A Computational Simulation of Aerodynamic Drag Reductions for Heavy Commercial Vehicles

C. Pevitt¹, H. Chowdury¹, H. Moriaand² and F. Alam¹

¹School of Aerospace, Mechanical and Manufacturing Engineering, RMIT University, Bundoora, Melbourne, VIC 3083, Australia
²Department of Mechanical Engineering, Yanbu Industrial College, Kingdom of Saudi Arabia

Abstract

Heavy commercial vehicles are known for being extremely inefficient compared to other ground vehicles, partly due to high aerodynamic drag. This is a result of their un-streamlined body shape. A large commercial vehicle travelling at 100 km/h consumes approximately 52% of the total fuel to provide the power required to overcome the aerodynamic drag. The primary objective of this study is to determine the aerodynamic impact of various fuel saving devices used in heavy commercial vehicles. To measure the aerodynamic drag produced by the vehicle, an experimental and computational simulation study was undertaken using a 1/10th scale model of a Mack 600R class 8 tractor-trailer. The aerodynamic drag on the base vehicle with external attachments (i.e., front fairing, side skirting and gap filling) was measured for a range of vehicle operating speeds and yaw angles. The configurations used were chosen as they required minimal modification of the vehicle and could be implemented on existing commercial trucks. This paper focuses on the validation of the experimental work through computer simulations on a baseline vehicle configuration. As well as this, the simulations will be used to further predict the expected aerodynamic effects and possible drag reductions with various add-ons and configurations. The findings indicate that a significant drag reduction between 20% and 35% can be achieved depending on the modifications and cross wind conditions. It was found that the full-skirting (using the front fairing, side skirting and gap filling) has maximum impact while only front fairing has lowest impact on aerodynamic drag reduction overall.

Introduction

Heavy commercial vehicles are aerodynamically inefficient compared to other ground vehicles due to their un-streamlined body shapes. Large commercial vehicles travelling at 100 km/h consumes approximately 52% of the total fuel to provide the power required to overcome the aerodynamic drag [4].

It has been researched that on average a heavy commercial vehicle’s annual mileage will vary between 130,000 km and 160,000 km [3]. Due to such a high mileage, any reduction of aerodynamic drag will result in dramatic fuel savings and reductions in greenhouse gas emission. Despite significant work over the decade to develop various fuel saving mechanisms for commercial vehicles, there is still large potential for further developments and reductions in aerodynamic drag.

Many modern trucks today already have a range of drag reducing mechanisms in place to reduce fuel usage [5]. A common and simple method of drag reduction is the addition of external modifications to the truck. Many of these devises help to streamline the truck, improving its aerodynamics. Many of these additions aim to not affect the frontal area of the truck but rather improve their streamline aerodynamics.

Aerodynamic drag on a semi-trailer truck typically accounts for about 75-80% of the total resistance to motion at 100 km/h [5]. As a result the possible reduction in aerodynamic drag could contribute significantly to the fuel efficiency of a truck, as well as the reduction of greenhouse gas emissions.

Previous studies have found that a fuel reduction of as little as 1% (typically 0.1L/100km) could result in savings as much as US $30 million annually [7]. In recent times the reduction of fuel consumption has become more relevant with increasing fuel prices and the further consumption of oil reserves. In 2009 it was found that over 1.3 trillion litters of petrol and diesel was consumed by road vehicles [7]. This also relates to high levels of pollution (CO₂), from the burning of fossil fuels.

Many current designs for drag reduction on commercial vehicles are not well studied or documented. Much of the research today is being done on reducing drag on newly designed trucks [5-6]. As a result little work is being done on current designs. Due to the high number of trucks already on the road today, as well as the fact that many of these older designs are still being sold, it is imperative to find ways to reduce the drag on these designs. The primary purpose of this work is to undergo research on the effects of modifications on currently designed trucks. Studies were completed on a 1/10th scale Mack 600R class 8 tractor-trailer. Studies were completed both experimentally and through the use of computational simulations. This paper will focus on the validation of the computer simulations as well as its further predictions of possible drag reductions. The modifications that will be considered will look at fairings on the front of the truck, different sized skirtings on the side of the truck and the effect of changes in the gap between the truck cabin and trailer.

Experimental Procedure

The RMIT Industrial Wind Tunnel was used to measure the aerodynamic changes in drag with different test sections for the truck. The tunnel is a closed return circuit wind tunnel with a turntable to simulate the cross wind effects. The rectangular test section dimensions are three meters wide, two meters high and nine meters long, and the tunnel’s cross sectional area is six square meters. The wind tunnel was found to run with a turbulence intensity of around 1.5 and the walls have a maximum boundary layer of 15cm [1]. More details regarding the wind tunnel can be found by Alam, 2010 [1].

The test vehicle was mounted on a sting with the JR3 multi-axis load cell, also commonly known as a 6 degree of freedom force-torque sensor made by JR3, Inc., Woodland, USA. The sensor was used to measure all three forces (drag, lift and side forces) and three moments (yaw, pitch and roll) at a time. The data was recorded for 10 seconds and time average with a frequency of 20 Hz ensuring electrical interference is minimised. Multiple data sets were collected at each speed and the results were averaged, minimising possible errors in the raw experimental data. Tests
were conducted at a range of wind speeds from 40 km/h to 120 km/h under four yaw angles 0°, 10°, 20° and 30° to simulate the crosswind effects. Yaw angle (ψ) can be defined as the anticlockwise angle between the vehicle centreline and the mean direction of airflow experienced by the vehicle. Five different modifications to the truck were tested throughout the experiments. The different design layouts are seen in Figure 1 and the full dimensions of the standard truck are seen in Figure 2.

Figure 1. Different modification add-ons used in this study, a) Standard Configuration b) Fairing c) Gap Filled d) Full Skirting e) Part Skirting

Simulation Procedure

The computational fluid dynamics (CFD) simulations were performed using ANSYS CFX software. The simulations were designed as a tool to validate the wind tunnel data. The model replicated the experimental model as well as the dimensional restrictions of the wind tunnel. The simulation trucks are a simplified design of the experimental replicas. As a result their validity is expected to be somewhat limited. While at the same time their accuracy is expected to be enough that they will be capable of validating wind tunnel tests, predict additional flow phenomena and determine expected drag reductions. The simplified models are seen in Figure 1.

The setup of the simulation model can be seen in Figure 3. The inlet and outlet represent the airflow through the wind tunnel, walls and floors are dimensioned to the size of the wind tunnel to restrict flow. The truck model is then represented as a solid model removed from the main domain. The simulation model represents the experimental truck without the testing sting. As the truck is quite low to the floor of the wind tunnel, it is expected that the effects of the sting should be minimal to the overall drag on the model.

Figure 2. Standard model configuration of scaled truck (cm)

Results and Discussion

In this paper, only drag force (F_D) data and its dimensionless quantity drag coefficient (C_D) are presented, as it is the reduction of drag and its corresponding reductions in emissions that are of interest. The aerodynamic drag depends on the size of a vehicle (projected frontal area, A), the drag coefficient (C_D) which is a measure of the flow quality around the vehicle, the square of the vehicle speed (V) and the air density (ρ), as expressed in equation (1).

$$C_D = \frac{F_D}{\frac{1}{2} \rho V^2 A}$$

From initial findings, the component of drag on each of the truck configurations indicated the same trends between both the CFD and Experimental Fluid Dynamics (EFD) results. The results show that the highest levels of drag occur consistently for all speeds on the baseline configuration. A noticeable reduction in the total aerodynamic drag over the truck was noted for; the fairing, closure of the gap, part skirting and full skirting. The fact that both sets of results indicated the same trend is a sign that they are following the same flow development and representative of the same characteristics of the drag over the truck.
In order to look at the results in more detail, to determine the level of drag reduction for each of the configurations, a clearer representation can be seen from the $C_D$ values. Figure 4 represents the drag coefficient values for both the CFD simulations (Figure 5A) and the EFD results (Figure 5B). From Figure 5 it is clear to see the differences between the two sets of data. Despite the differences in the results it is still clear to see that the results indicate the same trend in the drag values with increasing speed. This is promising and means the results may be capable of providing accurate drag characteristics. From the differences in the results, small variations in the trend of the drag with increasing speed were noted. The CFD results indicate a fairly horizontal and stable drag coefficient with increasing speed, with a slight upwards trend. While the EFD results indicate the same horizontal and stable drag coefficient with a slight downwards trend.

There are many different reasons for these variations, though the most evident would derive from the simplifications in the CFD model. Many of the fine detail on the front of the truck were removed for the CFD model to improve simplicity. This was decided as the model was designed as a validation and prediction tool only. Additionally higher accuracy would greatly increase the computational power and mesh density required for precise results.

Another key factor that would affect the true drag characteristics of the flow resides in the number of EFD testing speeds completed. For these preliminary findings the truck was tested in the wind tunnel for six different speeds, while the CFD model was run for a range of twelve speeds. As a result the CFD findings provide a greater overview of the total pressure distribution, in the drag direction. It is expected that with greater detail in the CFD model and a larger range of testing points in the EFD data, the two sets of results would not only show the same key characteristics but also indicate the same detailed trends in drag variations with increasing speeds.

It is also important to note that despite these differences, on such a small scale the magnitude of the changes in the drag coefficient for both the CFD and EFD results are around 7%. This indicates that slight variations in the results will greatly affect how they appear to trend with increasing speeds. It is important to look at the differences in the results closer, as well as comparing initial predictions of drag reduction from both the CFD and EFD results.

Figures 5 and 6 demonstrate this comparison and confirm the previous results by comparing the baseline model and configurations at a yaw angle of $0^\circ$. Figure 5A shows the drag coefficient as a function of speed for different test configurations at $\psi = 0^\circ$. Figure 5B shows the drag coefficient as a function of speed for the baseline configuration for varying yaw angle ($\psi$).

Despite slight variations in the CFD and EFD results it can be seen that both represent the same general trend and drag characteristics over the truck. From both sets of results it can be seen that there are clear increases in drag with changing yaw angles. This is expected as a change in yaw angle represents an increase in the equivalent cross wind on the truck, in turn increasing the area in line with the flow. Both sets of data indicate a fairly constant and stable $C_D$ value for increasing speed and overall are in good agreement. The next step is to determine if they are both representative of each other when predicting the amount of drag reduction for changing configurations. In order to make these comparisons the range of configurations were compared at $0^\circ$ yaw angle for both the CFD and EFD results.

In order to make the comparisons needed the amount of drag reduction for each of the configurations was compared to the baseline model at a yaw angle of $0^\circ$. As shown in Table 1 both the CFD simulations and the EFD data provide a similar magnitude of drag reduction. Both sets of data show an agreement within 3%. Based on the percentage reduction for each of the configurations it is expected that the CFD models are capable of representing the characteristics of the flow around the truck. They are also fully capable of predicting the changes in drag and determine possible drag reductions for each of the configurations. This would be further confirmed with the addition of flow visualisation for both the CFD and EFD model.

<table>
<thead>
<tr>
<th>Add-ons</th>
<th>$C_D$ CFD</th>
<th>$C_D$ EFD</th>
</tr>
</thead>
<tbody>
<tr>
<td>Fairing</td>
<td>20%</td>
<td>18%</td>
</tr>
<tr>
<td>Full Skirting</td>
<td>30%</td>
<td>27%</td>
</tr>
<tr>
<td>Part Skirting</td>
<td>24%</td>
<td>25%</td>
</tr>
<tr>
<td>Gap Filled</td>
<td>24%</td>
<td>26%</td>
</tr>
</tbody>
</table>
As the CFD models indicated that they were capable of representing the predicted drag reductions, the next step is to utilise them to gain a full collection of data to predict possible drag reductions across a range of yaw angles. The drag coefficients for each of the configurations across the range of angles and yaw angles. The results were found to be in agreement with predicted drag reductions, with an average deviation of 2%. Both sets of data predicted the same trends and drag characteristics for each of the configurations. As a result it was deemed that the CFD simulations could be validated and utilised for further testing. This would help to determine predictions in drag reduction capabilities for a range of different configurations and yaw angles.

It was found that even simple modifications such as a fairing on the front of the truck are capable of reducing drag by as much as 20% at a 0° yaw angle. Additionally under small cross winds, as little as 5° yaw angle, with the addition of a full skirting along the truck, aerodynamic drag can be reduced by over 35%. These findings indicate that there is high potential for aerodynamic drag reductions in existing heavy commercial vehicles. These drag reductions could be capable of leading to sizable reductions in fuel costs and CO₂ emissions. With trucks being the result of around 20% of global warming emissions [2], a reduction of even 20% in aerodynamic drag is sizable.

There is clear potential for drag reductions in heavy commercial vehicles. As a result it has been decided that further research should be made to further understand the drag reduction capabilities of modifications on heavy commercial vehicles. This would include looking at additional modifications such as rear fairings, streamlining surfaces and merging current designs together. Additionally to better understand the flow characteristics and causes behind the achieved drag reduction capabilities, a range of flow visualisations should be made. This would help to determine how each configuration results in a drag reduction. By better understanding the reasons behind it and how each configuration affects the flow, it may be possible to develop more efficient designs, catered to the direct effects of the aerodynamic drag. It is also recommended that for future work, experimental testing utilising such techniques as hot wire should be used to further validate the CFD results. It is expected that this will better relate the true flow characteristics of the two sets of results.

References


