CFD analysis of aerodynamic drag reduction in heavy vehicles by changing its frontal area

Yuvaraj C,
Bcrows Technologies, Chennai, India
Gopinath S*
School of Mechanical Engineering, Lovely Professional University, Phagwara, India

Abstract

Now days, heavy-truck fuel quality has been an overwhelming concern for production and construction engineers. The best way to lower fuel consumption is to boost vehicle aerodynamic efficiency. This can be accomplished by reducing the drag, as the drag coefficient is directly proportional to the consumption of fuel. Design engineers attempting to improve the performance of the heavy vehicle by manipulating different parameters such as engine parameters, weighing, rolling resistance and aerodynamic drag. Efforts were made in this project to increase aerodynamic performance by changing the container's frontal area. Computational modeling was conducted at different speeds (50km / hr, 60km / hr, and 70km / hr) by modifying the container's frontal area in heavy vehicles. Specific truck geometries were made using CATIA V5 and the simulations were made using the software program ANSYS CFX. Results were obtained and comparative studies were made. As a result of comparisons between various designs, the cowl of 2h dimension shows better results in reducing the drag when compared with the other designs.

Key words: Cowl dimension, drag coefficient, frontal area, aerodynamic performance

Introduction

There's a increase in demand these days to reduce the consumption of car petrol. Heavy heavy vehicles such as buses have low fuel economy because of the high aerodynamic drag. New research has shown aerodynamic engineering to be one of the most efficient means of reducing fuel consumption. For a big car travels at a speed of 100 km/h so the aerodynamic drag would be decreased by 50 percent of the overall power.

In his thesis, Azim, A.F.A, outlines the study of the aerodynamic resistance of the truck and points out the drag coefficient values without adding any add-ons and add-ons, and finally finds that add-ons help increase the fuel performance by reducing the drag of the vehicle. Strain, size, path lines, streamlines, and coefficient of drag values were found with support in this study. Richard M. Wood et al., Bauer suggested the production of some aerodynamic drag reduction machines which are simple and low cost. The increase in fuel consumption has everything to do with the drag reduction. [1], in their work. Prakash explained about fuel-reduction technologies. With trucks it is considered that aerodynamic enhancement is one of the most successful techniques when it comes to saving fuel. Specific aerodynamic trailer structures and aerodynamically formed trailers have been studied computationally to analyze their effect on vehicle traffic. Comparisons were made using flow transfer between the vehicle, the trailer, the tractor frame and the cab undercarriage. Abdel Latif, A Toro has sought to reduce the drag force put on tractor trailers by modifying the rear portion of the carrier trailer. Three different design changes were tested, observed and even the total reduction in drag force was assessed. The calculated drag coefficient values can be compared with contrasts between the Reynolds number and the model in the real world. Noger, et al explored the impact of experimentally and in theory changing the aerodynamic shape of a tractor trailer sized model on its aerodynamic drag. Using modification added to the body of the vehicle, they also enhanced its aerodynamic properties. Sun, et al proposes an effective numerical model for obtaining the flow structure and the forces surrounding a Spoiler passenger vehicle based on the Computational Fluid Dynamics (CFD) method. Spoiler deployment results in a reduction in head-on wind of
2.02 per cent drag coefficient and 14.06 per cent lifting coefficient according to their report. The drag force can then be limited by using add on vehicle equipment. We can also improvise the fuel economy; it can make a passenger car more efficient.

II. Modeling and Simulations:

The heavy truck is built using CATIA V5 3D modeling tools. The truck was initially built with no modifications. Finally, we made the modified truck designs for our comparative analysis.

The model was imported to ICEMCFD software for meshing. Meshing was unstructured because of the complexity of truck model.

The meshed model was imported to CFX and the boundary conditions were given. The analysis was carried out at different velocities such as 50km/hr, 60km/hr, and 70km/hr by changing the frontal area of the container in heavy vehicles. The frontal area is modified in reference with the “h” value of the carrier.

III. Results and Discussions

The result obtained from the base truck was used for the comparative analysis of modified truck models. The pressure contour, velocity contour, turbulent kinetic energy, streamline pattern was observed for each case. The analyses were done for three different velocities 50 km/hr, 60km/hr and 70 km/hr.
Figure 3: Pressure contours for different trucks at velocity V=50km/hr
Pressure Contour at velocity 60 km/hr:

Figure 4: Pressure contours for different trucks at velocity V = 60 km/hr

Pressure Contour at velocity 70 km/hr:
Figure 5: Pressure contours for different trucks at velocity $V=70$ km/hr

Velocity Contour at velocity 50 km/hr:

- Without cowl
- With 0.5 h
- With 1h
- With 2h

Figure 6: Velocity contours for different trucks at velocity $V=50$ km/hr
Figure 7: Velocity contours for different trucks at velocity V=60 km/hr

Figure 7: Velocity contours for different trucks at velocity V=70 km/hr
Turbulent Kinetic Energy Contour at velocity 50 km/hr:

Without cowl

With 0.5 h

With 1h

With 2h

Figure 8: Velocity contours for different trucks at velocity V=70km/hr

Figure 9: Turbulent Kinetic Energy contours for different trucks at velocity V=50km/hr
Turbulent Kinetic Energy Contour at velocity 60 km/hr:

- Without cowl
- With 0.5 h
- With 1 h
- With 2 h

Figure 10: Turbulent Kinetic Energy contours for different trucks at velocity V=60km/hr

Turbulent Kinetic Energy Contour at velocity 70 km/hr:

- Without cowl
- With 0.5 h
- With 1 h
- With 2 h
Drag Coefficient:

Our project’s key goal is to decrease the drag coefficient. For each case the drag coefficient value was determined and the diagram was drawn between the values of the velocity and the drag coefficient. It is well known from the graph that the surface adjustment will reduce the drag-coefficient values.

![Figure 11: Turbulent Kinetic Energy contours for different trucks at velocity V=70km/hr](image)

In the graphical representation, the velocity is considered along the horizontal axis and the drag coefficient values are considered along the vertical axis. From the graphical representation it is clearly understood that the drag coefficient value is greater for the base model without cowl and it decreases for modified truck models.

IV. Conclusions

This research is an effort to validate the probability of developing the aerodynamics of a truck that reduces fuel consumption in return. Changes in truck design may increase the aerodynamic properties. Aerodynamic properties will naturally reduce the drag that occurs in the vehicles. The analyses were made for four different designs including the base model at three different velocities. The pressure distribution, velocity distribution, turbulent kinetic energy distribution and the streamline pattern were observed at the end of the analyses. The model with 2h cowl shows efficient results in reducing drag when compared with the other models.

V. References