Impacts of The Gap Size Between Two Bluff bodies on The Flow Field Within the Gap

T. Charles, Y. Lu and Z. Yang
Department of Engineering,
College of Engineering and Technology
University of Derby,
Derby, UK
Contact: t.charles@derby.ac.uk

Abstract

When two bluff bodies is in close tandem, i.e, the distance between blocks D is less than half of the height of the body H, the flow field is very similar to that in the gap between tractor and trailer in a truck, and hence understanding such a flow field would help to reduce the gap drag of a truck. This paper presents a numerical study of the flow field in the gap between two identical cubes in tandem arrangement, in particular, focusing on the impact of gap size on the flow field within the gap and around the two cubes. Simulations have been carried out for four different gap sizes. The numerical model has been validated first against a test case before further studies are carried out to study the impact of gap width on the flow field. The predicted mean velocity profiles compare well against the experimental data for the validation test case. Detailed analysis has been carried out to reveal the change of the flow fields when the gap size changes, leading to a better understanding of the drag coefficient variations for the four cases studied.

Nomenclature

D (m)	gap between two cubes
H(m)	height of the cube
x(m)	streamwise distance
y(m)	vertical distance
L(mm)	length of validation test case
l(mm)	width of validation test case
h(mm)	height of validation test case
u(m/s)	mean velocity

Introduction:

Heavy trucks are considered aerodynamically inefficient compared to other ground vehicles due to their un-streamlined body shapes with more than of 60% engine power being required to overcome the aerodynamic drag at 60m/hr. In contrast, a passenger car under the same driving conditions, requires approximately 4 times less of engine power to overcome the drag. Therefore aerodynamics drag reduction is hugely important for trucks. In order to reduce aerodynamic drag we need to understand the drag distribution and the associated flow field. The percentage of drag distribution generated from a truck is usually split as: the front face of a tractor generates a drag of 25%, the gap between the tractor and trailer generates a 20% drag with the rear of the trailer generating another 25% drag, and the rest 30% of the total drag is due to the underside of the truck [1]. This vehicle type is in use for transporting majority of the goods across a country an appreciable drag reduction has been achieved by modifying the design or by adding simple ad-on devices [2].

An experimental study on the aerodynamic drag of a simplified tractor-trailer was carried out and it was found that a significant drag reduction could be achieved when two different drag-reducing devices were employed [3]. There have been many other studies on the use of drag-reducing devices but this study will be focusing on the flow field around two bluff bodies in close tandem, which is similar to the flow field in the gap between tractor and trailer in a truck. Many studies have been conducted for two-dimensional bluff bodies and a review has been provided by Rockwell [4]. Sakamto and Haniu [5] carried out an experimental study on the drag variation of two square cylinder in a tandem arrangement and identified a sudden increase in the drag at a certain spacing between the two cylinders, denoted as the critical spacing. Chen and Liu [6] showed that the spacing has a significant impact on the flow field around two square cylinders in a tandem arrangement. Mahbub *et al.* [7] also identified the existence of a critical spacing for two square cylinders in a tandem arrangement at which a sudden change in the drag occurred. Kim *et al.* [8] conducted a detailed experimental study on two square cylinders in a tandem arrangement and the velocities, turbulence intensities, Reynolds shear stresses and turbulent kinetic energies were measured using PIV. Their results confirmed the findings of previous studies that the flow characteristics can be divided into two drastically different flow patterns separated by a critical spacing.

There have been relatively less studies on three-dimensional bluff body geometries [9] and the flow field around three-dimensional bluff bodies are inherently very complex with the formation of spanwise and streamwise vortices, which interact with each other too. It was shown in [9] that the structure of the turbulent field in the cavity region was significantly different from that in a two-dimensional case.

The current work focus on the detailed study of the flow fields in the gap region between two identical cubes in tandem arrangement, specifically the effect of gap size (spacing) on the flow field inside the gap and around the two cubes. The improved understanding of the flow field in the gap region will lead to a better understanding of the aerodynamic drag generation mechanism.

Methodology and computational setup

Numerical simulation setup is based on the experimental study carried out by Martinuzzi and Havel [9]. Two identical cubic blocks of height H = 40 mm are arranged in tandem which is placed in a wind tunnel. Various gap sizes between two blocks, D/H = 0.25, 0.5, 0.75, 1, have been used to investigate the effect of gap size on the flow field. The same computational domain is used for each case with 2H upstream and 7H downstream. The width and height of the domain are 12H and 6H. Simulations have been performed actually in half of the domain since the RANS approach is employed in the current study so that the simulated averaged flow field is symmetric about the central plane in the span wise direction. The working fluid is air and the inlet velocity is 8.8m/s, corresponding to a Reynold number of 22,000 based on the cube height. An appropriate polyhedral mesh of about 1.5 million cells has been generated with a good resolution in the wake, boundary layer, the gap and around the bluff body regions.

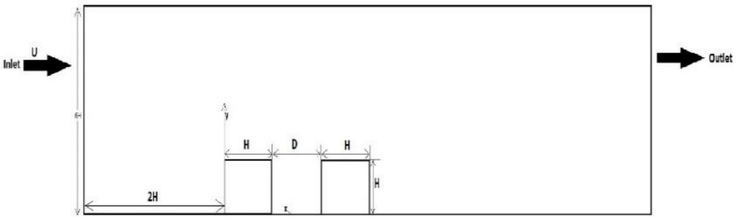


Figure 1: Schematic drawing on the symmetry plane of two cubic tandem obstacles subjecting to air flow in a windtunnel. Cubic obstacle size H = 40.0 mm and. D is the gap between two obstacles and D/H varies as 0.25, 0.50, 0.75 and 1.0.

The CFD code used in the current study is STAR_CCM+ and the well tested k-ϵ turbulence model is chosen with all y+ wall treatment - a hybrid wall treatment combining high y+ wall treatment for coarse meshes, and the low y+ wall treatment for fine meshes.

Validation

The numerical model used in the current study was firstly validated against an experimental study [10], where two rectangular obstacles of 88 mm(h) x 90 mm(l) x 59 mm(L) are in tandem arrangement. The gap between two obstacles is half its height. The inlet velocity is 8.0 m/s, corresponding to a Reynolds number of 46,000. Figure 2 shows the predicted mean streamwise velocity profile on the symmetrical plane at position of 30 mm away from back of the 1st obstacle and the experimental data at several positions. It can be seen that the predicted velocity profile compares well with the experimental data at the same streamwise location denoted as open circles in the figure. The experimental data also show that the velocity profiles at different streamwise positions are quite similar too.

Results and Discussion

a). General flow features

Using the case of D/H = 0.5 as an example, Figure 3 show the flow field on the symmetry plane and it can be seen that very complex flow structures are generated. The flow separates at the leading edge of the upstream obstacle and reattches to top edge of the downstream obstacle forming a single semi elliptical shaped recirculation region at the top of both the obstacles.

This acts like a single rotating vortex at the top face of both the obstacles and it seems that there is not a strong interaction between this vortex and the flow inside the gap.The center of this vortex is approximately 0.01m above the downstream obstacle and slightly closer to the upstream obstacle in the streamwsie direction. In the wake region behind the second obstacle, a typical vortex behind the downstream obstacle is observed which is commonly reported in the single obstacle and backward facing step problem. The flow field in the gap region is quite complex a large vortex fromed in the top half occupying almost 80% of the total gap region, while some small flow structures formed in the bottom half near the wall.

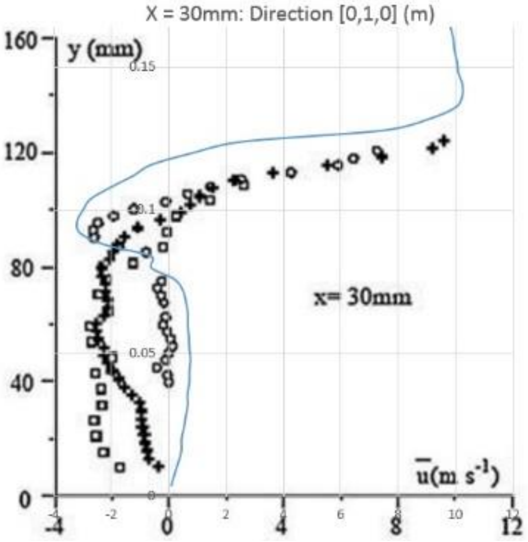


Figure 2:Mean velocity profile at one longitudinal section, x=30mm in the symmetry plane in validation study.

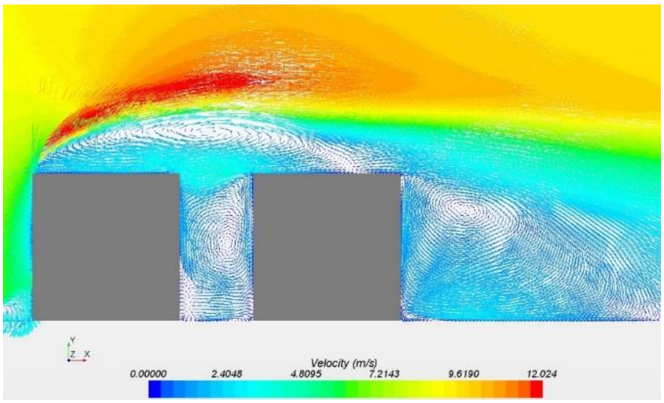


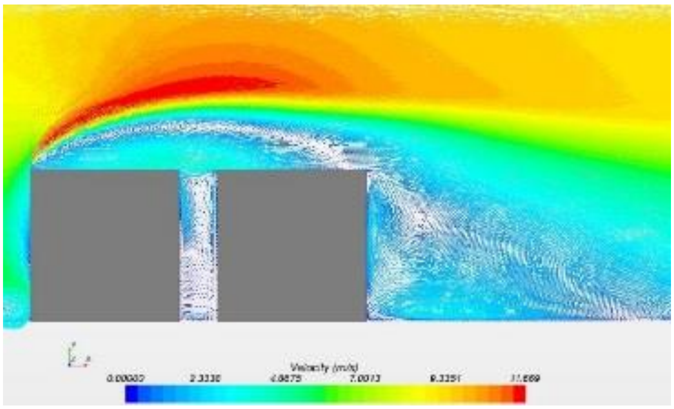
Figure 3 Mean flow velocity vector plot on the symmetry plan around two tandem obstacles.

b). Effects of the gap size on the flow pattern.

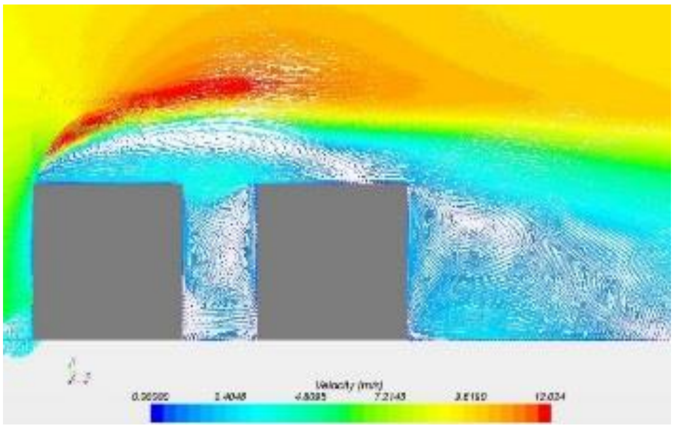
When the gap size between two tandem obstacles changes, the flow patterns within the gap changes too. In the case of D/H = 0.25, the primary recirculation zone on top of the twin obstacles and the wake flow behind the second obstacle look similar to those in the case of D/H = 0.5. However, the large vortex inside the gap region changes significantly as it becomes much smaller and moves upwards, occupying less than 50% of the total gap region. More small flow structures form at the bottom half of the gap region compard with the D/H = 0.5 case. For both cases there is no strong interaction between the primary vortex on top of the twin obstacles and the flow inside the gap region.

However, when the gap size increases to D/H = 0.75, it can be seen that there is much more interaction between the primary vortex on top of the twin obstacles and the flow inside the gap region as more flow enter the gap region. The single domiant vortex clearly visible inside the gap region for the previous cases (D/H = 0.25, 0.5) is hardly observable anymore and the flow becomes more complex.

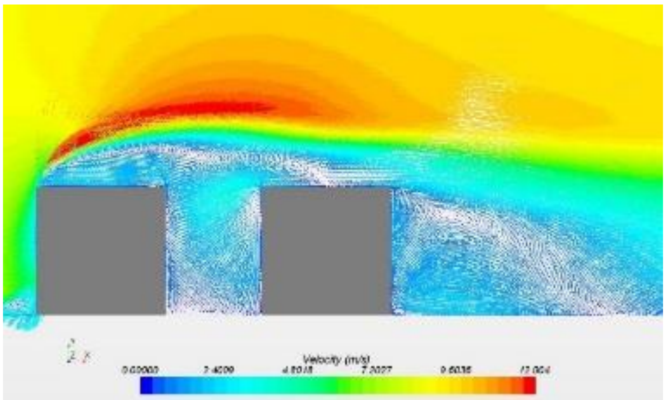
There is a significant change in the the primary recirculation zone on top of the twin obstacles as it nearly breaks into two resirculation zones. The flow in the wake region behind the second obstacle looks similar to those in the previous cases. when the gap size increases further to D/H = 1.0, the primary vortex on top of the twin obstacles breaks into two or three vortices, leading to a much stronger interaction between the flow above the obstacles and the flow inside the gap region as not only more flow enters the gap region but also penetrate more deeply into the gap region. Nevertheless the gross flow features in the wake region behind the second obstacle more or less the same as those in the other three cases.



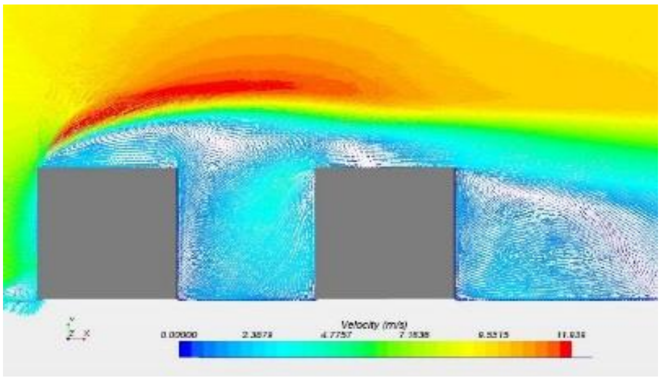
(a)



(b)



(c)



(d)

Figure 4: Comparison of flow pattern within the gap region for different gap size.

Figure 5 shows the predicted streamwise velocity profiles inside the gap region on the symmetry plane at x = 0.0045 m, 0.005m away from the backward face of the upstream obstacle. When D/H = 0.25 in the top half of the gap region the velocity is positive and the reverse flow happens in the bottom half and when the gap size increases to D/H = 0.5 it is almost the opposite as reverse happens mainly in the top half of the gap region while the velocity is positive in the bottom half. When D/H = 0.75 there is hardly any forward flow as the velocity is negative acorss the whole gap height apart from a tiny region at the bottom where velocity is positive. When the gap size increases further to D/H = 1.0 reverse flow occurs across the whole gap height which is consistent with the flow field presented in Figure 4 as more flow enter the gap region and penetrate deep into the region.

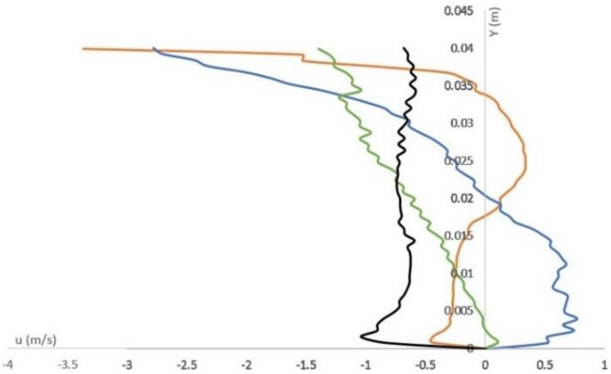


Figure 5 Mean Velocity plot on the symmetry plane at x = 0.045m (a) D/H = 0.25 (orange), 0.5 (blue), 0.75 (green) and 1 (black)

c). Effects of the gap size on the drag coefficient of the downstream cube.

As mentioned in the introduction that the motive for this study is towards a better undertsnading of the flow field in the gap region and around the cubes, leading to a better understanding of aerodynamic drag generation mechanism of the trailer of a truck. Hence the focus in the current study will be only on the drag coefficient of the downstream cube. The drag coeffcients for all four cases are shown in Figure 6 and it can be seen the current predictions compare well the results obtained by Martinuzz and Havel [9] for the range of the gap size studied. It can also be seen that the drag coefficient changes little for the four different gap sizes studied although it seems that the flow fileds have changed. This may indicate that the so called critical spacing as mentioned before has not been reached yet and hence in the current study there is no sudden jump in the drag coefficient. However, the results obtained by Martinuzz and Havel [9] do not show a sudden jump in the drag coefficient either, just a gradual increase when the gap size incrases fruther from D/H = 1. This is quite different from many previous studies for two-dimensional cases mentioned above where a sudden increase in the drag coefficients is clearly observed. Further investigation is definitely needed for three-dimensional cases with a wider gap size range to clarify it and this will be carried out in the near future.

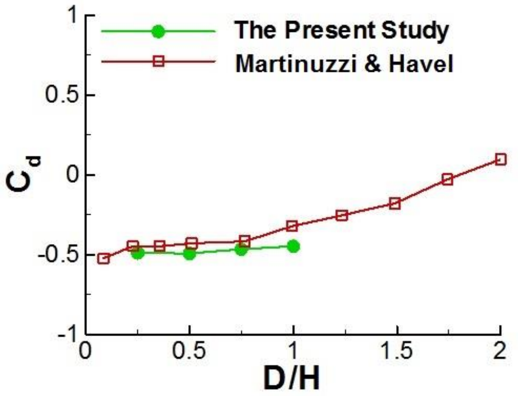


Figure 6: Drag coefficient of downstream cube

Conclusion

This paper describes a numerical study of the flow field in the gap between two identifical cubes in tandem arrangement. The numerical results are first validated against experimental data and the predicted results compare well the experimental data, demonstrating the accuracy of the numerical model used in the current study. Four further simulations have been carried out for four different gap sizes and the main findings from this study are:

- Complex flow fields have been visualized showing various vortices formed at different regions.
- The flow structures in the wake region behind the downstream cube is more or less the same for all the four cases studied (D/H = 0.25, 0.5, 0.75, 1.0).
- A single recirculation zone is formed on top of the two cubes when D/H = 0.25 and 0.5, and the interaction between the flow in this recirculation zone and the flow inside the gap region is very weak.
- The single recirculation zone starts to break up when the gap size increases further and when D/H = 1.0, it breaks into two or three vortices and a strong interaction is observed between the flow on top of the two cubes and the flow inside the gap region.
- The drag coefficient changes little with the four gap sizes studied but no jump has been observed.

References:

[1] R.M. Wood, “A discussion of a heavy truck advanced aerodynamic trailer system,” Presented at *Int. Symp. Heavy Veh. Weights Dimens.*, 9th, University Park, PA, 2006

[2] H. Choi, J. Lee and H. Park, “Aerodynamics of Heavy Vehicles,” *Ann. Rev. Fluid Mech.*, vol. 46, pp. 441-468, 2014.

[3] J.W. Allan, “Aerodynamic Drag and pressure measurement on a simplified tractor-trailer model, ”*Journal of Wind Engineering*, vol. 9, pp. 125-136, 1981.

[4] D. Rockwell, “Vortex body interactions,” *Ann. Rev.Fluid Mech.*, vol. 30, pp. 199-229, 1988.

[5] H. Sakamoto and H. Haniu, “Effects of free stream turbulence on characteristics of fluctuating forces acting on two square prisms in tandem arrangement,” *J. of Fluids Eng., Transcations of ASME,* vol. 110, pp. 140-146, 1988.

[6] J.M. Chen and C.H. Liu, “Observations of hysteresis in flow around two square cylinders in a tendem arrangement,”*Wind Eng. & Ind. Aerodyn.*, vol. 90, pp. 1019-1050, 2002

[7] M.D. Mahbub, M. Moriya, H. Sakamoto and K. Takki, Supression of fluid forces acting on two square prisms in a tandem arrangement by passive control of flow, *Fluids & Structures*, vol. 16, pp. 1073-1092, 2002.

[8] M.Y. Kim, D.K. Kim, S.H. Yoon and D.H. Lee “Measurement of the Flow field around two square cylinders in a tandem arrangment,” *Journal of Mechanical Science and Technology*, vol. 22, pp. 397-407, 2008.

[9] R.J. Martinuzzi and B. Havel, “Vortex shedding from two surface mounted cubes in tandem,” *International journal of Heat and Fluid flow, vol. 25,* pp. 364-372, 2004.

[10] N.M. Said, H. Mhiri, C. Philippe, G.L. Palec and B. Philippe, “Wind tunnel investigation and numerical simulation of the near wake dynamics for rectangular obstacles,” *Environmental Engineering Science,* vol. 25, pp. 1037-1060, 2008.