Proceedings of the
International Conference on Mechanical Engineering and Renewable Energy 2017
(ICMERE2017) 18 – 20 December, 2017, Chittagong, Bangladesh

ICMERE2017-PI-295

DESIGN OF AN AERODYNAMIC TRUCK MODEL WITH LOW DRAG FORCE CO-EFFICIENT

Khondakar Sami Ahamad[1], Md. Jonayed Hasan[2], Jamal Uddin Ahamed[3]

[1,2)] Department of Mechanical Engineering
Chittagong University of Engineering & Technology, Chittagong-4349, Bangladesh
samiahamad10079@gmail.com[1], jonayed.me@gmail.com[2*], jamal@cuet.ac.bd[3]

Abstract*- The aerodynamic drag force of truck can be reduced by changing the shape or by using effective aerodynamic edges. In this research, an aerodynamic truck model is designed with low drag force parameters. At first a simple truck model is designed using Solidworks. Then Solidworks Flow Simulation which is an intuitive CFD tool is used to figure out the places where the effects of drag force can be existed. In order to run flow simulation decorously, all the important parameters such as unit system, analysis type, fluid type, initial and ambient conditions, geometry resolution have been selected cautiously. This same procedure has also been applied on five other truck models. The modified simple truck model with a triangular shape fairing, base plates at the end of the carrier box and having under carriage gap treatment which shows probably the best output among our six models, so we have considered this model for analyzing the change of drag force at different velocity. From overall analysis, it is evident that modified simple truck model with a triangular shape faring has low drag force and drag force co-efficient but considering the other aspects this research shows that our recommended the aerodynamically modified truck model is the best option because that model is designed similar to the size of a heavy duty truck but it's drag force and drag force co-efficient is very low and nearly close to the output of the small size truck models. It has also the ability of carrying huge amount goods which is the main purpose of a truck.*

Keywords: CFD, drag reduction, triangular shape fairing, base plates, gap treatment.

1. INTRODUCTION

One of the most important environmental issues within the automotive industry today is to reduce the fuel consumption and emissions. Together with the increased fuel price this has resulted in a "green race" within automotive companies in order to stay competitive, and the development of fuel efficient products has escalated. This is, among other methods, achieved by improving efficiency of the engine, reducing rolling resistance and improving aerodynamics [1]. When a truck runs on the highway, it loses 65% of its total energy to overcome the drag forces that oppose the truck's motion. The goal of this project is to modify the shape of the truck by using effective aerodynamic edges and reduce the drag force co-efficient. Drag force due to the combine effects of the wall shear stress and pressure forces, which is define by

$$F_d = C_d \frac{1}{2} \rho V^2 A \quad (1)$$

Here, C_d is the drag coefficient, ρ is the density of air, V is the velocity, and A is the cross-sectional area. We will determine some major target areas on a truck that contribute to the overall drag force. In our project, we will design some effective aerodynamic edges for those areas and reduce the drag force co-efficient. [2]

Computational fluid dynamics, usually abbreviated as CFD is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyze problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved. Ongoing research yields software that improves the accuracy and speed of complex simulation scenarios such as transonic or turbulent flows. Initial experimental validation of such software is performed using a wind tunnel with the final validation coming in full-scale testing [3]

In fluid dynamics, drag (sometimes called air resistance, a type of friction, or fluid resistance, another type of friction or fluid friction) refers to forces acting opposite to the relative motion of any object moving with respect to a surrounding fluid. This can exist between two fluid layers (or surfaces) or a fluid and a solid surface. Unlike other resistive forces, such as dry friction, which are nearly independent of velocity, drag forces depend on velocity. Drag force is proportional to the velocity for a laminar flow and for a squared velocity for a turbulent flow. Even though the ultimate cause of a drag is viscous friction, the turbulent drag is independent of viscosity.

Aerodynamic drag consists of two components, pressure drag (force acting normal to surface) and friction drag (force acting tangential to surface). Friction drag is due to shear stress between the fluid and the surface, whereas pressure drag is due to a pressure difference between the front and the rear of the body. For a truck, and other blunt bodies, the pressure drag is the most dominating one and contributes to more than 90 percent of the total drag [6]. In addition to the tractor front, the regions that represent the main drag-contributing areas around a truck are: the gap between the tractor and the trailer, the base wake behind the trailer and the undercarriage [7].

2. PROCEDURE

2.1 DESIGN MODEL TRUCKS IN SOLIDWORKS

Solidworks was used for designing Truck models of different shapes and sizes. Solidworks is a worldwide used software in field of three dimensional object drawing. Designing the outer shape of the model trucks the main concern of this research. Two types of truck models (simple and heavy duty) were designed and both types are modified with different aerodynamic edges. One completely conceptual aerodynamic truck was also designed having the size of a heavy duty truck.

2.2 FLOW SIMULATION STEPS

There are some initial steps which are needed to be define for running the flow simulation properly. Those steps are shown in Figure 1.

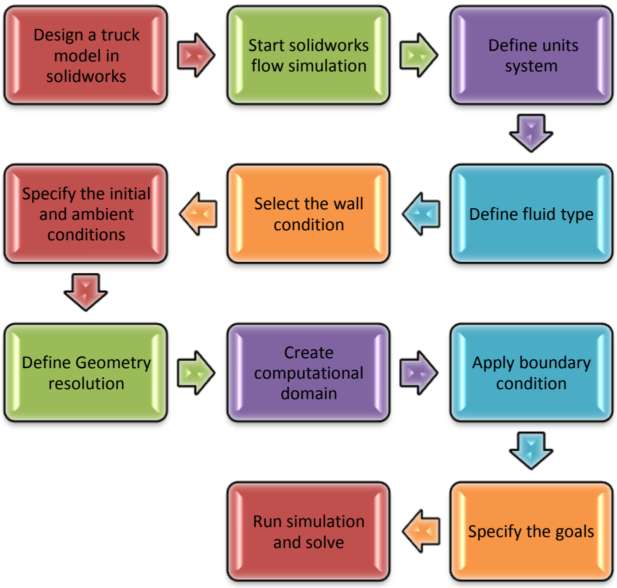

Fig. 1: Simulation procedure.

For every model, individual flow simulation project file was created before running the simulation. SI unit system is used and the fluid type is obviously air. Pressure and temperature is environmental but the velocity of air flowing over the truck body has average speed of the truck. The concept of this simulation is totally inverse of the reality. In real condition air is stationary and truck moves through the air but here in this simulation truck in stationary and air is flowing over the truck creating the lift force. After defining the initial conditions for simulation, domain size, and mesh were being generated which are the most important features in operating CFD process. Proper domain and fine mesh are essential for better result. The solidworks Flow Simulation technology is based upon the use of Cartesian-based meshes and Meshing Technology is one of the key elements of the CAD/CFD Bridge for CAD-embedded CFD. Solidworks Flow Simulation employs transport equations for the turbulent kinetic energy and its dissipation rate, using the k-ε model. Global goals were selected which were essential for analysis of aerodynamic drag. An equation of calculating drag force co-efficient also implemented in defining simulation steps. These procedure is flowed in different configurations of the truck model and results were collect after finishing each simulation.

Table 1: System information

Product	Flow Simulation 2013 SP0.0. Build: 2177
Computer name	DELL-PC
User name	DELL
Processors	Intel(R) Core(TM) i5-2450M CPU @ 2.50GHz
Memory	3990 MB / 8388607 MB
Operating system	Windows 7 (Build 7600)
CAD version	SolidWorks 2013 SP0.0
CPU speed	2501 (775) MHz

Table 2: Domain size

X min	4.715 m
X max	14.671 m
Y min	12.901 m
Y max	22.695 m
Z min	6.705 m

Table 3: Ambient Conditions

Thermodynamic parameters	Static Pressure: 101325.00 Pa Temperature: 293.20 K
Velocity parameters	Velocity vector Velocity in X direction: 0 m/s Velocity in Y direction: 0 m/s Velocity in Z direction: 36.00 m/s
Turbulence parameters	Turbulence intensity and length Intensity: 0.10 % Length: 0.050 m

Table 4: Formula, Dimensionality & Use in convergence

Formula	GG Force(Z) 1*2/(1.204*3.85*3.58*36^2)
Dimensionality	Force
Use in convergence	On

2.3 IDENTIFYING THE TARGETED AREAS OF MODIFICATION

Targeted areas of modification are found by running a test simulation over a usual shape truck and figure 2 shows the pressure and velocity distribution over the truck body where disturbances of pressure and velocity can be easily understand. From the output of flow simulation some major areas were identified which will reduce the pressure drag over the truck significantly. Those areas are also shown in figure 2.

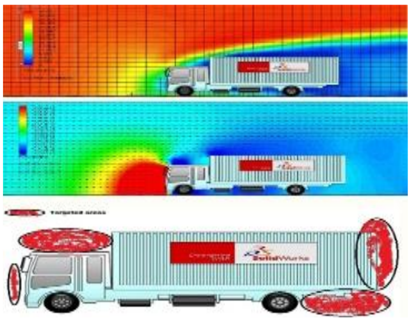

Fig. 2: Velocity and pressure distribution over model truck.

2.4 MODIFICATION OF A SIMPLE TRUCK

A triangular fairing over the upper position of the frontal part of the truck was used in the modified version of the simple truck model. A gap treatment at the under carriage portion of the truck and two triangular shape base plates which are known as the vortex stabilizer was also used for aerodynamic improvement. All this aerodynamic improvements are shown in the figure 3.

Fig. 3: Modified truck model (I)

Some trucks use fairings of different shapes. A most commonly used fairing over truck is shown in figure 4 along with other aerodynamic modifications of previous model in order to compare the advantages of both types of models.

Fig. 4: Modified truck model (II)

2.5 MODIFICATION OF HEAVY DUTY TRUCK

Figure 5 shows the modification over the design of the heavy duty truck using a rectangular type fairing with the upper edge of the frontal side of the fairing is being filleted and also changed the shape of the carrier box making the end edge curving downward. Sealing the gap between the truck frontal part and the carriage part and under carriage design treatment were also applied for aerodynamic improvement. A modern concept of using skirt below the carriage part is also applied in this version.

Fig. 5: Heavy duty truck with modified model.

2.6 CONCEPTUAL AERODYNAMIC TRUCK MODEL

Applying modern aerodynamic techniques, a new conceptual model truck was designed which is shown in figure 6. The size of this truck is same as heavy duty truck but its external body shape is completely aerodynamic. This aerodynamic truck has all the aerodynamic design treatment like use of fairing, gap treatment, under carriage design treatment, reshaping the carrier box.

Fig. 6: Aerodynamic truck model

3. RESULT AND DISCUSSION

By running the flow simulation over each model each truck model, different kinds of information, data and graphical plots were collected and analyzed on basics of aerodynamics. To understand results we study velocity, pressure and streamline based results in graphical mode.

3.1 VELOCITY DISTRIBUTION OVER MODIFIED MODEL TRUCKS

Figure 7 shows the velocity distribution over a truck model which have been modified by using fairing over the truck and tail plates on the end parts of the truck carrier box and sealed the under-carriage gap. it represents that using of these aerodynamic edges and under carriage gap treatment reduce the velocity drop over the truck simple truck model.

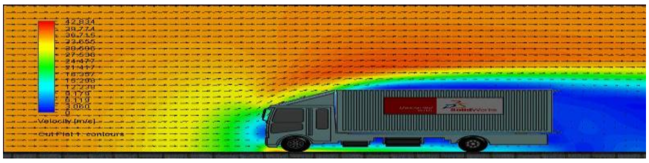

Fig. 7: Velocity distribution Modified simple truck model (I).

The figure 8 below shows another type of modification of a simple truck model in which the fairing is different than the previous one. This type of fairing is shown over cargo trucks on the road. Although this modification is less efficient than the previous because it reduces less velocity drop than the previous one.

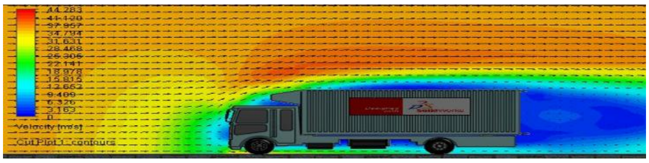

Fig. 8: Velocity distribution over modified truck model (II).

3.2 VELOCITY DISTRIBUTION OVER HEAVY TRUCK MODEL

This type of truck model is much more lager than simple truck model. So, the velocity drops over this kind of truck is also huge on different areas. The velocity distribution presented in figure 9 shows the velocity drop over the heavy-duty truck model.

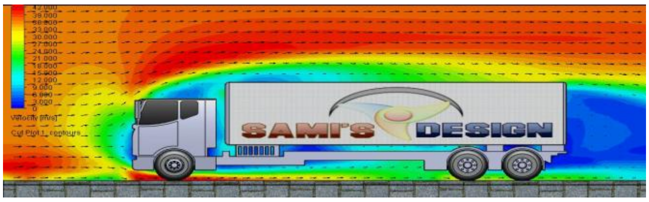

Fig. 9: Velocity distribution over heavy truck model.

3.3 VELOCITY DISTRIBUTION OVER MODIFIED HEAVY TRUCK MODEL

Modifications over heavy duty truck model significantly reduces velocity drop at frontal part and tail part which is portrayed in figure 10. Which has a better aerodynamic results.

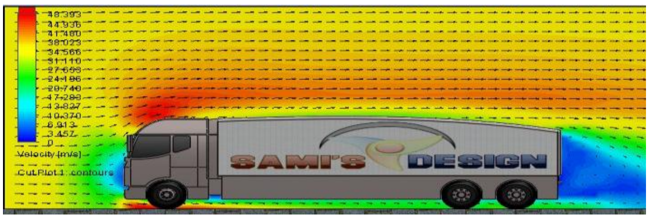

Fig. 10: Velocity distribution over modified heavy truck model

3.4 VELOCITY DISTRIBUTION OVER MODIFIED AERODYNAMIC TRUCK MODEL

In figure 11 Aerodynamic truck model shows less velocity drop than previous models.

Fig. 11: Velocity distribution over modified aerodynamic truck model

3.5 PRESSURE DISTRIBUTION OVER MODIFIED SIMPLE TRUCK MODELS

In figure 12 Pressure distribution over simple modified truck models are in which it is clearly presented that modified truck model faces less pressure drag than the modified one and also in case of tail part the modified version of the trucks significantly low pressure drop which also reduces vortex effect.

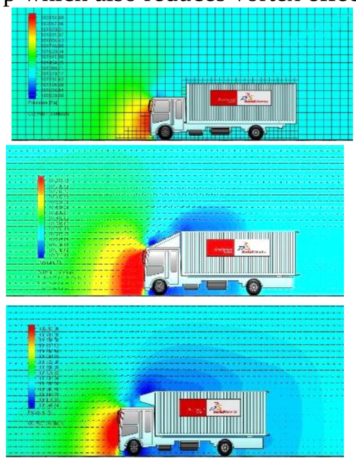

Figure 12: Pressure distribution over modified simple truck models.

3.6 PRESSURE DISTRIBUTION OVER HEAVY-DUTY TRUCK AND MODIFIED VERSION OF HEAVY DUTY TRUCK

Heavy duty truck faces huge amount of pressure differences due to its shape but modification of the body structure using aerodynamic edges reduce pressure differences significantly which is shown in figure 13.

Figure 13: Pressure distribution over heavy duty truck and modified version of heavy duty truck

3.7 COMPARISON OF DRAG FORCE CO-EFFICIENT

Figure 14 shows below the co-efficient of drag forces for the different configurations of the truck models. First three truck models are actually small sized truck models. These trucks have low co-efficient of drag. Among the above-mentioned model trucks, the first modified truck model with rectangular shape fairing have low drag force co-efficient. For this reason, first modified truck model is better than others. But overall aerodynamic truck is the best because it is a heavy-duty truck but it has comparatively low co-efficient of drag and even nearly as much as small size trucks.

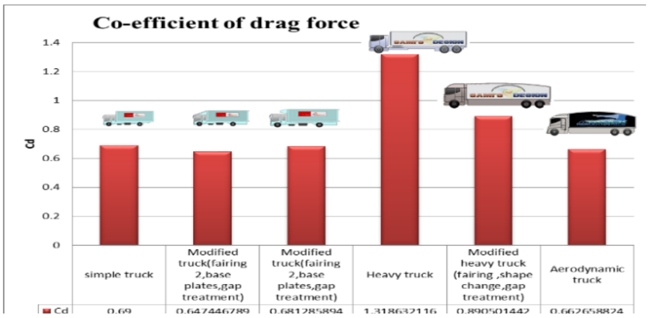

Fig. 14: Comparison of drag force co-efficient

3.8 CHANGING OF DRAG FORCE WITH CHANGE OF VELOCITY OVER AN AERODYNAMIC TRUCK MODEL

Aerodynamic truck model which shows comparatively the best output among our six models, so we consider this model for analyzing the change of drag force at different velocity. We consider the velocity 33, 36, 37, 40, 43, 45 meter per second which have been given as input velocity in solidworks flow simulation 2013 and we collected the drag force data. From the graph in figure 15, it seems that with the increase of velocity the drag force also increases. Figure 16 and figure 17 shows velocity distribution and streamline regarding same aerodynamic model which is visual representation of that curve which have been produced from the analysis of the simulation.

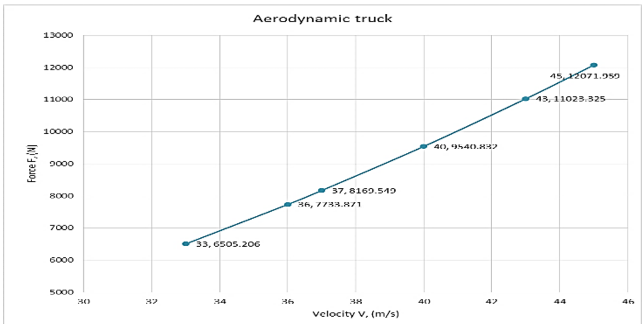

Fig. 15: Graphical representations of velocity vs drag force over an aerodynamic truck.

Fig. 16: velocity distribution over the aerodynamic truck model at various velocity

Fig. 17: Flow trajectories showing velocity flow over aerodynamic truck model

4. CONCLUSION

A reasonable comparison of drag force co-efficient among various model trucks along with conceptual aerodynamic truck has been attempted. From the simulation result it is evident that Aerodynamic truck model is much more effective than other truck models. Although the first modified truck model which is shown in Figure 3 has the lowest drag force co-efficient but that truck is a small size truck. Our proposed aerodynamic truck is heavy duty truck model hence it has the drag force co-efficient nearly close to the small size truck model. In last it was important to show that CFD tool is good approach for aerodynamic analysis and also Solidworks flow simulation takes complexity out of computational fluid dynamics and makes it easier to analysis. For future work this design should be made more realistic considering practical aspects.

5. ACKNOWLEDGEMENTS

Heartiest gratitude to Dr. Jamal Uddin Ahamed for his scholarly and patient guidance throughout this research.

6. REFERENCES

[1] C. hakansson, M. J. lenngren, "CDF Analysis of Aerodynamic Trailer Devices for Drag Redution of Heavy Duty Truck" chapter 1, Department of Applied Mechanics, Division of Vehicle Engineering and Autonomous System, Chalmers University of technology Goteborg, Sweden 2010.

[2] K. Ihlen, K. Toro Ryan, B. White, Professor Roger Gans, "Trucking On Drag Coefficients of Tractor Trailer Trucks" page 2, ME 241 May 2, 2007.

[3] E. Michael, "The Dawn of Fluid Dynamics: A Discipline between Science and Technology", Wiley. p. ix. ISBN 3-527-40513-5.

[4] Y. Donald, F. Bruce, R. Munson, Theodore H. Okiishi, Wade W. Huebsch , " A Brief Introduction to Fluid Mechanics (5 ed.)", John Wiley & Sons. p. 95, ISBN978-0-470-59679-1

[5] R.M. Wood, "Impact of advanced aerodynamic

technology on transportation energy consumption", SAE, 2004-01-1306, 2004.
[6] R. Schoon, F.P. Pan, "Practical devices for heavy truck aerodynamic drag reduction", SAE, 2007-01-1781, 2007.
[7] R. M. Wood and Steven X. S. Bauer, "Simple and Low-Cost Aerodynamic Drag Reduction Devices for Tractor-Trailer Trucks", SOLUS – Solutions and Technologies 2003-01-3377.

7. NOMENCLATURE

Symbol	Meaning	Unit
F_d	Drag Force	(N)
C_d	Drag Force Co-efficient	Dimensio -nless
V	Velocity	m/sec
T	Temperature	(K)
P	Pressure	(Pa)
A	Cross sectional area	m^2