A Study into the Influence of the Car Geometry on the Aerodynamic Transient Effects Arising in a High Rise Lift Installation

Hayder Al-Jelawy[1], Stefan Kaczmarczyk[1], Seyed Mirhadizadeh[1], Dhirgham AlKhafaji[2]

[1]School of Science and Technology, Department of Engineering and Technology, The University of Northampton, UK.
[2]College of Engineering, The University of Babylon, Iraq.

Keywords: Computational fluid dynamics, car geometry, turbulent flow, computer simulation, aerodynamic performance

Abstract: One of the main goals in designing a high-speed lift system is developing a more aerodynamically efficient car geometry that guarantees good ride comfort and reduces energy consumption. In this study, a three-dimensional computational fluid dynamics (CFD) model has been developed to analyse an unsteady turbulent air flow around two cars moving in a lift shaft. The paper is focused on transient aerodynamic effects arising when two cars pass each other in the same shaft at the same speed. The scenarios considered in the paper involve cars having three different geometries. Aerodynamic forces such as the drag force that occur due to the vertical opposite motions of the cars have been investigated. Attention is paid to the airflow velocity and pressure distribution around the car structures. The flow pattern in the boundary layer around each car has been calculated explicitly to examine the flow separation in the wake region. The results presented in the paper would be useful to guide lift designers to understand and mitigate the aerodynamic effects arising in the lift shaft.

1 INTRODUCTION

The fast development of super-high-speed elevators has been facing significant challenges related to aerodynamic problems, such as high air resistance to the car movement, vibration of the lift car, excessive pressure fluctuation and noise generated inside the car as it is moving along the shaft. These problems occur because of the high-speed air flow around the moving car which could be increasing around the sharp edges. The ride quality is very important for the passengers' safety and comfort. Accordingly, it is essential to understand the aerodynamic forces and mitigate their effects.

In their work, Matsukara, Y. et al. [1] and Teshima, N. et al. [2], studied two types of noise. They stated that the mechanical noise is much smaller than the aerodynamic noise for high-speed lifts. In order to reduce the aerodynamic noise, Matsukara used a streamlined cover at both sides of the lift car (top and bottom). On the other hand, Toshima studied the impact of removing the apron which has to be installed at the bottom of each lift car due to legal requirements. Eventually, the noise was reduced in a range of (4.1 – 4.3 dB (A)) in Matsukara's work. Teshima was able to reduce the aerodynamic noise by producing a guide plate for the apron. They carried out two experiments in wind tunnels where the cars are stationary facing dynamic air.

Bai, H. et al. [3], have drawn attention to the fact that using wind tunnels is considered to be a total deviation from the real situation where the car and the air are dynamic. Thus, four different shapes of moving cars inside a cylindrical hoistway were studied. According to the consideration of average pressure difference, they considered the proper car shapes to be parabolic, spherical, conical and cylindrical respectively. This work did not take into account the fact that real cars and hoistways are rectangular in shapes, as they established that both the car and the hoistway are cylindrical in shape.

A numerical simulation has been done by Shi, L. et al. [4]. Their work was focused on a two-dimensional model of unsteady turbulent boundary layer flow around a lift car passing a counter-weight in the same shaft with different velocities and horizontal gaps. They found a severe increase of aerodynamic forces when the car passes the counter-weight.

Based on the 2-Dimensional work of Wu, R. et al. [5], the Coriolis force is much smaller than the lateral aerodynamic buffeting force when two conveyances pass each other.

In 2015, Wu, R., et al. [6], simulated a 3-Dimensional work to compare the lateral aerodynamic buffeting force and the clearance size of two kinds of rope-guided conveyances (mine lift and mine cages). Their study shows that the aerodynamic buffeting effect is directly proportional to the clearance size between the conveyances.

Mirhadizadeh, S. et al. [7], developed a computational software platform for high-speed lift systems by using MSC Dytran solver. Their model predicted the aerodynamic interactions in high-rise high-speed lift systems by utilizing CFD and Multibody Dynamics techniques.

According to the previous studies, the aerodynamic performance of high-speed lifts has become very important as lifts are getting faster in order to achieve the best design. Due to the aerodynamic influence, a three-dimensional aerodynamic model is developed in order to have a good understanding of lift cars passing each other in one shaft with different geometries at the same speed and with the same horizontal clearance between them.

2 GOVERNING EQUATIONS

A three-dimensional incompressible flow has been considered in order to have a better understanding of the flow. The transport flow variables are governed by two basic physical principles which are the conservation of mass and momentum. These variables are the pressure (p) and the flow velocity (u_i). Mathematical statements of the fluid physical principles are called Navier-Stokes (N-S) equations and shown in equations (Eq. 1, Eq. 2).

$$\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0. \qquad i = 1,2,3. \tag{1}$$

$$\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (u_i \rho u_j)}{\partial x_j} + \frac{\partial p}{\partial x_i} - \frac{\partial \tau_{ij}}{\partial x_j} - F_i = 0 \qquad i,j = 1,2,3. \tag{2}$$

Where ρ is the mean mass density, F_i represents the body forces, p is the pressure and τ_{ij} is the shear stress in the fluid.

3 THE MODEL DESIGN AND COMPUTER SIMULATION

Three simplified CAD designs of a lift car are shown in Figure 1. It shows the geometries of lift cars. The lift's design depends on the shape of the host building, according to Bai et al. [1], most of the lift cars and hoistways are rectangular in shape. Therefore, this study considers the rectangular shape to be investigated with different top and bottom aerodynamic shroud shapes of the lift cars.

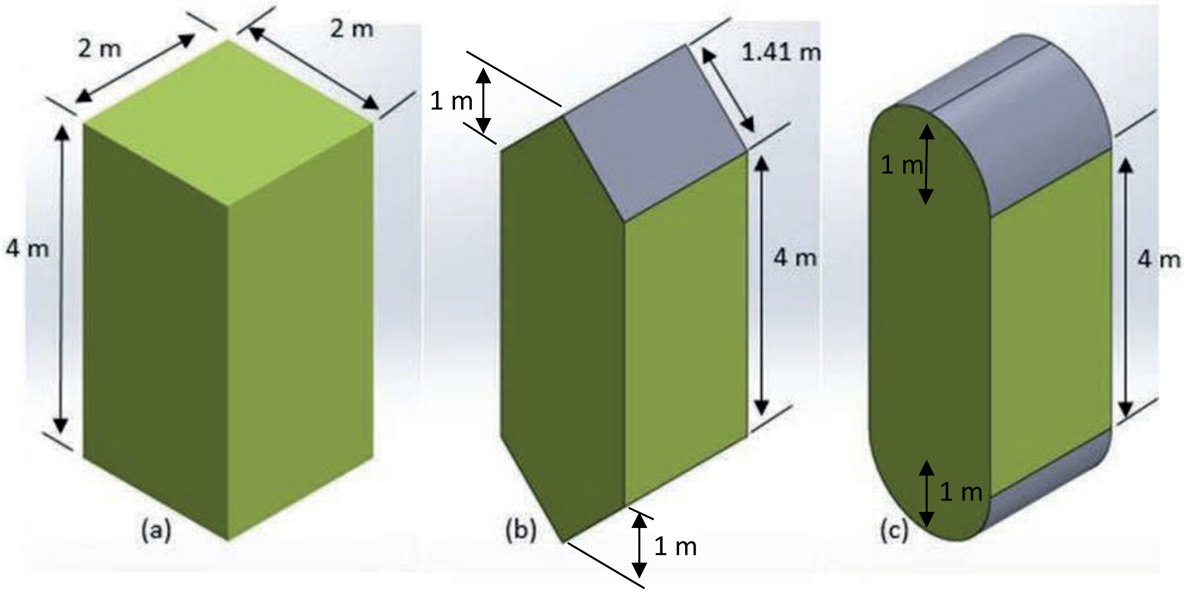


Figure 1 Schematic layout of three different shapes of the shroud (a) flat; (b) triangular; (c) hemi-cylindrical

The computer simulation has been implemented in MSC Dytran commercial software system [8]. The system's fluid solver based on the Finite Volume Method (FVM) is used to generate Eulerian mesh which is then used to model the dynamic motion of the air around the lift car.

The cars are considered to be rigid bodies with masses 2000 kg each. The air is an ideal gas with properties as follows: density is 1.2041 $kg\ m^{-3}$, specific heat ratio is 1.401 at 20 $°C$ and the gas constant is 287 $J\ kg^{-1}K^{-1}$. In order to have a simple interpolation between the grids, the Cartesian square grids have been applied in order to reduce the computer time. The interface between the car and the air grids moves at the same speed. In the present simulation, accurate flow simulations have been done by taking into account the grid resolution. The unsteadiness of the flow has been resolved by setting the integration time step at $1 \times 10^{-4}\ s$.

The total number of grid points is approximately 127,000. Lifts are located 12m vertically apart from each other in order to reduce the computational cost. Figure 2 shows the cross-sectional area of the hoistway/cars layout.

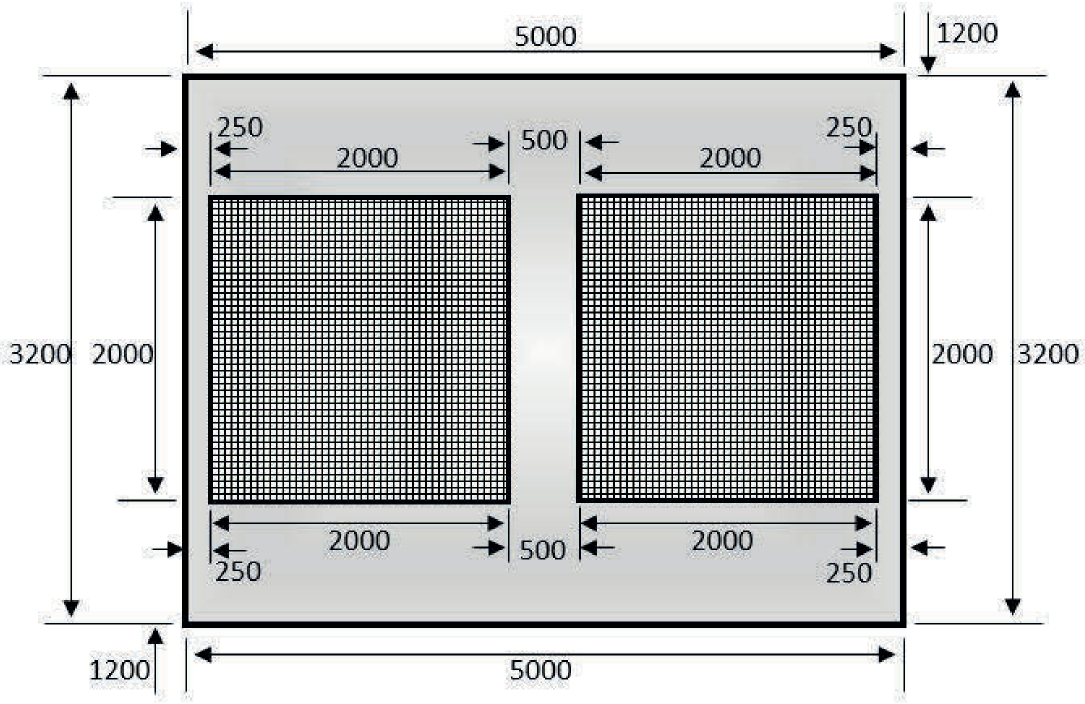


Figure 2 Cross-sectional area of the simplified cars/hoistway model (top view), all dimensions in mm

4 RESULTS AND DISCUSSION

To clarify the results, several time stations for the vertical opposite motion of lifts are illustrated in figure 3. The steady-state solution is applied as an initial condition at $t = 0.0$ when the lifts start moving. The lifts will be aligned side by side at $t = 0.3$, and the crossing event will be finished at $t = 0.6$. Each car moves at the speed of 20 $^{m}/_{s}$ in vertical opposite directions.

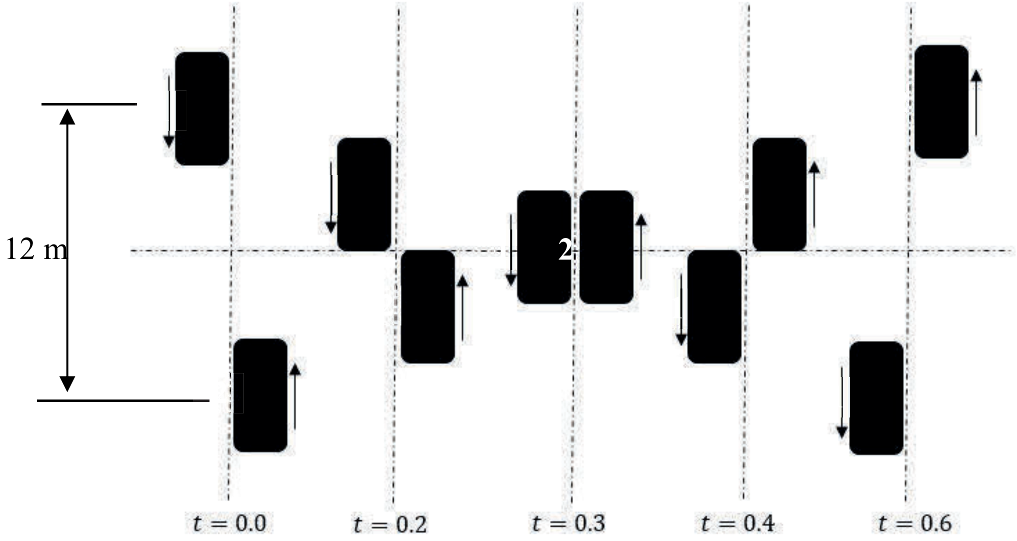


Figure 3 The time sequence of the relative position of each lift

Figure 4 illustrates the air field and its velocity profile around each car. Also, it shows the pressure distribution over the cars' bodies. Both cars are moving inside the hoistway at the same speed (20 m/s) so that Mach number is 0.06. The hoistway height is 90 m, and the lateral (horizontal) distance between the two lifts is 0.5 m (see figure 2). The Reynolds number (Re) based on the car width scale is 2.66×10^6 which is calculated as follows:

$$Re = \frac{\rho V L}{\mu} \tag{3}$$

where:

ρ: the air density ($1.2041\ kgm^{-3}$) (at $20\ °C$)

V: the lift car velocity ($20\ ms^{-1}$)

L: the lift car width ($2\ m$)

μ: the dynamic viscosity of air ($1.81 \times 10^{-5}\ kgm^{-1}s^{-1}$)

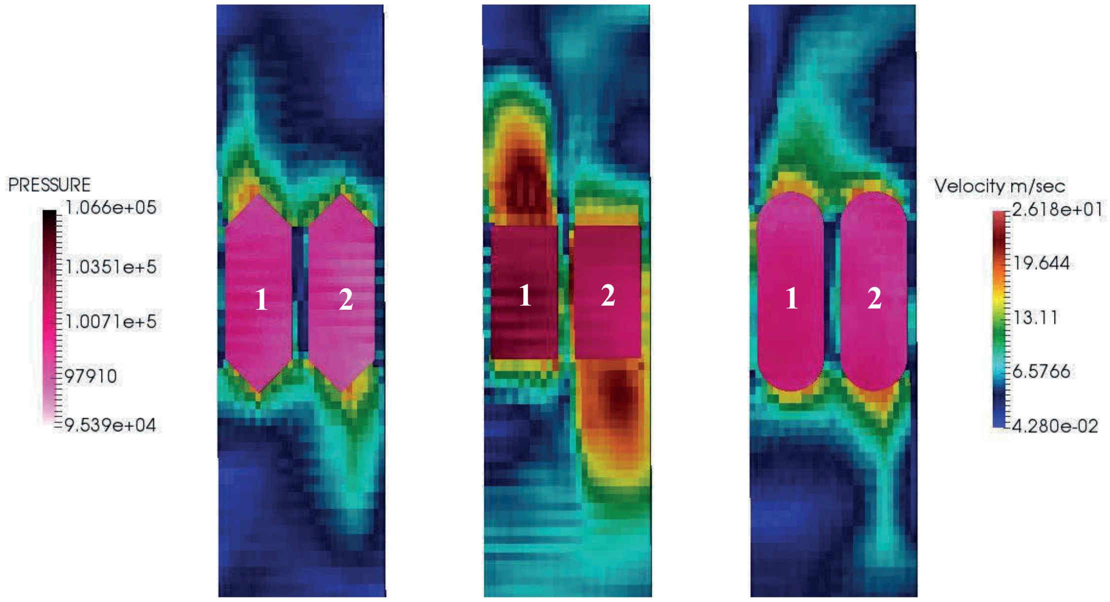
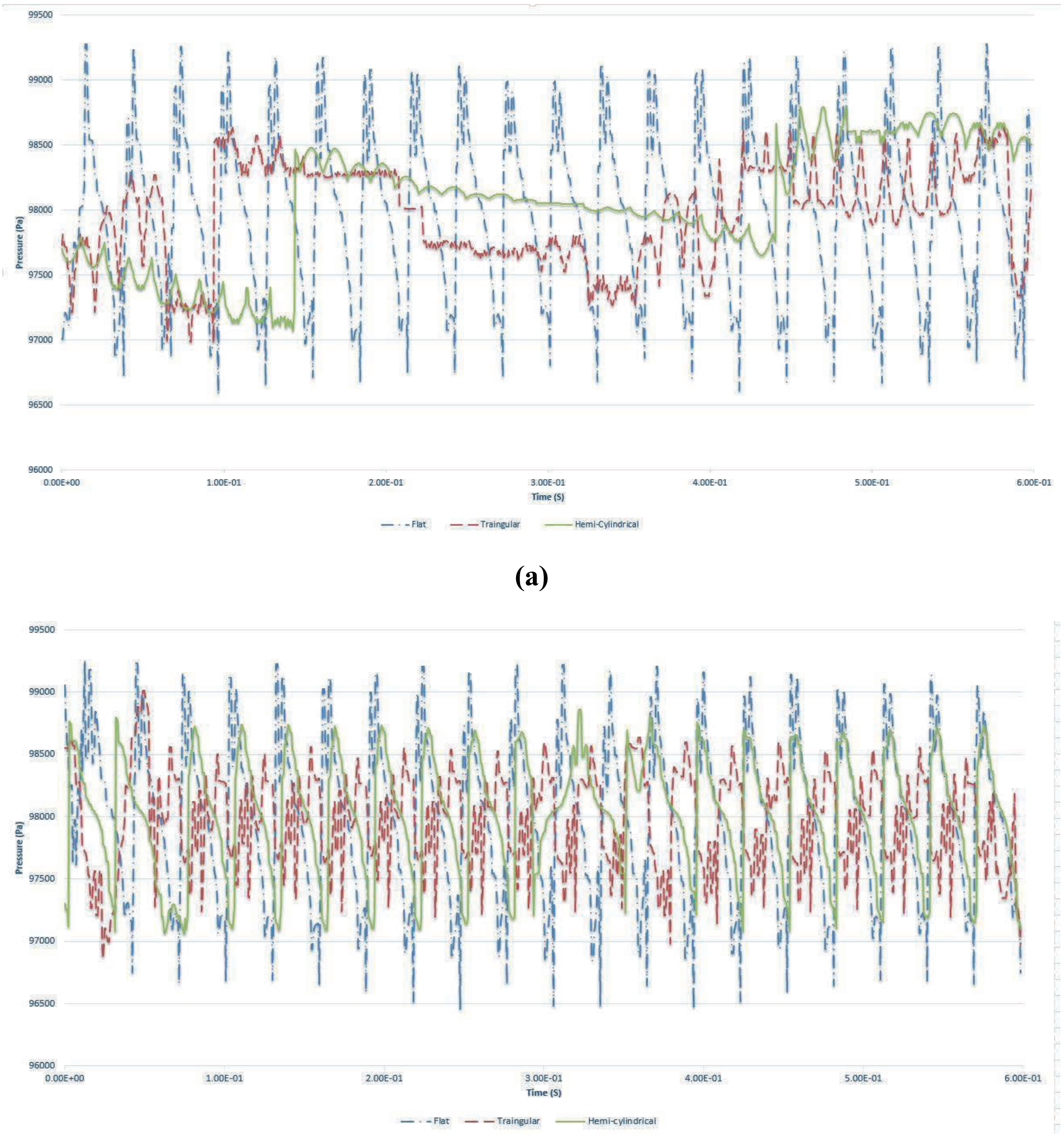


Figure 4 Side view of the flow field around the lifts and the pressure distribution at $t = 0.3$.

The pressure load fluctuations in the gap between the two cars may cause noise and vibrations. In the simulation the gamma low equation of state has been applied in order to estimate the initial value of the pressure as 103.1 kPa [9]. Figure 5 shows the overall pressure acting upon the side walls of the lift cars. The pressure would be building up and reaching its highest values during the crossing event. In the scenario considered in this simulation study the event starts when the cars are 12 m apart from each other and ends after the cars have passed each other and are again separated by the same distance. Consider, for instance, the time instant when the two lifts are alongside each other ($t = 0.3$). For lift 1 with triangular shrouds, the maximum air pressure acting on the side wall is then 97.7 kPa, and the maximum air pressure acting on the side walls of the lift with hemi-cylindrical and flat shrouds is 98.2 kPa and 99.1 kPa, respectively.

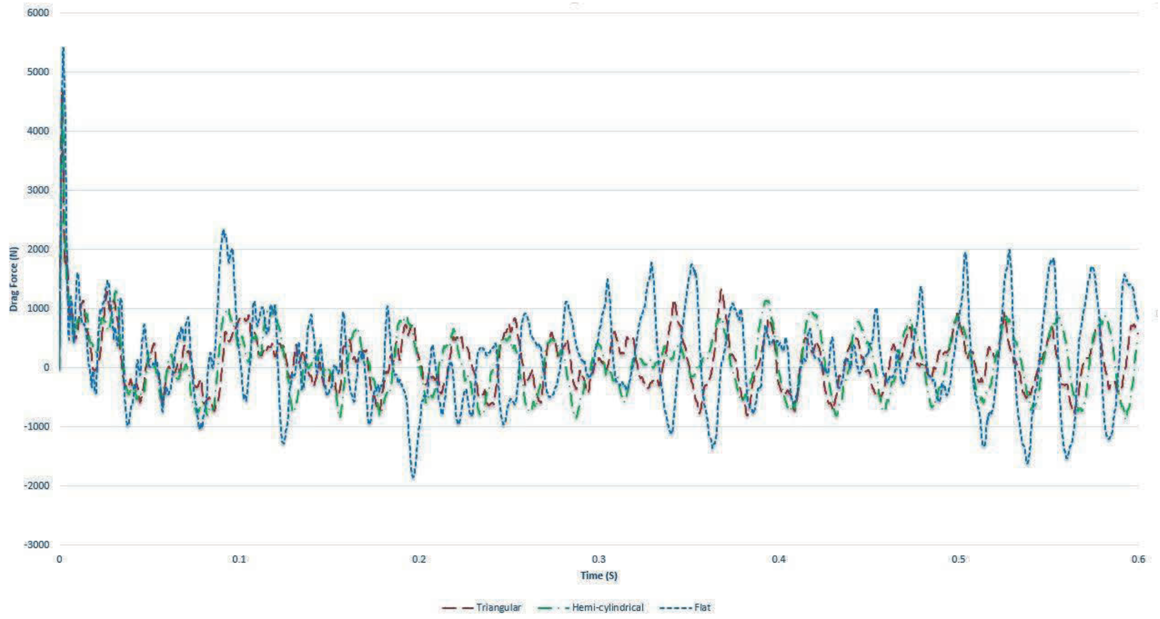


(a)

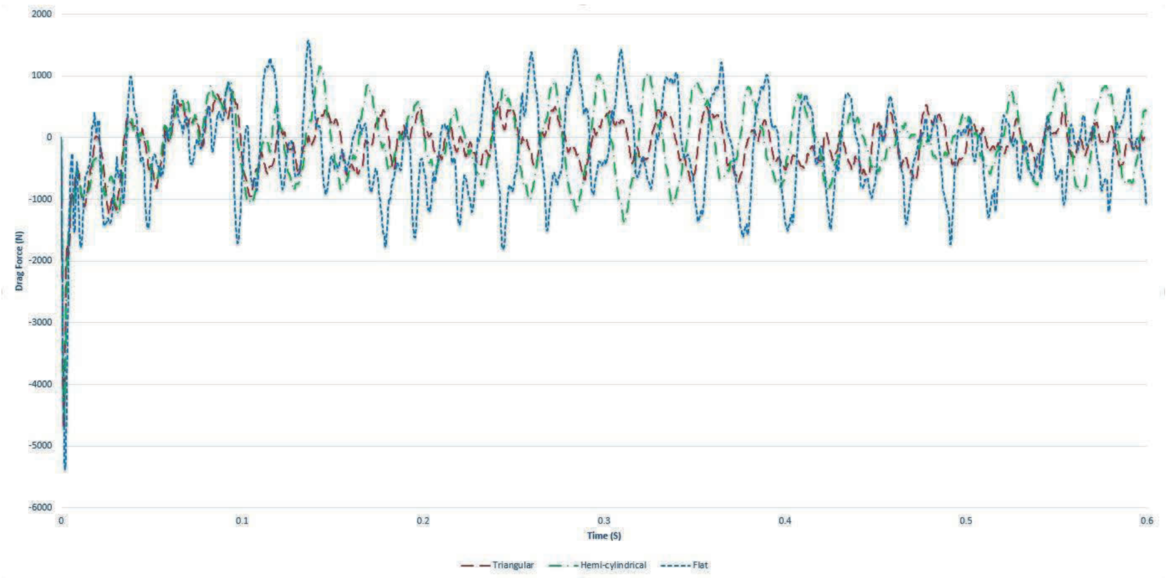
(b)

Figure 5 Pressure fluctuation over the side walls of each lift (a) at lift 1 side wall facing lift 2. (b) at lift 2 side wall facing lift 1.

One of the main forces acting on the lift body is the drag force. The time histories of the aerodynamic drag force are shown in Figure 6. The drag forces have been determined through a simulation test with the gravity effects removed. It is clear that the drag force fluctuation acting on the lift with triangular shrouds is less than the other lifts.



(a)



(b)

Figure 6 the time history of the aerodynamic drag forces (a) at the top of lifts 1 at the bottom of lifts 2; (b) at the bottom of lifts 2

The resultant overall forces that act on the coupling surface due to the fluid effects are shown in Figure 7. The fluctuation shows that the highest forces occur when the two cars pass each other. It is clear that the resultant force acting on the triangular lifts approaches approximately 23 kN. On the other hand, the highest forces affecting the other shapes are 46.8 kN and 44.9 kN for the flat and hemi-cylindrical lifts respectively.

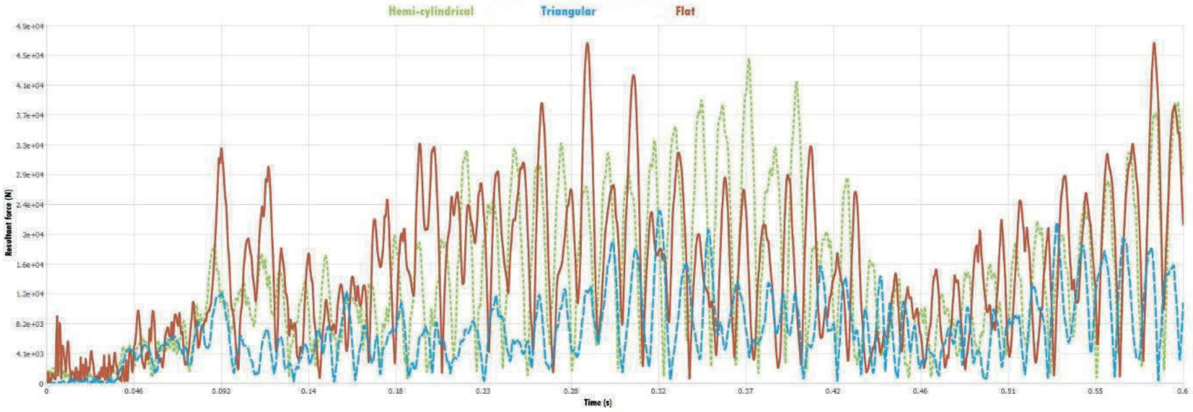
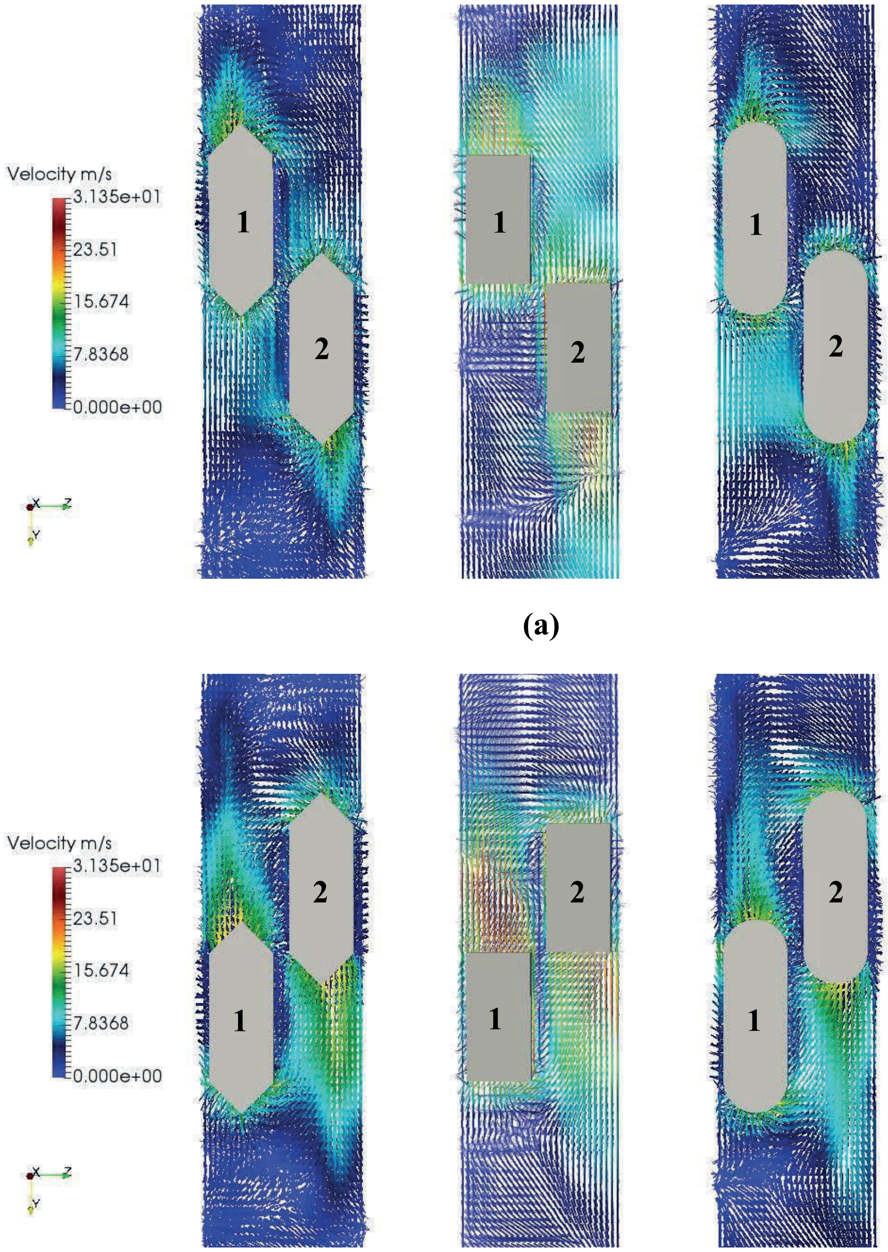


Figure 7 The overall forces acting on the coupled surfaces of the lift cars

The drag force will also have an effect on the flow in the wake region of each lift. In this kind of engineering problem, predicting and modelling the shear layer separation is essential because it has a significant impact on the opposite moving bodies due to the vortex shedding in its wake region. The air flow patterns and velocities are illustrated in Figure 8. The diagrams presented in this figure show that the turbulence behind the ‘flat’ lifts is much higher than the turbulence corresponding to the other shapes. Furthermore, the flow behind the ‘triangular’ lift 1 tends to reattach at the wall side rather than to the side of the moving lift 2. Thus, lift 2 will experience less turbulence during the crossing event.



(a)

(b)

Figure 8 The air flow patterns and its velocity in the hoistway while the lifts pass each other (a) at $t = 0.2$ (b) at $t = 0.4$

5 CONCLUSION

The very fast development of the construction of high-rise buildings raises an essential need for the design of high-speed lifts. The aerodynamic performance of these lifts has been discussed in the paper. Attention has been paid to the scenario in which two lifts are passing each other in the same hoistway. The flow field, pressure distribution, velocity, drag forces and the flow patterns have been studied. It was revealed that the lift car geometry design plays a significant role in the aerodynamic performance of the lift itself. From the aerodynamics point of view, the results also indicate that the triangular shape of the lift's top and bottom shroud would be the best design in comparison with the flat and the hemi-cylindrical shapes.

REFERENCES

[1] Matsukura, Y., E. Watanabe, Y. Sugiyama and O. Kanamori, (1992). New mechanical techniques for super high-speed elevators. Elevator Technology. 4, Proceedings of ELEVCON'92. Amsterdam, the Netherlands, pp. 174-181.

[2] Teshima, N., Miyasako, K. & Matsuda, H., (1992). Experimental and numerical studies on ultra-highspeed elevators. Elevator Technology, 4, pp.276–285.

[3] Bai, H., Shen, G. and So, A., (2005). Experimental-based study of the aerodynamics of super-high-speed elevators. Building Services Engineering Research and Technology, 26(2), pp.129–143.

[4] Shi, L., Liu, Y., Jin, Z., Cao, Z., (2007). Numerical Simulation of Unsteady Turbulent Flow Induced by Two-Dimensional Elevator Car and Counter Weight System. Journal of Hydrodynamics, Ser.B, 19(6), pp.720–725.

[5] Wu, R., Zhu, Z., Chen, G., Cao, G. and Li, W., (2014). 1263. Simulation of the lateral oscillation of rope-guided conveyance based on fluid-structure interaction. Journal of Vibroengineering, 16(3).

[6] Wu, R., Zhu, Z. & Cao, G., (2015). Computational Fluid Dynamics Modeling of Rope-Guided Conveyances in Two Typical Kinds of Shaft Layouts. PloS one, 10(2), p.e0118268.

[7] Mirhadizadeh, S., Kaczmarczyk, S., Tongue, N., Al-Jelawy, H., Feldhusen, P., Delk, W., Anderson, K., Dudde, F., (2015). Modelling and Computer Simulation of Aerodynamic Interactions in High-Rise Lift Systems. In The 5th Symposium on Lift and Escalator Technologies. Northampton, UK, pp. 117–123.

[8] MSC. Software Corporation, (2013). Dytran Explicit Solution for Transient Structural Dynamics and Fluid-Structure Interaction. Product Datasheet-Dytran^{TM}.

[9] Powell, K.G., Roe, P.L., Linde, T.J., Gombosi, T.I. and De Zeeuw, D.L., (1999). A solution-adaptive upwind scheme for ideal magnetohydrodynamics. Journal of Computational Physics, 154(2), pp.284-309.

BIOGRAPHY

(1) Mr. Hayder Al-Jelawy

Hayder has a master’s degree in Mechanical Engineering from the University of Technology in Iraq. His expertise is in the area of applied mechanics and computational fluid dynamics. Currently, Mr. Hayder is a PhD. Student at the University of Northampton. He is also an associate member of the Institution of Mechanical Engineers and in the Institute of Physics.

(2) Professor Stefan Kaczmarczyk

Stefan Kaczmarczyk has a master’s degree in Mechanical Engineering and he obtained his doctorate in Engineering Dynamics. He is Professor of Applied Mechanics and Postgraduate Programme Leader for Lift Engineering at the University of Northampton. His expertise is in the area of applied dynamics and vibration with particular applications to vertical transportation and material handling systems. He has been involved in collaborative research with a number of national and international partners and has an extensive track record in consulting and research in vertical transportation and lift engineering. Professor Kaczmarczyk has published over 90 journal and international conference papers in this field. He is a Chartered Engineer, being a Fellow of the Institution of Mechanical Engineers, and he has been serving on the Applied Mechanics Group Committee of the Institute of Physics.