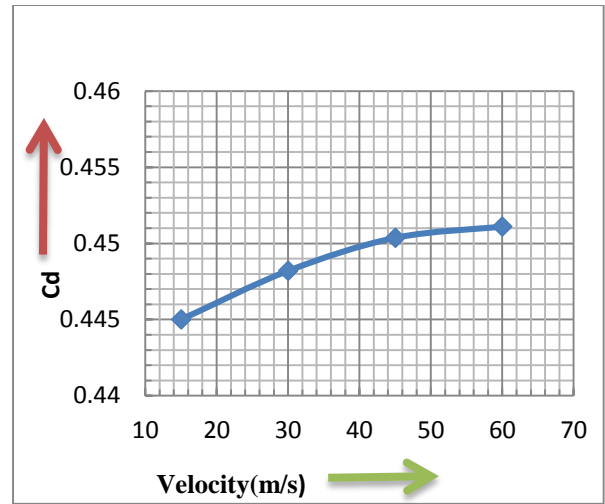Fig. 6: Variations of drag Co-efficient with Velocity (m/s) for model-1

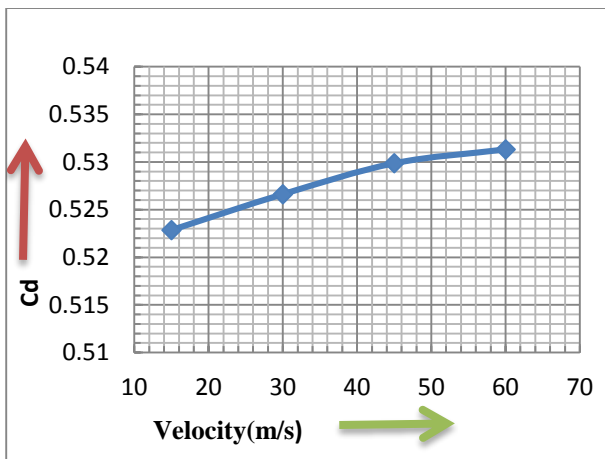Fig. 7: Variations of drag Co-efficient with Velocity (m/s) for model-2

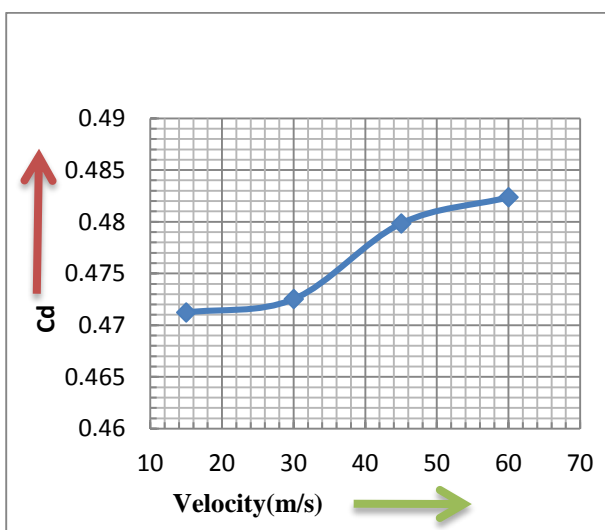Fig. 8: Variations of drag Co-efficient with Velocity (m/s) for model-3

Fig. 9: Variations of drag Co-efficient with Velocity (m/s) for model-4

In figure 10 the compression of drag co-efficient between model-1 and model-2 is 4.041%.At model-2 the frontal area of model-1 is replaced by a triangular area. The difference of drag co-efficient between model-1 and model-3 is 13.555%. In model-3 a curved type spoiler is used there. The difference of drag co-efficient between model-1 and model-4 is 18.36%.At this model the curved type spoiler is replaced by a triangular type spoiler. For verification the result a paper, "Numerical Drag Reduction studies of Generic Truck Models using Active flow Control"[2] is studied. A model is studied of the paper and simulates according to the papers conditions. The results of the paper and the simulated result is comparing in figure 11. All the simulations are performed at the same procedure.
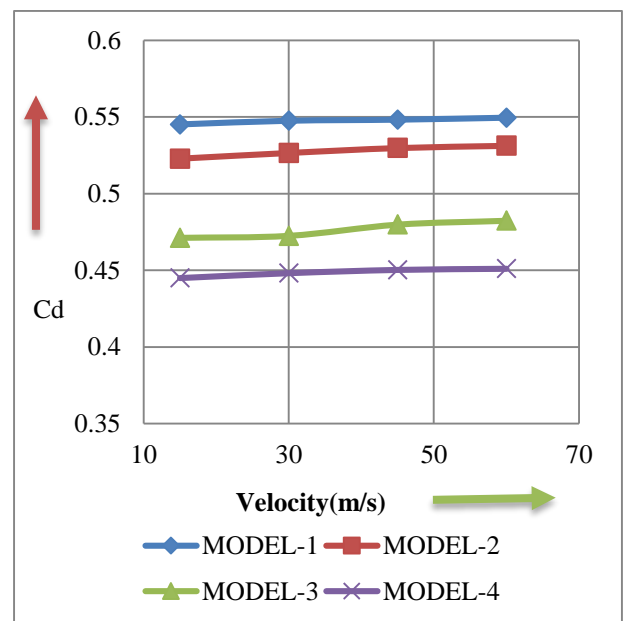

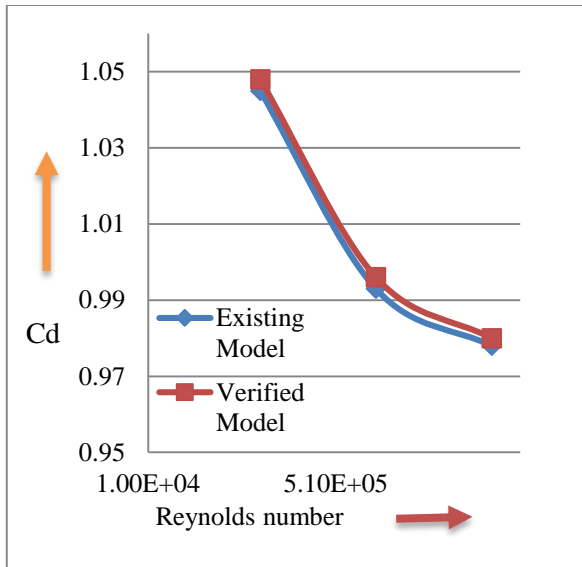Fig.10: Comparison of Drag co-efficient among different model.

Symbol	Meaning	Unit
C_d	Drag Coefficient	Dimensionless
F_x	Drag force along X-axis	(N)
V_∞	Free steam velocity	(m/s)
ρ_∞	Density of the air	(Kg/m^3)
A	Frontal area of the vehicle	(m^2)

Fig.11: Compression between existing model and verified model

5. ACKOWLEDGEMENT

6. REFERENCES

[1] Md Hasan Ali, Mohammad Mashud, Abdullha Al Bari, Muhammad Miaba-Ul Islam, "Aerodynamic Drag Reduction of a Car by Vortex Generation", *International Journal of Mechanical Engineering*, ISSN-2277-7059, Volume-2 Issue 1

[2] Miles Bellan, Jonathan Naber Ramesh Agarwal, " Numerical Drag Reduction Studies of Generic Truck Models Using Active Flow Control",*39th AIAA Fluid Dynamics Conference 22-25 June ,san Antolo , Texas*.

[3]E.M Wahab, H. Al-Marzooqi, M.Shaath, T.El-Dhmashawy, "Aerodynamics Drag Reduction for ground Vehicles Using Lateral Guide Vanes", Vol, 4(2)-June 2012

[4] Dr. P.N Mod, S.M Seth, *Hydraulics and Fluid Mechanics*: Standard Publishers, 1998

[5]Dr.R.K BANSAL, *A TEXT BOOK OF FLUID MECHANICS AND HYDRAULIC MECHANICS*: LAXMI PUBLICATIONS (P) LTD, 2010