A numerical investigation on the impacts of voids combinations on natural ventilation of high-rise residential building

Hamza Laloui, Noor Hanita Abdul Majid and Aliyah Nur Zafirah Sanusi
Department of Architecture and Environmental Design, International Islamic Univeristy Malaysia, Kuala Lumpur, Malaysia

Received 2 November 2020
Revised 27 December 2020
Accepted 4 January 2021

Abstract

Purpose – This paper aims to investigate the impacts of introducing voids combinations on natural ventilation performance in high-rise residential building living unit.

Design/methodology/approach – This study was carried out through field measurement and computational fluid dynamics methods. The parameters of the study are void types and sizes, and a wind angle was used to formulate case studies.

Findings – The results indicate that the provision of a single-sided horizontal void larger by 50% increase the indoor air velocity performance up to 322.37% to 0.471 m/s in the living unit and achieves the required velocity for thermal comfort.

Originality/value – Passive design features are the most desirable techniques to enhance natural ventilation performance in the high-rise residential apartments for thermal comfort and indoor air quality purposes.

Keywords Numerical simulation, Built environment tectonics and technologies, Air velocity, Field measurements experiment, Voids combinations, Wind driven ventilation, High-rise residential building, Computational fluid dynamics (CFD)

Paper type Research paper

1. Introduction

Adopting natural ventilation in high-rise residential buildings is considered as an effective feature to improve thermal comfort and indoor air quality, and minimize total energy consumption (Daniel and Williams, 2007; Fung and Lee, 2015). However, due to the high temperature and humidity levels in tropical climates, natural ventilation is deemed insufficient to provide thermal comfort in high-rise residential buildings compared to low- and medium-rise residential buildings (Ahmad *et al.*, 2017; Santamouris, 2016).

Wind-driven ventilation proved to be a reliable and effective cooling strategy for the internal spaces compared to buoyancy-driven ventilation that requires specific design configurations and is not significantly effective due to the low-temperature difference between the internal and external space in hot and humid regions (Allard and Ghiaus, 2012; Liu *et al.*, 2009, p. 136). On the other hand, different factors affect wind-induced ventilation performance in high-rise residential buildings such as opening configurations, ventilation mode (cross and single-sided ventilation) (Aflaki *et al.*, 2019; Chu and Chiang, 2014; Omrani *et al.*, 2017). Among these parameters, Omrani *et al.* (2017) clarified that cross-ventilation

Open House International
© Emerald Publishing Limited
0168-2601
DOI 10.1108/OHI-11-2020-0157

mode is more effective than single-sided ventilation in high-rise residential building where it can improve the indoor air velocity and the airflow distributions.

Achieving thermal comfort and air quality through natural ventilation is challenging and complicated due to different factors such as low wind speed circulation in hot and humid climate regions (Aflaki *et al.*, 2016); the location of high-rise residential buildings in the high-density urban area reduces the wind speed circulation, and the limited design variation of these buildings characterized with living units connected to the exterior environment only by one façade and rely mainly on single-sided ventilation mode (Farea, 2012). Moreover, limited design variations complicated the application of low residential building features such as wind tower, wing wall and wall groves (Aflaki *et al.*, 2019). Particular design features were proposed to enhance ventilation performance in high-rise residential buildings such as horizontal voids (Sapian, 2004), vertical voids (Moosavi and Mahyuddin, 2013), balconies (Omrani *et al.*, 2017) and ventilation shafts Prajongsan and Sharples (2012).

Voids in high-rise residential buildings are classified into vertical and horizontal voids (Farea, 2012; Ismail, 1996; Sapian, 2004). A vertical void is "air well" or "atria" found mostly in plan characterised with deep plans to provide suitable daylights and natural ventilation, and to reduce the solar gain effects (Gaber *et al.*, 2015; Elotefy *et al.*, 2015; Ismail, 1996). Chan (2014) classified horizontal voids in the high-rise residential buildings into four main categories:

(1) single sky garden;
(2) linked sky garden;
(3) duplex sky garden; and
(4) balcony type sky garden, and have various advantages related to the environment, social life and ecology.

From an environmental perspective, the horizontal voids can enhance the ventilation performance and the airflow circulation for the adjacent spaces (Chan, 2005; Tony, 2013). Previous studies clarify that introducing voids in high-rise residential buildings can enhance natural ventilation. Chiang and Anh (2012) using the computational fluid dynamics (CFD) simulation method suggested that the vertical void in 11-story residential building connected with horizontal voids located in the ground and top levels improves the air velocity (m/s) to 38% to 1.43 m/s for the living units situated in higher levels. Farea *et al.* (2015), using CFD simulation, revealed that high-rise building models with vertical void linked with horizontal voids located at the bottom and the intermediate levels induce the average air velocity (m/s) inside vertical void by 97%. A study by Pei-chun *et al.* (2016) proposed that the segmentation of the light well in a high-rise office building by horizontal voids in the building increases the airflow rate in the vertical void.

It can be concluded that previous studies focused on the impacts of voids combinations on natural ventilation in simplified buildings geometries. On the other hand, the impacts of design features (horizontal voids position and type) with other factors such as the effects of the horizontal void size, wind directions, surrounding environment and internal space analysis were not investigated. The objective of this study is to investigate the effects of these factors on wind-induced ventilation performance. This study used field measurement to collect data from a living unit in high-rise residential buildings in Kuala Lumpur, Malaysia and CFD simulation to investigate building models designed with horizontal voids (double and single-sided) with different size connected with vertical and their effects on indoor air velocity (m/s) performance.

In this study, air velocity (m/s) was used as the main parameters in natural ventilation evaluation. Air velocity is the main factor influencing thermal comfort level. Achieving the optimal indoor thermal comfort in the hot and humid climate like Malaysia requires air velocity/speed that varies between 0.2 and 1.5 m/s (ASHRAE, 2011). The Malaysian standard (MS:2680, 2017) set suitable air velocity varied from 0.25 to 0.5 m/s and from 0.5 to 1.0 m/s. Therefore, air velocity is used as an indicator of natural ventilation performance in the selected living unit of high-rise residential building under different variables.

2. Research methodology and procedure

2.1 Field measurement experiment

The field measurement experiment was carried out in a living unit of a 29-story high-rise residential building located in Kuala Lumpur, Malaysia. The selected living unit is located at the western end of the block B at the Level 11 precisely 36.3 m above the grade plane and orientated 71° from north to the western direction perpendicular to the south-west orientation as presented in Figure 1.

The living unit has a dimension of 7.85 × 13.96 × 3.3m (width × length × height) representing the worst scenario in terms of orientation toward the prevailing winds with external faced oriented to the south-west (SW) characterized with the lowest external wind speed. Figure 2 illustrates the location of the living unit at Level 11 below the middle of the building height allows examining the wind effects on lower levels compared to the higher levels precisely at 2/3 (stagnation point), where units at this height receive higher wind speed to get the real situation of the living unit situated in non-recommended orientation and level toward the external wind.

The block dimension is 26.3 × 42.5 × 101.5 m (width × length × height) and contains two identical attached vertical voids with a similar dimension of 10.88 × 11.77 × 101.5 m (width × length × height). Both attached vertical voids are connected to double-sided horizontal voids with dimensions of 7.85 × 13.96 × 6.6 m (width × length × height) located at Levels 15 and 23, as presented in Figure 2.

The selected living unit is connected directly to the first attached vertical void oriented to the south-east through two windows with a dimension of 0.5 × 1.45 m at the main entrance and the second with a dimension of 0.5 × 2.85 m at the kitchen. The field measurement is

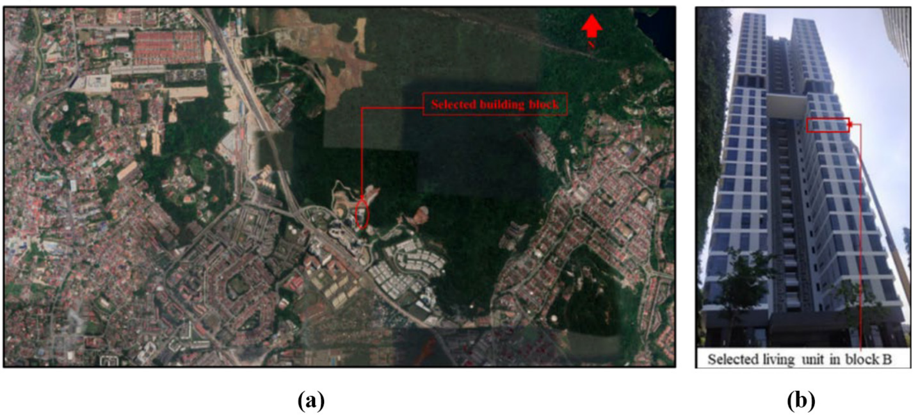


Figure 1. (a) Location of the case study residential building block, Kuala Lumpur, Malaysia, (b) the case study living unit in block B

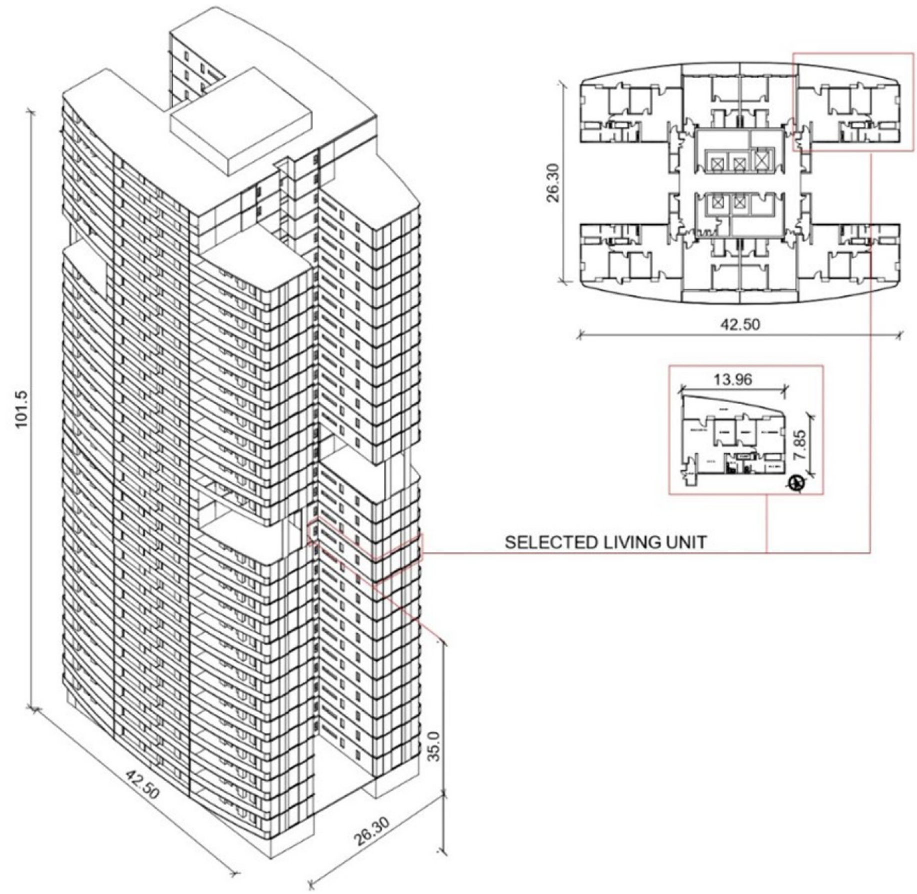


Figure 2.
Dimension details of the case study residential building and the selected living unit

conducted in the living room, the main entrance on the balcony and window connected to void.

2.1.1 Field measurement setup and the measuring instruments. The current research is limited to wind-driven ventilation because the relative humidity and air temperature are excluded from the data collection. Only internal air velocity (m/s) were measured at different positions in the living unit. Figure 3 demonstrates the positions of a total of three recording instruments positioned in the living unit at different places with similar height. The internal air velocity (m/s) was conducted using three similar anemometers (Kestrel 4000 weather tracker) named according to their position in the living unit, namely, P1, P2 and P3. All the instruments were calibrated from the factory and function appropriately for the experiment objectives. Thus, they are suitable and reliable to conduct field measurements. The (Kestrel 4000 weather tracker) can measure the air velocity that varies between 0 and 40.0 m/s with an accuracy of ± 0.03 m/s with capable of storing up 2000 data point for every measurement with store rate varies from 2 s to 12 h.

All three sensors were installed at the same height of 1.6 m above the unit floor level as the typical human height based on the recommendations of Omrani *et al.* (2017). The

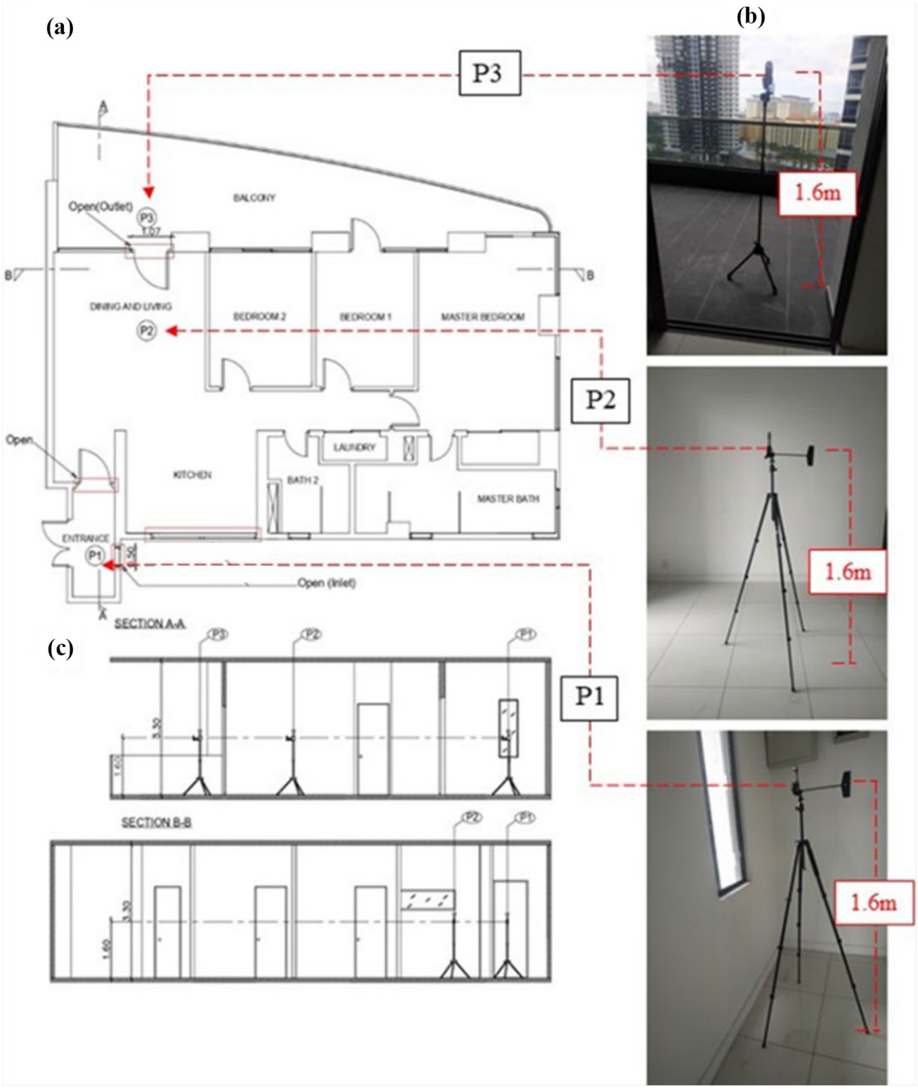


Figure 3.
(a) Location of the measurement instruments, (b) the measurement instruments in the living unit, and (c) section of the living units with the height of the instruments

anemometer P1 was located at the balcony in front of the sliding door to record the external wind speed entering the living unit, P2 was located at the living room in the same line (parallel to P1), and P3 was located at the window connected to the vertical void (as presented in Figure 3). Table 1 illustrates the details of the manometers of the experiment.

The data storage interval was set for 5 min for all the anemometers as suggested by Muhsin *et al.* (2016) over from 4th April 2019 until 5th June 2019, the data of 24 h is taken as suggested by previous research of Aflaki *et al.* (2019).

All doors and windows remained closed during the experiment except the balcony's sliding door, main entrance door, kitchen window and the window connected to the void. All the cooling and heating mechanical systems were not working to avoid any effects on the data, and no occupants were living in the unit during the experiment.

The meteorological data from the Malaysian metrological weather department (Malaysian Meteorological Departement, 2019) of the nearest weather station to the case study building was used as reference weather conditions, the Petaling Jaya weather station height is 10 m and situated around 15 km from the case study building which considered as an acceptable distance for the data collection (Malaysian Meteorological Departement, 2019). This study was limited to the wind-induced ventilation; thus, the temperature and relative humidity were excluded, and wind speed was used for the CFD validation and the indoor air velocity performance calculation.

2.2 Numerical simulation

2.2.1 Computational fluid dynamics building model geometry. CFD simulation is divided into two main phases:

(1) CFD building model validation (CVM); and

(2) Suggested CFD building models with similar void dimensions to the field measurement and complied with requirements of the Malaysian Uniforms Building by-Laws (UBBL, 2015).

Omrani *et al.* (2017) clarified that a fully open balcony type has an insignificant influence on the airflow movement and the internal air velocity. Accordingly, the CFD validation model (CVM) has been simplified and reproduced numerically using 3DSmax software without the existing balconies to reduce the computational cost without affecting the CFD simulation solution.

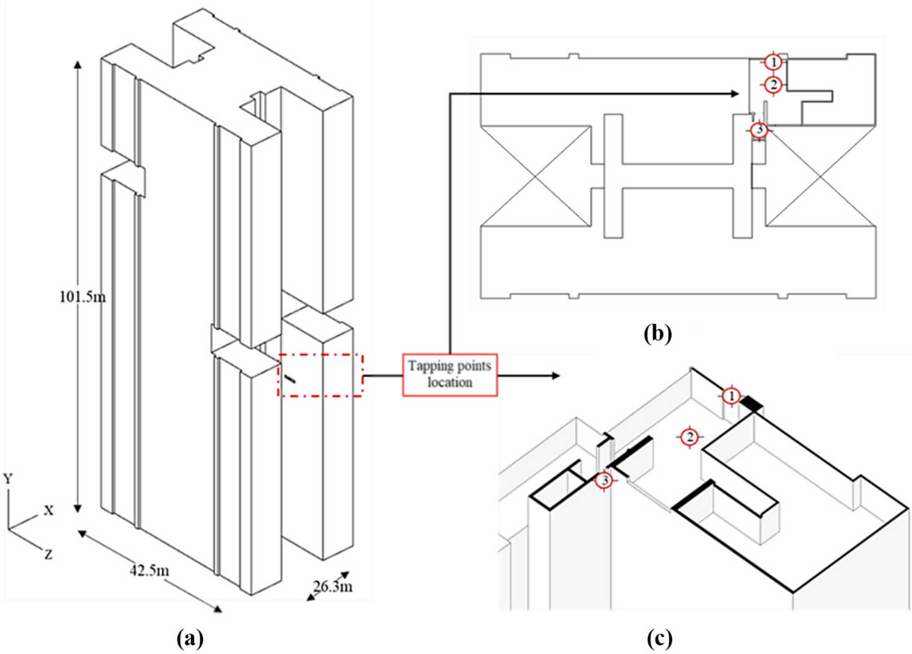
As illustrated in Figure 4, the internal of the living unit in the (CVM) is simplified (the doors, window and the separation walls, kitchen details). The walls thickness set to 0.10 m, similar to the real case to reduce the computational costs and the number of the cells generated during the simulation without affecting the accuracy of the results as recommended by previous studies (Aflaki *et al.*, 2019). The points are located inside the living unit as in the field measurement.

To evaluate the impacts of the voids combination on indoor ventilation performance, the building models were classified based on the type and the size of horizontal voids; thus, the simulated models were generated with different horizontal voids types and size. Two different horizontal voids sizes were generated and expressed as percentages of the living unit height (0 and 50%) in addition, two horizontal voids types were defined, namely, single-sided and double-sided horizontal voids.

All the CFD models with different horizontal voids sizes, the ventilation of cross ventilation strategy was applied in the living unit where the two openings similar in the

Table 1. Summary of the field measurement instrumentation

Sensors No.	Instrument	Location	Height (m)	Parameters	Accuracy	Range
P1	(Kestrel 4000 weather tracker)	Main entrance	1.6 m	Internal air velocity	± 0.03 m/s	0.4 to 40.0 m/s
P2	(Kestrel 4000 weather tracker)	Balcony	1.6 m	Internal air velocity	± 0.03 m/s	0.4 to 40.0 m/s
P3	(Kestrel 4000 weather tracker)	Living room	1.6 m	Internal air velocity	± 0.03 m/s	0.4 to 40.0 m/s



Notes: (a) CFD valadation model axonometry; (b) CFD living unit plan; (c) CFD living unit plan axonometry

Figure 4. CFD validation model with the points located in the living unit

surface opening of 1.5 m^2 as recommended by the UBBL (2015) as the minimum opening 5% required in the living units rooms. The openings are situated on the windward and leeward walls connected directly with the outdoor environment and the central vertical void space.

Figure 5 and Table 2 illustrate the details of the generated building models. All the voids configurations were evaluated under cross-ventilation mode.

2.2.2 Computational domain and numerical grid. As suggested by previous studies to captivate the impacts of the surrounding environment (Bharat and Ahmed, 2012; Guo *et al.*, 2015; van Hooff and Blocken, 2010), the obstacles were considered in the computational domain. In this study, the external environment was included in the simulation to capture precisely and evaluate the real wind flow effects accurately on the case study building, and the indoor ventilation performance, the surrounding obstacles around the building model within a range of 200 m were taken in consideration. Thus, the case study building model was placed in the domain adjoining to high rise residential building models with a height of 100 m situated at the South-West (SW) direction. The terrain was considered flat in the generated domain, as the terrain profile where the case-building model located has a small slope to reduce the computational time and the airflow complexity (as illustrated in Figure 6).

The computational domain for the CFD simulation is generated according to the suggestions of the best practice guideline (Franke *et al.*, 2007). The domain dimension is created according to the building model height (H). The upstream set to 5H, downstream, the lateral side and height, was set to 10, 4 and 6 H. The computational domain dimension is 1819 × 1086 × 609 m (length × width × height). Figure 10 illustrates the surrounding buildings models and the validation building model within the computational domain. The

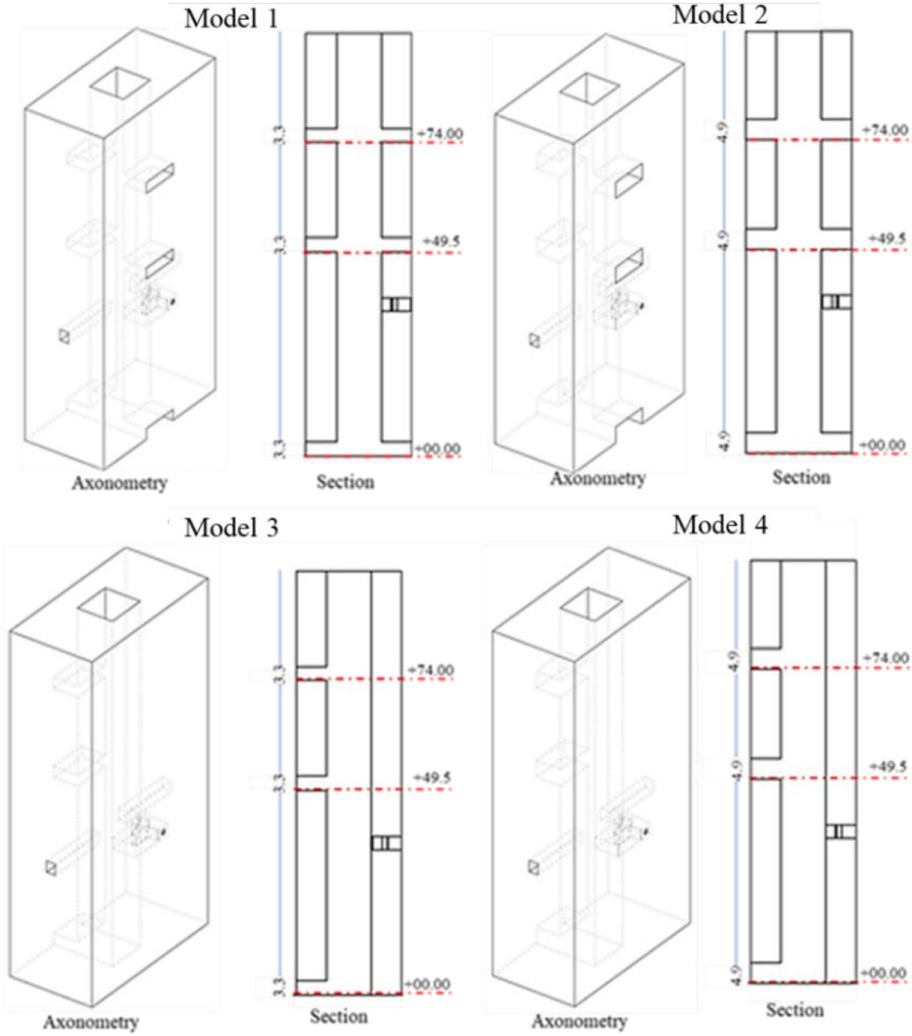
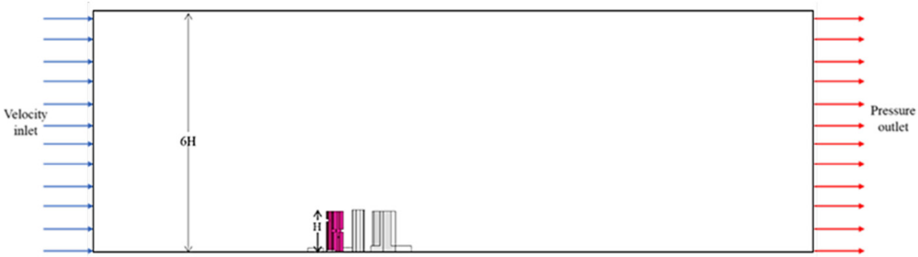


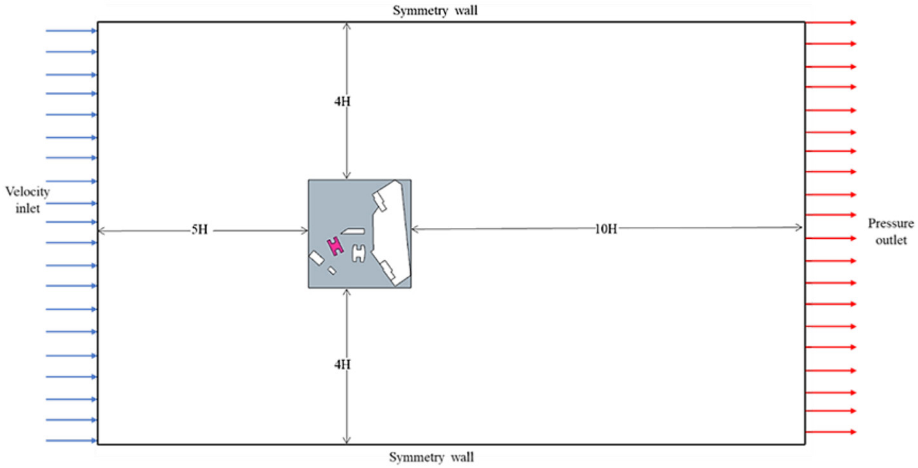
Figure 5.
CFD building models for natural ventilation evaluation: Model 1, Model 2, Model 3 and Model 4

Table 2.
Detail CFD building models: Model 1, Model 2, Model 3 and Model 4

Building model categories	Models	Horizontal void size (%)	Horizontal opening dimensions (length ×height)	Remarks
1. Building with double sided horizontal voids	Model 1	0	11.98 × 3.3	/
	Model 2	50	11.98 × 4.9	/
2. Building with single sided horizontal voids	Model 3	0	11.98 × 3.3	/
	Model 4	50	11.98 × 4.9	/



(a)



(b)

Figure 6. (a) Lateral and (b) top view of the computational domain for the CFD validation model

building surfaces blockage ratio and the domain cross-section are 1.69%, which is less than 3% recommended by a previous study (Tominaga *et al.*, 2008).

CFD simulation solution accuracy is affected fundamentally by the type and the grid resolution, namely, the number of cells generated in the whole computation domain and the grid cells dimension (Meng *et al.*, 2018). The computational domain grid can be classified into two main categories: structured grids and unstructured grids (Meng *et al.*, 2018). "Unstructured" grids connect the grid cells through points in different polyhedral forms as tetrahedra, primes, pyramids or hexahedra form in the three-dimensional model.

Unstructured grid with tetrahedral cell reported to be widely used in the CFD simulation of the internal air velocity and flow around the building models (Izadyar *et al.*, 2020; Muhsin *et al.*, 2017; Omrani *et al.*, 2017). Two growth rates of 1.15 and 1.2 were applied for mesh density in a radius of 42 and 200 m of the case study building model. A refinement method of face and edges was applied to the building model wall surfaces and openings. Maximum mesh sizes of 0.13 m were applied to the living unit, and the adjacent corridor surfaces and openings. The maximum mesh size applied for the building surfaces adjacent to the opening was 1.4 and 0.13 m at the surface next to the living unit and corridor with a minimum edge length of 3.93e-002 m.

As clarified by Meng *et al.* (2018), mesh sensitivity analysis allows identifying and eliminating the impacts of the generated meshes on the results of the CFD simulation and reducing the computation time (Montazeri and Blocken, 2013). Thus, the numerical simulation results are compared to each other to verify the (CVM) solution. Five meshes, namely, very coarse mesh, coarse mesh, medium mesh, fine mesh and very fine mesh, with 0.174, 0.722, 1.6, 4.4 and 10.2 million elements respectively, are generated CFD validation model using the tetrahedral method. The refinement ratio is the ratio between the number of the mesh cell elements in the fine mesh Δ_{fine} and the coarse mesh Δ_{coarse} as shown in the following equation [equation (1)] (Celik, 2008).

$$r = \left(\frac{\Delta_{\text{fine}}}{\Delta_{\text{coarse}}}\right)^{\frac{1}{3}} \tag{1}$$

As suggested by Celik (2008), recommended the refinement ratio to be higher than 1.3 to allow the discretization to be determined from the other error sources. Table 3 represents the elements of the grid for all generated meshes Δ_1, Δ_2, Δ_3, Δ_4 and Δ_5 with the refinement ratio r_{21}, r_{32}, r_{32}, r_{43} and r_{54}.

Air velocity (m/s) in the measured points P1, P2 and P3 from the CFD simulation results have been analysed for the mesh sensitivity, which was conducted based on five different meshes. A 27%, 11% differences were found between the very coarse, coarse and the medium mesh and 7, 6 difference between the fine, very fine and the medium mesh. Thus medium-mesh consisted of 1.6 million elements considered to provide a grid-independent solution and was selected for further simulation analysis.

2.2.3 Boundary condition. The inlet of the boundary condition is set to have an atmospheric boundary layer (ABL) wind profile using the power law equation [equation (2)], and the obtained data were used to the boundary condition inlet.

$$\frac{V_z}{V_{\text{ref}}} = \left(\frac{Z}{Z_{\text{ref}}}\right)^{\alpha} \tag{2}$$

where V_z(m/s): The wind speed at height Z(m), V_{ref} (m/s): The reference wind speed at the reference height Z_{ref} (10 m), α: The exponent that represents the terrain roughness. Taking into account the location of the case study building in Kuala Lumpur within large surroundings obstacles (50 m in height), α was set to 0.4, as recommended by Cermak *et al.* (1999).

Before the CFD model simulation investigation of the building models, the CFD building model was validated against the field measurement data explained in Section 3.1. For the CFD model validation, the boundary inlet for the ABL was set according to the meteorological weather station wind data of the reference velocity and wind direction during the same time of field measurement (Malaysian Meteorological Departement, 2019). For the

Table 3.
Generated mesh type with the refinement ratio

Mesh type					Refinement ratio (r)			
Very coarse	Coarse mesh	Medium mesh	Fine mesh	Very fine mesh	r_{21}	r_{32}	r_{43}	r_{54}
Δ_1	Δ_2	Δ_3	Δ_4	Δ_5	1.60	1.31	1.39	1.32

building models investigation the reference velocity V_{ref} (m/s) was set to 1 m/s according to the annual average wind speed extracted from the meteorological weather station of Petaling Jaya (Malaysian Meteorological Departement, 2019) at reference height (10 m) (meteorological weather station height). Turbulence intensity was set to medium intensity of 5% for the inlet domain.

For all the building models simulation, the boundary condition outlet was set to pressure outlet with relative pressure of zero 0 (Pa). The top and lateral boundaries were set to symmetry. Wall boundary conditions were applied to the buildings surfaces and the ground, as suggested by the best practice guideline (Franke *et al.*, 2007). walls were set to no-slip wall with no additional surface roughness in the boundary condition walls, as illustrated in Figure 7.

The wind directions were set under two wind directions, which are perpendicular wind conditions (represent the northeast (N-E) wind direction and second is oblique wind condition represents the east(E) wind direction, which represents the dominant wind directions. It is important to note that both wind directions were set under similar boundary conditions.

2.2.4 Computational fluid dynamics model and solver control setup. The CFD software Ansys CFX 18.1 used to conduct the simulation. RANS turbulence model was widely used in investigating wind-induced ventilation in residential buildings models due to its capacity to predict the airflow with less computational time compared to the other turbulence models spatially LES turbulence model (Toja-Silva *et al.*, 2015). Among the numerous RANS turbulence models, namely, standard k-ε (SKE), RNG k-ε, realizable k-ε (RLZ), SST k-⊠ and Reynolds stress model RSM (van Hooff *et al.*, 2017). The two-equation standard K- ε model was the most common and extensively used RANS simulations model (Cheng *et al.*, 2003). Standard K–ε turbulence model offers a good prediction of the internal and external airflow characteristic around the physical models of medium and high residential buildings (Abdul Razak1, Noor Hanita1, 2012; Ai *et al.*, 2011; Montazeri and Blocken, 2013; Muhsin *et al.*, 2017; Muhsin *et al.*, 2017; Prajongsan and Sharples, 2012), for that, it has been selected for the current study. The standard K-ε turbulence models stability offers reliable and accurate airflow prediction results with acceptable computational time (Yim *et al.*, 2009; Yoshie *et al.*, 2007).

The spatial discretization was set to second-order, the convergence was assumed to be reached when all the residuals assumed to reach the convergence target of 10^{-4}. According to Different studies (Farea *et al.*, 2015; Montazeri and Blocken, 2013; Muhsin *et al.*, 2017) that recommended that achieve a convergence solution, the number of iteration reach down to 0,0001 is suitable. Table 4 shows the selected values applied to solver control.

3. Computational fluid dynamics simulation results

3.1 Computational fluid dynamics building model validation

The data obtained from the field measurements were compared with the CFD validation model simulation results for cross-ventilation mod. The CFD model validation was conducted under the perpendicular wind direction to the experiment living unit aperture. Thus, the data of the internal air velocity from the north east wind direction were extracted. Average time of 5 min intervals was used for the numerical simulation validation. The obtained data of the external wind from the reference weather station were applied for the inlet of the boundary condition. A time average of 5 min intervals for all the data at each of three measuring points in the living unit was compared to the results of the points located at the same coordinates in the CFD model units.

Figure 8 illustrates the deviation (%) between the field measurement experiment data and the CFD simulation of the building validation model of air velocity (m/s) in the points. The deviation for P1, P2 and P3 is 5.308, 3.921 and 9.825%, respectively. The range of

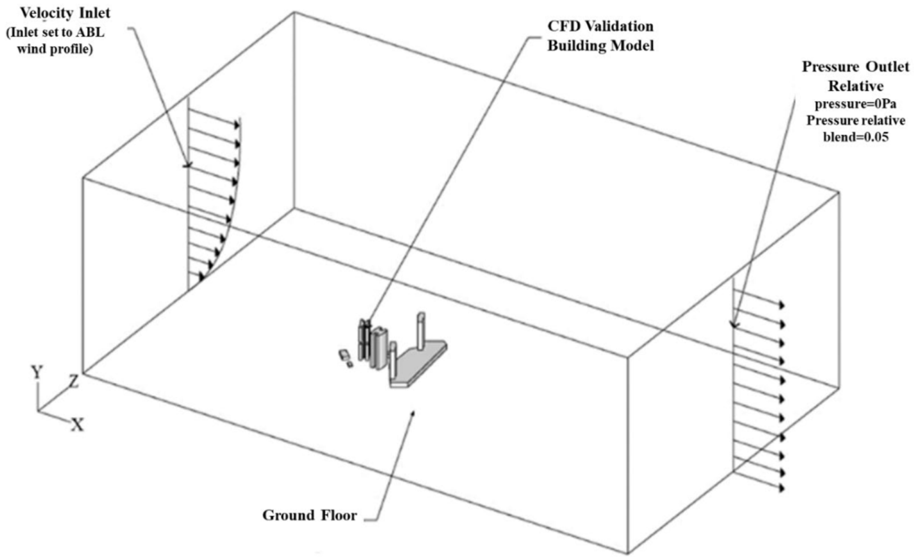


Figure 7.
Boundary condition setup to generate atmospheric boundary layer (ABL) wind profile

Table 4.
Selected values of the solver control setups

Basic settings	Advection scheme	Turbulence numeric	Convergence control	Fluid timescale control	Convergence criteria
Values	Blend factor = 1.0	Second order	Minimum iteration = 1 Maximum iterations = 1000	Timescale control = automatic	Residual type = RMS Residual target = 10–4

deviation per cent is varied in different studies (Meng *et al.*, 2018; Montazeri and Blocken, 2013; Muhsin *et al.*, 2017). The discrepancy between the simulation results and the measured air velocity was due to the CFD simulations steady-state assumption (Omrani *et al.*, 2017), the CFD model lacks the ability to predict the changes in the internal airflow movement pattern (Lo *et al.*, 2012). Thus, the CFD solution using a grid with 1.6 million cells revealed a strong agreement with the field measurement experiment data, predicted and calculated the internal air velocity (m/s) in the unit with an acceptable range of deviation. Review of the previous studies showed that there is no particular acceptable range of deviation in the CFD simulation; however, most of the studies found that the acceptable range is around 10% (Meng *et al.*, 2018; Montazeri and Blocken, 2013; Muhsin *et al.*, 2017).

3.2 Computational fluid dynamics simulation of the external airflow

Simulation results in the computational domain revealed that the surrounding buildings influence the external wind flow speed (m/s) and air pressure (Pa) around the buildings envelope. Figure 9 illustrates the CFD simulation of the wind flow in the computational domain through a top view of the contour plan located at the height of 37.9 m at the same height of the measuring points. The wind flow from the North-East (NE) direction is coming from the inlet (The right side of the domain) set with atmospheric boundary layer wind profile, hitting the windward wall of the building models at an angle of 71³.

The envelope of the surrounding building models has apparent effects on the pattern of external airflow movement and air velocity magnitude (m/s) as a consequence decreasing the downstream velocity at the back of the buildings compared to higher upstream velocity at windward of the building envelope. Under the wind direction of North-East (NE), air velocity magnitude (m/s) and total pressure (Pa) were determined in the outlet, inlet of the living unit and the opening connected to the vertical void at the height of 1.6 m from living unit floor. Table 5 represents the selected parameters' value in the living unit apertures under the north east (NE) wind direction.

3.3 Evaluation of void configurations effects on natural ventilation performance

3.3.1 Results of internal air velocity inside the void space. Airflow simulation through the voids space showed that under North-East (NE) direction, airflow recirculation has emerged in the vertical void space of both Model 1 and 2 with a horizontal void larger by 50%. Larger, clear and multiply airflow recirculation zones showed in Model 1 with low airspeed (illustrated in dark blue colour), at the lower and middle floor levels compared to small airflow recirculation zone at the middle levels in Model 2. In Model 1, at the horizontal voids level, the airflow has emerged parallel to the double-sided horizontal voids opening with high speed (illustrated in light green colour) (as presented in Figure 10).

Under East (E) direction, airflow recirculation appears in Models 1 and 2 with a large recirculation zone in Model 2 at top levels above the top horizontal void, near the outlet of the vertical void and small recirculation zones near the vertical void walls. Airflow recirculation appeared in all models; however, the differences are in the recirculation magnitude, where it is higher under the East (E) direction in Model 2. According to Farea *et al.* (2015), these turbulences that emerged near the horizontal voids internal opening occurred due to non-parallel external wind hitting the front internal walls of the central vertical void.

Figure 11 illustrates the airflow pattern in the single-sided horizontal void models. All the building models have similar airflow patterns at North-East (NE) direction with small airflow recirculation at the lower levels near the ground floor horizontal where the airflow speed (m/s) increased in the middle and the top levels. Under East (E) wind direction, both Models 3 and 4 showed similar airflow patterns inside the vertical void where large airflow recirculation appeared at the lower and middle levels with low airspeed (illustrated in dark blue colour), the airflow is parallel to the walls at higher levels. Model 4 showed large airflow

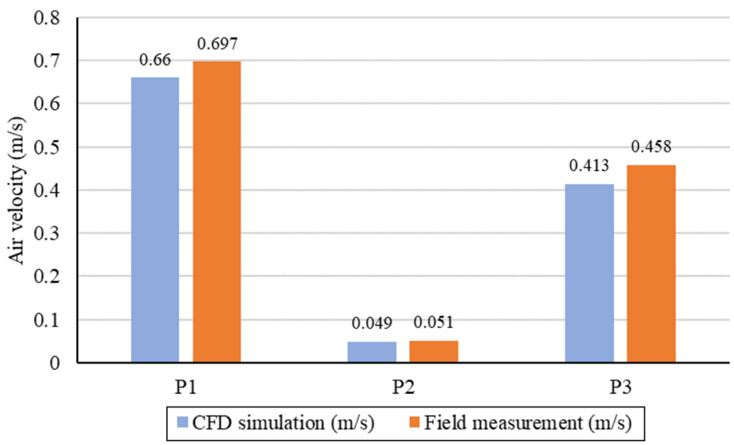


Figure 8.
Comparison of the internal air velocity (m/s) between the CFD simulation and the field measurements data under north east wind direction

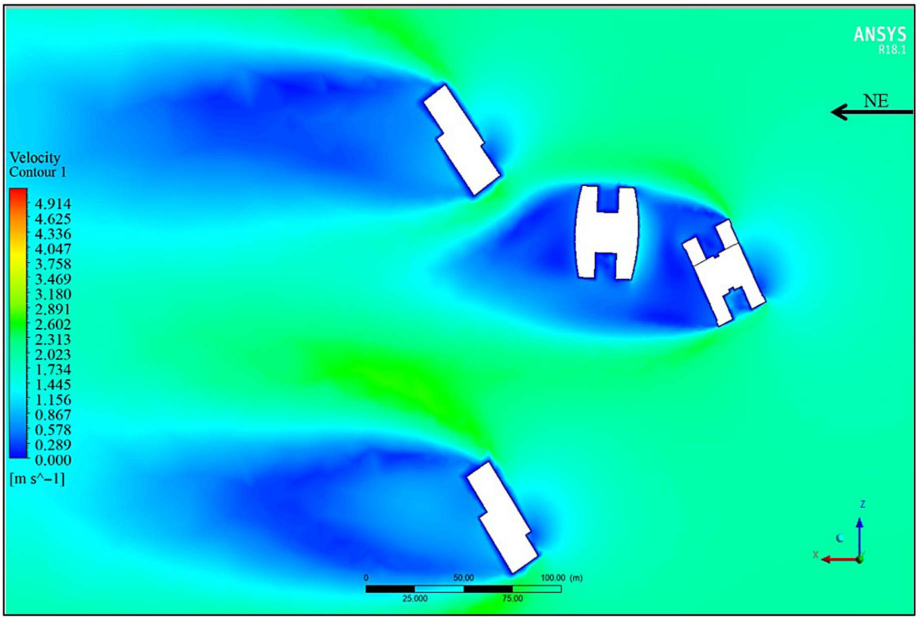


Figure 9.
Top view of velocity contour (m/s) in CFD computational domain at the height level of 37.9 m

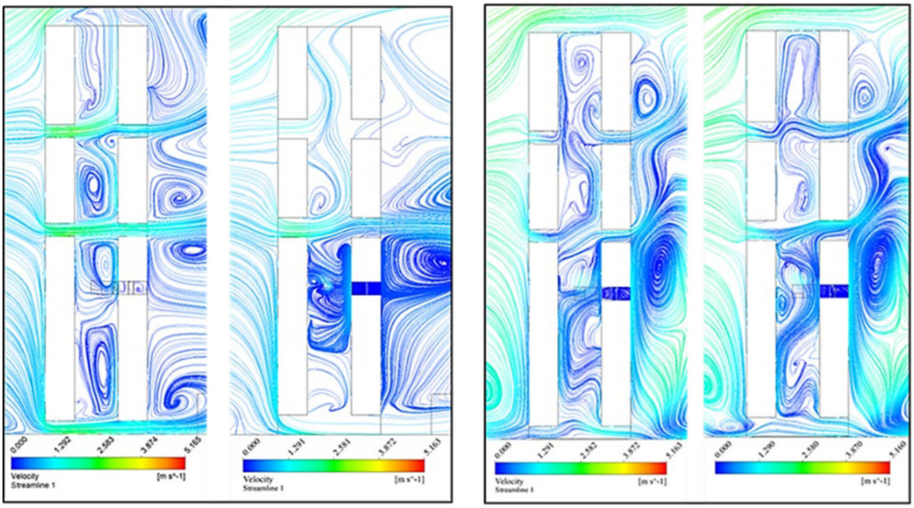
Table 5.
Comparison of the CFD simulation results at the CFD model apertures under North East wind direction

Values	Inlet aperture	Outlet aperture	Void aperture
Velocity magnitude (m/s)	0.4	0.402	0.605
Absolute pressure (Pa)	101323.695	101323.906	101323.641

circulation with higher speed at the level of horizontal void opening (74 m) compared to Model 3.

Under wind direction facing the lateral walls of the single-sided horizontal building models, it was found that airflow speed (m/s) decreased in all models due to the airflow recirculation that mainly appeared at the level of the single-sided voids. Caused by the wind flow hitting the lateral walls in the vertical central void and above the single-sided horizontal voids openings in the leeward wall of the front block and the windward wall of the front block particularly in lower levels.

3.3.2 Results of internal air velocity inside the living unit. Further analyses were carried out to investigate the impacts of voids combinations and wind direction on the internal air velocity in the living unit. The contour of the air velocity presented in Figure 12 illustrates the airflow inside the living units' internal space. The pressure differences between the inlet and the outlet caused the airflow to circulate inside the living unit space due to the higher air pressure (Pa) at the inlet connected directly with the internal vertical void space compared to lower air pressure (Pa) at the outlet.



Model 1 Model 2

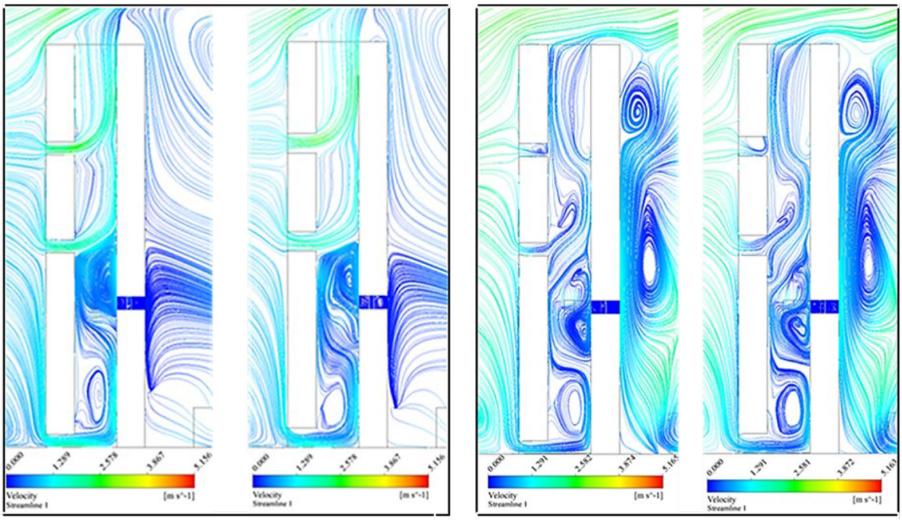
(a)

Model 1 Model 2

(b)

Notes: (a) North east; (b) east

Figure 10.
Velocity streamlines in Models 1 and 2 under (a) north east wind direction and (b) east wind direction



Model 3 Model 4

(a)

Model 3 Model 4

(b)

Notes: (a) North east; (b) east

Figure 11.
Velocity streamline in test configuration 3 and 4 under (a) north east wind direction and (b) east wind direction

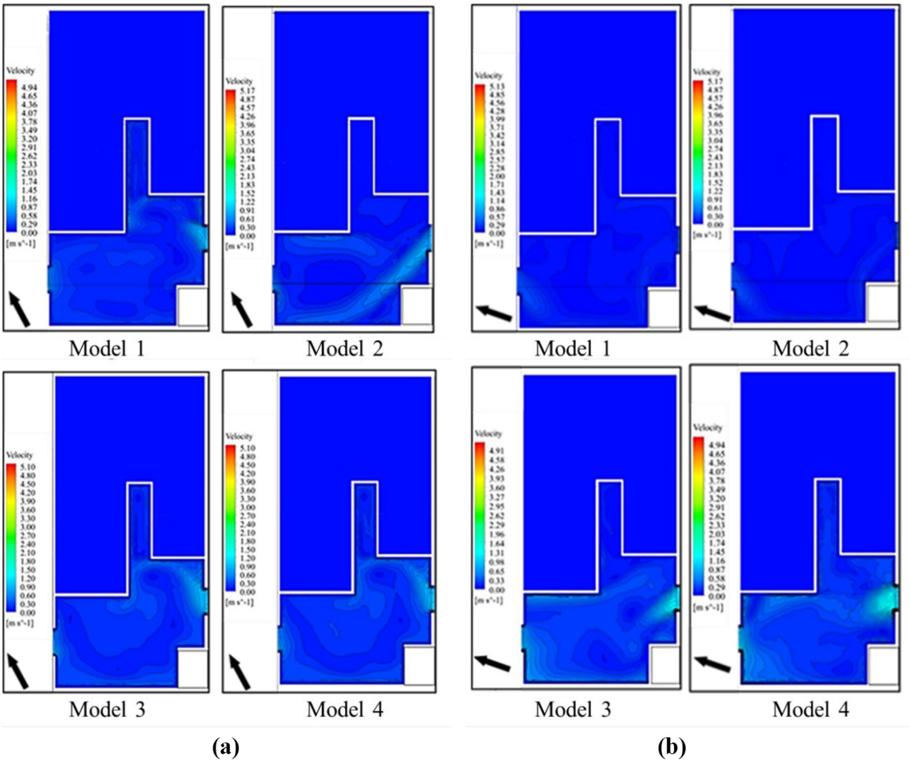


Figure 12.
Air velocity contour in the selected living unit in the CFD building models under east (E) and north east (NE) wind direction

Notes: (a) East direction; (b) north east direction

The quantitative measurement of the internal air velocity (m/s) confirms the living units velocity contours descriptive analysis. For north east (NE) wind direction, the indoor air velocity has the increase in building models with single-sided horizontal voids, the indoor air velocity at Model 4 with single-sided horizontal void larger by 50% increased by 322.37% reaching 0.471 m/s, while in Model 3 increased by 254.25% to 0.395 m/s. The indoor air velocity decreased in both models with a double-sided horizontal where it has the lowest decrease in Model 1 by 6.93% to 0.103 m/s, followed by model 2 by 23.78%, reaching 0.085 m/s compared to the reference building model.

For east (E) wind direction, indoor air velocity increased in all building models. The highest indoor air velocity increased was recorded in Model 4 with single-sided horizontal voids larger by 50% where it increased by 71.59% reaching 0.13 m/s followed by Models 3, 1 and 2 by 56.56, 14.01 and 8.57% reaching 0.118, 0.086 and 0.082 m/s, respectively.

Based on the simulation results, it was found that introducing voids combinations improve the indoor air velocity and hence provides better indoor thermal comfort in the living units of high-rise residential buildings in hot and humid climates. Different guidelines and regulations set minimum requirements regarding the air velocity for suitable thermal comfort. Lechner (2014) indicated that suitable air velocity for thermal comfort varies between 0.25 to 1.3 m/s. According to Ghiaus and Allard (2012), achieving optimal indoor thermal comfort in the hot and humid climate like Malaysia requires air velocity between 0.2

and 1.5 m/s. The Malaysian standard (MS:2680, 2017) set suitable air velocity varied from 0.25 to 0.5 m/s and from 0.5 to1.0 m/s. Thus, this study shows that introducing voids combinations, particularly vertical void combined with single-sided horizontal voids, can achieve the acceptable indoor air velocity range required for thermal comfort.

4. Conclusion

Field measurement was conducted in a high-rise residential unit to record the internal air velocity. The collected data revealed a strong agreement with the CFD validation simulation results. A detailed investigation of natural ventilation performance was performed. Building models were suggested based on various parameters, including horizontal voids connected to vertical void: horizontal voids size and wind direction. As indicated, all the building models share a similar vertical void size that complies with the Malaysian building code (UBBL, 2015). The following results were found:

The atmospheric boundary layer (ABL) simulation revealed that the surrounding buildings had marked impacts on the external wind flow characteristic around the buildings envelope in addition to the airspeed and pressure around the inlet and outlet of the living unit resulting of change in the internal airflow characteristic and air velocity performance in the internal space of the living unit. Thus, generating the ABL in the computational domain need to take into account the terrains type, location of the case study building within the surrounding buildings to determine the accurate values of air velocity and pressure at the living unit apertures and consequently evaluate the airflow and the internal air velocity performance accurately.

The simulation results of the proposed voids combination design feature revealed that the horizontal void type, size and position along the building height could significantly influence the void spaces airflow characteristic. The single-sided horizontal void produces better airflow quality characterized with parallel pattern and higher speed, particularly at higher floor levels compared to the double-sided horizontal voids that showed the worst airflow quality with airflow recirculation zones emerged at most of the floor levels with low airspeed. For that reason, the void configuration can affect the internal air velocity performance in the unit of the high-rise residential building directly.

The simulation results revealed that building models with single-sided horizontal voids outperformance models with double-sided horizontal voids under oblique and perpendicular wind angles regarding achieving required internal air velocity for thermal comfort varied from 0.2 to 1.3 m/s. Simulation results revealed that increasing the size of the single-sided horizontal voids by 50% in Model 4 increased significantly indoor air velocity by 322.37% reaching to 0.471 m/s under (NE) wind direction and by 254.25% to 0.395 m/s in Model 3 meeting the requirements of the guidelines and under east (E) direction by 71.59 and 56.56% to 0.13 and 0.118 m/s, respectively. However, it does not reach the requirement. On the other hand, indoor air velocity achieved the lowest indoor air velocity, where it decreased by 23.78% to 0.085 m/s in Model 2 with double-sided horizontal voids larger by 50%, which is far away so close to the minimum requirements.

This study aims to provide a comprehensive understanding of the impacts of voids combinations on internal air velocity performance in the unit of high-rise residential buildings in the hot and humid climate and can be useful as further knowledge to be considered by the minter of housing and local government and the building guidelines makers in Malaysia such as the (Uniform Buildings By-Laws) and for architects during the early stages of the design process.

Based on this study, further limitations needed to be addressed in future works. This study is limited to wind-driven ventilation and excluded the impacts of air temperature and buoyancy-driven ventilation on indoor ventilation performance that worth evaluating in future works. In further studies, turbulence models such as RANS (RNG $k-\varepsilon$, realizable $k-\varepsilon$, SST $k-\omega$ and Reynolds stress model) are recommended to be tested for the natural ventilation performance investigation. Unstructured grid with tetrahedral cell forms was used for mesh sensitivity analysis, the CFD model validation and the evaluation of the voids configurations; the structural grid with hexahedral cells is worth examining in further studies. Further studies are recommended to be conducted to investigate the voids combination impacts on indoor ventilation performance in a living unit with single and cross-ventilation modes located at different orientations and heights.

References

Aflaki, A., Mahyuddin, N. and Baharum, M.R. (2016), "The influence of single-sided ventilation towards the indoor thermal performance of high-rise residential building: a field study", *Energy and Buildings*, Vol. 126.

Aflaki, A., Hirbodi, K., Mahyuddin, N., Yaghoubi, M. and Esfandiari, M. (2019), "Improving the air change rate in high-rise buildings through a transom ventilation panel: a case study", *Building and Environment*, Vol. 147, pp. 35-49.

Ahmad, T., Aibinu, A. and Jamaluddin, M. (2017), "The effects of high-rise residential construction on sustainability of housing systems", *Procedia Engineering*, Vol. 180, pp. 1695-1704.

Ai, Z.T., Mak, C.M., Niu, J.L. and Li, Z.R. (2011), "The assessment of the performance of balconies using computational fluid dynamics", *Building Services Engineering Research and Technology*, Vol. 32 No. 3, pp. 229-243.

Allard, F. and Ghiaus, C. (Eds) (2012), *Natural Ventilation in the Urban Environment: Assessment and Design*, Routledge.

Bharat, A. and Ahmed, A.S. (2012), "Effects of high rise building complex on the wind flow patterns on surrounding urban pockets", *International Journal of Engineering Research and Development*, Vol. 4 No. 9, pp. 21-26.

Celik, I.B. (2008), "Procedure for estimation and reporting of uncertainty due to discretisation in CFD applications", *Journal of Fluids Engineering*, Vol. 130 No. 7, p. 78001.

Cermak, J.E., Jack, E. and Isyumov, N. (1999), *Wind Tunnel Studies of Buildings and Structures*, American Society of Civil Engineers.

Chan, S. (2014), *Social Performance of Communal Sky Garden in a Dense Urban City*, Hong Kong, (April).

Chan, S. (2005), "Communal sky gardens for high-rise residential buildings", *In the 2005 World Sustainable Building Conference*, Vol. 2005, pp. 2274-2279.

Cheng, Y., Lien, F.S., Yee, E. and Sinclair, R. (2003), "A comparison of large eddy simulations with a standard k-ε Reynolds-averaged Navier-Stokes model for the prediction of a fully developed turbulent flow over a matrix of cubes", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 91 No. 11, pp. 1301-1328.

Chiang, W.H. and Anh, N.D. (2012), "Natural ventilation inside courtyard-apartment building in Taiwan", *Fourth German-Austrian IBPSA Conference*, Berlin, pp. 392-399.

Chu, C. and Chiang, B. (2014), "Wind-driven cross ventilation in long buildings", *Building and Environment*, Vol. 80, pp. 150-158.

Daniel, E. and Williams, F. (2007), *Sustainable Design Ecology, Architecture, and Planning*, Vol. 53. John Wiley and Sons.

Elotefy, H., Abdelmagid, K.S.S., Morghany, E. and Ahmed, T.M.F. (2015), "Energy-efficient tall buildings design strategies: a holistic approach", *Energy Procedia*, Vol. 74.

Farea, T.G., Ossen, D.R., Alkaff, S. and Kotani, H. (2015), "CFD modeling for natural ventilation in a lightwell connected to outdoor through horizontal voids", *Energy and Buildings*, Vol. 86, pp. 502-513.

Farea, T.G. (2012), "Common configuration of light-well in high- rise residential buildings in Kuala Lumpur", *4th International Network for Tropical Architecture (iNTA) National University of Singapore*, pp. 1-8.

Franke, A., Hellsten, H. and Schlünzen, B.C. (2007), *Best Practice Guideline for the CFD Simulation of Flows in the Urban Environment. COST Action 732: Quality Assurance and Improvement of Microscale Meteorological Models (2007)*, Hamburg, Germany (2007), (April 2016).

Fung, Y.W. and Lee, W.L. (2015), "Identifying the most influential parameter affecting natural ventilation performance in high-rise high-density residential buildings", *Indoor and Built Environment*, Vol. 24 No. 6, pp. 803-812.

Gaber, T., Remaz, D., Alkaff, S. and Kotani, H. (2015), "CFD modeling for natural ventilation in a lightwell connected to outdoor through horizontal voids", *Energy and Buildings*, Vol. 86, pp. 502-513.

Ghiaus, C. and Allard, F. (2005), *Natural Ventilation in the Urban Environment: Assessment and Design*, Bath Press, Bath.

Guo, W., Liu, X. and Yuan, X. (2015), "A case study on optimisation of building design based on CFD simulation technology of wind environment", *Procedia Engineering*, Vol. 121 No. July, pp. 225-231.

Ismail, A.M. (1996), "Wind-driven natural ventilation in high-rise office buildings with special reference to the hot-humid climate of Malaysia", Doctoral dissertation, Cardiff University.

Izadyar, N., Miller, W., Rismanchi, B. and Garcia-Hansen, V. (2020), "A numerical investigation of balcony geometry impact on single-sided natural ventilation and thermal comfort", *Building and Environment*, Vol. 177, p. 106847.

Lechner, N. (2014), *Heating, Cooling, lighting: Sustainable Design Methods for Architects*, John Wiley & Sons.

Liu, P., Lin, H. and Chou, J. (2009), "Evaluation of buoyancy-driven ventilation in atrium buildings using computational fluid dynamics and reduced-scale air model", *Building and Environment*, Vol. 44 No. 9, pp. 1970-1979.

Lo, L.J., Banks, D. and Novoselac, A. (2012), "Combined wind tunnel and CFD analysis for indoor airflow prediction of wind-driven cross ventilation", *Building and Environment*.

Malaysian Meteorological Department (2019).

Meng, F., He, B., Zhu, J., Zhao, D., Darko, A. and Zhao, Z. (2018), "Sensitivity analysis of wind pressure coefficients on CAARC standard tall buildings in CFD simulations", *Journal of Building Engineering*, Vol. 16, pp. 146-158.

Montazeri, H. and Blocken, B. (2013), "CFD simulation of wind-induced pressure coefficients on buildings with and without balconies: validation and sensitivity analysis", *Building and Environment*, Vol. 60, pp. 137-149.

Moosavi, L. and Mahyuddin, N. (2013), "Thermal performance of atria: an overview of natural ventilation effective designs", *Renewable and Sustainable Energy Reviews*, Vol. 34, pp. 654-670.

MS:2680 (2017), *MS 2680: 2017 Energy Efficiency and Use of Renewable Energy for Residential buildings - Code of Practice*, Department of Standards Malaysia, Malaysia.

Muhsin, F., Mohammad Yusoff, W.F., Mohamed, M.F. and Sapian, A.R. (2016), "The effects of void on natural ventilation performance in multi-storey housing", *Buildings*, Vol. 6 No. 3, p. 35.

Muhsin, F., Yusoff, W.F.M., Mohamed, M.F., Rasani, M.R.M. and Sapian, A.R. (2017), "Potential of voids to enhance natural ventilation in medium cost multi-storey housing (MCMSH) for hot and humid climate", *ARPN Journal of Engineering and Applied Sciences*, Vol. 12 No. 10, pp. 3137-3144.

Muhsin, F., Yusoff, W.F.M., Mohamed, M.F. and Sapian, A.R. (2017), "CFD modeling of natural ventilation in a void connected to the living units of multi-storey housing for thermal comfort", *Energy and Buildings*, Vol. 144, pp. 1-16.

Omrani, S., Garcia-Hansen, V., Capra, B. and Drogemuller, R. (2017), "Natural ventilation in multi-storey buildings: design process and review of evaluation tools", *Building and Environment*.

Pei-Chun, L., Ford, B., Etheridge, D., Liu, P., Ford, B. and Etheridge, D. (2016), "A modelling study of segmentation of naturally ventilated tall office buildings in a hot and humid climate tall office buildings in a hot and humid climate", *International Journal of Ventilation*, Vol. 3315 No. October.

Prajongsan, P. and Sharples, S. (2012), "Enhancing natural ventilation, thermal comfort and energy savings in high-rise residential buildings in Bangkok through the use of ventilation shafts", *Building and Environment*, Vol. 50, pp. 104-113.

Santamouris, M. (2016), "Cooling the buildings – past, present and future", *Energy and Buildings*, Vol. 128, pp. 617-638, doi: 10.1016/j.enbuild.2016.07.034.

Sapian, A. (2004), "Possibilities of using void to improve natural cross ventilation in high rise low cost residential building (HRLCRB)", Doctoral dissertation, Universiti Teknologi Malaysia.

Toja-Silva, F., Peralta, C., Lopez-Garcia, O., Navarro, J. and Cruz, I. (2015), "Roof region dependent wind potential assessment with different RANS turbulence models", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 142, pp. 258-271.

Tominaga, Y., Mochida, A. and Yoshie, R. (2008), "AIJ guidelines for practical applications of CFD to pedestrian wind environment around buildings", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 96 Nos 10/11, pp. 1749-1761.

Tony, I. (2013), "Sky-garden design in high-density high-rise residential development", *Urban Density and Sustainability*, pp. 1-8.

UBBL (2015), *Uniform Building by Laws 1984, as at 1st November 2013*, International Law Book Service, Malaysia.

van Hooff, T. and Blocken, B. (2010), "Environmental modelling and software coupled urban wind flow and indoor natural ventilation modelling on a high-resolution grid: a case study for the Amsterdam ArenA stadium", *Environmental Modelling and Software*, Vol. 25 No. 1, pp. 51-65.

van Hooff, T., Blocken, B. and Tominaga, Y. (2017), "On the accuracy of CFD simulations of cross-ventilation flows for a generic isolated building: comparison of RANS", *Building and Environment*, Vol. 114.

Yim, S.H.L., Fung, J.C.H., Lau, A.K.H. and Kot, S.C. (2009), "Air ventilation impacts of the 'wall effect' resulting from the alignment of high-rise buildings", *Atmospheric Environment*, Vol. 43 No. 32, pp. 4982-4994.

Yoshie, R., Mochida, A., Tominaga, Y., Kataoka, H., Harimoto, K., Nozu, T. and Shirasawa, T. (2007), "Cooperative project for CFD prediction of pedestrian wind environment in the architectural institute of Japan", *Journal of Wind Engineering and Industrial Aerodynamics*, Vol. 95 Nos 9/11, pp. 1551-111578.

Further reading

American, S., Of, H., R. and A.-C, E. (2011), *ASHRAE 55-2011: Thermal Environmental Conditions for Human Occupancy*, Vol. 2011, *ASHRAE*, *Atlanta*.

Kotani, H., Narasaki, M., Sato, R. and Yamanaka, T. (2003), "Environmental assessment of light well in high-rise apartment building", *Building and Environment*, Vol. 38 No. 2, pp. 283-289.

Mohamed, M.F., King, S., Behnia, M. and Prasad, D. (2013), "Coupled outdoor and indoor airflow prediction for", *Buildings Using Buildings*, Vol. 3 No. 2, pp. 399-421.

Omrani, S., Garcia-Hansen, V., Capra, B. and Drogemuller, R. (2017a), "On the effect of provision of balconies on natural ventilation and thermal comfort in high-rise residential buildings", *Building and Environment*, Vol. 123.

Omrani, S., Garcia-Hansen, V., Capra, B.R. and Drogemuller, R. (2017), "Effect of natural ventilation mode on thermal comfort and ventilation performance: Full-scale measurement", *Energy and Buildings*, Vol. 156.

Sapian, A.R., Abdul Majid, N.H. and Hokoi, S. (2012), "Natural ventilation around open ground floors with pilotis in high-rise residential buildings in tropical areas: harmonisation of modern and traditional housing in tropical areas", *Archi-Cultural Translations through the Silk Road 2nd International Conference*, *Mukogawa Women's Univ.*, *Nishinomiya*, pp. 219-224.

Corresponding author
Hamza Laloui can be contacted at: laloui.hamza@live.iium.edu.my