Evaluation of Vehicle Platooning Aerodynamics using Bluff Body Wake Generators and CFD

Hesham M. Ebrahim
Department of Mechanical & Construction Engineering
Northumbria University
Newcastle, UK
Sayed.Ebrahim@northumbria.ac.uk

Robert G. Dominy
Department of Mechanical & Construction Engineering
Northumbria University
Newcastle, UK
Robert.Dominy@northumbria.ac.uk

Pak S. Leung
Department of Mechanical & Construction Engineering
Northumbria University
Newcastle, UK
PS.Leung@northumbria.ac.uk

***Abstract*—Recent developments in sensing and communications between vehicles (V2V) and their surroundings have provided the technology to allow cars to operate autonomously or semi-autonomously in closely spaced 'platoon' formation without the risk of collision. This is known to reduce the aerodynamic drag and thus consequently limits the energy consumption and associated emissions. Although wind tunnel investigations have been performed to mimic platoon operations, most experimental evaluations of multiple vehicles in platoon are severely compromised by the restricted length of the wind tunnel test section. Therefore, the model scale must be reduced which decreases the measurement accuracy. The innovative solution presented here is to reproduce the flow structure that is created by a leading road car through the use of a 'bluff-body wake generator' with a much reduced length which eliminates the need to decrease the scale of the following test model. Validated computational fluid dynamics (CFD) data and analysis are presented to evaluate an optimized design of a wake generator based on the Ahmed model [1] and the effect of inter-vehicle spacing on the aerodynamic characteristics of the following vehicle. It is shown that accurate reproduction of the wake is possible at half the characteristic length, thus correctly determining the flow impact on the downstream model. This demonstrates that the bluff body wake generator provides a reliable approach that allows platooning studies to be performed without sacrificing aerodynamic resolution.**

I. INTRODUCTION

The aerodynamic benefits associated with vehicle platooning have been investigated over two decades. Most publications including [2] [3] [4] [5] [6] focused on either drag measurements or fuel consumption for a single vehicle geometry in different driving conditions (i.e. driving speeds, inter-vehicle distances, passing manoeuvers etc.) using common testing techniques. Normally, the conclusions would suggest that the vehicles drag or fuel efficiency is proportional to the spacing and thus keeping it to a minimum is desirable. Crucially however, the influence of vehicle spacing that relates to driver comfort (e.g. buffeting and panel vibration), cabin noise and vehicle stability when situated within the highly turbulent flow generated by cars in platoon have been usually neglected. Also, very little information has been published regarding the physical sources of the changes in flow structure that occur in platoon.

In recent years aided by the development of both V2V and LIDAR technologies, vehicle platooning has garnered more interest and has become a fundamental part of a development strategy to reduce greenhouse gas emissions [7]. Platooning investigations have evolved with more studies focusing on up-to-date vehicle models placed in different arrangements and configurations, which represent more realistic on-road situations [8]. As a reflection of that, Schito and Braghin [9] used idealized geometries that embody a typical hatchback, sedan, van and a truck to show the influence of car shape and inter-vehicle distances on drag reduction. They discovered using both wind tunnel and CFD techniques that the degree of drag reduction was dependent on the vehicle arrangement, inter-vehicle distance and drag coefficient in isolation. This means that the leading vehicle size and geometry can influence the overall performance gained by the platoon which is problematic, as vehicle arrangement is highly unpredictable due to the range of different vehicle geometries and sizes on the road, yielding a large number of possible arrangements. This suggests that studying two vehicles of similar geometry in tandem is insufficient to characterise the diverse aerodynamic interactions possible on the road. Essentially, a requirement to develop more representative reference models or bluff body wake generators that can reproduce similar wake structures to varying vehicle geometries and sizes should be established.

Previously, the experimental studies have been compromised by reduced model scale required to accommodate multiple vehicles within the confines of the wind tunnel test section, which limits the aerodynamic resolution and further reduces the Reynolds number. Previous studies conducted by Wilson [10], Dominy [11] and Newbon [12] all addressed this problem by using a short bluff body to replicate the wake structure from the leading vehicle without losing significant test section length. These wake generators contained the main features of the original vehicle rear-end and were used to investigate the aerodynamic performance of race cars as they are driven in close proximity in tandem. In these investigations the wake generators proved the capability to recreate the wake structure correctly for NASCAR and Formula One models. However no studies were conducted for passenger vehicles and reference models.

The objectives of the study presented in this paper are as follows: Firstly, a thorough CFD investigation to characterise the geometrical limits of wake generators based on the Ahmed model. Secondly, to validate a platoon simulation against experimental measurements and thirdly, to investigate the performance of the optimised wake generator in recreating the wake predictions found in the wind tunnel experiments.

II. CFD STUDY

A. Optimisation of Bluff Body Wake Generator

In order to establish the wake characteristics of a full length Ahmed model (Fig. 1) a series of CFD simulations were conducted using the software package, STAR-CCM+. The Ahmed model was based on a 25° slant angle and the simulation parameters are a replica of the experimental conditions conducted by Lienhart et al. [13]. Following a mesh sensitivity analysis and a comparison of turbulence models the adopted CFD settings for the optimisation of the wake generators are as shown in TABLE I. The Ahmed model geometry was modified to achieve the minimum characteristic length possible without inducing spurious changes to the wake structure. Three different lengths were generated (depicted in Fig. 2) corresponding to 0.75L, 0.5L and 0.4L of the original full length (i.e. 1L). For simplicity, the mounts on the Ahmed model lower surface were removed to eliminate any other geometrical change beside the length. Note the full length case was necessary to re-simulate; as removing the mounts could induce changes on the flow along the underbody.

TABLE II summarizes the drag coefficient values (for a frontal area of 0.112 m^2) that indicate only small deviations arising from the change in characteristic length. This agreement in drag coefficient is not surprising as the frontal area of the model remained constant and the wake structure relatively similar. However, the lift coefficient increased significantly due to the pressure difference along the body's upper and lower surfaces as shown in Fig. 3.

TABLE I. SIMULATION SETTINGS FOR THE WAKE GENERATOR

Modelling Parameters	**Adopted Settings**
Reynolds Number (based on length)	Re = 2.784×10^6
Grid Topology	Trimmed hexahedral mesh
Number of Cells	4.6×10^6
Domain (L_f/L_R/W/H)	3L/9L/2L/1.4L
Near Wall Treatment	Hybrid all y+ mesh
Time	Steady State
Viscous Regime	Turbulent
RANS Model	K-Epsilon Realizable (RKE)

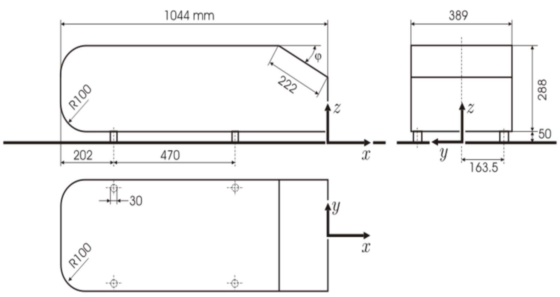

Fig. 1. The original Ahmed Model geometry with mounts [1]

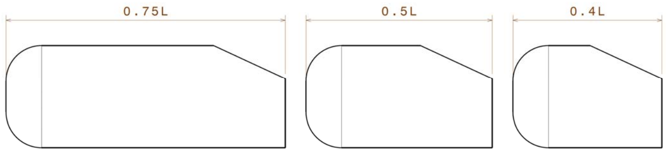

Fig. 2. The Ahmed model modified geometry without mounts

The changes in the wake structure are minimal for the 0.75L and 0.5L cases, with the velocity distribution (Fig. 4) showing similar trends to the simulated 1L case and experiment. Minor flow reversal is over-predicted by the turbulence model near the ground plane (between x/L=0.2 and x/L=0.4) as a result of the flow interaction between the flow diffusion from the Ahmed model underbody and the counter-rotating eddies at the vertical base. This interaction causes semi-periodic build up and collapse of the near wake, splitting off a packet of low pressure from the rotating eddies and convect along the ground plane. Such interaction has been previously discovered by Sims-Williams [14] during unsteady wind tunnel investigations of the flow around a 25° slant angle, and in these cases is surprisingly captured by the turbulence model. The simulated 0.4L case displayed a large exaggerated wake, influenced by the fast moving flow over the front radii and the sudden step change of the model height across the slat angle. This projects a large pressure drop across the model (Fig. 3), and therefore will not be considered further.

TABLE II. FORCE COEFFICIENT COMPARISON

Force Coefficient	**Exp.**	**1L**	**0.75L**	**0.5L**	**0.4L**
C_D	0.290	0.301	0.289	0.307	0.296
C_L	0.345	0.324	0.343	0.431	0.382

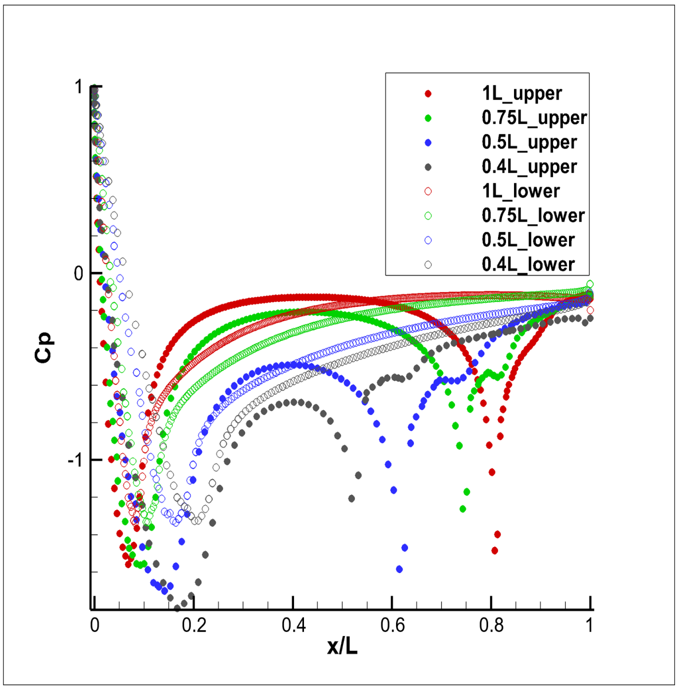

Fig. 3. The surface pressure distribution along the centreline

The secondary flow vectors of the 1L Ahmed model presented in Fig. 5 reveal that the exclusion of the mounts has no effect on the vortex centre generated at the C-pillars, although an additional vortex is produced that draws in the flow from the C-pillar vortices; rotating in a similar direction. This vortex is centred below the counter-rotating vortex and is inherently promoted by the stronger rotation of the C-pillar vortices along the centerline, and the increased velocity gradients at the model underbody due to the absence of the mounts. Consequently, this vortex restricts the height of the wake at the centerline and alters the closure of the wake at the base. It can be argued whether the correlation in drag coefficient is sensitive to the wake structure or only dependent on the near-wake prediction and surface pressure distribution. As for the 0.75L wake generator, the vortex produced is more closely related to experiment despite the lower portion of the wake being slightly inflated due to the increment in velocity at the underbody. The 0.5L wake generator appears to diffuse the flow from the underbody marginally faster towards the top right corner in comparison to 1L and 0.75L. This thins the wake in a more pronounced "bow" fashion.

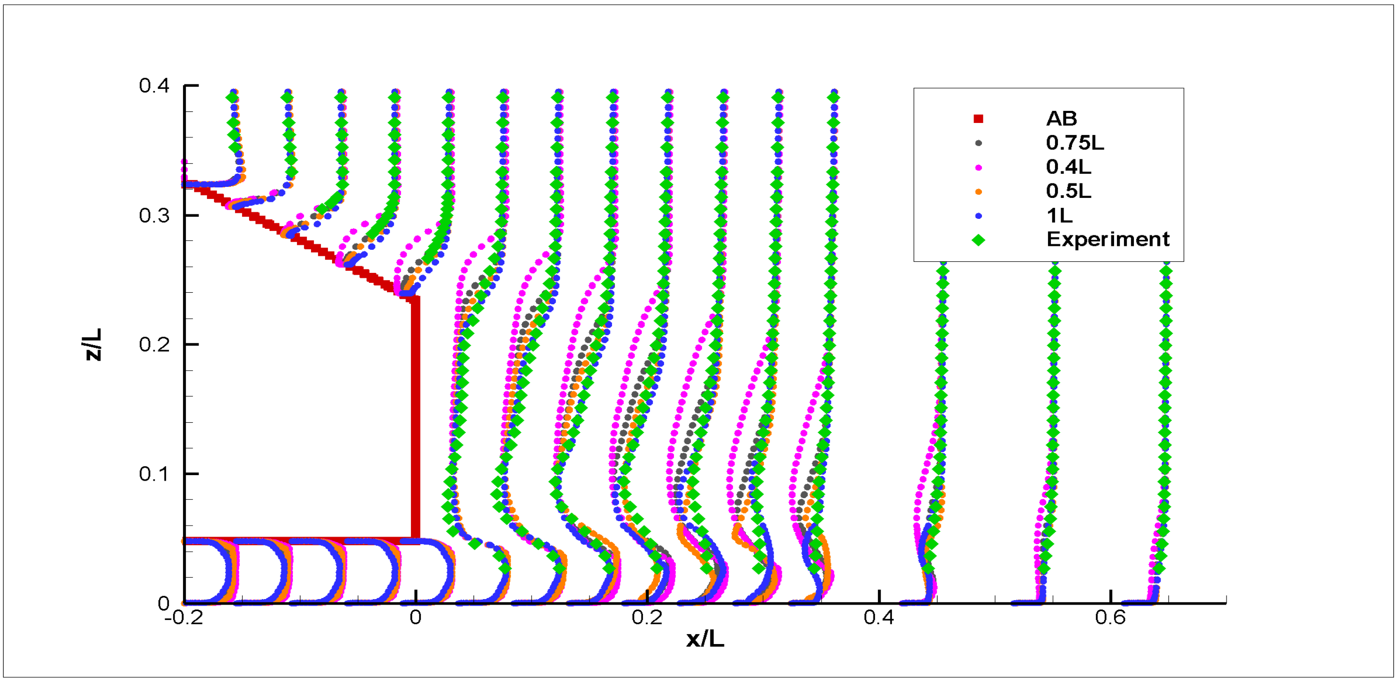

Fig. 4. Streamwise velocity distribution at the symmetry plane [13]

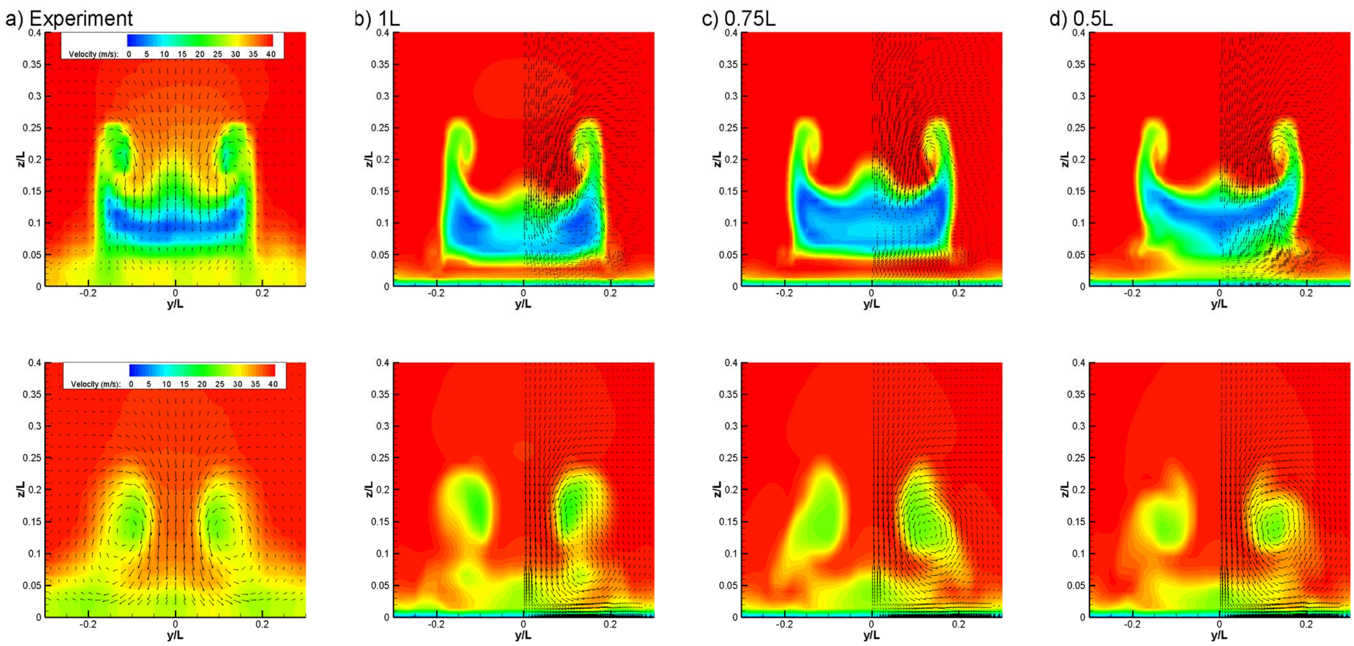

Fig. 5. Secondary flow vectors at x/L=0.192 (top), x/L=0.479 (bottom) [13]

Further downstream (x/L = 0.479), as shown in Fig. 4 (bottom), differences occur in the precise location of the vortex core for the 1L case due to the induced vortex centred below the C-pillar vortices as discussed earlier. This widened the flow rotation around the two vortex centres, compressing the velocity gradient around the vortex core and shifting it upwards from its original course. As for the 0.75L and 0.5L cases, the trailing vortices clearly match the experimental trend with a velocity increase in the lower perimeter of the vortices and a cleaner flow along the underbody due to the absence of the mounts.

Despite the minor differences in the secondary flow vectors described above, it is clear that the primary flow features are successfully recreated by the bluff body wake generators. This analysis concludes that the drag coefficient and the wake structure remains comparable to the experiment for both the 0.5L and 0.75L cases, therefore the 0.5L wake generator will be used for the platooning simulations to follow.

B. Wake Generator In Platoon

Placing two (or more) Ahmed models in tandem at close proximity may influence the familiar wake structures and aerodynamic forces; altering the overall aerodynamic performance. It is therefore necessary to compare the wake generator effects on a following model in comparison to a full length leading model. The experimental results conducted by Pagliarella [15] were used as a reference to develop the numerical simulations. The experimental parameters can be found in [16] and have been summarised in TABLE III. Note all unspecified simulation parameters are as previously set.

For the validation of the bluff body wake generator, only two inter-vehicle spacings were used. The closest spacing being 0.125 of the vehicle length (0.125L) and the furthest spacing being the full vehicle length (1L). For the 0.125L spacing, Fig. 6, reveals that the surface pressure acting on both the leading and following vehicles was predicted reasonably well for the full length Ahmed model and the optimised wake generator. However, CFD under-predicted the forebody pressure magnitude on the following vehicle centre and over-predicted the pressure over the lower section. Some changes in pressure on the backlight of the leading model were also apparent. These discrepancies are highly related to the flow instability in those regions as outlined experimentally by [15].

Nonetheless, it is clear that the step change of the model height across the slat angle causes the flow to accelerate with evidence that the C-pillar vortices are formed by the separating shear layers at the backlight sides (Fig. 6). However, the formation of those vortices is considerably inhibited in comparison to the Ahmed model in isolation due to the flow impingement on the following model forebody sides and the back pressure acting on the leading model rear that induces flow separation (Fig. 7). This inevitably reduced the downwash that aids flow reattachment for a slant angle of 25^{o}. Such observations are consistent with Pagliarella [15] whom analysed the flow experimentally. Fig. 8 indicates that the flow diffusion from the underbody is restricted; due to the presence of a following model that reduces the lower eddy recirculation

TABLE III. SIMULATION SETTINGS FOR THE PLATOON FORMATION

Modelling Parameters	Adopted Settings
Model Scale	¾ scale
Reynolds Number	Re = 1.85×10^{6}
Domain (L_f/L_R/W/H)	3L/9L/4.23L/2.82L
Turbulent Intensity	1.8%

(behind the lead model) and minimises the mass flow rate channeled under the following model triggering detachment.

At 1L spacing, the numerical simulation depicted by Fig. 9 showed close agreement to the experiment with the surface pressure acting on the following model being almost identical to the experiment using both the full length and wake generator models. Despite the minor differences in pressure magnitude, the formation of C-pillar vortices is no longer compressed by the back pressure and appears to regain energy showing a similar trend to an Ahmed model in isolation. This can be attributed to the increased inter-vehicle spacing. The greater drag reduction for the leading model has undoubtedly reduced as the longitudinal vortices and large recirculating region at the base has redeveloped as shown in Fig. 10 and Fig. 11. This equally influenced the following model and allowed the low pressure at the leading edge to recover which causes drag reduction in comparison to the 0.125L spacing case.

In both scenarios, replacing the leading model by the optimised wake generator produced almost identical wake measurements on the downstream model (Fig. 8 and Fig. 11). Also, the numerical simulation was able to predict the flow topology at 1L spacing more accurately than at 0.125L spacing where the wake mixing is prominent.

Generally, the drag coefficient remained comparible for the leading model at both inter-vehicle spacings. However, a drag variation of 16 and 50 drag counts was measured for the following full length model at 0.125L and 1L spacings respectively (TABLE IV). It is well known that CFD over-predicts the drag coefficient in comparison to wind tunnel measurements and in this case was demonstrated by the increased magnitude of surface pressures. Nonetheless, the results obtained confirm the drag reduction benefits associated with vehicle platooning; indicating an overall reduction in drag as the distance between the vehicles is reduced. The trend also illustrates that the performance is mainly gained by the leading model for both cases whilst disadvantagous for the trailing model. Regardless of that, the study confirms the applicability of wake generators in replicating the wake structure and pressure forces that acts on the following model, which will be adopted in the wind tunnel test programme to follow.

TABLE IV. SUMMARY OF DRAG COEFFICIENT

Vehicle Spacing	0.125L		1L	
Vehicle Arrangement	***Leader***	***Follower***	***Leader***	***Follower***
Experiment [17]	0.105	0.405	0.279	0.315
CFD Full Model	0.091	0.421	0.269	0.365
CFD Wake Generator	0.094	0.402	0.250	0.361

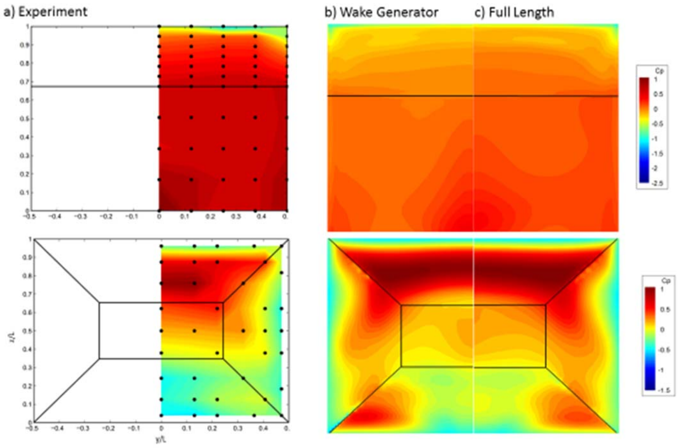

Fig. 6. Comparison of the surface pressure distribution for 0.125L spacing of the leading model backlight (top), following model forebody (bottom) [16]

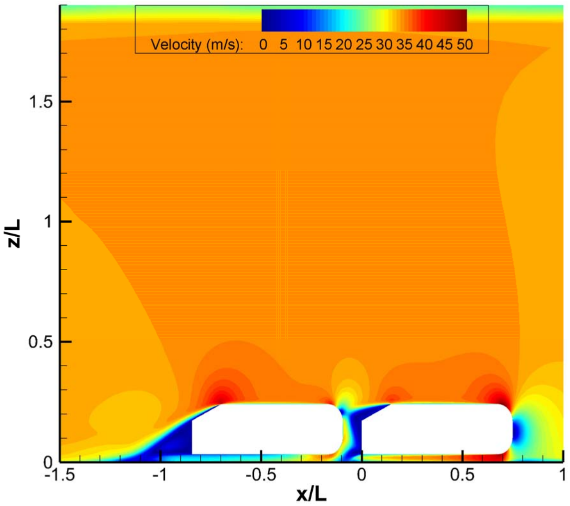

Fig. 7. Streamwise velocity distribution at the symmetry plane for full length leading model at 0.125L spacing

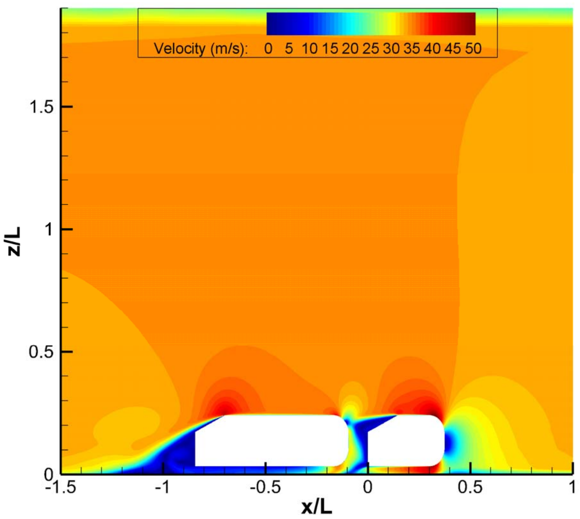

Fig. 8. Streamwise velocity distribution at the symmetry plane for an optimised leading wake generator at 0.125L spacing

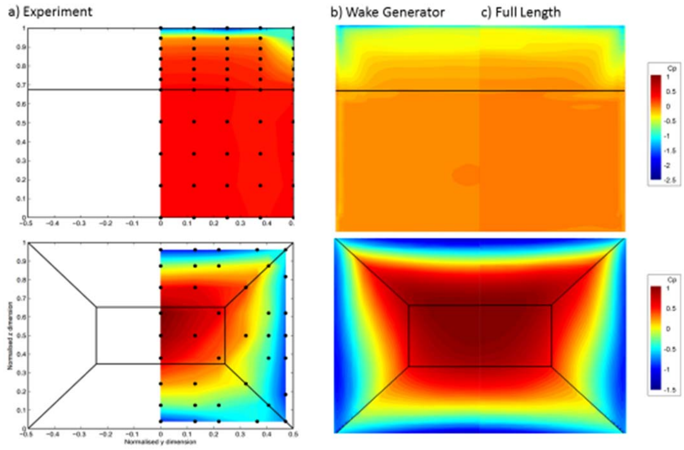

Fig. 9. Comparison of the surface pressure distribution for 1L spacing of the leading model backlight (top), following model forebody (bottom) [16]

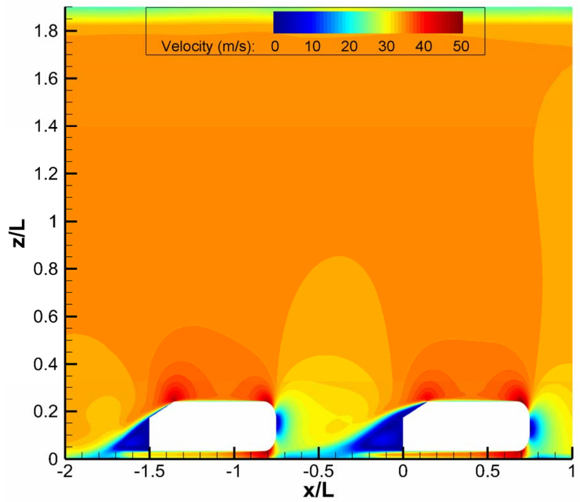

Fig. 10. Streamwise velocity distribution at the symmetry plane for full length leading model at 1L spacing

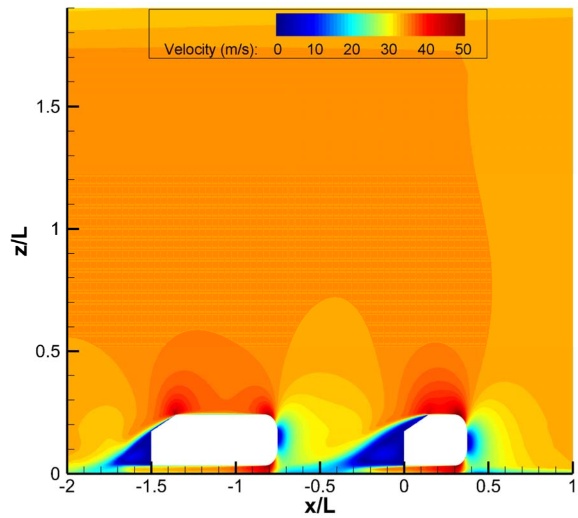

Fig. 11. Streamwise velocity distribution at the symmetry plane for an optimised leading wake generator at 1L spacing

III. Conclusion

The optimisation process of a bluff body wake generator based on the Ahmed model has been demonstrated. It was found that the wake generator is capable of recreating accurately the wake structure that is dominated by a pair of vortices generated at the C-pillars with only half the characteristic length. Further reduction to the characteristic length resulted in a changed wake structure and was therefore not considered further.

CFD comparison between the optimum wake generator and a full length Ahmed model for two inter-vehicle spacing configurations revealed very consistent measurements. For both 0.125L and 1L inter-vehicle spacings, the wake generator showed very similar flow characteristics acting upon the following model. Although slight variation in the pressure magnitude was detected in relation to the experiment, the drag coefficient corresponded well with 16 and 50 drag count increase for the 0.125L and 1L cases respectively.

Despite the inaccuracies of CFD it was proven that it is capable of developing a wake generator that provides a reliable approach to simulate platoon cases in the wind tunnel without sacrificing aerodynamic resolution.

Acknowledgments

The authors gratefully acknowledge the support of CD-Adapco in particular Konstantinos Karantonis for supplying individual license to assist this project and Prof. Simon Watkins for sharing the experimental results conducted by Dr. Riccardo Pagliarella.

References

[1] S. Ahmed, G. Ramm and G. Faltin, "Some Salient Features of the Time-Averaged Ground Vehicle Wake," *SAE International,* no. 840300, 1984.

[2] M. Zabat, S. Frascaroli and F. Browand, "Drag Measurements on a Platoon of Vehicles," Presented to the California Department of Transportation, 1994.

[3] A. Chen, O. Savas and K. Hedrick, "Transient Vehicle Aerodynamics in Four-Car Platoons," California PATH, 1997.

[4] P. Hong, B. Marcu, F. Browand and A. Tucker, "Drag Forces Experienced by Two, Full-Scale Vehicles at Close Spacing," *SAE International,* no. 980396, 1998.

[5] B. Browand, "Aerodynamic Forces Experienced by a 3-Vehicle Platoon in a Crosswind," *SAE International,* no. 1999-01-1324, 1999.

[6] F. Browand, J. McArthur and C. Radovich, "Fuel Saving Achieved in the Field Test of Two Tandem Trucks," *California PATH Program,* no. "ECB-ITS-PRR-2004-20", 2004.

[7] A. Davila, E. del Pozo, E. Aramburu and A. Freixas, "Environmental Benefits of Vehicle Platooning," *SAE International,* no. 2013-26-0142, 2013.

[8] A. Davila, E. Aramburu and A. Freixas, "Making the Best Out of Aerodynamics: Platoons," *SAE International,* no. 2013-01-0767, 2013.

[9] F. Braghin and P. Schito, "Numerical and Experimental Investigation on Vehicles in Platoon," *SAE International,* no. Politecnico di Milano, pp. SAE Paper No. 2012-01-0175, 2012.

[10] M. Wilson, R. Dominy and A. Straker, "The Aerodynamic Characteristics of a Race Car Wing Operating in a Wake," *SAE International,* no. 2008-01-0658, 2008.

[11] R. Dominy and G. LeGood, "The use of a Bluff Body Wake Generator for Wind Tunnel Studies of NASCAR Drafting Aerodynamics," *SAE International,* no. 2008-01-2990, 2008.

[12] J. Newbon, R. Dominy and D. Sims-Williams, "Investigation into the effect of the wake from a generic formula one car on a downstream vehicle," *The International Vehicle Aerodynamics Conference,* pp. 213-223, 2014.

[13] H. Lienhart, C. Stoots and S. Becker, "Flow and Turbulence Structures in the Wake of a Simplified Car Model (Ahmed Model)," Lehrstuhl für Strömungsmechanik (LSTM), Germany, 2002.

[14] D. Sims-Williams and B. Duncan, "The Ahmed Model Unsteady Wake: Experimental and Computational Analyses," *SAE International,* no. 2003-01-1315, p. , 2003.

[15] G. Vino, "An Experimental Investigation into the Time-Averaged and Unsteady Aerodynamics of a Simplified Passenger Vehicle in Isolation and In Convoys," PhD Thesis, RMIT University, 2005.

[16] R. Pagliarella, "On the Aerodynamic Performance of Automotive Vehicle Platoons Featuring Pre and Post-Critical Leading Forms," School of Aerospace, Mechanical and Manufacturing Engineering RMIT University, 2009.

[17] R. Pagliarella, S. Watkins and A. Tempia, "Aerodyanmic Performance of Vehicles in Platoons: The Influence of Backlight Angles," *SAE International,* no. 2007-01-1547, 2007.