Pertanika J. Sci. & Technol. 25 (S): 227 - 238 (2017)

SCIENCE & TECHNOLOGY

Journal homepage: http://www.pertanika.upm.edu.my/

CFD Analysis on Indoor Temperature and Velocity: Effects of Incident Wind Angle and Outlet Position

Ali, N. M., Abd Razak, A.*, Mohamad, M. F. and Bahsan, R.

Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM), 40450 Shah Alam, Selangor, Malaysia

ABSTRACT

Ventilation is the process of exchanging air in a closed space to provide good indoor air quality. Computational Fluid Dynamics (CFD) has become one of the most important and reliable tools to assess natural ventilation. This paper presents the effect of wind direction on temperature and velocity inside a building with outlet windows at leeward wall and side wall. The CFD simulation was performed using Reynolds-Average Navier-Stoke (RANS) approach with k-ε model. The selection of standard k-ε model is due to suitability of this model to perform the wind speed profile and temperature profile inside the building. Validation of cross-ventilation is performed based on earlier Particular Image Velocimetry (PIV) measurements and shows very minimal discrepancy between CFD and PIV result. The result of velocity and temperature shows that the wind speed and temperature inside a building strongly depends on the incident winds angle and outlet opening of the building.

Keywords: CFD, outflow position, temperature, velocity, wind direction

INTRODUCTION

Cross-ventilation and natural ventilation research has gained popularity in 1973 due to oil crisis (Kotani, Goto, Ohba, & Kurabuchi, 2009). Natural ventilation provides an opportunity to enhance comfort and health and as well as produce a sustainable built environment. Many researches have worked on how to improve the indoor natural ventilation performance by analysing different parameters and building characteristics. For example, Ohba et al. (2001) conducted a wind tunnel experiment to study the characteristic of air flow inside a simplified model with cross-opening. Furthermore, Seifert, Li, Axley and Rösler (2006), and Derakhshan and Shaker (2016) performed CFD analysis on an isolated building and investigated the

ARTICLE INFO

Article history:
Received: 05 January 2017
Accepted: 17 January 2017

E-mail addresses:
merlisa@salam.uitm.edu.my (Ali, N. M.),
azlirazak@salam.uitm.edu.my (Abd Razak, A.),
faizal3744@salam.uitm.edu.my (Mohamad M. F.),
rosnadiah@salam.uitm.edu.my (Bahsan, R.)
*Corresponding Author

ISSN: 0128-7680

effect of building configuration (opening location, wall porosities, wall thickness and height to width ratio of opening) and wind direction on ventilation performance. Despite many studies using isolated building, Mohammad Kasim, Zaki, Mat Ali, Ikegaya and Abd Razak (2016) investigated the effect of surrounding building and various opening on ventilation performance. Additionally, there are few study on naturally ventilated building in more complex geometry such as terrace house (Liyana, Abd Razak, Wan Jabarudin, & Harith, 2015; Mohamad, Hagishima, Tanimoto, Ikegaya, & Omar, 2013), high rise apartment (Hyun, Park, & Augenbroe, 2008) and shopping mall (da Graça, Martins, & Horta, 2012).

Computational Fluid Dynamics (CFD) has been used extensively in the research of cross-ventilation of building as confirmed from many CFD studies published in the last 25 years (Jiang & Chen, 2002; Karava, Stathopoulos, & Athienitis, 2011; Kato, Murakami, Mochida, Akabayashi, & Tominaga, 1992; Perén, van Hooff, Leite, & Blocken, 2016; Ramponi & Blocken, 2012a; Seifert, Li, Axley, & Rösler, 2006; Tan & Glicksman, 2005). In CFD analysis, the accuracy and reliability of the turbulence model are the main concern. Large Eddy Simulation (LES) is basically more accurate than Reynolds-Averaged Navier-Stoke (RANS). However, 32 out of 39 studies on natural ventilation used RANS (Ramponi & Blocken, 2012b), proving that it is still the most popular approach. Due to this fact the present study uses RANS models for cross-ventilation flow in a simple isolated building.

In spite of the presence of numerous studies, the effect of various ventilation strategies, such as single-sided ventilation, cross-ventilation, effect of aspect ratio and outlet opening location on isolated building from a viewpoint temperature has not been fully explained. The present study aims to investigate the effect of incident wind angle and outlet position on temperature and velocity by means of CFD simulation. The paper is organized as follows: the numerical setup and validation is described in section 2 and results are presented in section 3. Finally, in section 4 is the conclusion.

MATERIALS AND METHOD

Model Geometry and Parameters

Table 1 shows the geometry of the model used in this simulation. A simulation model was sized at 7.0 m × 5.0 m × 4.0 m (length × width × height) as shown in Figure 1 and Figure 2. Two small openings with dimension 1 m × 1 m (width × height) were installed at the middle of windward, leeward wall (Figure 1) and adjacent wall (Figure 2). The wall porosities (the ratio of the area of opening to the area of the façade wall, *hereafter* w.p) is 5% except for side-wall outlet. For side-wall outlet the w.p is 3.6% due to the larger façade wall compared to the opening area. In this study, the effect of outlet location is more important than the effect of w.p, therefore the w.p change when the location changed. For both configurations the building consists of two florescent lamp which produced the heat emission of about 22 W/m2 (standard heat emission as refer to Al-Shemmeri (2011)). Figure 3 shows the wind direction of both configurations. In this study, four incident wind angles are considered, θ = 0°, 15°, 30° and 45° from the centre line of the horizontal plane (x-y plane).

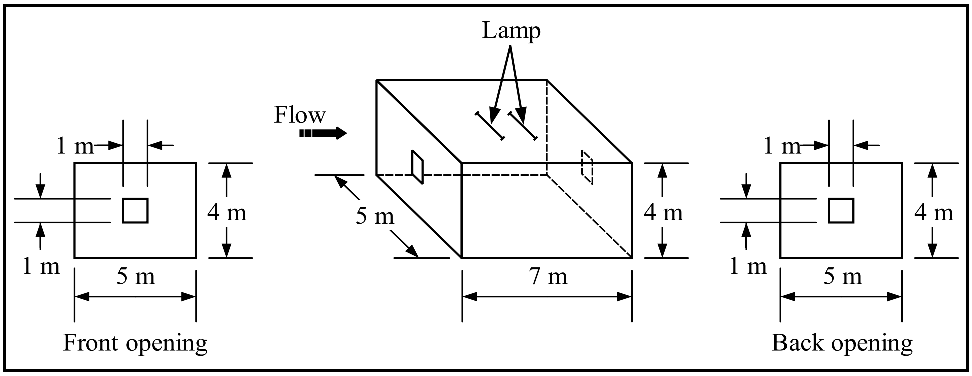


Figure 1. Schematic drawing of leeward outlet building

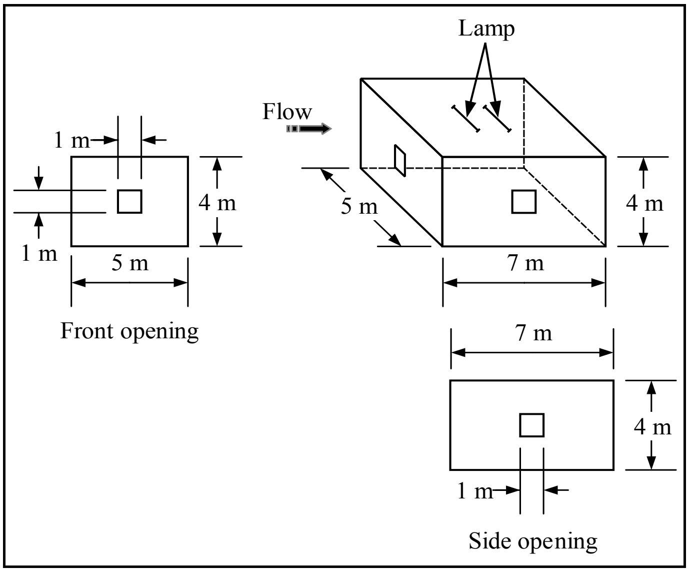


Figure 2. Schematic drawing of side-wall outlet building

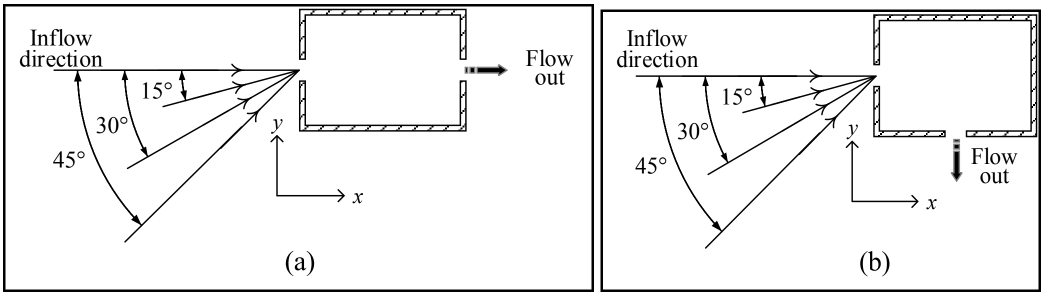


Figure 3. The angle of inflow direction and out flow direction: (a) leeward outlet; and (b) side-wall outlet

Table 1
Geometrical setup for simulation

Configuration	Case	Wind direction	Outlet opening size	Inlet wall porosity (A_{open}/A_{wall})	Outlet wall porosity (A_{open}/A_{wall})	Room dimension D × W ×H
Leeward outlet	LW0	0°	1m ×1m	5%	5%	7m × 5m × 4m
	LW15	15°				
	LW30	30°				
	LW45	45°				
Side-wall outlet	SW0	0°	1m ×1m	5%	3.6%	7m × 5m × 4m
	SW15	15°				
	SW30	30°				
	SW45	45°				

Numerical Setup and Model Validation

Considering the computational cost and the requirement mentioned by Ramponi and Blocken (2012b), RANS approach of standard two-equation k-ε model was chosen for this study. The simulation were performed by commercial CFD software ANSYS Fluent (Fluent, 2015). All simulation was conducted in an isothermal condition where the effect of heat transfer from outside was negligible. The detail of mathematical model of governing equation can be found in (Cheung & Liu, 2011). The upstream mean wind speed (u_{ref}) is 3 m/s, which is to reflect the weak wind condition of Class 1 in Malaysia. The inlet temperature from the ambient air was measured as 306 K and applied at the inlet opening of the flow.

Figure 4 shows the horizontal profiles of the normalised stream wise mean wind speed, measured along the centreline of the building derived from the current analysis and PIV measurement (Karava, Stathopoulos, & Athienitis, 2011). The values were normalised using the upstream mean wind speed, u_{ref}. The profiles obtained by present simulation analysis show excellent agreement with the measurement data (Karava, Stathopoulos, & Athienitis, 2011) with small discrepancies near the inlet and outlet. This is due to the reduced reliability of PIV closed to the wall (Ramponi & Blocken, 2012a). The validation result indicated that the boundary condition and standard k-ε model selected in this study was sufficient to capture the flow profile inside the building with isolated condition. Further analysis was applied by adding the effect of heat load from the florescent lamp to investigate the temperature profile for thermal comfort study.

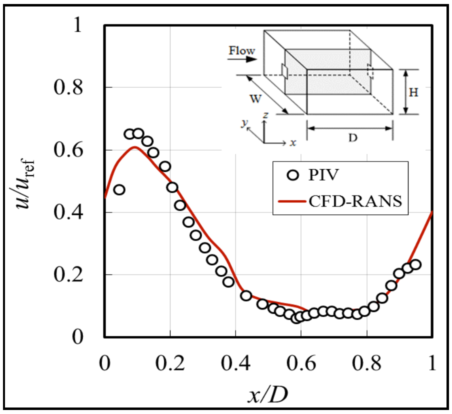


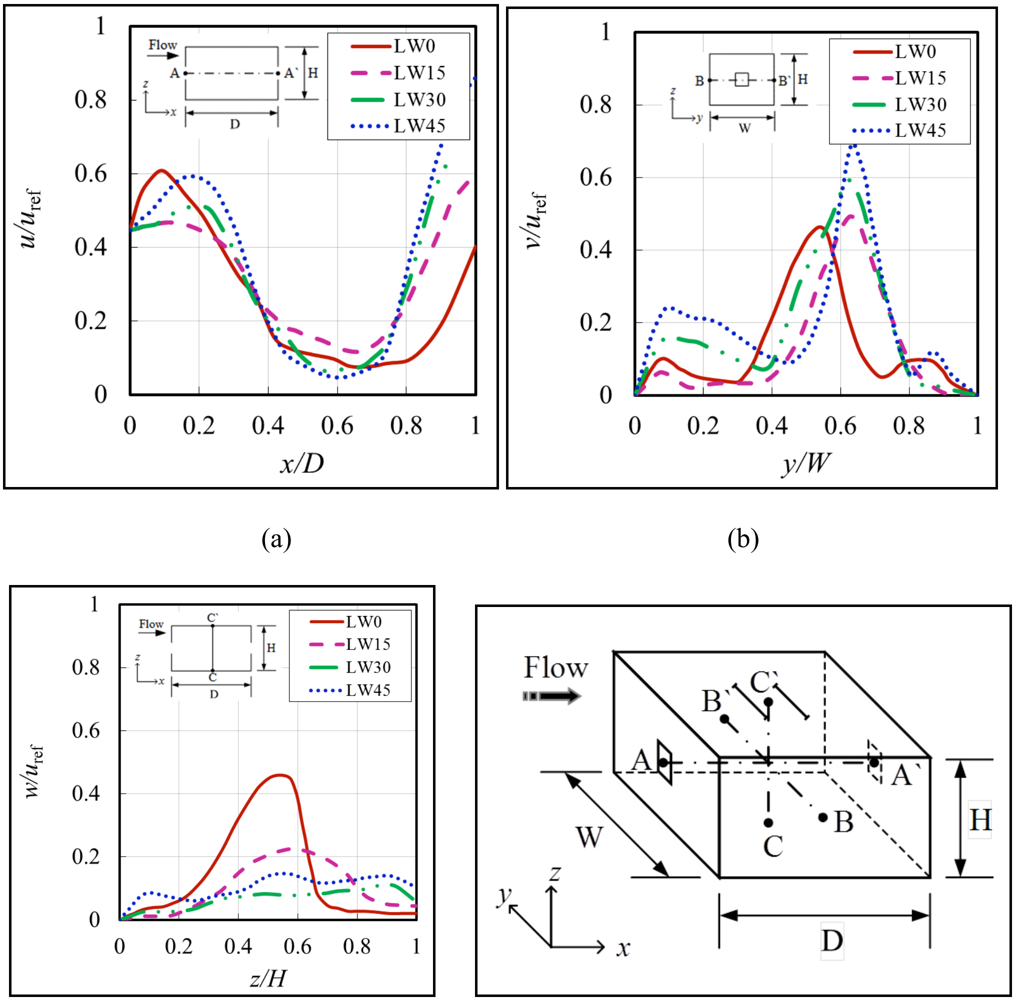
Figure 4. Validation result of streamwise wind speed ratio along the centreline: Comparison between PIV data from Karava, Stathopoulos and Athienitis (2011) and CFD result of current simulation setting

RESULTS AND DISCUSSION

Indoor Air Flow Characteristics

Figure 5 and Figure 6 show the normalised mean wind speed four different incident wind angles which are 0°, 15°, 30° and 45° with leeward opening. The profile of normalised mean wind speed (u/u_{ref}, v/u_{ref} and w/u_{ref}) was measured along centreline of stream wise (Figure 5(a) and Figure 6(a)), span wise (Figure 5(b) and Figure 6(b)) and vertical (Figure 5(c) and Figure 6(c)). The mean wind speed of the x-component for leeward wall outlet indicated that the variation was very small for all incident wind angles. This condition is similar to that of Meroney (2009). In the case of side-wall outlet, the variation was very large especially for 0° incident angle compared to other incident angles. The incident angle of 30° shows the lowest wind speed among other incident angles. Inconsistencies of x-component mean wind speed characteristic between these two configurations indicate that the mean wind speed is less dependent on incident angle for stream wise flow.

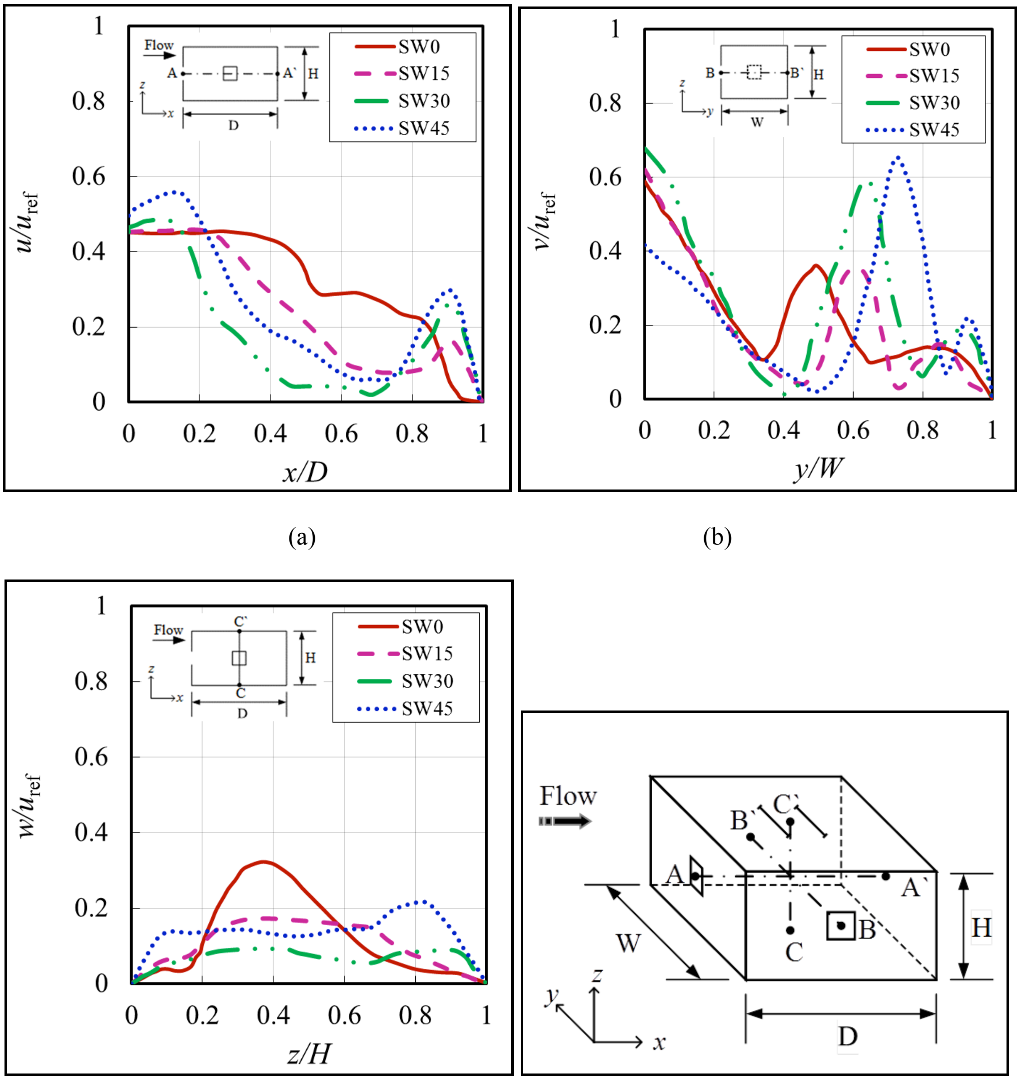
Normalised mean wind speed in y-component (v/u_{ref}) shows similar trend for $y/W > 0.3$ in both configurations. Huge discrepancies between these two configurations was observed closer to point B. This was due to the effect of the opening for side-wall configuration. The velocity close to point B was influenced by the flow that discharged from the building. In addition, we can conclude that the velocity of y-component is not only influenced by incident wind angle but by the position of opening as well. In y-component peak value of mean wind speed ratio of 45° incident angle is higher than other angle. However, for the z-component the 0° incident angle is more dominant. The same situation occurs in both configurations. Higher incident angle increases the velocity inside the building because the velocity which that enters the building tangent to the streamline along the y-axis. Therefore, it does not affect the vertical profile significantly as shown in Figure 5(c) and Figure 6(c).



(a) (b)

(c) (d)

Figure 5. Analysis of normalised mean wind speed profile by variation of incident wind angle for leeward outlet configuration along: (a) stream wise centreline (A – A); (b) span wise centreline (B – B`); (c) vertical centreline (C – C`); and (d) is schematic diagram of measurement line



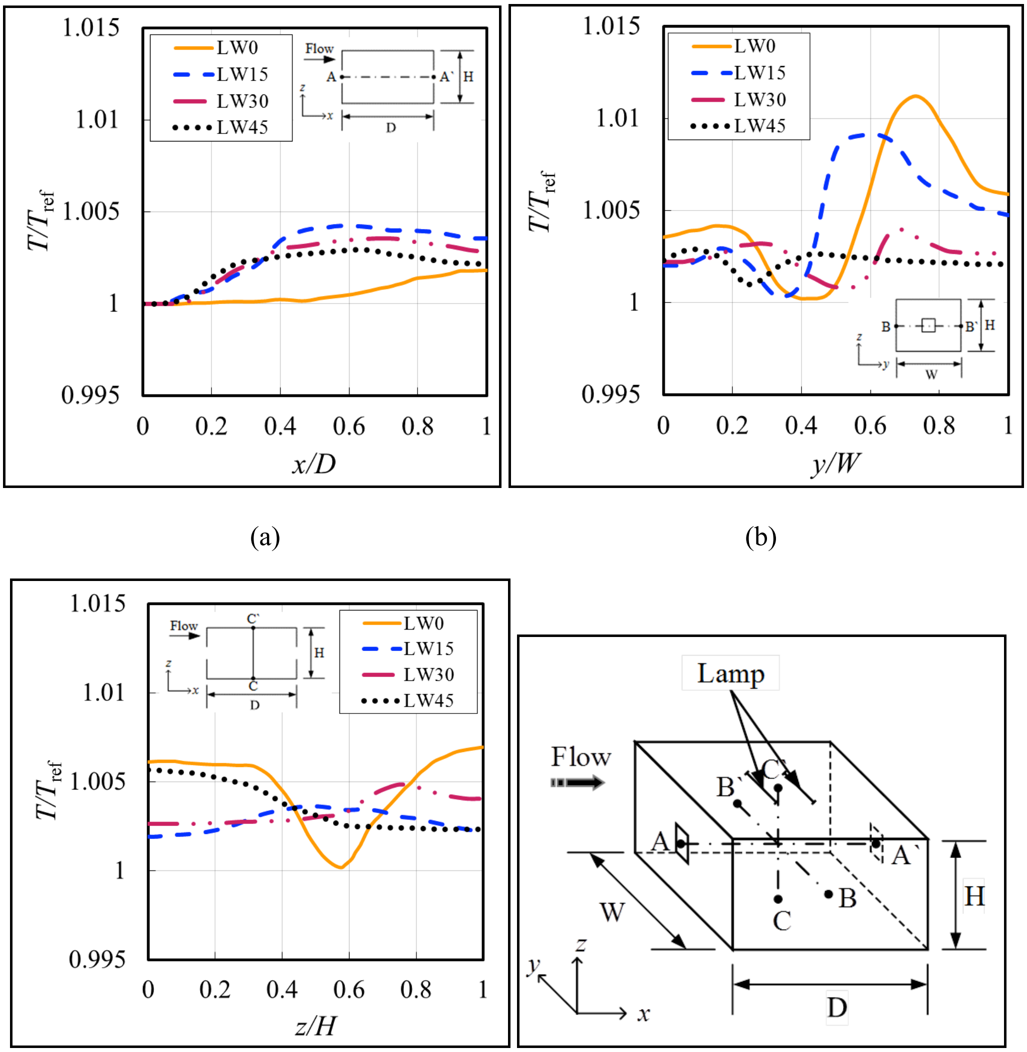
(a) (b)

(c) (d)

Figure 6. Analysis of wind speed ratio profile by variation of incident wind angle for side-wall outlet configuration along: (a) stream wise centreline (A – A); (b) span wise centreline (B – B`); (c) vertical centreline (C – C`); and (d) is schematic diagram of measurement line

Normalised Temperature Profile

Effect of incident wind angle on temperature variation for leeward configuration. The qualitative analysis of normalised temperature profile inside the building with leeward outlet is presented in Figure 7. The normalised temperature profiles measured at the centreline of stream wise, span wise and vertical. The sensitivity test was conducted for incident wind angle $\theta = 0°$, 15°, 30° and 45°. The effect of incident wind angle differs accordingly to the *x*-, *y*- and *z*-direction. Along the centreline of *x*-, *y*- and *z*-direction the lowest temperature recorded at θ = 0°. However, the highest temperature was captured at θ = 15° for x-direction and θ = 0° for *y*- and *z*-direction. The effect of heat gain from fluorescent lamp was the main contribution for these inconsistencies. In addition, the contribution of heat gain from fluorescent lamp significantly affects the temperature profile. Figure 5 (a – c) and Figure 7 (a – c) shows that the higher the wind speed the lower the temperature. This can be explain by the fact the wind speed is important tool in transferring heat and providing thermal comfort especially outdoors (Abd Razak, Hagishima, Awang Sa, & Zaki, 2016).

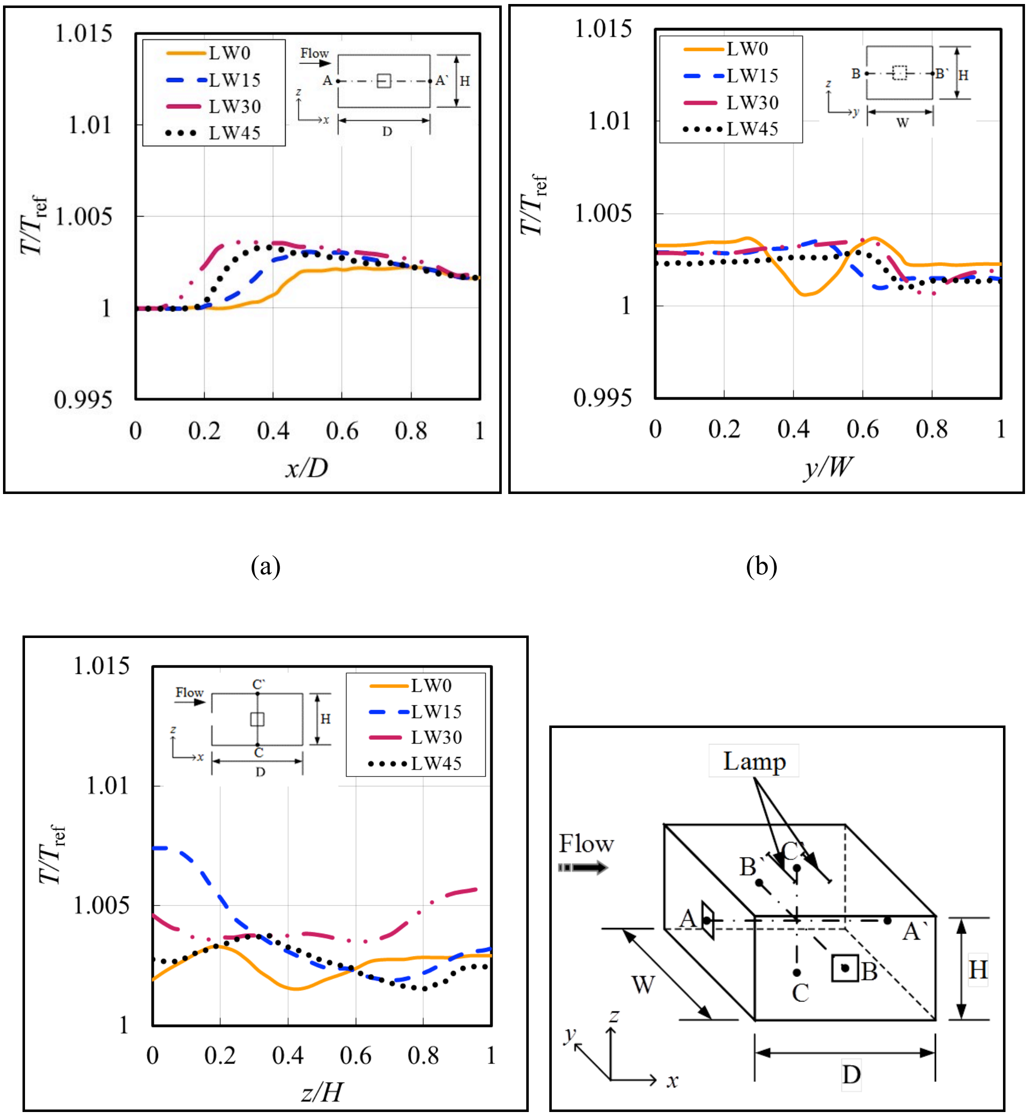


(a) (b)

(c) (d)

Figure 7. Analysis of temperature ratio profile by variation of incident wind angle for leeward outlet configuration along: (a) streamwise centreline (A – A); (b) spanwise centreline (B – B`); (c) vertical centreline (C – C`); and (d) is schematic diagram of measurement line

Effect of incident wind angle on temperature variation for side-wall configuration. This subsection investigated the effect of incident wind angle. Figure 8 shows the normalised temperature profile inside the building with side-wall outlet. In general effect of incident wind angle on temperature profile significant for vertical air temperature. Closer to the bottom (Figure 8(c)), the highest temperature occurs at θ = 15° and the lowest temperature occurs at θ = 0°. When the air temperature reaches the top at $z/H = 1$, the highest temperature occurs when the θ = 30°and the lowest temperature observed at θ = 45°. Figure 8a shows the temperature



(a) (b)

(c) (d)

Figure 8. Analysis of temperature ratio profile by variation of incident wind angle for side-wall outlet configuration along: (a) streamwise centreline (A – A); (b) spanwise centreline (B – B`); (c) vertical centreline (C – C`); and (d) is schematic diagram of measurement line

variation on incident wind angle at $0.2 < x/D < 0.5$. Furthermore, along the span wise (Figure 8(b)) the θ = 0° the temperature drops when it faces incoming flow and rises before it declines to constant temperature until reaches the wall. Similarly, for the leeward outlet configuration, the temperature was strongly influenced by wind speed. As example, the temperature along the ventilation flow path was found to be lower compared to surrounding.

Comparison between opening configurations on temperature variation. Figures 7 and Figure 8 show the temperature variation for two different opening configurations (Table 1) namely leeward outlet and side-wall outlet respectively. Generally, the side-wall outlet has better ventilation performance compared to leeward outlet. Quantitative analysis on the temperature values indicated that side-wall outlet has lower temperature along *x*-, *y*- and *z*-direction compared to leeward outlet. Even though the opening-to-wall ratio for side-wall outlet lower than leeward wall, this parameter was not significant.

CONCLUSION

The effects of incident wind angle and outlet position were examined in this study by solving the cross-ventilation using RANS approach with standard k-ε model. This study leads to the following conclusions: (1) The leeward outlet is more suitable for the purpose of pollutant removal but for thermal comfort the side-wall is more suitable; (2) The incident wind angle with θ = 45° provided a better solution for thermal comfort compared to other incident angle. In addition, θ = 0° can be considered as alternative for the case of inlet and outlet opening parallel as shown in Figure 2; (3) The temperature profile inside the building has strong correlation with mean wind speed that entering and leaving the building (ventilation rate); and (4) Current finding can be used as preliminary evaluation in building design of the inlet and outlet of cross-ventilation in a naturally ventilated building.

ACKNOWLEDGEMENTS

The authors of this work would like to express their sincere gratitude to Faculty of Mechanical Engineering, Universiti Teknologi MARA (UiTM) for facilitating this research. This research was financially supported by grant-in aid for scientific research (600-RMI/FRGS 5/3 (87/2015)) from the Ministry of Education, Malaysia.

REFERENCES

Abd Razak, A., Hagishima, A., Awang Sa, Z. A., & Zaki, S. A. (2016). Progress in wind environment and outdoor air ventilation at pedestrian level in urban area. *Applied Mechanics and Materials, 819*, 236-240.

Al-Shemmeri, T. (2011). *Energy Audits: A Workbook for Energy Management in Buildings.* Wiley.

Cheung, J. O., & Liu, C.-H. (2011). CFD simulations of natural ventilation behaviour in high-rise buildings in regular and staggered arrangements at various spacings. *Energy and Buildings, 43*(5), 1149-1158.

da Graça, G. C., Martins, N. R., & Horta, C. S. (2012). Thermal and airflow simulation of a naturally ventilated shopping mall. *Energy and Buildings, 50*, 177-188.

Derakhshan, S., & Shaker, A. (2016). Numerical study of the cross-ventilation of an isolated building with different opening aspect ratios and locations for various wind directions. *International Journal of Ventilation,* 1-19. doi: 10.1080/14733315.2016.1252146

Fluent. (2015). Ansys – Fluent User's Guide. Retrieved on 30 October, 2015, from http://www.arc.vt.edu/ansys_help/flu_ug/flu_ug.html

Hyun, S., Park, C., & Augenbroe, G. (2008). Analysis of uncertainty in natural ventilation predictions of high-rise apartment buildings. *Building Services Engineering Research and Technology, 29*(4), 311-326.

Jiang, Y., & Chen, Q. (2002). Effect of fluctuating wind direction on cross natural ventilation in buildings from large eddy simulation. *Building and Environment, 37*(4), 379-386.

Karava, P., Stathopoulos, T., & Athienitis, A. (2011). Airflow assessment in cross-ventilated buildings with operable façade elements. *Building and Environment, 46*(1), 266-279.

Kato, S., Murakami, S., Mochida, A., Akabayashi, S. I., & Tominaga, Y. (1992). Velocity-pressure field of cross ventilation with open windows analyzed by wind tunnel and numerical simulation. *Journal of Wind Engineering and Industrial Aerodynamics, 44*(1), 2575-2586.

Kotani, H., Goto, T., Ohba, M., & Kurabuchi, T. (2009). Review of Cross-ventilation research papers-from the working group for natural ventilation and cross-ventilation of the architectural institute of japan. *International Journal of Ventilation, 8*(3), 233-241.

Liyana, T., Abd Razak, A., Wan Jabarudin, W. T., & Harith, Z. Y. H. (2015). Cross ventilation analysis on the effect of setback distance of a terraced house using PIV. *Jurnal Teknologi, 76*(5).

Meroney, R. N. (2009). CFD prediction of airflow in buildings for natural ventilation. *Proceedings of the Eleventh Americas Conference on Wind Engineering, Puerto Rico*.

Mohamad, M. F., Hagishima, A., Tanimoto, J., Ikegaya, N., & Omar, A. R. (2013). Wind-induced natural ventilation in typical single storey terraced houses in Malaysia. *13th Conference of International Building Performance Simulation Association*.

Mohammad Kasim, N. F., Zaki, S. A., Mat Ali, M. S., Ikegaya, N., & Abd Razak, A. (2016). Computational study on the influence of different opening position on wind-induced natural ventilation in urban building of cubical array. *4th International Conference on Countermeasures to Urban Heat Island*.

Ohba, M., Irie, K., & Kurabuchi, T. (2001). Study on airflow characteristics inside and outside a cross-ventilation model, and ventilation flow rates using wind tunnel experiments. *Journal of Wind Engineering and Industrial Aerodynamics, 89*(14), 1513-1524.

Perén, J. I., van Hooff, T., Leite, B., & Blocken, B. (2016). CFD simulation of wind-driven upward cross ventilation and its enhancement in long buildings: Impact of single-span versus double-span leeward sawtooth roof and opening ratio. *Building and Environment, 96*, 142-156.

Ramponi, R., & Blocken, B. (2012a). CFD simulation of cross-ventilation flow for different isolated building configurations: Validation with wind tunnel measurements and analysis of physical and numerical diffusion effects. *Journal of Wind Engineering and Industrial Aerodynamics, 104*, 408-418.

Ramponi, R., & Blocken, B. (2012b). CFD simulation of cross-ventilation for a generic isolated building: Impact of computational parameters. *Building and Environment, 53*, 34-48.

Seifert, J., Li, Y., Axley, J., & Rösler, M. (2006). Calculation of wind-driven cross ventilation in buildings with large openings. *Journal of Wind Engineering and Industrial Aerodynamics, 94*(12), 925-947.

Tan, G., & Glicksman, L. R. (2005). Application of integrating multi-zone model with CFD simulation to natural ventilation prediction. *Energy and Buildings, 37*(10), 1049-1057.