THE EFFECTS OF AIR FLOW IN THE WAKE OF A LARGE VEHICLE ON TRAILING A PASSENGER CAR

Shahrin Hisham Amirnordin[1], Wan Saiful-Islam Wan Salim[2], Suzairin Md Seri, Mohd Faiz Ariffin, Akmal Nizam Mohamad, Hamimah Abd Rahman, Ishkrizat Taib, Ahmad Jais Alimin

[1],[2]Automotive Research Group,
Faculty of Mechanical and Manufacturing Engineering,
Universiti Tun Hussein Onn Malaysia,
86400 Parit Raja, Batu Pahat, Johor.

Email: [1]shahrin@uthm.edu.my, [2]wsaiful@ uthm.edu.my

ABSTRACT

Road driving condition under drafting is known to have an influence on the aerodynamic forces of the vehicle. Large vehicles such as busses and trucks traveling at high speeds give results in the formation of a large turbulent flow in the wake region. This turbulent flow is very unsteady in nature hence its influence on the air flow within its vicinity will also be unsteady. This paper investigates the relative values of drag and lift forces acting on a passenger car trailing a large vehicle (drafting) under unsteady conditions. The simulation is conducted using Computational Fluid Dynamics software, FLUENT for a two-dimensional flow domain at Re $3.18x10^6$ for a trailing distance of 0 to 30 meters. The unsteady effect is studied at 15 time intervals for each time step. Turbulence is simulated using the Reynolds-Average Navier Stokes (RANS) k-ε model. Results show that aerodynamically, the critical drafting distance is between three to five meters where the lowest drag is found to occur at three meters. The results show the suitable distance for drafting which may serve as useful information for vehicle fuel economy and stability.

KEYWORDS: *aerodynamics, drafting, drag, lift.*

1.0 INTRODUCTION

Drafting or slipstreaming is a technique where two vehicles or objects align in a close group reducing the overall effect of drag due to exploiting the lead object's slipstream. Especially when high speeds are involved, large vehicles such as busses and trucks traveling at high speeds results in the formation of a large turbulent flow in the wake region. This turbulent flow is very unsteady in nature hence its influence on the air flow within its vicinity will also be unsteady. In high speed roads such as the national highways where large vehicles travel at high speeds, it is important for us to understand the extent as to how the air flow behavior around these large vehicles affect other vehicles. In aerodynamics study, experiments mainly involve the measurement of the aerodynamic coefficients and flow visualization over vehicles. The same measurements and flow visualizations can also be done using numerical methods through Computational Fluid Dynamics (CFD) without having to undergo rigorous and costly wind tunnel tests.

Ahmed (1981) performed a series of wind-tunnel experiments in order to examine the wake structure around typical automobile geometries. The study focused on the time averaged structure obtained from visualizations of flow in the wake region for smooth quarter scale automobile models. Experiments were also performed with a bluff-body, "generic" vehicle geometry where pressure measurements, wake surveys and force measurements for different angles of base-slant were presented. Results indicated that almost 85% of total aerodynamic resistance is contributed by pressure drag and most of this drag is generated at the rear end.

Han (1989) performed a numerical study over a three-dimensional bluff-body in proximity to the ground. Due to the lack of detailed velocity measurements over this body, this work compared qualitatively the formation of the vortices in the wake and the drag coefficient for different slant angles. The author used two different Reynolds-Average Navier Stokes (RANS) based turbulence models namely the k-ε and the Renormalization Group (RNG) k-ε. It was reported that the overall validity of the computations was dependent on the turbulence model and the accuracy of the discretisation scheme.

Sinisa and Davidson (2003) stated that the consideration was not only the drag and lift coefficients that describe the aerodynamic properties of the body, but also the flow structures responsible for these properties. The forces acting on the surface of the body as a result of the surface pressure were studied both time averaged and instantaneously in the study. So, while the flow around the bodies is highly unsteady, our knowledge of this flow is based primarily on experimental and numerical studies of time averaged observations. The study has shown that the instantaneous flow is very different from the time averaged one, not only in the wake region but also along the entire body.

Further research by Watkins and Vino (2008) studied traffic vehicle spacing influence on drag coefficient of a vehicle. The authors confirmed that drag can be related to the longitudinal distances between trailing vehicles and that in general, drag reduction is significant as the vehicles the distance between the vehicles decrease. As a result, the concept of travelling in a convoy is being explored among trucks as the drag reduction between 10-40% was observed.

This paper investigates the effects of drafting distance between a hatchback passenger car when it is trailing a large vehicle. The analysis is conducted using CFD software, FLUENT in two-dimensional using RANS k-ε turbulence model. Attention is given on the drag and lift coefficient of the rear vehicle and its relation to the air flow structure due to the presence of upstream wake from 0.1 to 30 m of drafting conditions.

The paper is organized as follows. Section 2 presents the geometrical model of the car and its computational domain. Details of boundary conditions and solver set up are also presented, together with grid independence study. Section 3 discusses the result obtained in the CFD calculation which is based on the time-averaged and visualisation of instantaneous flow around the body with comparison to previous research work. This section starts on the isolated model before proceeds to the drafting conditions. Drag and lift coefficient of the trailing vehicle is presented and discussed, together with pressure distribution and velocity vector before the concluding remark in Section 4.

2.0 METHODOLOGY

The present numerical study is conducted on a two-dimensional (2D) car model as shown in Figure 1. This is important since three-dimensional (3D) unsteady calculation involves highly intensive task compared to 2D treatment (Benazza *et.al.*, 2007). Due to the nature of turbulence study in drafting condition, transient simulation able to provide more accurate treatment of the boundary layer and large scale boundary layer (Frank & D'Elia, 2004).

The body takes the shape of a hatchback passenger car with dimensions of 3.75 m in the stream wise direction and 1.55 m in height. A model of a passenger bus is also constructed and located upstream from the car in the flow domain. A bus-shaped with 10 m in length and 3.4 m in height is a simplified model proposed by Sanisa and Davidson (2003) which takes the square shape at its rear end with the ground clearance at 0.29 m. The purpose is to simplify the case and to focus more on the aerodynamic forces and flow structure (Sanisa and Davidson, 2003). The geometry of the computational domain is given in Figure 2 which shows the bus in its computational domain with 20 m in height and 70 m in length which accounts for six times the overall length of the bus with one length upstream and five lengths downstream. The projected area of the car in the mainstream direction is 1.55 m in height which is equivalent to 7.75 % of blockage ratio.

To limit the total number of elements, the computational domain was constructed with unstructured Tri-Mesh (with meshing spacing 0.05 and the mesh type was TGrid). Figure 3 shows grid distribution of the computational domain showing the complexity of the mesh used with number of mesh were up to 79,761.

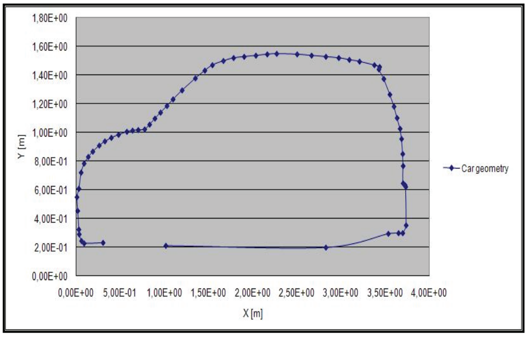

Figure 1 2-Dimensional geometry of the passenger car

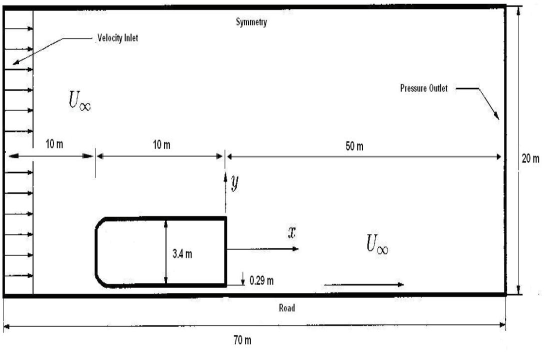

Figure 2 Geometry of computational domain

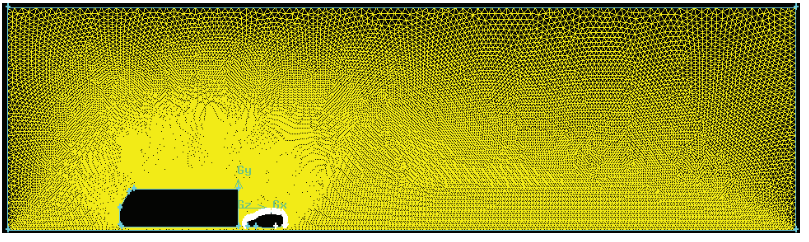

Figure 3 Grid distribution of computational domain

Initially, the simulation of the bus and car was conducted separately for validation and grid independence study. Afterwards, the passenger car was positioned at different locations within this distance in the trailing wake as shown in Table 1. The resulting drag and lift forces at these different locations were obtained. Since the simulation was unsteady, the drag and lift forces were determined at several time steps to see whether there are any obvious fluctuations of C_D and C_L.

Table 1 Drafting Position Of The Car

Spacing distance (m)	*0.1*	*0.5*	*1.0*	*2.0*	*3.0*	*4.0*	*5.0*	*10.0*	*20.0*	*30.0*

A pressure based solver was used along with k-ε turbulence model. Standard wall functions were used for near wall treatment. The transient computation was carried out at a time step of 0.01s. FLUENT solver was used to solve the Navier-Stokes equations for two-dimensional unsteady incompressible flow shown below.

Continuity equation.

$$\left(\frac{\partial \rho}{\partial t}+\frac{\partial u_i}{\partial x_i}\right)=0 \tag{1}$$

Momentum equation.

$$\rho\left(\frac{\partial u_i}{\partial t}+u_j\frac{\partial u_i}{\partial x_j}\right)=-\frac{\partial P}{\partial x_i}+\frac{\partial}{\partial x_j}\tau_{ij} \tag{2}$$

where,

$$\tau_{ij}=2\mu S_{ij}-\delta_{ij}\frac{2}{3}\mu\frac{\partial U_k}{\partial x_k}-\rho\overline{u_i' u_j'} \tag{3}$$

The grid independence study of the passenger car was conducted for computational grids consisting of 54,672, 67,216 and 79,761 elements and was carried out for drag coefficients. No significant change in drag was observed for the 67,216 and 79,761 grids.

3.0 RESULTS AND DISCUSSION

The results discussed in this section is the averaged value over several cycles of the unsteady condition of the CFD calculation, unless it is stated otherwise.

Detail analysis is conducted on the isolated model of the bus and the passenger car. Figure 4 depicts the velocity vector of the bus. It shows the air flows produce two counter rotating vortices behind the bus body which is typical for separating flows behind a rectangular bluff body. Top vortex (Vx_1) is in clockwise direction while bottom vortex (Vx_2) is in counterclockwise direction, similar to numerical work in Frank & D'Elia (2004) and Lui & Moser (2003). The combinations of large vortex and separation cause a large effect on the drag force. The size of the wake produced by the bus is approximately 15 m in length. It is much longer compared to the Ahmed model (with rear slant angle at 35^o) which gives velocity deficit after more than 4 m (Lui & Moser, 2003). As a result, the simplified bus model imposed longer wake region compared to Ahmed model. It gives better drag reduction on the trailing car as the main idea behind drafting is to take advantage of this low pressure region inside the wake (within the said length) which should result in less pressure drag on the trailing vehicle.

The geometry of the trailing vehicle is modeled according to a hatchback type passenger car as shown along with pressure contour in Figure 5. High pressure is formed in front of the passenger's ca. The model used in this analysis produces higher drag coefficient than car-shaped notchback. The results are similar with typical pressure distribution of the car in any open literature (Munson *et.al*., 2010).

The characteristic of unsteady flows around the model is observed as shown in Figure 6. The car produces counter rotating vortices at the near end similar to that produced by the bus. The wake structures behind the body are influenced by high pressure that occurs in front of body. It is consistent with the results obtained by experimental work conducted in 2001 in which intensive investigation on the unsteady structures in the wake was conducted on hatchback car models (Sims-Williams *et.al*., 2001).

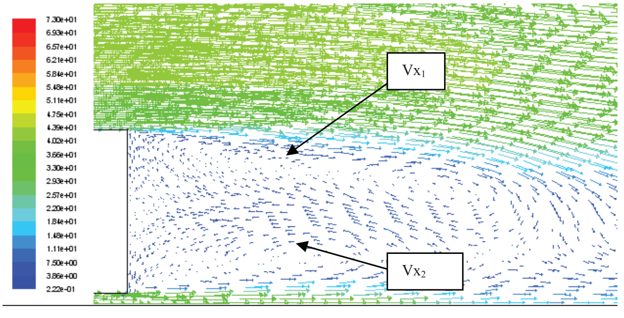

Figure 4 Velocity vectors behind the bus

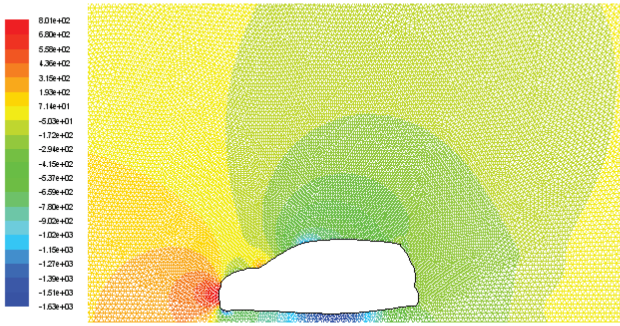

Figure 5 Contour of static pressure for isolated car

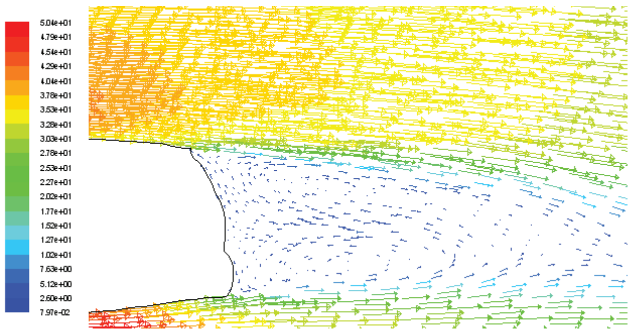

Figure 6 Velocity vectors behind the passenger car

Figure 7 and 8 are averaged drag and lift coefficient of the car at various spacing condition. In Figure 7, 0.322 is the value obtained by the car as an isolated model. Here, the main interest is the variation in coefficient with vehicle spacing. The graph indicates the relative value of drag decreases from 0.1 to 5.0 m. Obvious drag reduction starts at 1.0 m (111 %) up to 5.0 m. However, the lowest drag occurs at 3.0 m which gives the values -0.136. At 4.0 and 5.0 m, C_D values are -0.113 for both position. This gives the drag reduction at 3.0 m is 142 % and 135 % for both 4.0 and 5.0 m spacing distance. The result shows that the passenger car seems to be sucked forward at this close distance. Then, the value of drag coefficient suddenly increases as much as 0.065 when the drafting position is increased to 10.0 meters and continuous increment up to 30 m. The increment of this value is caused by a change of pressure force produced around the body surface. In other word, drafting position at 3.0 to 5.0 meter is a critical position to be discussed further.

In Figure 8, lift coefficients shows large increment as the vehicle spacing increases up to 5.0 m distance. As the spacing increases further, the values slowly reach the steady value of isolated case. This is consistent with the results obtained for drag coefficient showing the critical distance from 0.1 to 5.0 m.

Figure 9 describes velocity vector during drafting condition on the passenger car. It shows five rotating vortices in the wake region with one additional clockwise rotating vortex formed on the roof of the trailing vehicle (numbered as “3”). The direction of rotation in vortex 2 and 3 plays an important role in reducing the drag of the trailing vehicle.

In Figure 10, at 3.0 meter distance, the strong trailing vortices from the downstream rotate counterclockwise through in front of the car body. The lower vortex upstream of the trailing vehicle decreases in size thus reducing the suction effect, hence the higher drag. The entire observation for this analysis shows the drag coefficient is the lowest at 3.0 meter of the drafting distance. Based on the pressure distribution in Figure 5, high pressure is produced at the top and behind the passenger car while low pressure is produced in front of the body. At 5.0 meter, the pressure force produced acting on the entire car body. The effects of distribution show the higher pressure is produced at front and behind the body.

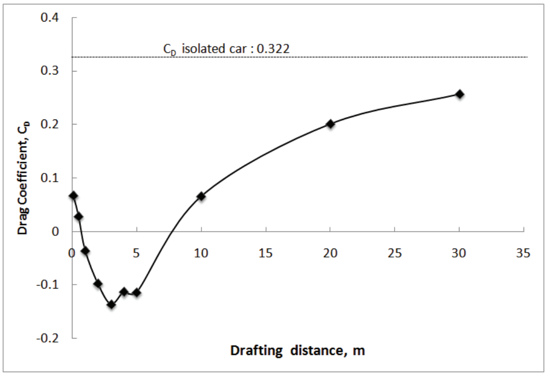

Figure 7 Drag coefficient of car at drafting distance

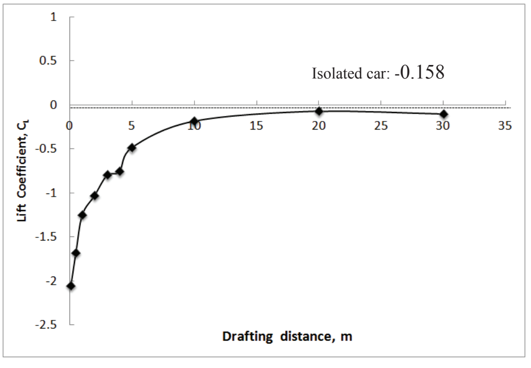

Figure 8 Lift coefficient of car at drafting distance

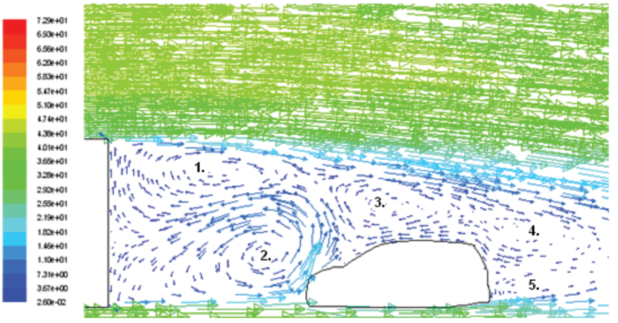

Figure 9 Enlarged view of velocity vectors at the passenger car surface

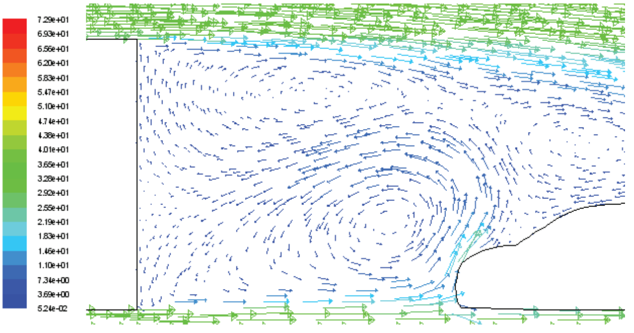

Figure 10 Difference of velocity vectors in the wake between 3.0 meter of drafting position

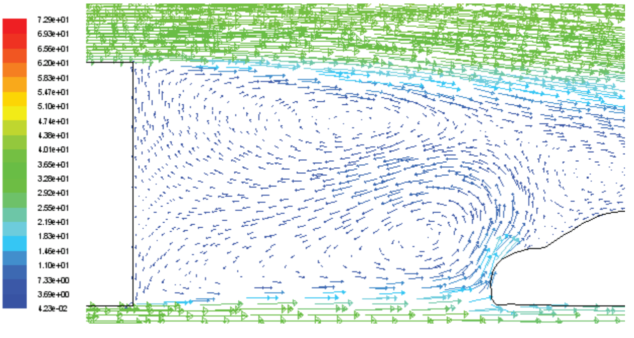

Figure 11 Difference of velocity vectors in the wake between 5.0 meter of drafting position

Drafting has been recognised as a method in drag reduction (hence fuel consumption), smaller trailing vehicles is seen to have more advantages by driving within this safe distance. Further study is necessary to investigate the effects on the combined drag of several vehicles trailing large vehicle.

Changes in lift is also noted for the same trailing distance from 3 to 5 m. It agrees to previous experimental work (Watkins & Vino, 2008) (via surface and off-body visualization and velocity and flow mapping) . It is believed the effects are due to the influence of the rear vortices (Watkins & Vino, 2008). Overall shows lift coefficient is also reduced in which it improves more on the vehicle stability. Thus, drivers can take the opportunity by staying in a farther distance from 3 to 5 m by having the advantage of less fuel consumption and improvement on vehicle stability.

4.0 CONCLUSION

Drafting position at 3.0 to 5.0 meter is a critical drafting where the air flows from the bus causing the drag coefficient obtained suddenly increase as much as 142 %. In this case, this distance is an ideal position in terms of fuel efficiency and safety to the passenger car while trailing behind a large vehicle. At critical distance the flow and the wake structure is highly unsteady in nature and further investigation into this phenomenon is required, especially in flow visualization field.

5.0 ACKNOWLEDGEMENT

The authors would like to thank the Ministry of Higher Education, Malaysia for supporting this research under the Fundamental Research Grant Scheme (FRGS) Vot. 0729.

6.0 REFERENCES

A. Benazza, E. Blanco and M. Abidat. 2007. 2d Detached-Eddy Simulation around Elliptic Airfoil at High Reynolds Number. *Journal of Applied Science.* Vol. 7. Issue 4. pp.547-552.

B.R. Munson, D.F. Young and T.H. Okiishi. 2010. Fundamentals of Fluid Mechanics. 6th Edition. John Wiley.

D.B. Sims-Williams, R.G. Dominy and J.P. Howell. 2001. An Investigation into Large Scale Unsteady Structures in the Wake of Real and Idealized Hatchback Car Models. *SAE Internationals*. 2001-01-1041.

G. Franck and J. D'Elia. 2004. CFD modeling of the Flow around the Ahmed Vehicle Model.

K. Sanisa and L. Davidson. 2003. Numerical Study of the Flow around a Bus-Shaped Body. *Transaction Journal of the ASME.* Vol.125.

S.R. Ahmed. 1981. Wake Structure of Typical Automobile Shapes. *Trans. ASME. J. Fluids Eng*. 103. 162-169.

S.R. Ahmed. 1983. Influence of Base Slant on the Wake Structure and Drag of Road Vehicles. *Trans. ASME. J. Fluids Eng*. 105. 429-434.

S. Watkins and G. Vino. 2008. The Effect of Vehicle Spacing on the Aerodynamics of a Representative Car Shape. *Journal of Wind Engineering and Industrial Aerodynamics* 96. 1232-1239.

T. Han. 1989. Computational Analysis of Three-Dimensional Turbulent Flow around a Bluff Body in Ground Proximity. AIAA J. 27(9). 1213-1219.

Y. Lui and A. Moser. 2003. Numerical Modeling of Airflow over The Ahmed Body. *Proceedings of the 11th Annual Conference of the CFD Society of Canada.* 507-512.