Wind-driven natural ventilation study for multi-storey residential building using CFD

M. F. Mohamed¹, ², S. King², M. Behnia³, D. Prasad² and J. Ling³

¹Universiti Kebangsaan Malaysia, Bangi, Malaysia
²The University of New South Wales, Sydney, Australia
³The University of Sydney, Australia

ABSTRACT: Computational Fluid Dynamics (CFD) has become an attractive alternative application tool to investigate outdoor and indoor airflows for buildings in comparison to other airflow prediction models. However, CFD comes with two main setbacks: it requires a large amount of computation and its results are often difficult to assess and require a validation process. In the case of a multi-storey residential building, it is always a limitation faced by researchers to simultaneously simulate both outdoor and indoor airflows of the building, due to its large scale and high complexity.

This is a preliminary study in which various CFD setups are investigated prior to subsequent investigation of wind-driven ventilation performance in a multi-storey residential building. The objectives of this study are: a. to simultaneously predict outdoor and indoor airflows using CFD; b. to investigate various CFD setups as an optimization process and c. to analyse the accuracy of the CFD results against existing wind tunnel data and empirical models as a validation process. This study found that CFD can provide reasonably accurate results for prediction of coupled outdoor and indoor airflows for a multi-storey building, but it comes with some limitations.

Keywords: CFD validation, ventilation performance, multi-storey building

INTRODUCTION

Residential buildings are responsible for approximately 12% of the energy use in Australia. For greenhouse gas emissions, residential buildings produce around 10% of the emissions. The energy use and greenhouse gas emissions are expected to increase as the number of new residential building grows, where it is predicted that the average energy used by each homes in Australia to grow more than 55% between 1990 and 2020 (Homes, 2010). Of the 12% energy consumption used in residential buildings, 38% is used for heating and cooling, making it the largest energy use in homes (Heating and cooling, 2010). The statistic shows that energy consumption for residential sector is associated with the need to provide thermal comfort for the occupants. One of the approaches to provide adequate indoor thermal comfort is by improving ventilation performance in buildings to avoid a need for mechanical cooling. While improving natural ventilation is important during the overheated season, limiting unnecessary ventilation during winter season is also vital to avoid energy wastage.

During the earliest stage in a building design process, the ability to predict indoor airflow for healthy indoor environmental quality is crucial to guarantee a proposed building design can optimize all implemented passive design strategies: for example, to predict that adequate fresh air is provided for indoor air quality, and to design indoor airflow pattern so that higher air velocity is provided for physiological cooling in parts of the space where occupants may be more active. The prediction models should be able to predict at a reasonable accuracy, and with the ability to capture changes made in the building design and, at the same time, flexible so as to be able to predict various design alternatives. There are various airflow prediction models which can be adopted for this purpose. Among all available models, computational fluid dynamics (CFD) is an attractive alternative in comparison to other models such as small scale physical models (wind tunnel experiment) and empirical models. This is largely because CFD is able to provide comprehensive information through post-processing for airflow pattern, air velocity and ventilation rates. CFD models have been increasingly used in building design; however, CFD comes with two main setbacks which are: it requires large amount of computation and its results are often difficult to assess, and require validation processes. In a large complex geometry building, such considerations always hinder the use of CFD, especially when it involves coupled indoor and outdoor airflows simulation.

It is not common to find coupled simulation of outdoor and indoor airflows in large scale buildings due to the very great computational effort which results from the need to simulate the huge outdoor environment necessary to correctly suggest a realistic atmospheric boundary layer (ABL) wind profile, as well as a need to simulate openings and the indoor environment which require a very small meshes. An example of such simulation, but for a small building, is a simple outdoor and indoor airflow study by Yang (2004), in which a single 6m cube was simulated with a very fine meshing. Recently, a research by van Hooff and Blocken (2010) has attempted to simulate outdoor and
indoor airflows in a much larger building configuration and urban domain with various buildings in its proximity. These studies suggest that with aggressive CFD software development, together with greater affordability of better computer facilities, in the near future, CFD simulation of coupled outdoor and indoor airflows should be a common approach to study ventilation performance for buildings. The main advantage of the approach is that it gives much more reliable outdoor and indoor airflows data, in comparison to a simplified approach which, in an attempt to reduce computational effort, simulates outdoor and indoor airflows separately. Therefore, the objectives of this paper are:

- to investigate various CFD setups as an optimization process and;
- to analyse the accuracy of the CFD results against existing wind tunnel data and empirical models as a validation process.

1. METHODOLOGY

1.1. Research methodology

This study uses CFD to create a numerical simulation for outdoor and indoor airflows for a building; however, due to uncertainty of the CFD outcomes, existing wind tunnel experiment data as well as empirical models are used for validation. For the purpose of this study, existing data from two completed wind tunnel tests are used. They are full-scale wind tunnel experiment completed by Larsen (2006), and scaled wind tunnel experiment by Ismail (1996).

The validation process starts with the work by Larsen (Simulation 01) to investigate pressure distribution and indoor ventilation rate for a single room building (Figure 1). Various CFD simulation setups are investigated and tested; and an optimised and reasonably accurate setup are selected to suit on an investigation of a multi-storey residential building. Then, the CFD setup obtained from Simulation 01 is tested on the work by Ismail (Simulation 02), where the accuracy of CFD in simulating coupled outdoor and indoor airflows for multi-storey building is examined. After that, the results from CFD simulations are compared against results obtained from combination of the wind tunnel experiment and empirical models. The number of element in CFD meshing is used as limitation for this study, where the limit used to simulate 36 unit rooms with balconies is to be around 24 million elements.

![Figure 1: The process adopted which consists of Simulation 01 and Simulation 02.](image)

1.2. Computational Fluid Dynamics (CFD)

CFD can be divided into three categories, depending on how it solves the turbulent flows. They are: 1. Direct numerical simulation (DNS) 2. Large Eddy Simulation (LES), and 3. The Reynolds Averaged Navier-Stokes equations with turbulence models (RANS) (Malkawi and Augenbroe, 2003). Direct numerical simulation requires the largest computer capacity and computing time, followed by LES, leaving RANS to be the most reasonable in term of requirements on computer capacity and computing time. This study only uses a single RANS model which is the standard k-ε turbulence model. The model is selected due to its robustness, though it may result in less accuracy in predicting vortex shedding. The meshing type used for this study is limited to tetrahedron meshing due to its flexibility and ability to mesh complicated building configurations which include various protrusion and opening configurations. This study is also limited to steady-state simulation with the minimum blend factor of 0.75 and convergence residual target value of 0.0001. This study uses commercial code CFD software (Ansys CFX 12.0). All the simulations are performed under isothermal conditions and buoyancy-driven ventilation is not taken into consideration.

1.3. Empirical Models

\[
\begin{align*}
Q_1 &= C_v \cdot A \cdot \frac{V}{2} \\
Q_2 &= C_d \cdot A \cdot \frac{(2 \Delta P/p)^{1/2}}{2} \\
Q_3 &= C_d \cdot (V^2 - \Delta C_{p2})^{1/2} \\
Q_4 &= 0.5 \cdot A \cdot V - 0.5 \cdot A \cdot (0.001 \cdot V^2 + 0.01)^{1/2} \\
Q_5 &= A \cdot (C_n, V^2, C_{dol})^{1/2} 
\end{align*}
\]

Where,\n\[
\begin{align*}
Q_1 &= \text{Ventilation rate, (m}^3\text{/s)}; \\
Q_2 &= \text{Ventilation rate (specifically for single-sided ventilation strategy), (m}^3\text{/s)}; \\
C_v &= \text{Opening effectiveness, (dimensionless)}; \\
A &= \text{Free area of an opening for single-sided ventilation and inlet opening for cross ventilation, (m}^2\text{)}; \\
V &= \text{Outdoor reference free wind speed, (m/s)}; \\
C_d &= \text{Discharge coefficient of opening, (dimensionless)}; \\
\end{align*}
\]
\[
\Delta P = \text{Pressure difference between inlet and outlet, (N/m}^2)\\
\rho = \text{Air density, (kg/m}^3)\\
\Delta C_{p1} = \text{Pressure coefficient difference (dimensionless)}\\
V_o = \text{Effective air velocity, (m/s)}\\
C_n = \text{Constant values of 0.0012 (windward), 0.0026 (leeward) and 0.0012 (parallel), (dimensionless)}\\
C_{o2} = \text{Pressure coefficient, (dimensionless)}\\
\]

Eqn. 01, Eqn. 02, Eqn. 03, Eqn. 04 and Eqn. 05, as listed above, can be used to predict ventilation rate for wind driven ventilation. Eqn. 01 is the most simplified equation, while Eqn. 02 requires values for pressure on the wall of a closed building. For cross ventilation strategy, ASHRAE (1997) indicates that the \(C_n\) values are 0.5 to 0.7 for perpendicular winds; and 0.25 to 0.35 for diagonal winds. Larsen (2006) found the values of \(C_n\) to be around 0.5 for perpendicular flow and around 0.48 for a diagonal wind. This is based on a completed cross ventilation study with small opening dimensions of 0.15m x 0.68m with porosity of 0.95%. According to Givoni (1998), the value for \(C_o\) is 0.7 for perpendicular wind direction with a direct cross ventilation and without any obstruction along airflow path. While, according to Aynsley et al (1977), if the opening size is less than 10% of the area of facade, the \(C_o\) value is between 0.50 to 0.65.

In the case of single-sided ventilation strategy, the value for \(C_n\) is 0.025 (Awbi, 1991). Eqn. 05 is derived by de Gibbs and Phaff (1982) measurements taken on the first floor of existing buildings of up to 4 floors high. Larsen (2006) derived a new equation for single-sided ventilation which includes wind direction as in Eqn. 04. The values for pressure coefficient used by Larsen for the equation is taken from Liddament (1996), where the values are 0.7 (windward, \(0^\circ\)), 0.2 (leeward, \(0^\circ\)), 0.35 (windward, \(45^\circ\)) and -0.4 (leeward, \(45^\circ\)).

Based on the above equations, it can be observed that empirical models provide a large range of prediction values. This could be due to the investigation approaches, such as differences in external airflow characteristics, as well as building and opening configurations. Therefore, these models are limited to certain building configurations only, and thus may result in great inaccuracy if used inappropriately. Based on completed comprehensive literature review of empirical models, it also can be concluded that no equations are available specifically to predict ventilation rates for single-sided ventilated multi-storey buildings, except Eqn. 02 and Eqn. 03, which are based on pressure difference. But this may lead to a misleading result if wind pressure is found to be similar, or if the pressure difference is too small. This paper will not discuss any further the accuracy of CFD prediction of single-sided ventilation, but does include CFD prediction of ventilation rates for single-sided conditions for comparison to cross ventilation. Further investigation of this ventilation strategy will be completed in a later study.

2. SIMULATION 01

Larsen (2006) completed a full-scale closed system wind tunnel experiment at the Japanese Building Research Institute (BRI), Tsukuba, Japan. The dimensions of the tested building model are 5.56m (width) x 3.00m (height) x 5.56m (depth). The thickness of the wall is 0.1m and the floor to ceiling height is 2.4m in which the indoor floor slab is raised by 0.5m. Various opening configurations, wind speeds and wind directions were tested by Larsen.

For this study, three wind directions are selected; they are \(0^\circ\), \(30^\circ\) (anti-clockwise), and \(180^\circ\). The selected wind speeds are 3 and 5m/s. For single-sided ventilation strategy, the building configuration is similar to Larsen’s model which is using a single opening with dimension of 0.86m (width) x 1.40m (height). For \(0^\circ\) wind angle, the opening is positioned at 0.54m distance from the right edge of windward wall, and 0.91m above the ground. For other wind angles, the model is rotated anticlockwise. In the case of cross ventilation strategy, Larsen used a much smaller size of opening in her investigation, thus it is not adopted in this study. Therefore, for the cross ventilation strategy, a similar size of opening as in single-sided ventilation strategy is maintained, but an inlet opening is located at the middle of windward façade and an outlet opening is located on the opposite (leeward) façade. The distance of opening from the ground is maintained. Since a study using different building configuration for cross ventilation strategy and ventilation prediction from wind tunnel experiment is not available, results obtained from empirical models are used for validation.

2.1. Simulation Setup

The main purpose of Simulation 01 is a mesh independence study based on pressure distribution of building facades and prediction of the ventilation rate. The CFD domain setting up for wind inlet and the domain cross section is similar to the wind tunnel; however, the downstream is created longer than the setting up in the wind tunnel to allow for the downstream airflow to be sufficiently developed (Figure 2).
Simulation 01 is divided into two (2) stages. The first stage (Stage 01) is simulation of external airflow; whereas the second stage (Stage 02) is the coupled outdoor and indoor airflow simulation. The purpose of Stage 01 is to establish through a grid independence study a sufficient meshing required for external airflow. Based on the grid achieved in Stage 01, the model is further developed in Stage 02 to include the indoor space together with an opening. The second grid independence study is completed in Stage 02 to obtain a suitable meshing for coupled outdoor and indoor flow. Two ventilation strategies are used in Stage 02: single-sided and cross ventilation. Firstly, simulations for single-sided ventilation strategy are run, and the results are compared with the results obtained from Larsen’s experiment. This is followed by simulations for the cross ventilation strategy with similar opening size but with 2 openings. Since the prediction of the cross ventilation strategy is validated with empirical models, the domain setup is changed to ensure that the blockage ratio is less than the maximum blockage recommended by others. Franke et al. (2004) recommend maximum 3%, while Kurotani and Sekine (1990) suggest a maximum of 3.57%, but with a preference to 1%. In this study, the blockage ratio adopted is 1.8% to avoid the effect of compressed flow.

2.2. Discussion and Findings
In Stage 01, four meshing setups are tested for 0°, 30° and 180° wind directions. The numbers of tetrahedron element for the simulations with 0° wind angle are around 210000 (1A), 350000 (1B), 680000 (1C) and 1410000 (1D). Similar meshing setup are applied to 30° and 180° wind angles, therefore the numbers of elements are almost similar. In this stage, the values for wind pressure on walls are compared. The values are taken at the middle of the location where an opening will be located at the later stage. For 0° wind angle, it is found that the pressure difference between mesh 1A and 1D is 4%, and the difference between mesh 1B and 1D is 1.3%, while, the difference between 1C and 1D is only 0.8%. In the case of 30° wind angle, the maximum pressure difference found is 3.3%. For wind angle of 180°, the largest pressure difference found is 2.9%. Thus, it can be concluded that the meshes used for this study are sufficiently grid independent since the results obtained from Mesh 1A and 1D show a very small changes in values. Therefore, it can be assumed that number of element as low as 210000 is sufficient to predict external airflow, unless greater accuracy is required.

In the second stage, the mesh setups in Stage 01 are combined with additional meshing at opening and indoor space. The numbers of tetrahedral element for simulations of 0° wind direction are 290000 (1E), 450000 (1F), 1350000 (1G) and 3530000 (1H) with the sizes of cell edge used at opening are 0.05m, 0.05m, 0.02m and 0.01m, respectively. Similar mesh setups are used for all wind angles. The final stage in Simulation 01 is to adopt mesh 1E for the cross ventilation strategy simulation.
For single-sided ventilation strategy with wind speed of 3m/s (see Figure 3), the airflow rates for wind angle of 0° for mesh 1E, 1F, 1G, and 1H are 0.095m³/s, 0.093m³/s, 0.100m³/s and 0.103m³/s, respectively. The result found in Larsen’s experiment is 0.124m³/s. Therefore, the simulation results under-predict by approx. 17% in comparison to data by Larsen, and over predict around 25% the results calculated using Eqn. 05 (0.083m³/s). However, the result agrees well with Eqn. 04 (0.105m³/s). In the case of 30° wind angle, the ventilations rates results range from 0.121m³/s to 0.127m³/s, respectively. The simulations are found to over-predict the ventilation rate by around 6%, in comparison to 0.117m³/s as found by Larsen. For the wind angle of 180°, Larsen’s experiment result is 0.103m³/s. The simulation airflow rate is far under-predicted, with inaccuracies up to 66%, where ventilation rates are predicted to range from 0.035m³/s to 0.045m³/s, with the coarsest mesh providing the worst result. The inaccuracies for the wind angle of 180° are expected, being due to limitations of the RANS standard k-ε turbulence model (Yim et al., 2009, Yoshie et al., 2007) to accurately predict airflows at areas with recirculation such as on leeward side of a building.

In the case of varying air velocities, CFD is able to predict changes in ventilation rate due to changes in outdoor airspeed at acceptable values. For 0° wind angle, Larsen found that outdoor air velocity increment from 3m/s to 5m/s results in 67% increase in ventilation rate, while CFD (mesh 1D) predicts increment of 68%, which is very close Larsen’s finding. For 180° wind angle, even though CFD under predicts the overall ventilation rate, it still can capture a change in ventilation rate due to increased wind speed but with slight over prediction, where it predicts 69% increment while experiment data shows 48% increment.

<table>
<thead>
<tr>
<th>Case</th>
<th>Wind angle (°)</th>
<th>Wind Speed (m/s)</th>
<th>Total Opening area (m²)</th>
<th>Cε values</th>
<th>Ventilation Rate (m³/s) based on Eqn. 01 and 02</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>6</td>
<td>3</td>
<td>1.2</td>
<td>0.50 - 0.70</td>
<td>1.71 – 2.53</td>
</tr>
<tr>
<td>2.</td>
<td>7</td>
<td>0</td>
<td>5</td>
<td>0.50 - 0.70</td>
<td>2.86 - 4.21</td>
</tr>
<tr>
<td>3.</td>
<td>8</td>
<td>45</td>
<td>3</td>
<td>0.25 - 0.48</td>
<td>0.90 - 1.50</td>
</tr>
<tr>
<td>4.</td>
<td>9</td>
<td>45</td>
<td>5</td>
<td>0.25 - 0.48</td>
<td>1.50 - 2.50</td>
</tr>
<tr>
<td>5.</td>
<td>10</td>
<td>45</td>
<td>5</td>
<td>0.25 - 0.48</td>
<td>3.01 – 5.00</td>
</tr>
<tr>
<td>CFDb</td>
<td>Wind angle (°)</td>
<td>Wind Speed (m/s)</td>
<td>Total Opening area (m²)</td>
<td>Mesh Setup</td>
<td></td>
</tr>
<tr>
<td>6.</td>
<td>11</td>
<td>0</td>
<td>3</td>
<td>1.2</td>
<td>1E</td>
</tr>
<tr>
<td>7.</td>
<td>12</td>
<td>0</td>
<td>5</td>
<td>1.2</td>
<td>1E</td>
</tr>
<tr>
<td>8.</td>
<td>13</td>
<td>45</td>
<td>3</td>
<td>1.2</td>
<td>1E</td>
</tr>
<tr>
<td>9.</td>
<td>14</td>
<td>45</td>
<td>5</td>
<td>1.2</td>
<td>1E</td>
</tr>
<tr>
<td>10.</td>
<td>15</td>
<td>45</td>
<td>5</td>
<td>2.4</td>
<td>1E</td>
</tr>
</tbody>
</table>

Table 1: Prediction of ventilation rate for cross ventilation strategy by empirical models and CFD simulations with setup mesh 1E.

For the cross ventilation strategy, CFD predictions are within the range of predictions as calculated using empirical models. Changes in ventilation rates due to increased wind speed predicted by CFD are also almost similar to results calculated using empirical models. The same applies to change in opening size, which is tested in Case 10 and Case 15.

From Simulation 01, it can be concluded that mesh 1E is suitable to be used as the guideline for the mesh setup in Simulation 02. It is important to note that comparison in term of ventilation rate between various wind angles is not possible using CFD since the accuracy of CFD simulation varies depending on wind angle, where the worst prediction is for an opening located in the leeward wall under single-sided ventilation strategy. Investigation in Simulation 01 also shows that CFD able to acceptably predict changes in ventilation rate due to changes in outdoor air velocity and opening size.

---

44th Annual Conference of the Architectural Science Association, ANZAScA 2010, Unitec Institute of Technology
3. SIMULATION 02

3.1. Simulation Setup

Figure 5: 12-storey building block with openings and 1.5m deep balconies (a), and atmospheric boundary layer (ABL) wind profile with mean speed exponent of 0.28(b). Number 1, 2 and 3 indicate the units which are investigated for ventilation rate.

The second wind tunnel experiment data is obtained from investigation by Ismail, using an open circuit wind tunnel facility. The model’s scale factor used by Ismail is 1:200. The real size building configuration is of 12-storey height with 4.0m floor to floor height in which to resemble apartment building with 36 units. The overall dimension of the building is 30m (width) x 50m (height) x 10m (depth), making the floor area of each unit 100m². Two model configurations are selected: a model with flat facade and a model with 1.5m deep balcony with both horizontal and vertical protrusions (Figure 5). In the wind tunnel experiment conducted by Ismail, a total of 72 pressure taps are located on the middle of each windward and leeward unit’s facade. For the purpose of this study, only one wind direction is selected, being normal to the windward facade (0°).

A building model with 1.5m deep balconies and ABL wind speed profile is shown in Figure 5. The domain setup for the simulation is similar to the wind tunnel setup, where the blockage ratio is of 1.875%. The distance of inlet from the facade of the building is 60m and the downstream length chosen for this study is 10 times the building height to allow the airflow behind the building to adequately develop. The atmospheric boundary layer (ABL) wind speed attempted in the experiment is 0.28 mean speed exponent of power law, which is between urban and suburban terrains.

The validation of the CFD results is divided into two stages. The first stage is on the external airflow only, while the second stage is again for coupled outdoor and indoor airflows. For the first stage, mesh 1A is adopted with some improvement on meshing near the ground. The wind pressure distribution data obtained from the wind tunnel experiment are compared with the predicted data by CFD simulations to ensure that the predicted data are within acceptable accuracy. In this stage, the flat façade as well as the façade with protrusions are tested to determine whether the CFD is able to predict changes in outdoor airflow due to the provision of protrusions at balconies. The wind pressure data obtained from the CFD simulations is also used for calculation of ventilation rate using empirical models.

After the CFD simulations for external airflow are validated, the second stage of Simulation 02 proceeds. In this stage, the validated setting up for external airflow is developed to include the indoor spaces, thus simultaneously predicting the outdoor and indoor airflows. Mesh 1E setup is adopted with maximum number of elements around 24x10⁶. For the cross ventilation strategy, two openings are provided at two opposite external walls (front and rear) at each unit. The dimension of each opening is 2.2m (width) x 1.15m (height), therefore the ratio of the openings to area of facade is 6.325%, except for Case 17 and 20 (see Table 3). The area of each of the openings is decided to be 2.53m² for two reasons: a. To ensure the ratio of the openings to wall to be less than 20% so that the porosity effect is insignificant, b. To obtain opening area approximately 5% of floor area as required by local authority (Australian Building Codes Board, 2009). In the case of single-sided ventilation, the second opening on the leeward side is removed, therefore only a windward opening exists. However, the opening is widened to double its size with the intention that the total opening area is similar to cross ventilation.

CFD predictions of ventilation rates for cross ventilation strategy are validated using an empirical model as in Eq. 2. It is expected that greater discrepancies exist between the prediction of the empirical model and CFD simulations for model with protrusions since none of the empirical models are derived from analysis which includes protrusion elements. Various factors could contribute to explain such discrepancies: the existence of small eddies as a result of wind incidence with balconies on the building’s facades; the balconies could act as wind scoops directing the incoming air into openings; and the balconies could become buffer spaces protecting the indoor environment from direct influence of prevailing wind (Mohamed et al., 2009). Therefore, in the case of the building model with protrusions, an assumption is made that the CFD predictions are acceptably accurate given that accurate results are achieved in all other cases.
3.2. Analysis and Findings

Table 2: Comparison of wind pressure values between wind tunnel (WT) experiment data and CFD results. The values are taken at tapping points at unit 1, 2 and 3.

<table>
<thead>
<tr>
<th>Unit</th>
<th>Flat Facades</th>
<th>Facades with Balconies</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Wind pressure, N/m$^2$</td>
<td>Wind pressure, N/m$^2$</td>
</tr>
<tr>
<td></td>
<td>WT</td>
<td>CFD</td>
</tr>
<tr>
<td>3</td>
<td>1.54</td>
<td>1.59</td>
</tr>
<tr>
<td>2</td>
<td>1.39</td>
<td>1.46</td>
</tr>
<tr>
<td>1</td>
<td>1.22</td>
<td>1.16</td>
</tr>
</tbody>
</table>

Table 2 shows the results of first stage of Simulation 02. It shows that CFD acceptably predicts the pressure distribution on the facades of the buildings, except that on the bottom leeward sides. It can be observed that the worst inaccuracy on the windward side is around 10% in comparison to the experimental data. The table also indicates that the worst prediction is at the bottom leeward side of the model, where the difference is over 70%. These results are expected, since the standard k-ε turbulence model unable to accurately predict the wake region behind bluff body (Yim et al., 2009, Yoshie et al., 2007). Therefore, it can be concluded that if CFD prediction of wind pressure data is to be used together with empirical models to predict cross ventilation strategy, it will under predict ventilation rate for units located at the bottom of the building.

Table 3 (below) shows the results of the second stage of Simulation 02. It indicates that CFD predictions for ventilation rate (cross ventilation strategy) of coupled indoor and outdoor airflows provide a good agreement with predictions gained from combination of wind tunnel data and empirical model, except at Unit 1 which is located at the bottom of the building. This is expected since the CFD prediction of pressure distributions also shows inaccurate prediction at the bottom of the model, especially on the leeward side. However, the combination of CFD predictions of wind pressure together with empirical models provides better agreement with wind tunnel experiment in comparison to CFD prediction of coupled indoor and outdoor airflows.

Table 3: Graph showing predicted ventilation rate using CFD and wind tunnel experiment (WT) for single-sided (SSV) and cross ventilation (CV) strategies.

<table>
<thead>
<tr>
<th>Case</th>
<th>Methods</th>
<th>Ventilation Strategy</th>
<th>Total opening area (m$^2$)</th>
<th>Unit 1</th>
<th>Unit 2</th>
<th>Unit 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>1.</td>
<td>16</td>
<td>WT + Empirical (C$_d$ = 0.5)</td>
<td>CV</td>
<td>5.06</td>
<td>2.30</td>
<td>2.46</td>
</tr>
<tr>
<td>2.</td>
<td>17</td>
<td>WT + Empirical (C$_d$ = 0.5)</td>
<td>CV</td>
<td>7.60</td>
<td>3.45</td>
<td>3.68</td>
</tr>
<tr>
<td>3.</td>
<td>18</td>
<td>CFD + Empirical (C$_d$ = 0.5)</td>
<td>CV</td>
<td>5.06</td>
<td>1.92</td>
<td>2.34</td>
</tr>
<tr>
<td>4.</td>
<td>19</td>
<td>CFD only</td>
<td>CV</td>
<td>5.06</td>
<td>1.69</td>
<td>2.14</td>
</tr>
<tr>
<td>5.</td>
<td>20</td>
<td>CFD only</td>
<td>CV</td>
<td>7.60</td>
<td>2.54</td>
<td>3.16</td>
</tr>
<tr>
<td>6.</td>
<td>21</td>
<td>CFD only</td>
<td>SSV</td>
<td>5.06</td>
<td>0.082</td>
<td>0.041</td>
</tr>
</tbody>
</table>

<table>
<thead>
<tr>
<th>Case</th>
<th>Methods</th>
<th>Ventilation Strategy</th>
<th>Total opening area (m$^2$)</th>
<th>Unit 1</th>
<th>Unit 2</th>
<th>Unit 3</th>
</tr>
</thead>
<tbody>
<tr>
<td>7.</td>
<td>22</td>
<td>WT + Empirical (C$_d$ = 0.5)</td>
<td>CV</td>
<td>5.06</td>
<td>2.21</td>
<td>2.32</td>
</tr>
<tr>
<td>8.</td>
<td>23</td>
<td>CFD + Empirical (C$_d$ = 0.5)</td>
<td>CV</td>
<td>5.06</td>
<td>1.94</td>
<td>2.36</td>
</tr>
<tr>
<td>9.</td>
<td>24</td>
<td>CFD only</td>
<td>CV</td>
<td>5.06</td>
<td>1.79</td>
<td>2.40</td>
</tr>
<tr>
<td>10.</td>
<td>25</td>
<td>CFD only</td>
<td>SSV</td>
<td>5.06</td>
<td>0.044</td>
<td>0.036</td>
</tr>
</tbody>
</table>

Figure 6: Middle sections and plans (2.0m above floor level) for unit 3 showing airflow patterns for models with (a) and without (b) balcony, at wind angle of 0°. Large white arrows show wind direction.
It is also found that CFD shows improvement of the ventilation rate with introduction of balconies, while the result from the wind tunnel experiment suggest that introduction of balconies result in reduced ventilation performance. This could be because of the calculation of indoor air flow for wind tunnel predictions are limited to a single wind pressure reading which is in the middle of the facade and away from the balcony, and thus unable to capture the changes in outdoor airflow due to wind incidence with protrusion elements of balconies, which CFD does. For a cross ventilation strategy, the balcony appears to be able to improve indoor ventilation performance, where the balcony acts as a wind scoop directing outdoor air into indoor spaces. CFD also acceptably predicts changes in indoor airflow patterns resulting from the provision of balconies, as can be observed in Figure 6. This is expected as the flat facade results in the direct influence of the overall outdoor airflow on indoor airflow patterns, while balconies act as an architectural element which changes the outdoor airflow characteristic close to the openings.

For a single-sided ventilation strategy, introduction of balconies reduces the ventilation performance for Unit 1 and Unit 2; however, it improves the ventilation performance of Unit 3. These could be due to balconies acting as a buffer protecting direct penetrations of prevailing wind where in the centre of the facade the wind remains near normal (Unit 1 and Unit 2), while with larger yaw higher on the facade, the existence of airflow circulation and small eddies within the balcony improve ventilation rates for the single-sided ventilation strategy (Unit 3). It is also found that, for the single-sided ventilation strategy with provision of balconies, the ventilation rates predicted by CFD are not portraying the increase of wind velocity due to ABL wind profile as in cross ventilation strategy. This can be observed in Case 25, where ventilation rates predicted for Unit 1, Unit 2 and Unit 3 are almost similar. The findings on single-sided ventilation prediction will not be further investigated in this study; however, it will be completed in later study.

CONCLUSION

Based on the investigation, the followings important conclusions on CFD are drawn:

a. Mesh independence study is an important part of CFD to ensure that accurate predictions are achieved. In the case of simulating coupled outdoor and indoor airflows of a tall building with design complexity, a validation process is crucial to avoid incorrect prediction, especially if computational power is a limitation.

b. CFD coupled indoor and outdoor airflows simulation can predict ventilation rate at acceptable accuracy, and is able to capture changes in wind speed and opening size. However, prediction of ventilation rate at various wind angles should not be compared to each other in term of its absolute values, especially for any single-sided ventilation strategy with openings on the leeward side. This is due to CFD with standard k-ε turbulence model being unable to correctly predict vortex shedding.

c. For a tall building with cross ventilation strategy, CFD coupled outdoor and indoor airflows simulation provides acceptable predictions of ventilation rate, except for units located at the bottom of the building.

d. If a study is only concerned with ventilation rate but not with indoor airflow pattern, for a simple shape building it would be sufficient to use CFD to only predict wind pressure data on the façade of the building and with the help from empirical models, ventilation rate can be calculated. However, in the case of a building with a complex façade treatment, it is important to apply coupled outdoor and indoor airflow simulation since pressure data is unable to capture local changes in wind direction and air turbulence.

e. CFD is an appropriate and reliable method of ventilation performance prediction for multi-storey buildings, where it able to predict ventilation rate and capture changes on indoor airflow pattern. Thus, CFD should be used by designers at the earliest stage of design to optimise ventilation performance, such as to design for indoor airflow pattern based on activities within apartments.

ACKNOWLEDGEMENT

The authors would like to thank Dr. T. Larsen and Dr. A. M. Ismail for providing us with information on their wind tunnel experiments.

REFERENCES


Websites
