International Journal of Advance Engineering and Research Development

Investigation of Modified Profile of Heavy Commercial Vehicle to Prune Aerodynamic Drag

D. M. Ekade[1], C. V. Patil[2]

[1]*Department of Mechanical Engineering Shri Sant Gajanan Maharaj College of Engineering Shegaon*
[2]*Department of Mechanical Engineering Shri Sant Gajanan Maharaj College of Engineering Shegaon*

Abstract: *The aerodynamics study for a heavy vehicles and the resulting the design changes could improve the fuel efficiency that can be substantial over a long duration, as well as the vehicle dynamics. In this study, CFD simulations for a truck were performed with an objective to reduce the drag coefficient. Four different configurations were considered such as a rear slant from 1/3rd of the trailer length and the rear slant from 2/3rd of the trailer length from rear end and covering the gap between the tractor and trailer. The studies were performed for a speed range of 20 kmph to 120 kmph to include the effects of real life scenarios. ANSYS FLUENT was chosen for the CFD simulations. Reynolds Averaged Navier-Stokes (RANS) approach was applied for this study, with Shear Stress Transport (SST) k-ω turbulence model was selected for the simulations. From the CFD simulations, all those three configurations provide drag reduction in comparison with the base model. Drag was reduced by ~20% when the gap between the tractor and trailer is covered. Other configurations provide drag reduction around 7% and 4% at highway speeds. The base model results were in agreement with the experimental results from the reference journal.*

Keywords- Heavy Commercial Vehicle, Aerodynamic Drag, CFD simulation, Drag Reduction, RANS (Reynolds Averaged Navier Stokes)

I. INTRODUCTION

Heavy commercial vehicles are considered aerodynamically inefficient compared to other ground vehicles due to their un-streamlined body shapes. A large commercial vehicle travelling at 100 km/h consumes about approximately 52% of the total fuel to provide power to overcome the aerodynamic drag [1]. In contrast, a car under the same driving conditions, consumes approximately 4 times less power to overcome drag. Generally, a heavy commercial vehicle's annual mileage can vary between 1,30,000 km and 1,60,000 km. Therefore, any reduction of aerodynamic drag will result in huge fuel savings and reduction of greenhouse gas emission.
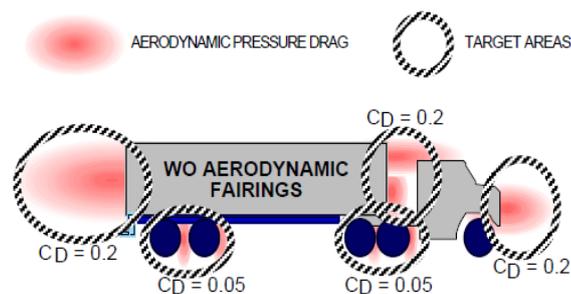

Fig1. The distribution of aerodynamic drag for standard class 8 trucks operating in zero crosswind condition.

Each area in Fig. 1 represents a portion of the total drag coefficient with respect to the impact on the pressure drag[6]. The target areas of the wheels equate to a Cd of 0.1. The gap between the tractor and the trailer is a 0.2 of the Cd. The frontal area is a 0.2 of the Cd. The area behind the trailer is the wake area, with a Cd of 0.2. The frontal area and the area behind the tractor trailer have the higher drag coefficients. The higher the drag coefficient, the higher the aerodynamic drag and vice versa as seen in the direct relationship between the two stated in Equation (1). The higher the drag force on the tractor-trailer, the more fuel is needed to move the vehicle. Therefore, reducing the drag coefficient will result in reduced fuel consumption.

Most of the class 8 trucks now a days are using external fuel saving devices without changing the projected frontal area of the truck, it is possible to modify the shapes of the truck including the container box in a more streamlined way. These external attachments can minimize aerodynamic drag based on their external shapes, sizes and placements. Aerodynamic drag (*D*) depends on the size of a vehicle (projected frontal area, *A*), the drag coefficient (*Cd*) which is a measure of the flow quality around the vehicle, and the square of the vehicle speed (*V*) as expressed in Eq. (1) where *ρ* is the air density.

$$D = 1/2 \; \rho \; V^2 \; C_d A. \tag{1}$$

II.   Literature Review

Harun chowdhury, Hazim Moria, Abdul Kadir Ali, Iftekhar Khan, Firoz Alam & Simon Watkins [1] investigated the impact of various fuel saving devices used in commercial vehicle. The aerodynamic drag on the baseline vehicle including different external attachments (i.e. front fairing, side skirting and gap filling)were measured for a range of vehicle operating speedsand yaw angles with different combinations. The reduction drag for combination of all attachments was about 26%.

Chaitanya Chilbule & Awadhesh Upadhyay [2] carried out numerical simulation on unmodified and modified profile. The modification are wind deflector on truck's cabin, vortex trap, miniskirt, vortex strake & aerodynamic revolute. The modification in the commercial truck resulted in 21% aerodynamic drag.

Fu Hung Hsu & Roger L. Davis [3] investigated for the drag reduction of a tractor trailer using the combined use of curved boat tail flaps with add on humps which had predicted drag reduction of 50.9%.

Christopher J. Roy & Jeffrey Payne [4] presented simulations for a flow over a simplified tractor trailer geometry at zero degree yaw angle.

Volodymyr Sirenko & Roman Pavlovsky [5] did the experimental investigation of a SUV in the wind tunnel Two passive devices rear screen which is plate behind the car and the rear fairing where the end is aerodynamically extended which reduced drag upto 6.5% for rear screen and 26% for rear fairing.

Richard M. Wood & Steven Bauer [6] presented three simple low cost aerodynamic drag reduction devices which are developed for the application to the trailer of the tractor trailer truck. These devices collectively reduced the drag by 30%.

Subrata Roy & Pradeep Srinivasan [7] tested the truck and other high sided vehicles without and with different deflector.

Cheng Jing & Wu Yun Zhu [8] studied different aerodynamic drag reduction devices (wind deflector & dome) for heavy commercial vehicle. The use of these devices reduced the aerodynamic drag by 10%.

Li Song & Zhang Ji Cheng [9] carried out numerical simulation of truck with different add on which reduced the aerodynamic drag by 8.9%.

III.   Problem Statement

A heavy commercial vehicle' aerodynamics that was studied [1] experimentally along with CFD simulations was considered for this work. In that work, the authors had investigated multiple drag reduction techniques such as the inclusion of Skirting to Truck, addition of fairing and filling gap between the cabin and the trailer. The present work will focus on reduction of drag using the modifications on the trailer geometry and gap covering between tractor and trailer.

In order to study the drag reduction, a total of four different geometrical changes to the truck were considered. These are listed below

1) Base Model
2) Configuration A – Introducing Rear Slant from $1/3^{rd}$ of the Trailer length
3) Configuration B – Introducing Rear Slant from $2/3^{rd}$ of the Trailer length
4) Configuration C – Covering the gap between the tractor and trailer

Any design changes to the Truck geometry must be functional for a wide range of operating conditions. So, the study had considered the flow speeds from 20 km/hour till 120 km/hour with an interval of 20 km/hour. Such a detailed study would result in the investigation flow physics for the entire operating range of a typical truck.

The project work was focused on numerical simulations based on CFD methods. Experimental studies were not considered for this project work and the same shall be taken up as a future work. All the CFD simulations were performed using ANSYS FLUENT, a finite volume based commercial CFD solver.
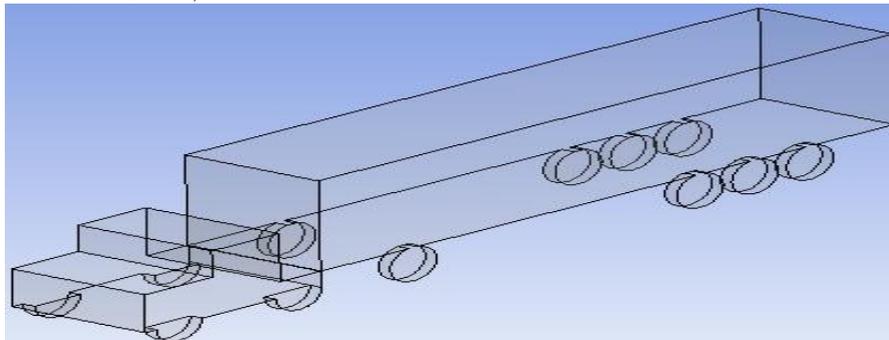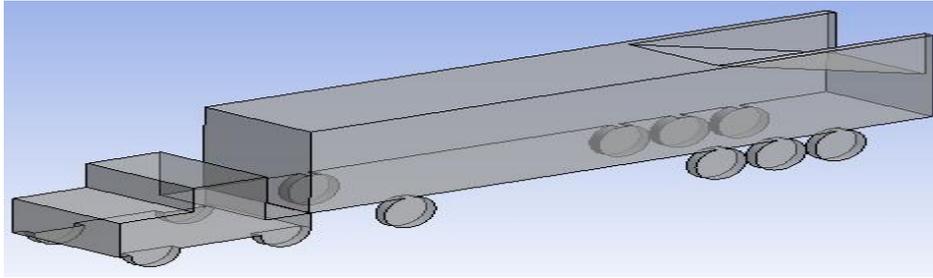
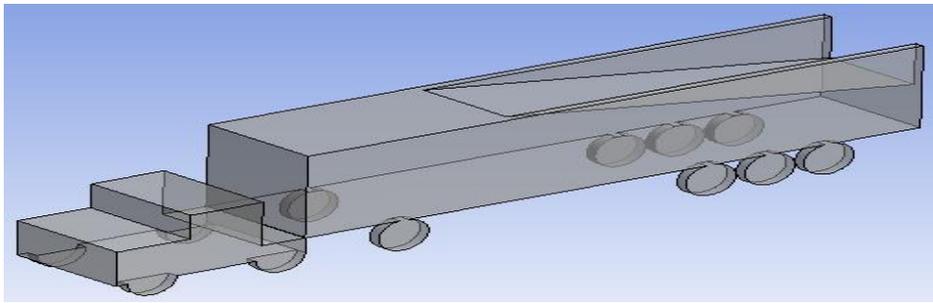

Fig.2 Base Model
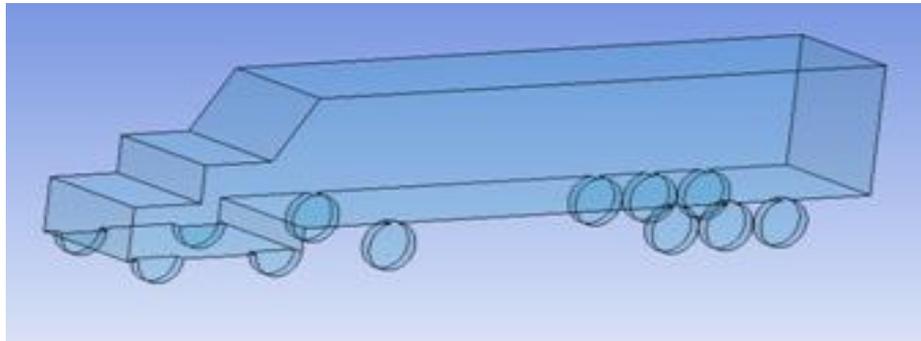
Fig.3 Configuration A

Fig.4 Configuration B

Fig5. Configuration C

IV.   CFD Methodology

ANSYS Design Modeler and ANSYS Workbench Mesher were utilized for the pre-processing activities such as geometry creation and meshing. Hexahedral mesh elements were generated for the computational domain with 10 layers of prisms for a total height of 3 mm to resolve the boundary layer effects in the simulations.
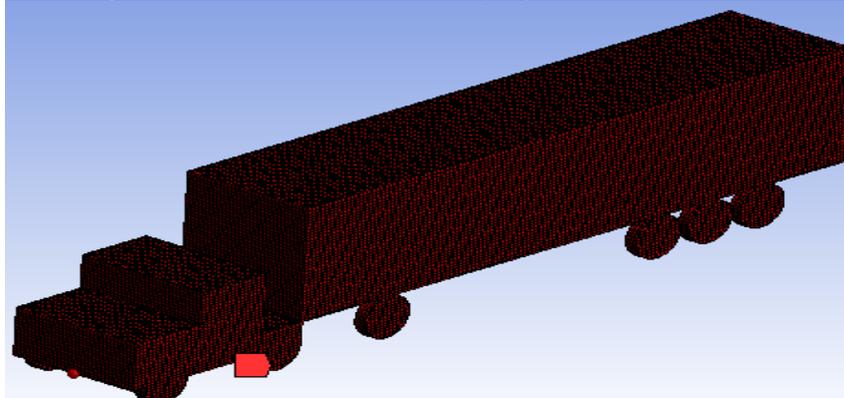

Fig.6 Meshing of Base model

The scope of the present work was limited to investigate the steady state external aerodynamics around the trucks. The transient effects and the un-steadiness can be considered for a separate research work in the future. So, the 3-Dimensional Steady solver in ANSYS FLUENT was used for these CFD Simulations. Reynolds Averaged Navier Stokes (RANS) approach with Shear Stress Transport (SST) k-ω turbulence model was selected for the simulations.

The fluid flow inlet face was modeled using the 'velocity-inlet' boundary condition with the corresponding truck speed. The flow leaving the computational domain was modeled using the 'pressure-outlet' boundary condition with the specification of gauge pressure (atmospheric pressure). The road surfaces along with the side and top surfaces of the computational domain were modeled as 'Adiabatic, Stationary walls with No-Slip conditions'. A similar wall setting was applied to the Truck's surfaces as well. The simulations were considered to be converged based on the governing equation residuals and the force balance over the truck surface, monitored for each iteration.

V. Results and Discussions

The drag coefficient for the base model was in the range of 0.85. From the authors [1] experimental results, there's a variation of ~6%. So, it can be concluded that this numerical results are validated.

Table1. Comparison of EFD and CFD in terms of drag coefficient

Speed	EFD	CFD
20	0.800	0.850
40	0.805	0.859
60	0.790	0.860
80	0.800	0.862
100	0.805	0.859
120	0.790	0.852
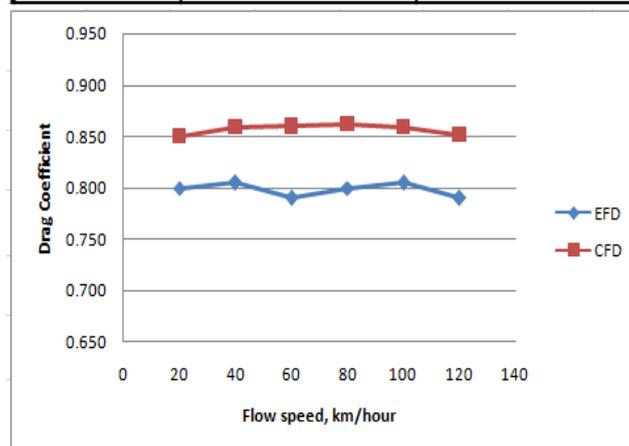
Fig.7 Graphical representation of EFD and CFD for base model

Now, the simulations with the truck geometrical modifications were carried out. The drag coefficient for each case had been plotted.

Table 2. Coefficient of drag value for different configurations

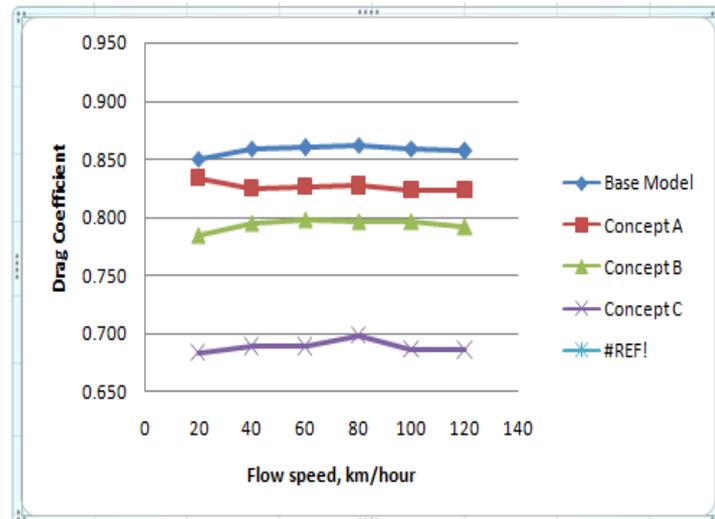| Speed	Base Model	Concept A	Concept B	Concept C
20	0.850	0.834	0.785	0.684
40	0.859	0.825	0.795	0.689
60	0.860	0.826	0.798	0.689
80	0.862	0.827	0.797	0.698
100	0.859	0.823	0.797	0.687
120	0.857	0.823	0.792	0.686

Fig.8 Graphical representation of drag coefficient for different configurations

All the three configurations reduce the drag in reference to the base model. However, the reduction for the gap coverage concept between the tractor and trailer results in mush lower drag among other configurations. The introduction of rear slant on the trailer geometry has a positive effect on drag reduction. This was evident from the drag coefficient curves.

Of these two configurations, the drag reduces when the rear slant had been introduced further upstream (configuration B). A set of velocity contours, with a range of 0 – 20 m/s, at the mid-section of the Truck had been provided for each case for studying the flow profile.
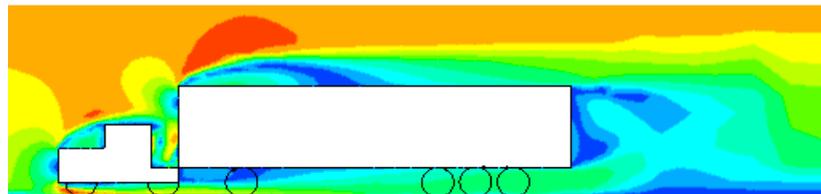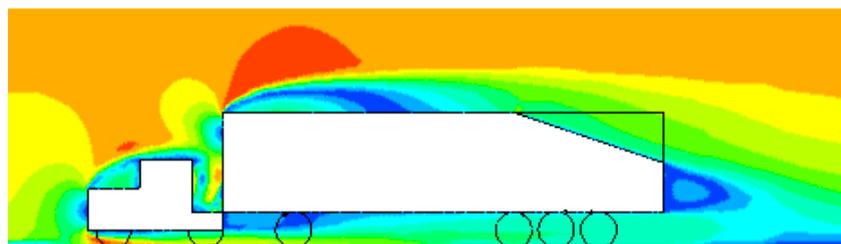

Fi.9 Base Model – Velocity Contours

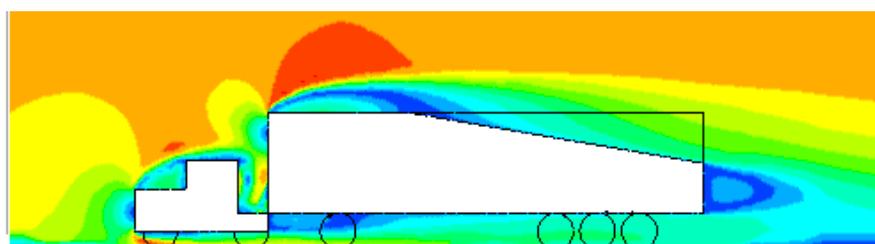Fig.10 Configuration A – Velocity Contours
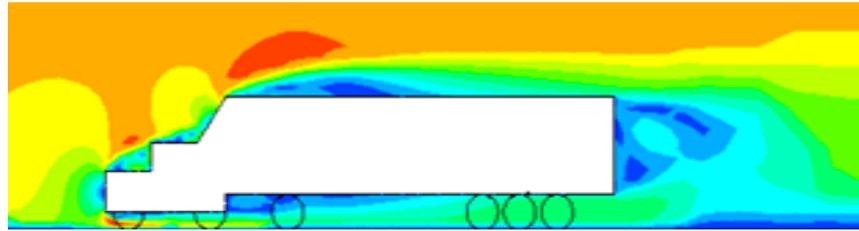

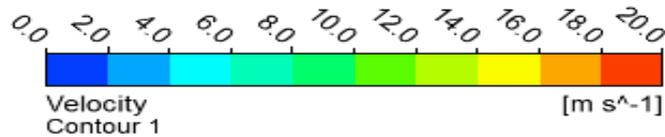Fig.11 Configuration B – Velocity Contours

Fig.12 Configuration C – Velocity Contours

VI. Conclusions

Drag reduction for a heavy commercial vehicle was investigated based on the computational fluid dynamics (CFD) methods.

1) The base model drag coefficients predicted in these CFD simulations were in line with the previous experimental results.

2) All the three suggested models provide reduction in drag coefficient. Among those, configuration C reduces the drag up to 20%

3) The introduction of rear slant at $2/3^{rd}$ from the rear end in the container also had a positive impact on the drag reduction. the drag values reduces up to ~7%.

4) The introduction of rear slant at $1/3^{rd}$ from the rear end in the container also had a positive impact on the drag reduction. the drag values reduces up to ~4%.

References

[1] Harun Chowdhury, Hazim Moria, Abdulkadir Ali, Iftekhar Khan, Firoz Alam and Simon Watkins "A study on aerodynamic drag of a semi-trailer truck" Sciencedirect Elsevier Procedia Engg. 56 (2013) 201-205.

[2] Chaitanya Chilbule, Awadhesh Upadhyay, Yagna Mukkamala "Analyzing the profile modification of truck- trailer to prune the aerodynamic drag and its repercussion on fuel consumption" Sciencedirect Procedia Engineering 97 (2014) 1208 – 1219.

[3] Fu-Hung Hsu, Roger L. Davis "Drag reduction of tractor-trailers using optimized add-on devices" ASME August2010, Vol. 132 / 084504-1-6.

[4] Christopher J. Roy, Jeffrey Payne, Mary McWherter- Payne "RANS simulations of a simplified tractor/trailer geometry" ASME September 2006, Vol. 128 / 1083-1089.

[5] Volodymyr Sirenko, Roman Pavlovs'ky, Upendra S. Rohatgi "Methods of reducing vehicle aerodynamic drag" ASME 2012 BNL-98039-   2012-CP.

[6] Richard M. Wood & Steven X. S. Bauer "Simple and low-cost aerodynamic drag reduction devices for tractor-trailer trucks" SAE 2003-01-3377.

[7] Subrata Roy & Pradeep Srinivasan External flow analysis of a truck for drag reduction SAE 2000-01-3500.

[8] Chen Jing, Wu Yun-zhu "Research on aerodynamic drag reduction of heavy commercial vehicle" IEEE 2010   V5-191-194.

[9] Li Song, Zhang Jicheng, Liu Yongxue, Hu Tieyu "Aerodynamic drag reduction design of van body truck by numerical simulation method" IEEE ICDMA 2011 168-171.