International Journal of Engineering Research ISSN:2319-6890)(online),2347-5013(print)
Volume No.4, Issue No.11, pp : 613-617 01 Nov. 2015

Aerodynamic Simulation of A Truck To Reduce The Drag Force

[1]Shreenidhi R Kulkarni, [2]Abhishek Kothari, [3]Chetan Pavate, [4]Chinmay Dodmani

Dept. of Mechanical Engg, KLS GIT, Belgaum, Karnataka, India
. srkulkarni@git.edu,abhikothari28@gimail.com, chetikiip@gimail.com,chinmaydodmani@gimail.com

Abstract: This paper deals with aerodynamic simulation of an existing Tata 1613 truck in ANSYS FLUENT. The original truck was modelled in CATIA V5 analyzed for coefficient of drag and hence the drag force and fuel consumption was determined. The truck was then modified and reanalyzed for the same. The results seem promising and a reduction in drag force and hence fuel consumption was reduced by 43%.

Key words: Coefficient of Drag, Drag Force, Fuel consumption, Fuel economy, Aerodynamics of Truck/

1. Introduction

Aerodynamics is the study of effect of air flow around an object. The effect is studied considering one of the two cases namely stationary air and moving object or moving air and stationary object. Generally the parameters under considerations are coefficient of drag and coefficient of lift by which drag force and lift force can be calculated respectively. In air borne vehicles, the aim is to increase the lift force and in land borne vehicles the aim is reduce the drag force. In our work we consider one such land borne vehicle that is a truck. For the progress of developing nation like India, logistics is one of the key factors. Through decades, better aerodynamically designed trucks have evolved but the new aerodynamically designed trucks turn out to be expensive with respect to local buyers and small scale transportation companies. This calls for an aerodynamic study of an existing truck model so as to improve the model design by making small modification for better performance. Hence we choose TATA 1613 truck model for our analysis. Lot of work has been done related to this field. Different methods have been used and suggested to reduce the drag force in land borne vehicles. Kalpesh Vaghela [1] has optimized the roof fairing angle to reduce the drag force for heavy duty trucks by conducting tests using suction type wind tunnel. The tests were conducted for 10°, 13°, 17.5° and 20° angle of roof fairing and the drag reduction was more for 17.5° as compared to other the angles.

Harun Chowdhurt et.al [2], have conducted a series of tests on a semi trailer truck for different combination of vehicle speed and yaw angles on a baseline truck with various attachments. The results show an increase in drag force for yaw angle up to 5° and thereafter decrease in drag force. It is also shown that using only front fairing the drag can be reduced up to 17%. Further reduction of drag up to 26% is achieved by covering various gaps in the truck.

Subrata Roy and Pradeep Srinivasan [3] have carried out a fully three dimensional near field flow on a truck like bluff body (unmodified and modified) for CFD analysis. The theoretical modeling is done using Navier Stokes equation and k-ε turbulence model. The authors have also explained convergence of results and its importance. Two cases of wind flow have been considered that is normal flow and cross flow. Results show 30% drag reduction. An empirical equation to determine fuel consumption using drag force is also presented. It should be noted that the authors suggest use of more practical truck model so that the results can be commercially useful.

Chaitanya Chilbule et. al [4], have carried out numerical analysis on a basic truck model with trailer using ANSYS FLUENT. The modified model of the base truck consists of wind deflector and six vortex trap panels giving a reduction of drag by 5% and 3% respectively. A total of 21% drag reduction was estimated. Shear Stress Transportation (SST) turbulence model was used for analysis. The paper also talks about the fuel economy of 4 litters due to reduction of drag force obtained using an empirical relation developed by Subrata Roy and Pradeep Srinivasan in their paper [3] by conducting experiments on diesel powered engine. This equation will used in our work to find the fuel economy. The model was also analyzed by including rare view mirrors which increased the drag by 3.22%.

Xiaolong Yang and Zihui Ma [5], have studied two passive methods for drag reduction for the truck model that is attaching fairwater and cylinder to the truck. In fairwater method the shrink angle was varied and analyzed and in cylinder method the diameter of cylinder was varied. The results show that the drag force was reduced by 22.8% and 7% respectively. Keeping the shrink angle constant the cylinder diameter was varied the drag force optimized and the total drag reduction obtained 24.8%. The authors have also explained the cause of the pressure distribution on the face of the truck.

K. Durga Priyanka and Dr. B. Jayachandraiah [6] have done CFD analysis on a heavy truck vehicle. The analysis was carried out a baseline truck model and truck with add on that is a spoiler, for different speeds. Results show decrease in drag coefficient. The authors also show that cross wind increases efficiency of truck.

It can be noted that most of the time for CFD analysis, a simplified model of the truck is used. In this paper we have modelled a more realistic truck model (Tata 1613 truck) and analyzed for aerodynamics. Our intention is to find coefficient of drag, drag force and hence fuel consumption of the original and the modified truck model, and to compare the obtained results. If the considerable reduction in drag force and fuel consumption is obtained due to modification in truck then recommend the modification to the local truck body builders.

2. The original truck model

To model the Tata 1613 truck CATIA V5 modelling software was used. The dimensions of the truck were measured on real truck approximately and model was built. For our analysis only the truck coach that is the driver cabin is considered as it is this cabin that experiences air resistance first. The overall dimensions of the truck cabin are:

Width of the truck cabin including fillets at the sides: 2440mm.
Fillet radius on the side: 100mm.
Height of the truck cabin including wooden board on top at the roof: 2775mm.
Height of the wooden board on top of cabin: 500mm.
Length of the truck cabin: 2060mm.
The 2D wire frame and 3D model of the truck are as shown in figure 1 and figure 2 respectively. It should be noted that the rare view mirrors are not included in the model

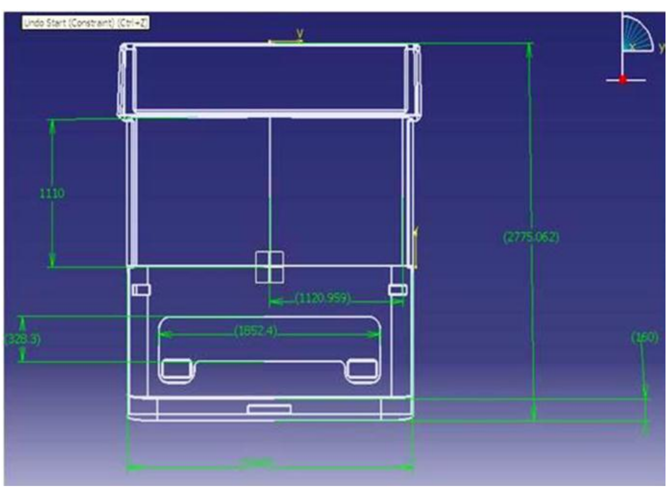

Figure 1:2D wireframe model of the original truck

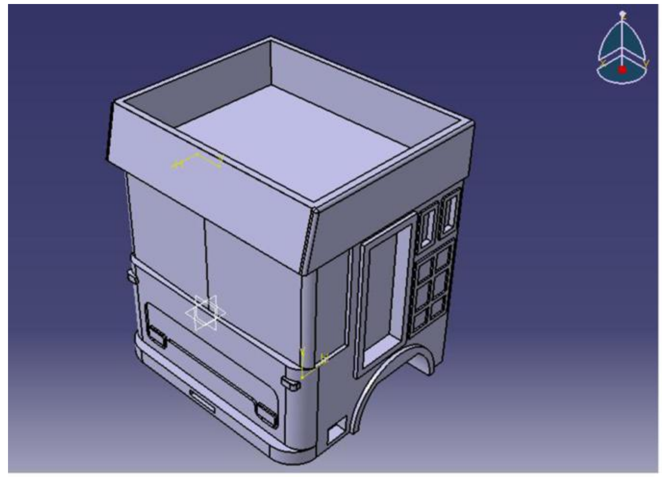

Figure 2: 3D model of the original truck

It can be seen from figure 2 that a pocket is formed on top of the roof by the wooden board surrounding the cabin. The space thus formed on top is generally used by the drivers for storage purposes. This truck model was created in CATIA V5 and was imported to ANSYS FLUENT by converting the part file in IGES format.

3. CFD Analysis

The truck model shown in figure 2 was imported in ANSYS FLUENT. A control volume or enclosure was created around the truck model by using enclosure option available in tools menu. It should be noted that the enclosure should be a minimum of 2-3 times the size of the target body (which in our case is the truck cabin) in all directions except on the bottom side where the truck is contact with the road. The truck cabin was then subtracted from the control volume using Boolean operation option. An automatic mesh was generated and then refined for element size. The mesh generated thus is unstructured. The inlet, outlet, road and wall parameters were defined. The boundary conditions were then defined along with properties of air. The boundary conditions, properties of air and few other important parameters are as stated below.

- Inlet Boundary conditions
  Turbulence intensity = 1% Viscosity ratio = 10%
  Air inlet velocity = 14 m/sec (50Kmph approx.)
  Velocity specification method = Magnitude and Direction
- Outlet boundary conditions
  Turbulence intensity = 5% Viscosity ratio = 10%
  Pressure= 0Pa
  Density of air= 1.225 kg/m^3

- Wall conditions
  Side and upper wall= No shear force
  Road= No slip
- Other parameters
  Turbulence model= k-ε
  Solver type= Pressure based
  Solution method= Pressure velocity coupling, 2^{nd} order upwind iterations
  Solution initialization type= Hybrid

With these parameters and boundary conditions the analysis were carried out. The obtained results are discussed later in this paper.

4. Modified truck model

The modified truck model is as shown in figure 3.

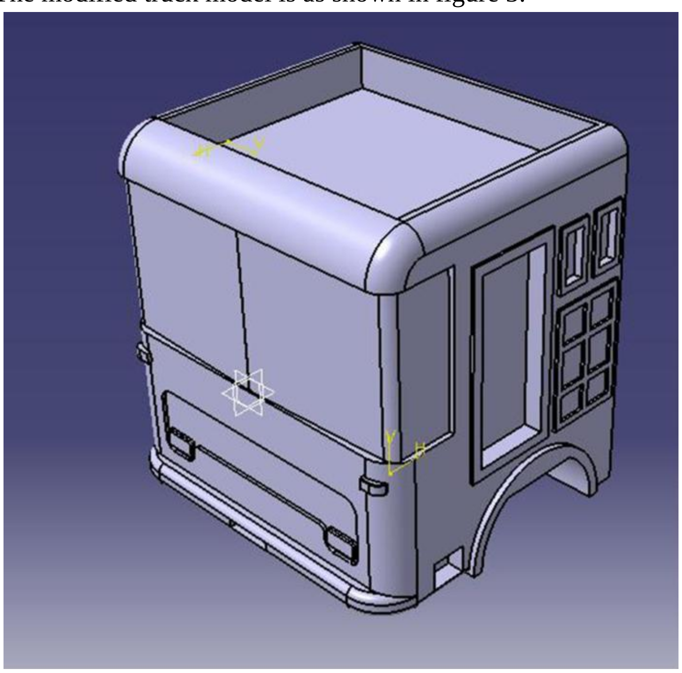

Figure 3: Modified model of the truck

The shape of the flat wooden board which forms the pocket on top is modified by rounding the board in front and on sides. Also the height of the wooden board is reduced by 200mm. this results in change in projected area of the truck when compared to original model of the truck. This modified model is imported to ANSYS FLUENT and analyzed for the aerodynamics. It should be noted that the method and procedure, properties of air, boundary conditions and rest of the parameters remain same as in case of analyzing original truck model. The obtained results for both original and modified truck model are presented in next section.

5. Results and Discussion

The Drag coefficient, Velocity streamline plot and Pressure distribution plot obtained for both original and modified truck model are as shown below.

Drag coefficient for original truck model: 0.85

Drag coefficient for modified truck model: 0.52The figure 4 and figure 5 below show velocity streamline plot and the pressure distribution plot for original truck model.

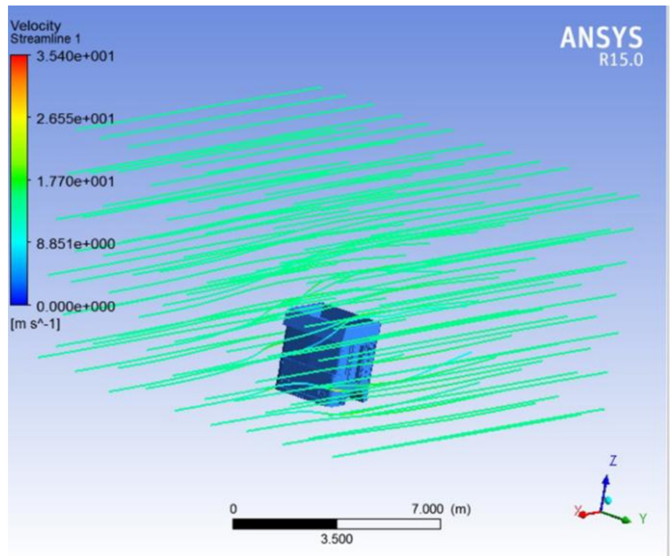

Figure 4: Velocity streamline plot for the original truck model

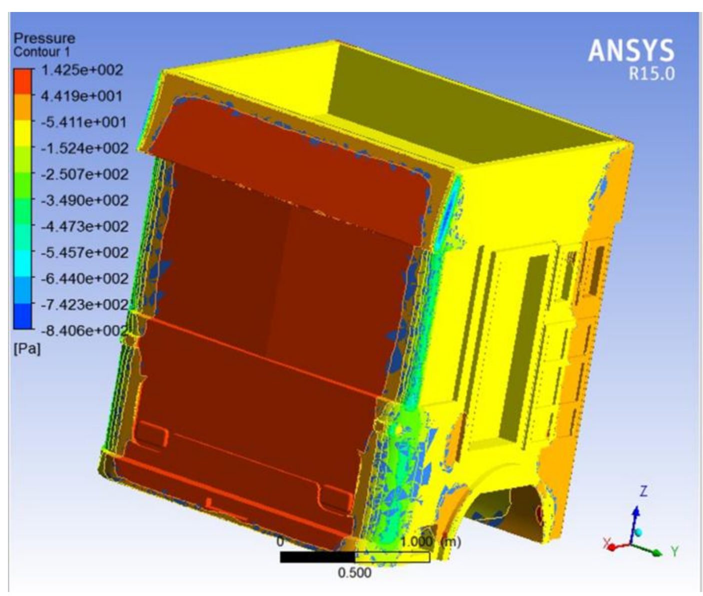

Figure 5: Pressure distribution plot for original truck model

The figure 6 and figure 7 below show the velocity streamline plot and the pressure distribution plot for modified truck model.

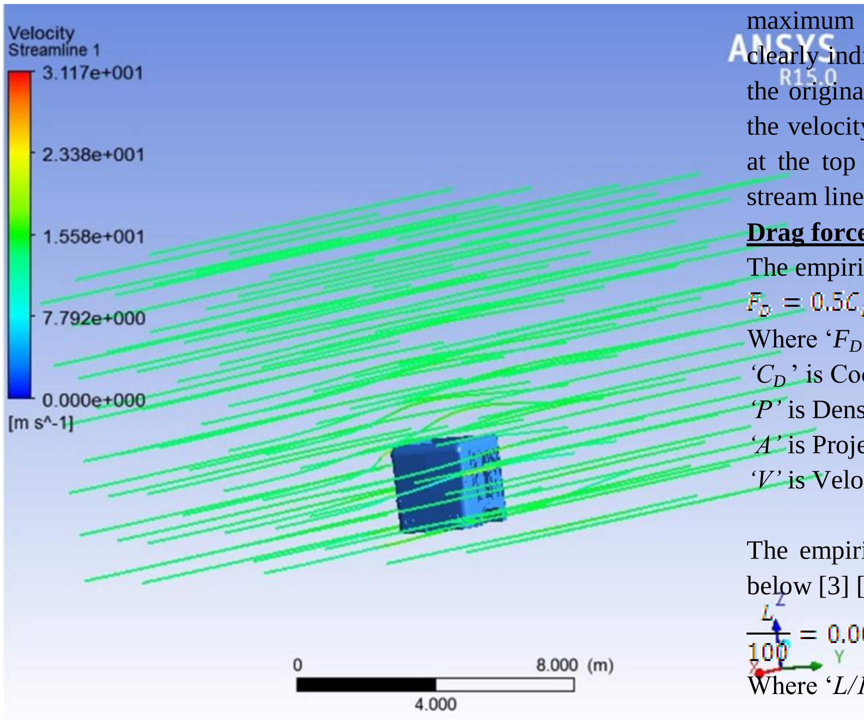

Figure 6: Velocity streamline plot for the modified truck model

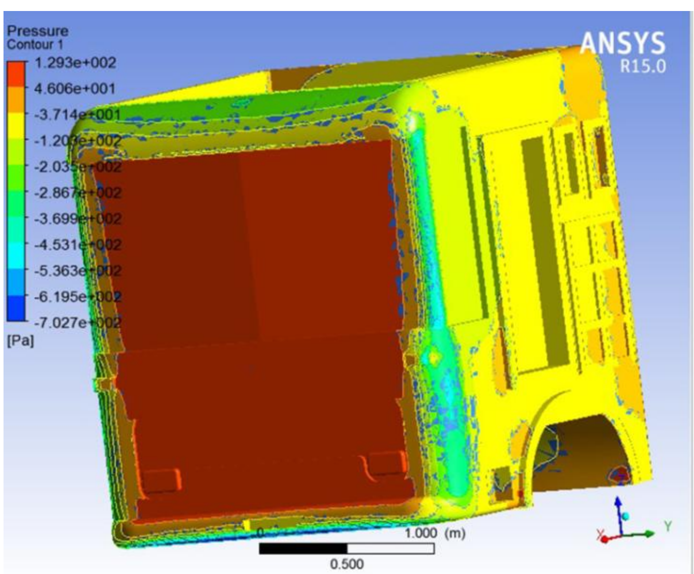

Figure 7: Pressure distribution plot for the modified truck model

It can be seen from figure 5 that the maximum pressure on the face of the truck is higher when compared with the maximum pressure of modified truck model in figure 7.This clearly indicates decrease in pressure due to the modification in the original truck model. It can also be seen from figure 6 that the velocity lines of modified truck model are more streamlined at the top near the wooden board when compared to velocity stream lines of original truck model in figure 4.

Drag force and fuel consumption

The empirical formula for the drag force is as given below.

$$F_D = 0.5C_D\rho AV^2 \quad \dots \quad \dots \quad \dots \quad (1)$$

Where ‘F_D ‘ is Drag force in *N*

‘C_D’ is Coefficient of Drag

‘*P*’ is Density of fluid (air) in *kg/m*^{3}

‘*A*’ is Projected Area in *m*^{2}

‘*V*’ is Velocity in *m/s*

The empirical relation to determine fuel consumption is given below [3] [4].

$$\frac{L}{100} = 0.008051F_D \quad \dots \quad \dots \quad \dots \quad (2)$$

Where ‘*L/100’* is the fuel consumed in Litters per 100 *Km*

- Original truck model
  The drag force is obtained by substituting the following values in equation 1.
  $C_D = 0.85$ $P = 1.225\ kg/m^3$
  $A = 2.775*2.44 = 6.771m^2$ $V = 14m/s$
  The Drag force thus obtained is $F_D = 690.929N$
  Fuel consumption is obtained by substituting the obtained drag force into equation 2. The fuel consumed is *L/100= 5.56*
- Modified truck model
  The drag force is obtained by substituting the following values in equation 1.
  $C_D = 0.52$ $P = 1.225\ kg/m^3$
  $A = 2.575*2.44 = 6.283m^2$ $V = 14m/s$
  The Drag force thus obtained is $F_D = 392.222N$
  Fuel consumption is obtained by substituting the obtained drag force into equation 2. The fuel consumed is *L/100= 3.15*

Net decrease in drag force= 43.23%
Net fuel saved-= 2.41 litters per 100 Km
A net reduction of 43.23% drag force and 2.41 litters of fuel economy is obtained which seem promising and notable

modifications. It should be noted that the height of the wooden board and hence the height of the truck was decreased.

6. Conclusion

The results obtained seem promising and the modification can be recommended to the local truck body builders. It can be seen that 2.41liters of fuel can be saved which surely will contribute towards green revolution. Further the work can be extended by carrying analysis on truck-trailer assembly including mirrors for varying speeds and wind directions.

References

i. Kalpesh Vaghela, “Optimization of Roof fairing angle to reduce the Aerodynamic Drag of Heavy duty truck” International Journal of Emerging Technologies in Computational and Applies Sciences, 13-322; © 2013.

ii. Harun Chowdhury et. al, “A study on aerodynamic drag of a semi-trailer truck” 5th BSME International Conference on Thermal Engineering, 1877-7058 © 2013.

iii. Subrata Roy and Pradeep Srinivasan, “External Flow Analysis of a Truck for Drag Reduction”, Society of Automotive Engineers, Inc, 2000-01-3500.

iv. Chaitanya Chilbule et. al, “Analyzing the profile modification of truck-trailer to prune the Aerodynamic drag and its repercussion on fuel consumption”, 12th Global Congress on Manufacturing and Management, GCMM 2014.

v. Xiaolong Yang and Zihui Ma, “Drag Reduction of a Truck Using Append Devicesand Optimization”, ParCFD 2013, CCIS 405, pp. 332–343, 2014.

vi. K. Durga Priyanka and Dr. B. Jayachandraiah, “Aerodynamic Improvement of a Truck Body by using CFD”, International Journal of Emerging Technology in Computer Science & Electronics ISSN: 0976-1353 Volume 13 Issue 2 – March 2015.