18th Australasian Fluid Mechanics Conference
Launceston, Australia
3-7 December 2012

A Computational Simulation of Aerodynamic Drag Reductions for Heavy Commercial Vehicles

C. Pevitt[1], H. Chowdury[1], H. Moriaand[2] and F. Alam[1]

[1]School of Aerospace, Mechanical and Manufacturing Engineering,
RMIT University, Bundoora, Melbourne, VIC 3083, Australia

[2]Department 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 faring, 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 (CO_2), 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 0º, 10º, 20º and 30º to simulate the crosswind effects. Yaw angle (ψ) can be defined as the anti-clockwise 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.

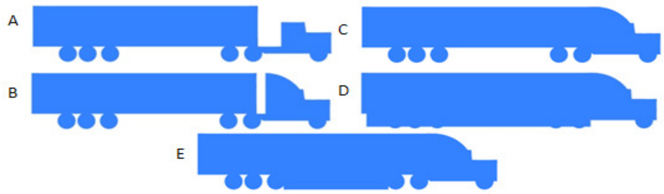

Figure 1. Different modification add-ons used in this study, a) Standard Configuration b) Fairing c) Gap Filled d) Full Skirting e) Part Skirting

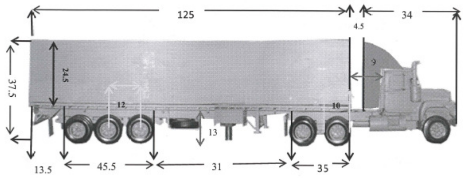

Figure 2. Standard model configuration of scaled truck (cm)

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 me 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.

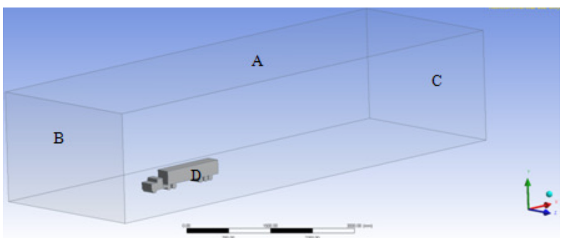

Figure 3. Computational Domain used in CFD simulations

Both mesh convergence and turbulence studies were performed on the simulation to help improve the quality of the results. It was found that the magnitude of the drag on the truck was highly dependent on the density and refinement of the mesh around the truck. The final mesh used for the simulations consisted on an unstructured hybrid mesh. The model has a structured inflation layer around the surface, with an initial cell height of 1E-4m, 28 layers and a growth rate of 1.18. This structured mesh then merges to an unstructured global mesh. This mesh is developed based on a proximity and curvature function around the truck with an accuracy of 0.5. The mesh was then created with an automatic development with a slow transition and fine grading. The global mesh has a max size of 0.1m and a growth rate of 1.15. The final Mesh has 2.3 million elements with average Y-Plus value of 1.5. This was mostly limited to by computational power of the computer. The final mesh can be seen in Figure 4.

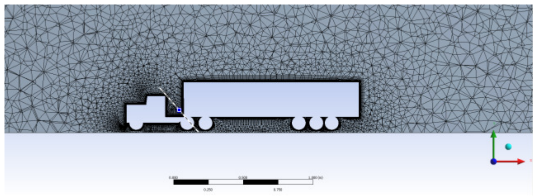

Figure 4. Final mesh around the baseline truck configuration

As well as the mesh convergence study, a range of simulations were run to find the most appropriate turbulence model. For the purpose of this work three different turbulence models were considered. These were the standard k-ε, k-ω, and the Shear Stress Transport (SST) equations. It was found that all of the turbulence models provided reasonably good agreement compared to experimental results for low speeds and at 0° yaw angle. As both speed and yaw angle increased, making the flow more complicated, both the k-ε and k-ω models began to struggle to correctly represent the true pressure distribution of the flow. As a result the SST turbulence model was used to run the full set of simulations on the truck.

The model was developed with the capability to change its yaw angle while keeping all parameters of the mesh development mentioned above constant. The simulations were run for 12 different speeds ranging from 11km/h to 122k/m with and average increment of 10km/h. Each simulation was then repeated for four different yaw angles, ranging from 0° to 15° with 5° increments. The direction of rotation was designed to match that of the experimental model, as previously mentioned.

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).

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

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, from largest to smallest reductions respectively, for both the simulation and experimental data. 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 its clear to see the differences between the two sets of data. Despite the difference 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 finding 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.

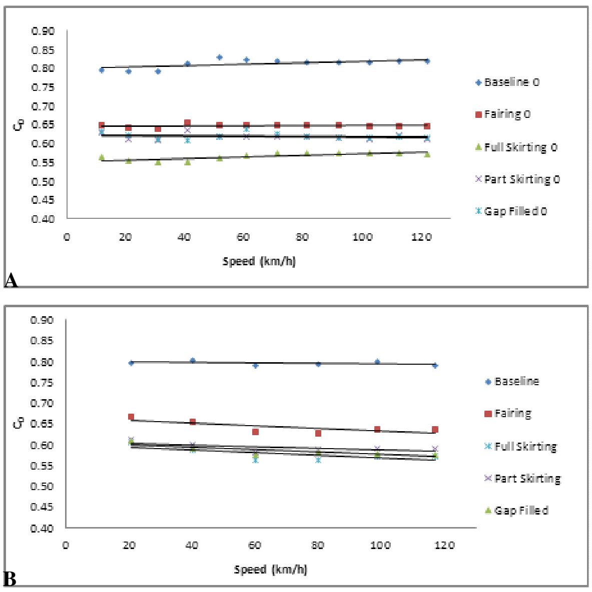

Figure 5. Drag coefficient as a function of speed for different test configurations at ψ = 0° A) CFD B) EFD

In order to determine how accurate the results can be it was decided to focus on the baseline configuration as a validation method. From this model it was possible to see where the CFD results varied from the EFD findings and determine how these differences can be interpreted and explained. Figure 6 is representative of the baseline model at a yaw angle of 0°, 5°, 10° and 15° for both the CFD and EFD testing. From these results it is clear to see the difference between the two sets of data. When only one model is compared, it is noted that the results are quite similar. As mentioned earlier, the EFD results have a limited number of test points, meaning they provide a smaller overview of the total flow. It can be seen that the EFD data indicates a small downwards trend with increasing speed while the CFD results indicates a much more horizontal results. This again could be put down to the CFD models simplified design and idealised flow conditions. At the same time there are many imperfections in the wind tunnel, which can lead to errors in the readings. These differences can help to explain why the CFD model indicates a more horizontal trend and the EFD results have a slight downwards trend. This would be better understood through the use of flow visualisation, which is planned for future work.

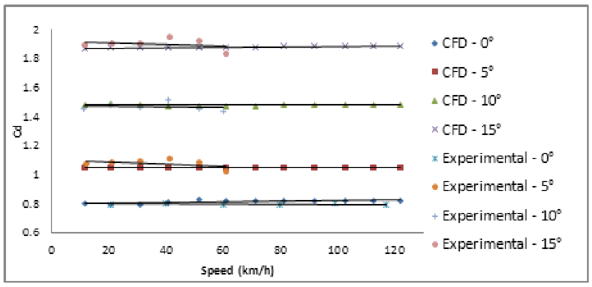

Figure 6. Drag coefficient as a function of speed for baseline configuration for varying yaw angle (ψ)

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° 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°. 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 1. Average percentage decrease of drag coefficient (C_D) for different modifications at ψ = 0º compared to baseline model

Add-ons	***C_D CFD***	***C_D EFD***
Fairing	20%	18%
Full Skirting	30%	27%
Part Skirting	24%	25%
Gap Filled	24%	26%

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 can be seen in Figure 7. From these results it was noted that the amount of drag varies greatly across each of the configurations and the magnitude is highly dependent on the cross wind (yaw angle) on the truck. In order to gain a clearer understanding of the predicted drag reduction under varying angles, a comparison of drag reduction percentage per angle for each configuration was made and can be seen in Figure 8.

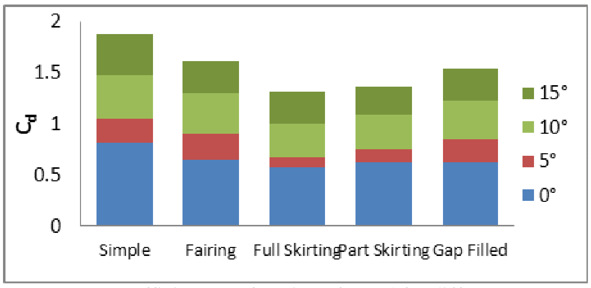

Figure 7. Drag coefficient as a function of speed for different modifications at $\psi = 0°$

From the drag reduction percentage in Figure 8 it can be seen that there are notable differences in each of the configurations and their drag reduction capabilities. Throughout all of the yaw angles it is noted that the full skirting model results in the highest drag reduction. With a reduction between 30% and 35% such an addition to the truck could relate to large cost savings in fuel plus a reduction in emissions. It is also noted that at a yaw angle of 0° the differences in each of the configurations is much less. This is a result of the aerodynamic drag being mostly affected by the frontal area of the truck. As a result additions such as the Fairing to the front of the truck were seen to reduce drag by as much as 20%, while skirtings are less critical at this angle.

It is important to note that unlike the full skirting along the side of the truck, the fairing is a much simpler addition and would result in fewer possibilities of damage in a real life application. Due to the large distances the truck would cover annually across a range of roads and surfaces, modifications close to the ground could easily take damage, reducing there effectiveness. It was also noted that at 0° yaw angle both the part skirting and gap filled models resulted in similar drag reductions. As mentioned earlier, this occurs due to the frontal area of the truck being the critical dimension affecting the aerodynamic drag. This means the addition of a small skirtings along the side of the truck do not affect aerodynamic drag as much as they will for larger yaw angles which is evident in Figure 8.

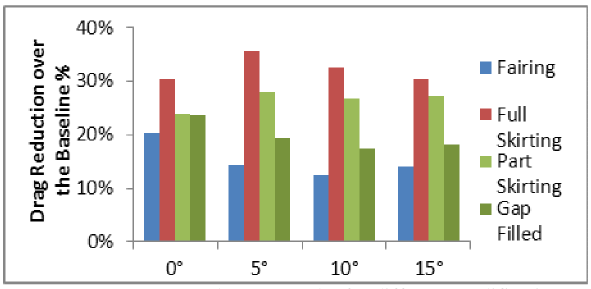

Figure 8. Drag percentage decrease under for different modifications, at varying yaw angles, compared to the baseline model

Conclusion

The initial comparison between the CFD simulations and the EFD data shows that the simulations were capable of accurately determining the changes in drag over the truck with varying configurations 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 CO_2 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

[1]Alam F, Chowdhury H, Moria H and Watkins S. Effects of Vehicle Add-Ons on Aerodynamic Performance. In: The Proceeding of the 13th Asian Congress of Fluid Mechanics (ACFM2010), Dhaka: Bangladesh University of Engineering and Technology; 2010, p.186-189

[2] California Air Resources Board Draft California Greenhouse Gas Inventory, November 17, 2007 [cited, August 2012] http://www.arb.ca.gov/cc/inventory/inventory.htm

[3] Cooper, K.R. (2004), Commercial Vehicle Aerodynamic Drag Reduction: Historical Perspective as a Guide. In The Aerodynamics of Heavy Vehicles: Trucks, Buses, and Trains. Lecture Notes in Applied and Computational Mechanics. Springer

[4] Cooper, K.R. (2003), Truck Aerodynamics Reborn- Lessons From the Past. SAE Technical Paper, 2003-01-3376, 9-19.

[5] Drollinger, R.A (1986), Heavy duty truck aerodynamics. SAE Technical Paper, 870001

[6] Hucho, W.H. (1998), Aerodynamics of Road Vehicles, 4th Edition. Society of Automotive Engineers, USA

[7] Schoon, R. E. (2007), On-road Evaluation of Devices to Reduce Heavy Truck Aerodynamic Drag, SAE Publication, SAE Paper No. 2007-01-4294.