Journal of Advance Manufacturing System and Technology
Volume 1, Issue 1, January-June, 2016

Parametric Optimization and Aerodynamic Analysis of Loaded Truck

Jaydeepsinh J Vaghela[1], Yuvrajsinh L Raol[2]
Research Scholar (M.E)[1], Asst. Professor[2],
Department of Mechanical Engineering, L.C. Institute of Technology, Bhandu, Mehsana
E-mail: jaydipvaghela3@gmail.com[1], yuvrajsinh.raol@lcit.org[2]

Abstract

To save energy and to protect the global environment, fuel consumption reduction is a primary concern of the modern truck manufacturers. Drag reduction is essential for reducing fuel consumption. Designing a vehicle with a minimized drag resistance leads to advantages of fuel economy and performance improvement. The shape is an important factor for drag reduction. To design a shape for an efficient truck that gives low resistance when moving forward, the most important functional requirement to consider is low fuel consumption. The resistance, termed as the drag force (or the drag coefficient in non dimensional terms), is a strong function of the shape of the truck. The main purpose of the project on which this paper is presented, was to reduce the drag co-efficient (CD) and drag force (FD) of a truck's body by improving the aerodynamic shape using CFD software (Autodesk Flow Design). The difference of FD as well as CD before and after the change in aerodynamic shape of truck is carried out at different speeds. We got the desirable results for the reduction of drag force - about 12 to 15%.

Keywords: *Optimization, Autodesk Flow Design, Drag Force, Drag Co-Efficient, Fuel Consumption, Non Dimensional, Drag Reduction ,Aerodynamics*

INTRODUCTION

• Aerodynamic means "Study of forces generated by motion of air on moving body."

• Applications : aerospace engineering, design of automobiles, ships, civil engineering, design of bridges, etc.

• Aerodynamic forces: Drag, Lift, Weight, Thrust.

• Computational Fluid Dynamics (CFD) is the science of predicting fluid flow, heat transfer, mass transfer, chemical reactions, and related phenomena by solving the mathematical equations which govern these processes using a numerical process.

Applications : Flow and heat transfer in industrial processes, Aerodynamics of ground vehicles, aircraft, missiles, ventilation, heating and cooling flows in buildings, film coating, thermoforming in material processing applications.[10]

LITERATURE SURVEY

Han T., (1989), "Computational Analysis of Three-dimensional Turbulent Flow around a Bluff Body in Ground Proximity", AIAA J. 27(9) 1213- 1219 [1]

Han (1989) performed a numerical study over a three dimensional bluff-body in proximity to the ground. Due to the lack of detailed velocity measurements over this body, this work compared the formation of the vortices in the wake and the drag coefficient for different slant angles qualitatively. The author used two different RANS based turbulence models namely the k-ε and the RNG k-ε. It was reported that the overall validity of the computations was dependent on the turbulence model and the accuracy of the discretization scheme. [1]

Sanisa K. and Davidson L., (2003). "Numerical Study of the Flow around a Bus-Shaped Body." Transaction Journal of the ASME.Vol.125 [2]

Sinisa and Davidson (2003) stated that the consideration was not only the drag and lift coefficients that describe the aerodynamic properties of the body, but also the flow structures responsible for these properties. The forces acting on the surface of the body as a result of the surface pressure were studied, both time averaged and instantaneously, in the study. So, while the flow around the bodies is highly unsteady, knowledge of this flow is based primarily on experimental and numerical studies of time averaged observations. The study has shown that the instantaneous flow is very different from the time averaged one, not only in the wake region but also along the entire body.[2]

Gokul Krishnan Rajamani, (2006), "CFD Analysis of Air Flow Interactions in Vehicle Platoons", School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, August 2006 [3]

Gokul Krishnan Rajamani (2006) studied time averaged characteristics of a simplified, generic passenger vehicle called the Ahmed car model. The 2D analysis of isolated model showed that the wake structure influences the drag coefficient of the model which is dominated by rear slant angles of the Ahmed body. Significant drag reduction was observed for 25˚ rear slant angle, due to reduced recirculation region behind the model. [3]

Shahrin Hisham Amirnordin et al., (2010). "Numerical Analysis on the Effects of Air Flow in the Wake of a Large Vehicle on Trailing a Passenger Car", Proceedings of MUCET 2010. [4]

Shahrin Hisham Amirnordin et al. (2010) performed a study which focusses on the drag and lift forces on a hatchback car trailing a large vehicle. He studied the effect of distance in the drag of a body at a Reynolds number of 3.65x105. He found out that aerodynamically optimum critical distance was between 4 to 5 m. Martin Bjerling et al (2011) conducted large eddy simulation (LES) on a vehicle platoon of four boxes.[4]

Mohamed-Kassim, Z., Filippone, A. (2010) Fuel Savings on a Heavy Vehicle via Aerodynamic Drag Reduction. Transp. Res. Part D Transp. Environ., 15(5):275–284 [5]

Muhammed-Kassim and Fillippone conducted a numerical study of fuel saving potential of various drag reducing retrofits. They found that vehicle parameters alone do not affect total drag; operational parameters have a large effect as well. It was found that using aerodynamic devices is optimal when these vehicles travel at high speeds on long haul driving cycles, as weight does not have any direct effect on drag. Driving through urban areas utilizes most fuel for acceleration and deceleration. These factors are important because every part of a truck contributes, positively or negatively, to the total drag of the truck. Commonly used retrofits are given in the following subsections. [5]

Lancer, W. R., Ross, J., Kaufman, A. E. (1991) Aerodynamic Performance of a Drag Reduction Device on a Full-Scale Tractor/Trailer. Aerospace Technology Conference and Exposition, p11 [6]

Lancer et al. studied the effectiveness of boat tailing on a tractor/trailer at NASA Research Centre wind tunnel. It was concluded that the modified flow due to the boat tails increased the pressure over the base area, hence reducing the aerodynamic drag. Khalinghi et al. extended the work done by Lancer and his team by doing the computational and experimental study on bodies with and without the drag reducing device. The computational fluid dynamics (CFD) simulations were done using both steady and unsteady Reynolds-Averaged Navier-Stokes (RANS) approach. Hot wire anemometry and Particle Imagine Velocimetry (PIV) were used to understand the wake flow better. It was found that the drag coefficient reduced by 6% for steady and 18% for unsteady CFD simulations; as compared to the experimentally measured value of 20%. They concluded that unsteady RANS is more valid and useful in studying wake flows. [6]

Modi, V. J., Hill, S. S., Yokomizo, T. (1995) Drag Reduction of Trucks through Boundary Layer [7]

Modi et al. did wind tunnel and full scale road tests to see the effect of 'fences' (boundary layer control method) on drag reduction. Wind tunnel testing done on a 1/6th scale model at subcritical Re showed drag reduction of 31.4% while the road tests showed the drag reduction of only 16.6%. The discrepancies in results were attributed to the much idealized circumstances in wind tunnel as compared to the roads; the wind tunnel model was not exactly same as the full scale model; the scaling of fence and their orientation for full scale trucks were not accurate; and that the number of fences used on the wind tunnel model and the full scale trucks were not the same. Also, it was found that the effectiveness of the 'fences' reduced at speeds lower than 30 km/h. The optimization in terms of location of fences was left as future work. [7]

Bruneau, C H., Mortazavi, I., Gilliéron, P. (2008) Flow Regularisation and Drag Reduction [8]

Bruneau et al. presented a new passive flow control method using a porous layer. In their paper, they explained the solid-porous-fluid physical models used; the setup of the boundary conditions for such flow; integration of Darcy equations with Navier Stokes equations and a penalization method applied. Numerical simulations were done to understand the passive flow control capability of porous layer at high Reynolds number and it was found to reduce vortex induced vibrations. Based on the location of porous layer on an Ahmed body, it was found that drag coefficient could be reduced by 40%. However, the drag reduction was not very significant in case of Ahmed body with a rear window. Bruneau et al. found that using a porous layer on the roof of the body modifies the shear forces, reducing the drag by 22%. The porous layer induces the Kelvin Helmotz instabilities which change the size of vortices on the top of the body, hence reducing the drag. However, their study was performed only on 2D body and concluded that more detailed understanding of vortex dynamics is required to be able to modify it substantially.

Chen et al. did a numerical study to see the effect of bionic riblets on drag reduction of a revolving body. It was found that the drag could be reduced by almost 9%. The basic reason of drag reduction was found to be reduction in skin friction drag due to control and correction of boundary layer. Sirovich and Karlson showed that by having ‘vee’ shaped protrusions, drag can be reduced by almost 10%. However, studying various protrusion heights, ‘vee’ angles and their effects was left as future work. Parker and Sayers [19] found that symmetric V groove riblets on a flat plate perform better at reducing drag than unsymmetrical U groves. These grooves suppress the streamwise vortices, turbulent mixing and hence turbulent shear stress. [8]

Park, H., An, N. H., Hutchins, N., Choi, K.-S., Chun, H. H., Lee, I. (2011) Experimental Investigation on the Drag Reducing Efficiency of the Outer-layer Vertical Blades. J. Mar. Sci.Technol., 16(4):390–401 [9]

Park et al. proposed a new passive control method to reduce form drag over 2-D blunt bodies. They found that by attaching small tabs on the trailing edge of a body, flow characteristics in wake changes, leading to the drag reduction. Both experimental and computational studies were done to comprehend the efficiency in reducing drag. Parametric studies were done by varying spacing, height and width of tabs at three different Reynolds numbers. It was found that the tabs reduce the strength of the vortices in the wake, leading to increase in base pressure (decreased drag). Norman and McKeon studied the effect on a stud (tab) on an otherwise smooth sphere using PIV (Fig. 3). It was found that in subcritical regime, with an isolated roughness added, the flow separation was delayed further downstream. The stud seems to increase the Reynolds stresses that lead to a decrease in the shear layer instabilities and hence, the narrower wake. However, in supercritical regime, it seems that the stud causes premature separation and hence, an increase in drag. The researchers also studied the effect of the size of stud (tab). Though only cylindrical tabs were used, the effect of other geometries was left as future work.[9]

OBJECTIVE

1. Find out FD and CD of simple truck model at various velocity of wind.
2. Improvement of aerodynamic shape of the truck model by drag reduction techniques and reducing the drag force as well as drag coefficient.

DESIGN & ANALYSIS

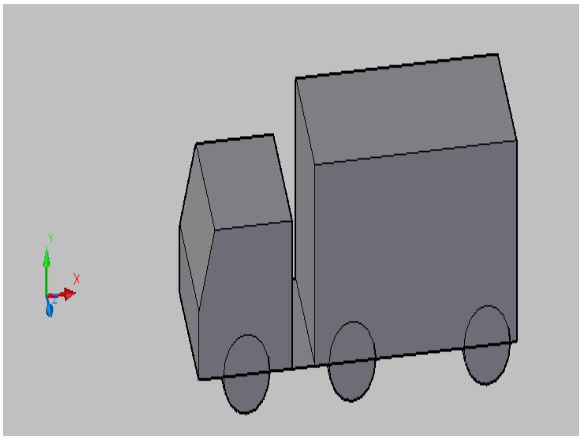


Fig.3 simple model of truck[11]

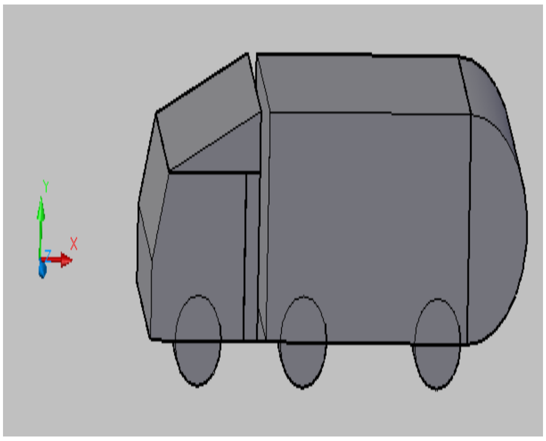


Fig. 4 modified truck after change shape[11]

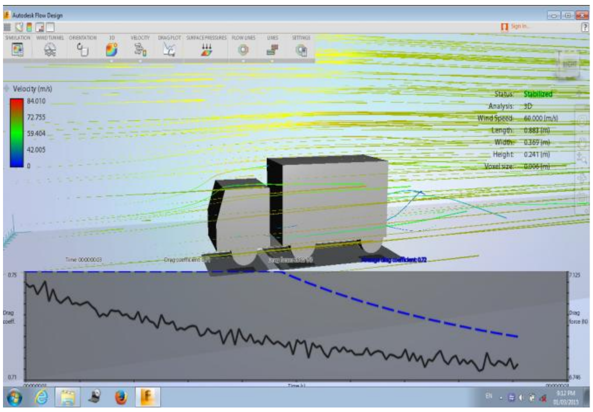


FIG.5 C_D & F_D at 80 km/h before shape change[11]

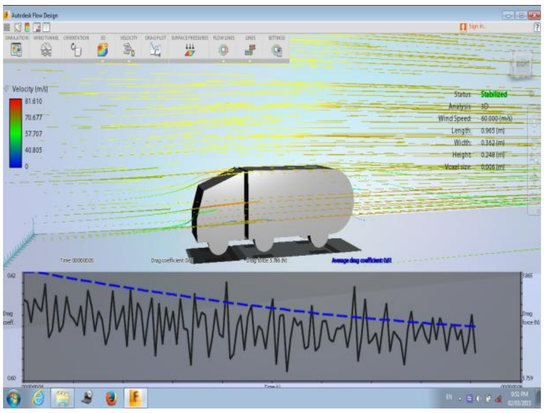


FIG.6 C_D & F_D at 80 km/h after shape change[11]

RESULT AND DISCUSSION

Table 1

Velocity of wind (km/h)	F_D (Before change in shape)	F_D (After change in shape)
60	6.802 N	5.786 N
80	12.272 N	10.368 N
120	19.389 N	16.203 N

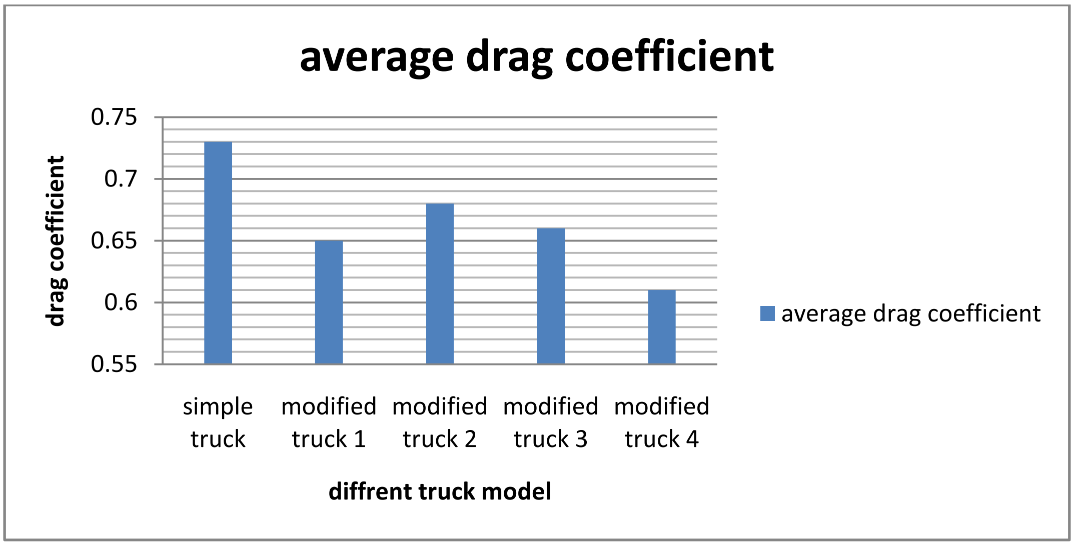


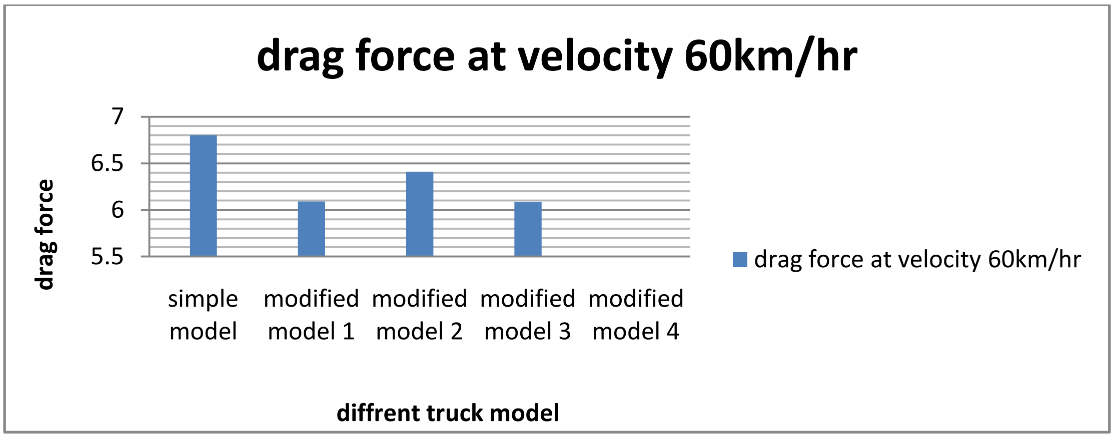
Fig 7

Table 2

Model	Drag coefficient, C_D
Simple model	0.73
Modified model	0.61

MANTECH Publications

Journal of Advance Manufacturing System and Technology
Volume 1, Issue 1, January-June, 2016



CONCLUSION

From the above table and graph, we show the difference of drag force as well as drag coefficient before and after the change of aerodynamic shape of Truck. The reduction of drag force as well as drag coefficient is about 12 to 15% from the above analysis.

We have completed successfully the analysis of truck in 3-D by using CFD software Autodesk Flow Design. By taking in account the boundary conditions; velocity, pressure, density of air and the frontal area of the truck, we can design the new shape with the help of drag reduction techniques of the truck in 2D as well as in 3D in the future. Thus, we will be able to produce the vehicle's body shape with optimum frontal area that will offer less resistance to the air moving in the opposite direction.

REFERENCES

I. Han T., (1989), "Computational analysis of threedimensional turbulent flow around a bluff body in ground proximity", AIAA J. 27(9) 1213- 1219

II. Sanisa K. and Davidson L., (2003). "Numerical Study of the Flow around a Bus-Shaped Body." Transaction Journal of the ASME.Vol.125

III. Gokul Krishnan Rajamani, (2006), "CFD Analysis of Air Flow Interactions in Vehicle Platoons", School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, August 2006

IV. Shahrin Hisham Amirnordin et al., (2010). "Numerical Analysis on the Effects of Air Flow in the Wake of a Large Vehicle on

Trailing a Passenger Car", Proceedings of MUCET 2010

V. Mohamed-Kassim, Z., Filippone, A. (2010) Fuel savings on a heavy vehicle via aerodynamic drag reduction. Transp. Res. Part D Transp. Environ., 15(5):275–284

VI. Lancer, W. R., Ross, J., Kaufman, A. E. (1991) Aerodynamic Performance of a Drag Reduction Device on a Full-Scale Tractor/Trailer.Aerospace Technology Conference and Exposition, p11

VII. Modi, V. J., Hill, S. S., Yokomizo, T. (1995) Drag reduction of trucks through boundarylayer

VIII. Bruneau, C H., Mortazavi, I., Gilliéron, P. (2008) Flow Regularisation and Drag Reduction

IX. Park, H., An, N. H.,Hutchins, N., Choi, K.-S., Chun, H. H., Lee, I. (2011) Experimental investigation on the drag reducing efficiency of the outer-layer vertical blades. J. Mar. Sci.Technol., 16(4):390–401

X. Wikipedia – The free encyclopedia

XI. www.autodesk.com

