Design configurations analysis of wind-induced natural ventilation tower in hot humid climate using computational fluid dynamics

Chin Haw Lim1, Omidreza Saadatian1, Kamaruzzaman Sopian1, M. Yusof Sulaiman1, Sohif Mat1, Elias Salleh1 and K.C. Ng2
1Solar Energy Research Institute (SERI). Level 3, Perpustakaan Tun Sri Lanang, National University of Malaysia (UKM), 43600 Bangi, Selangor, Malaysia; 2Mechanical Department, National University of Singapore (NUS), Singapore, Singapore

Abstract

Wind-induced natural ventilation tower is one of the effective devices in enhancing indoor air quality. It can be designed and integrated as part of building components. This paper investigates the performance of various design configurations of a wind-induced natural ventilation tower with the focus on Venturi-shaped roofs and towers. The Venturi-shaped roofs and towers are used to create negative pressure in order to enhance the extraction air flow rates of the wind-induced natural ventilation tower. The computational fluid dynamics (CFD) method is used to analyse each of the design configurations. The different design configurations are based on roof tilt angles, roofs' shapes, tower heights and shapes of the wind-induced natural ventilation tower. The parameters analysed are extraction air flow rates and air flow pattern. Based on the CFD simulation results of various design configurations, the 'biconcave'-shaped wind tower has the best design configuration with 14 568.66 m³/h extraction air flow rates at 0.8 m/s external wind velocity.

Keywords: air change rates; extraction air flow rates; computational fluid dynamics (CFD); Venturi shaped; wind-induced natural ventilation tower

Received 19 February 2013; revised 26 April 2013; accepted 28 April 2013

1 INTRODUCTION

Natural ventilation is considered as one of the effective passive methods to provide healthy and comfortable indoor building environment. This strategy can be used to achieve an acceptable indoor thermal environment in hot and humid climate [1]. Sufficient natural ventilation is vital to prevent moisture development in the air and reduce pollutants concentration effectively [2]. According to a ventilation rates and health review by a group of researchers, there is still a need to study the relationship between ventilation rates and health especially in diverse climate and locations in building [3]. The ventilation rate and air change rates are two important parameters for evaluating indoor air quality (IAQ). Studies conducted in Seoul, Korea, show that natural ventilation is more effective than mechanical ventilation [4]. Problem of a low performance of mechanical ventilation devices is normally due to their poor maintenance. Allard [5] suggested that natural ventilation is more cost-effective than mechanical ventilation in terms of capital cost, maintenance and operational costs. Future buildings performance analysis revealed that providing a comfortable indoor environment without heavy reliance on mechanical cooling will be a major challenge [6]. Yu et al. [7] discovered that building occupants who dwell in a natural ventilated indoor environment have a stronger capacity for physiological regulation of heat shock than the occupants who stay in an air-conditioned environment.

Two fundamental principles of natural ventilation are the stack effect and the wind-driven ventilation. The stack effect is caused by temperature difference between the indoors and outdoors of buildings, and it happens when the inside building temperature is higher than the outside temperature. Naghman and Yuehong [8] observed that the stack effect reduces when the temperature differences between the indoors and outdoors of buildings are small. In
hot and humid climate conditions, due to the low-temperature difference between the indoor and outdoor temperature, the stack ventilation method is insufficient to create higher ventilation rates to achieve good air changes for the building occupants [9].

One of the wind-driven natural ventilation devices that have been neglected in current building industry is the windcatcher or wind-induced natural ventilation tower [10]. Hughes and Cheuk-Ming [11] discovered that wind-driven ventilation provides 76% more internal ventilation than buoyancy effects. Despite that, group of researchers based in Malaysia [10] emphasized that there are still insufficient research and studies conducted on the wind-induced natural ventilation tower performance in hot and humid climatic conditions. Wind-induced natural ventilation is based on pressure differences created by the wind. One of the key design criteria that influence the pressure differences of the wind-induced natural ventilation tower is the aerodynamic design of the roof of the wind tower. Studies conducted by Lim et al. [12], Blocken et al. [13] and Van Hoof et al. [14] revealed the effectiveness of Venturi-shaped roofs in improving the extraction power generated by the wind-induced natural ventilation tower. Negative pressure can be created underneath the roof of a wind tower by utilizing a Venturi-shaped roof for the wind tower. Thus, the pressure differences between the wind tower roof and the interior of the building will drive the air to flow out from the building through the wind tower. This phenomenon is based on the Bernoulli principle that states that in an inviscid and incompressible flow, the sum of its kinetic and potential energy must remain constant. Hence, when there is an increase in the speed of the fluid the static pressure decreases. The Bernoulli equation illustrates that:

$$P + \rho g h + \frac{1}{2} \rho V^2 = \text{Constant}$$

where $P$ is the static pressure; $\rho$ the density of fluid; $V$ the fluid velocity; $g$ the acceleration due to gravity; $h$ the relative height.

Based on the concept of Venturi geometry, this paper discusses various design configurations of the wind-induced natural ventilation tower aiming to improve its performance. An optimum design configuration of the wind-induced natural ventilation tower is vital in obtaining maximum extraction air rate from the indoor building environment.

Hence, the objectives of this study are as follows:

- to explore various design configurations of the wind-induced natural ventilation tower;
- to analyse the extraction air flow rates of each design configuration;
- to compare the performance of different design configurations and identify the optimum design of the wind-induced natural ventilation tower.

## 2 RESEARCH METHODOLOGY

The application of the computational fluid dynamics (CFD) technique in the field of natural ventilation research is gaining more popularity due to the availability of high-performance computer and improvement in turbulence modelling [15, 16]. According to Perino [17], once a numerical model is validated, it can be used to optimize design strategies to achieve satisfactory IAQ in building. This research utilizes commercial CFD software known as FloVENT (version 9.3) for simulation of different wind-induced natural ventilation tower design configurations. To validate the accuracy of the FloVENT software, a full-scale wind-induced natural ventilation tower was constructed for field measurement and its field data were used for validation. The wind-induced natural ventilation tower was constructed on top of an experimental building at Green Technology and Innovation Park, National University of Malaysia, Bangi, Selangor as shown in Figure 1. The site is located in Malaysia with Latitude 2.93537 North and Longitude 101.78183 East. Malaysia is situated at the equatorial region with day time temperature ranging from 29 to 34°C, relative humidity (RH) of 70%–90% throughout the year [18]. The mean monthly surface wind speed is generally very weak throughout the year, which varies from 1 to 1.5 m/s [19].

Figure 1. Wind-induced natural ventilation tower.
To model the site atmospheric boundary layer conditions for the purpose of the CFD simulation, an anemometer was installed at the roof top of the experimental building to record the onsite wind velocity and direction data. The height of the anemometer from the ground level is 11.4 m (see Figure 2). The onsite-recorded wind data were used in the Log Law Model equation to ascertain the velocity at a reference height ($V_{ref}$) of 10 m. After establishing the $V_{ref}$ at 10-m height, the $V_{ref}$ was inserted into the FloVENT Boundary Layer Generator (FBLG) to generate the atmospheric boundary layer. The computation details will be discussed at the subsequent part of this paper. The FBLG is available online at Mentor Graphics website [20].

The atmospheric boundary layer was modelled as a series of fixed flows at the domain boundary.

The research methodology framework is illustrated in Figure 3. After the validation for the FloVENT software, it was used to simulate the various design configurations of the wind-induced natural ventilation tower. The CFD simulation results of the various design configurations of the wind-induced natural ventilation towers were then compared and analysed. Finally, the optimum design configuration of the wind-induced natural ventilation tower was identified.

Figure 3. Research methodology framework.

Figure 4. Cross-section of the experimental house and wind-induced natural ventilation.
Figure 5. Locations of sensors of the data acquisition system.

Figure 6. Site wind rose diagram and wind speed classification.

Design configurations analysis
THE EXPERIMENTAL HOUSE WITH WIND-INDUCED NATURAL VENTILATION TOWER

The wind-induced natural ventilation tower was built on a concrete flat roof of the experimental house. The total volume space of the experimental house is 232.76 m³. The first floor of the experimental house was raised above the ground level at 3.2 m on four concrete pillars. A staircase was built to connect the ground floor to the first floor level. The raised floor concept enables free flow of air below the first floor. The house was orientated along the North–South axis with the front facade facing the southern direction. The height of the wind-induced natural ventilation tower is 2.81 m with a Venturi-shaped roof geometry of 5.56-m width by 5.20-m length. A cross-section of the experimental house is shown in Figure 4.

DATA ACQUISITION SYSTEM

Altogether 13 units of sensors were installed at 6 different locations in the experimental house and wind-induced natural ventilation tower. The six locations were at the bottom of the wind tower roof, wind tower windows, middle section of the wind tower, lower section of the wind tower, middle section and front facade windows of the experimental house. The sensor types and locations are shown in Figure 5. The parameters for the data acquisition were air velocity (m/s), pressure (Pa), ambient temperature (°C) and RH%. The data logger installed was of Graphtec GL800 with 20 channels. The pressure sensor was of Piezo-resistive sensitive element type with measuring range of \(-500\) to \(+500\) Pa and a resolution of 1 Pa. The air velocity sensors were of the hotwire type with measuring range of \(0–20\) m/s and with a resolution of 0.01 m/s. The temperature sensors were PT100 Class ‘A’ element with measuring range from

---

Table 1. Atmospheric boundary layer (ABL) characteristic for different terrain roughness [21]

<table>
<thead>
<tr>
<th>Class</th>
<th>Terrain description</th>
<th>Z₀ (m)</th>
<th>α</th>
<th>Iₜ (%)</th>
<th>Exp.</th>
<th>Zₖ (m)</th>
</tr>
</thead>
<tbody>
<tr>
<td>1</td>
<td>Open sea, fetch at least 5 km</td>
<td>0.0002</td>
<td>0.1</td>
<td>9.2</td>
<td>D</td>
<td>215</td>
</tr>
<tr>
<td>2</td>
<td>Mud flats, snow, no vegetation, no obstacles</td>
<td>0.005</td>
<td>0.13</td>
<td>13.2</td>
<td>D</td>
<td>215</td>
</tr>
<tr>
<td>3</td>
<td>Open flat terrain; grass, few isolated obstacles</td>
<td>0.03</td>
<td>0.15</td>
<td>17.2</td>
<td>C</td>
<td>275</td>
</tr>
<tr>
<td>4</td>
<td>Low crops; occasional large obstacles, (x'/h &gt; 20)</td>
<td>0.1</td>
<td>0.18</td>
<td>27.1</td>
<td>C</td>
<td>275</td>
</tr>
<tr>
<td>5</td>
<td>High crops; scattered obstacles, residential suburban, (15 &lt; x'/h &lt; 20)</td>
<td>0.25</td>
<td>0.22</td>
<td>27.1</td>
<td>B</td>
<td>370</td>
</tr>
<tr>
<td>6</td>
<td>Parkland, bushes; numerous obstacles, (x'/h \approx 10)</td>
<td>0.5</td>
<td>0.29</td>
<td>33.4</td>
<td>B</td>
<td>370</td>
</tr>
<tr>
<td>7</td>
<td>Regular large obstacles coverage (dense spacing of low buildings, forest)</td>
<td>1.0–2.0</td>
<td>0.33</td>
<td>43.4</td>
<td>A</td>
<td>460</td>
</tr>
<tr>
<td>8</td>
<td>City centre with high and low-rise buildings</td>
<td>≥2.0</td>
<td>0.40–0.67</td>
<td>—</td>
<td>A</td>
<td>460</td>
</tr>
</tbody>
</table>

\(x'\) is the distance between the obstacles and the subject, whereas \(h\) is the height of the obstacles.

Figure 7. Wind profile of the site.

3 THE EXPERIMENTAL HOUSE WITH WIND-INDUCED NATURAL VENTILATION TOWER

4 DATA ACQUISITION SYSTEM
0 to 50°C and with a resolution of 0.1°C. All the sensors were connected to the data logger through RS232 channels. The sensors were calibrated by KIMO instruments in France before site installation and commissioning. The calibration certificates for the sensors were delivered together with the sensors. All the data were logged every 10 min intervals and stored in an USB memory drive. The data were then retrieved every 2–3 weeks for analysis. The data collection duration was from October 2010 to January 2011. All the measurements were taken with windows opened at the front of the experimental house and the top of the wind-induced natural ventilation tower. Left- and right-side windows were closed enabling the air movement to flow freely from the front of the experimental house and towards the wind tower.

5 SITE WIND DATA ANALYSIS

Based on the wind rose generated from the site wind data recorded from November to January 2011, the prevailing wind was seen to be blowing from the northern direction as shown in Figure 6. The rear façade of the experimental house was facing the prevailing wind. The overall wind speed analysis indicated that 64.3% were classified as calm days (<0.5 m/s) and 34.3% have wind velocity range from 0.5 to 2.1 m/s. Meanwhile, 1.3% of the days have wind velocity between 2.1 and 3.6 m/s and only 0.1% of the days have wind velocity between 3.6 and 5.7 m/s. Generally, the wind data analysis revealed that the site has low outdoor wind velocity. The mean wind velocity recorded by the anemometer at the height of 11.4 m was 0.85 m/s.
The Log Law Model equation that was used to determine the mean wind velocity ($V_z$) at 11.4-m height is as follows:

$$V_z = V_{ref} \left[ \log \left( \frac{Z}{Z_o} \right) \right] \left[ \log \left( \frac{Z_{ref}}{Z_o} \right) \right] / C_{20} / C_{21}$$

(2)

where $V_z$ is the mean wind velocity at height $Z$ (gradient wind), $V_{ref} = 0.85$ m/s (mean wind velocity at reference height $Z_{ref}$ of the anemometer), $Z = 370$ m (height for which the wind velocity $V_z$ is computed) (gradient height), $Z_o = 0.5$ (the roughness length of log layer constant). Different class types of the site are presented in Table 1.

The site falls under ‘Class 6: terrain type of Parkland, bushes; numerous obstacles, $x'/h \sim 10$’ is used for the computation.

Using Equation (2),

$$V_{114} = 0.85 \left[ \log \left( \frac{370}{0.5} \right) \right] / \left[ \log \left( \frac{114}{0.5} \right) \right]$$

(3)

$$V_{114} = 0.85 \left( \frac{2.87}{1.36} \right)$$

(4)

$$V_{114} = 1.79 \text{ m/s}$$

(5)

Rearranging Equation (2), the mean wind velocity at 10-m height can be found to be as follows:

$$V_{ref} = \frac{V_z}{\left[ \log \left( \frac{z}{z_o} \right) / \log \left( \frac{z_{ref}}{z_o} \right) \right]}$$

(6)

$$V_{10} = \left( \frac{2.06}{\left[ \log \left( \frac{370}{0.5} \right) / \log \left( 10 / 0.5 \right) \right]} \right)$$

(7)

$$V_{10} = \left( \frac{1.79}{\left( \frac{2.87}{1.36} \right)} \right)$$

(8)

$$V_{10} = 0.81 \text{ m/s}$$

(9)

The FBLG which is the web-based software was used to generate the boundary layer conditions [20]. The wind velocity $V_{10} = 0.81 \text{ m/s}$ was inserted into the FBLG for the generation of the boundary layer conditions. After generation of the boundary layer conditions, the file in ‘pdml’ format was then exported into FloVENT version 9.3 for CFD simulation. Figure 7 shows the wind profile of the site.
Validation is defined as a process for assessing simulation modelling uncertainty by using a benchmark of experimental data [22]. An exact full-scaled experimental house was modelled in the FloVENT CFD software. FloVENT uses Cartesian-type grid for meshing of the model. The total number of cells used for the model was 239,904 with the maximum grid cell aspect ratio of 1.89. The turbulence model used for the simulation was the standard Reynolds-average Navier–Stokes (RANS) k-ε turbulence model with global system setting of 1 atm datum pressure. According to Awbi [23], the standard k-ε turbulence model is the most widely used and developed the turbulence model. This turbulence model is able to predict reasonable results for airflow studies in buildings. The external ambient and radiant temperature was set at 33°C. The calculated solar intensity was fixed at 665 W/m² and the cloudiness index at 0.3. The overall solution control was set using an outer iteration of 1000, and fan relaxation was set at 1.0. The simulation was run until it reached convergence. Figure 8 shows the FloVENT simulation results and the locations of sensors.

In Figure 9, the empirical mean air velocity is compared with the results of the FloVENT CFD simulation. The percentage of...
root-mean square deviation (PRMSD) was used to determine the accuracy of the simulation.

The PRMSD equation used is as follows:

$$\text{PRMSD} = \sqrt{\frac{\sum_{i=0}^{N} \left( \frac{C_i - C_{cfd}}{C_i} \times 100 \right)^2}{N}}$$

where \(C_i\) is the empirical value; \(C_{cfd}\) the CFD value; \(N\) the number of values used.

The PRMSD between FloVENT CFD simulation and the empirical results shows 6.7% deviation. The PRMSD indicates that the FloVENT CFD simulation produces good agreement with empirical results. Following the satisfactory validation of the FloVENT CFD simulation result, the CFD software was used to simulate the various design configurations of the wind-induced natural ventilation tower.

7 DIFFERENT DESIGN CONFIGURATIONS OF WIND-INDUCED NATURAL VENTILATION TOWER

Overall, evaluations were carried out for 15 different design configurations of the wind-induced natural ventilation tower. The design configurations were assessed for different tower heights, tower shapes, roof tilt angles and roof shapes. Table 2 gives the variations of the 15 different design configurations of the wind-induced natural ventilation tower.

The configuration no. 2 and 15 were modelled in conformity with the experimental wind-induced natural ventilation tower and it serves as a benchmark for the other design configurations. All the CFD simulations were conducted using the same boundary conditions. The boundary conditions set in the FloVENT CFD software were as follows:

- external wind speed at 0.8 m/s;
- solar radiation at 600 W/m²;
- cloudiness index at 0.3 and
- ambient and radiant temperature at 33°C.

For the present simulation, the RANS k-ε turbulence model with the total cell grid of 800,000 cells were used. The opening area for the wind-induced natural ventilation tower remains constant at 3.4 m \(\times\) 3.45 m equals to 11.73 m² as shown in Figure 10a and b.

8 TOWER HEIGHTS CONFIGURATIONS

The FloVENT CFD simulation was conducted on four tower height variations with an increment of 1-m height for each variation. The other roof components and window opening sizes in the model house remain constant. The tower height variations are 1.86, 2.86, 3.86 and 4.86 m. Figure 10a indicates the tower height variation adjustment.

Referring to Figure 11, the highest extraction flow rate of 13,555.19 m³/h is generated by the shortest tower height of 1.86 m. This can be explained by the Bernoulli equation (1) expressed as

\[\text{Figure 14. (a) Biconcave-shaped tower; (b) cross-section of biconcave-shaped tower; (c) cone-shaped tower; (d) cross-section of cone-shaped tower; (e) bottle-shaped tower; (f) cross-section of bottle-shaped tower; (g) trumpet-shaped tower; (h) cross-section of trumpet-shaped tower; (i) rectangular-shaped tower; (j) cross-section of rectangular-shaped tower.}\]
Figure 15. CFD simulation results of the various design configurations of the tower shapes.

Figure 16. (a) Scalar field of the 'bottle'-shaped tower; (b) vector field of the 'bottle'-shaped tower; (c) scalar field of the 'biconcave'-shaped tower; (d) vector field of the 'biconcave'-shaped tower.
a conservation of energy as shown in Figure 12. Thus, when the air flows upwards (against the gravitational force) the potential energy reduces due to a decrease in the height. Consequently, the sum of the kinetic energy and pressure energy increases resulting in a higher extraction flow rate.

Figure 13a and b shows the CFD simulation results of 1.86-m tower height and Figure 13c and d shows the CFD simulation results of 4.86-m tower height. The wind velocity vector field for 1.86-m tower height shows less turbulence inside the tower compared with the tower height of 4.86 m. The higher the tower, more vortexes and turbulence are formed and hence dampen the extraction air flow rate.

Figure 14 shows the detailed design of the shapes of the towers.

All CFD simulations were conducted with the same boundary conditions. The boundary conditions were set in the FloVENT CFD software. The boundary conditions set were as follows:

- external wind speed at 0.8 m/s;
- solar radiation at 600 W/m²;
- cloudiness index at 0.3 and
- ambient and radiant temperature at 33°C

All simulations were conducted with the RANS k-ε turbulence model with the total cell grid of 800,000 cells. The height of the tower from the flat-roof level to the bottom of the tower roof is fixed at 2.86 m.

Figure 15 shows that the ‘biconcave’-shaped tower has the highest extraction flow rate amounting to 14,568.66 m³/h. This is due to the constriction at the middle of the tower, which allows the air velocity to increase and thus decreasing the pressure. The constriction at the middle location of the ‘biconcave’ tower has a ‘Venturi’ effect. The ‘Venturi’ effect was the main driving force for the high air velocity and extraction air flow rate at the lower opening part of the tower.

Figure 16a–d shows the scalar and vector fields of the FloVENT simulation results for ‘bottle’-shaped tower (the lowest extraction air flow rate performance) and ‘biconcave’-shaped tower (the highest extraction air flow rate performance). Figure 16c scalar field reveals that the higher air speed ‘biconcave’-shaped tower was due to the Venturi effect caused by the constriction at the middle of the tower. Figure 16d vector fields also reveals no vortex and hence smoother air flow in the

9 TOWER SHAPE DESIGN CONFIGURATIONS

This research considers five types of tower shape configurations for analysis. The tower shapes are ‘biconcave’-shaped tower, ‘cone’-shaped tower, ‘bottle’-shaped tower, ‘Trumpet’-shaped tower and ‘Rectangular’-shaped tower. The tower heights and opening area below the tower were maintained constant at 2.86-m height with the opening size area of 11.73 m², respectively. Figure 14 shows the detailed design of the shapes of the towers.

All CFD simulations were conducted with the same boundary conditions. The boundary conditions were set in the FloVENT CFD software. The boundary conditions set were as follows:

- external wind speed at 0.8 m/s;
- solar radiation at 600 W/m²;
- cloudiness index at 0.3 and
- ambient and radiant temperature at 33°C

Figure 17. (a) Roof tilt angles: 0°, 20°, 30° and 45°; (b) view of 0°-tilted roof and tower.
'biconcave'-shaped tower compared with the 'bottle'-shaped tower (see Figure 16b). Although there is also a constriction at the end of the 'bottle'-shaped tower, the high air velocity exiting the tower in the vertical direction is immediately dampen by the external wind flow in the horizontal direction below the tower roof. This situation is shown in Figure 16b, vector field of the 'bottle'-shaped tower. The 'biconcave'-shaped tower scalar field in Figure 16c also reveals a higher indoor air velocity compared with the 'bottle'-shaped tower.

10 TOWER ROOF TILT ANGLES CONFIGURATIONS

Another factor that influences the extraction air flow rate of the wind-induced natural ventilation tower is the tower roof tilt angles. This study covers four different roof tilt angles of the wind-induced natural ventilation tower namely 0° (no tilting), 20°, 30° and 45° tilt angles. If the tower roof angle is titled higher than 45°, there is a tendency of rain penetration into the house. Figure 17a shows the diagram of four different roof tilting angles, and Figure 17b shows the three-dimensional view of the 0°-tilted roof. The CFD simulations were conducted with other parameters fixed namely, the height of the wind tower is 2.86 m the roof size is 5.20 m × 5.56 m and both the bottom opening size of the wind tower is 3.4 m × 3.45 m.

The CFD simulation results reveal that the 45°-tilted roof angle has the highest extraction air flow rate of 9 796.90 m³/h compared with the other tilted roof angles of 0°, 20° and 30°. These extraction air flow rates are presented in Figure 18. The tilting angle in airfoil design is also known as 'angle of attack'. According to the Bernoulli’s principle, as the ‘angle of attack’ of the roof increases, the air velocity below the roof increases and hence the pressure decreases. Consequently, a higher extraction air flow rate is generated below the wind-induced natural ventilation tower.

From Figure 18, the extraction air flow rate increases almost double from 0°- to 20°-tilted roof angle, whereas the increment becomes more gradual when the tilted roof angles are between

![Figure 19. (a) Scalar field of 0°-tilted roof angle; (b) vector field of 0°-tilted roof angle; (c) scalar field of 20°-tilted roof angle; (d) vector field of 20°-tilted roof angle; (e) scalar field of 30°-tilted roof angle; (f) vector field of 30°-tilted roof angle; (g) scalar field of 45°-tilted roof angle; (h) vector field of 45°-tilted roof angle.](https://academic.oup.com/ijlct/article-abstract/10/4/332/2363393)
20° and 45°. If the roof 'angle of attack' becomes too great, the air movement below the roof surface begin to push down the movement of upwards air from the tower and when this situation occurs, the extraction air flow rate of the wind tower is further reduced.

Figure 19a–h shows the scalar and vector fields of air flow for tilted roof angle of 0°, 20°, 30° and 45°. There is a possibility that angle higher than 45° will force the external air to flow into the wind tower. Figure 19h revealed that vector field of 45° roof tilt angle is able to give an indication of a back flow possibility. This phenomenon was also described by Blocken and Carmeliet [24] as 'wind-blocking effect' that refers to the disturbance of the wind-flow pattern by building component that decreases the wind velocity.

11 ROOF SHAPE VARIATIONS

Only two types of roof shapes, namely the rectangular roof (experimental house) and round-shaped roof, were discussed in this study. Figure 20a and b illustrates the three-dimensional view of the rectangular- and round-shaped roofs. Figure 21a and b shows that both roofs have a similar roof cross-section profile.

Figure 21a and b shows the vector field of the CFD simulation results of the rectangular- and round-shaped roofs. Although the round-shaped roof (Figure 21b) generates slightly a higher extraction air flow rate compared with the rectangular-shaped roof, the vector field for the rectangular-shaped roof (Figure 21a) shows less turbulence and smoother air flow inside the interior of the house.

Figure 22 shows the extraction air flow rate of the rectangular- and round-shaped roofs. The CFD simulation produces the extraction air flow rate of the rectangular-shaped roof to be 10 911.72 m³/h, whereas the round-shaped roof is 11 148.19 m³/h. The percentage of a marginal increase between the rectangular- and round-shaped roofs is only 2.17%. The only other advantage of the round-shaped roof is that it can accommodate external wind from different directions.
12 CONCLUSIONS

This study explored various design parameters that influence the extraction air flow rate of the wind-induced natural ventilation tower. The study covered four main parameters namely the tower height, tower shapes, tilting angles of the tower roof and shapes. In the tower height study, the CFD simulations showed the shortest tower height has a better extraction air flow rate. This is mainly due to the upwards air flow (against the gravitational force). If the height \( h \) decreases, which equals to a reduction in the potential energy \( \frac{1}{2} \rho gh \), then the sum of the kinetic energy and pressure energy, which is \( \frac{1}{2} \rho V^2 + P \), will increase. Hence, the extraction air flow rate increases. For the tower shapes study, the ‘biconcave’-shaped tower has the highest extraction air flow rate of 14,568.66 m\(^3\)/h. This design configuration outperforms all the other design configurations covered in this work. The reason for the high extraction air flow rate is mainly due to the Venturi effect, which is the result from reduction in pressure from the double constriction of both the tower and the roof of the ‘biconcave’-shaped tower configuration. For the tower roof tilting angles, the 45° tilt angle of the tower roof generated the highest extraction air flow rate of 9,796.90 m\(^3\)/h compared with other roof tilting angles. Based on the CFD simulation vector field diagram of the 45° roof tilt angle, the exiting air flow velocity from the tower is dampened by the external wind flow from the roof. Any higher roof tilt angles than 45° might generate opposite air flow direction in the tower due to the ‘wind-blocking effect’ of the tower roof inclination which was first discovered by researchers, Blocken and Carmeliet [24]. The extraction air flow rate increases more than double (4,433.94 to 9,796.90 m\(^3\)/h) from 0° to 45° tilt of the tower roof angle. As for the roof shapes, we focus on two types, namely the rectangular- and round-shaped roof. Although both roofs have different shapes, their cross-section profile is similar. Therefore,
the CFD simulation only showed a marginal difference in the extraction air flow rate of only 2.17%. The round-shaped tower roof produces an extraction air flow rate of 11 148.19 m³/h which is higher than the rectangular-shaped tower roof which is 10 911.72 m³/h. The main advantage of the round-shaped roof is that it is able to cater for different external wind directions compared with the rectangular-shaped roof. It is also important to note that the limitation of this study is it did not consider the influence and effects of surrounding buildings and wind directions which are recommended for future research.

ACKNOWLEDGEMENTS

The authors would like to thank the Ministry of Higher Education of Malaysia for the financial assistance under Fundamental Research Grant (FRGS) for this research project. Without this research would not have been possible.

REFERENCES