Airflow patterns and pressure distributions for a cross-ventilated detached house analyzed by wind tunnel tests and CFD analysis

Tomohiro Kobayashi¹,³, Mats Sandberg², Hisashi Kotani¹, Leif Claesson², Toshio Yamanaka¹, and Kazunobu Sagara¹

¹Osaka University, Japan
²KTH Research School, University of Gävle, Sweden
³Japan Society for Promotion of Science, Japan

Corresponding email: kobayashi_tomohiro@arch.eng.osaka-u.ac.jp

ABSTRACT

Sealed building wind pressure coefficients and discharge coefficients of openings are generally used to predict building ventilation airflow rates. For the flow through large openings, however, it is well known that this method underestimates the flow rate. The ultimate goal of this work is to establish an improved prediction method based on energy balance inside a selected stream tube passing through/around the building. In this paper, wind tunnel tests and CFD analyses are conducted by using a simple-shaped detached house with a purpose to study the accuracy of the CFD simulation. Here three cases are studied varying the opening size. Finally, measured and simulated ground pressure distribution and airflow pattern are compared.

INTRODUCTION

With the aim to conserve nonrenewable energy reserves, the use of the renewable energy sources to control the building environment has been attracting practical and academic attention. In Japan, traditionally, wind-induced cross-ventilation has been a beneficial method to moderate the hot and humid environment in summer. In designing a building to be cross-ventilated by wind, the flow rate of a room must be known in the planning phase. In predicting the flow rate of the building, the orifice equation is generally used as:

\[ Q = (C_D A)_{\text{eff}} \sqrt{\frac{2}{\rho} (P_W - P_L)} \]  

(1)

where, \((C_D A)_{\text{eff}}\) is regarded as the connected value of the effective opening area of the cross-ventilation airflow path (e.g., \((C_D A)_{\text{eff}} = 1/(\sqrt{(1/C_{D1} A_1)^2 + (1/C_{D2} A_2)^2})\) for a path having two openings in series, where \(C_D\) is discharge coefficient and \(A\) is opening area for each opening). Here, the discharge coefficient is obtained from the chamber method [1]. \(P_W\) and \(P_L\) are the wind pressures on windward and leeward side of the building respectively, which are traditionally obtained from wind tunnel tests using a sealed building model. Yet it is well known that the predicted flow rate based on this equation could be much lower than the actual flow rate. This traditional method includes two implicit assumptions; (1) the resistance to airflow through the building is equal to the sum of the resistances through each opening, and (2) the dynamic pressure behind the leeward opening is always dissipated. The authors [2] have shown that both assumptions are not adequate when the openings are large. Many researchers have been working on this problem in recent years. For example, Ishihara [3]
explained this problem as *interference* of openings. He showed the resistance coefficients of openings could not be simply summed up and proposed the parameter named interference coefficient. Kurabuchi and Ohba et al. [4] suggested that the discharge coefficient be determined based on the *local dynamic similarity* at the opening using a *dimensionless internal pressure* parameter (Local Dynamic Similarity Model). They showed that this model could work well when the wind direction is not perpendicular to the opening [5]. Kotani and Yamanaka [6][7] also proposed a similar prediction method based on the composition of the vectors of normal component to the opening and parallel components to the wall of the sealed building to predict inflow direction. These improved prediction models modify the discharge coefficient but continue to use the sealed building wind pressure coefficients. However, it is questionable that the total pressure behind the building becomes the same as the wind pressure of the sealed building. Murakami, Kato, Akabayashi et al. [8][9] proposed the flow rate be predicted based on the balance of energy inside the stream tube passing through/around a building, whose concept was originally shown by Guffy and Fraser [10] for the flow in pipe. They showed the simplified energy balance equations from Navier-Stokes equation. Axley and Chung [11] also formulated a multi-zone flow network model based on almost the same theory. It is believed that these are rational models because they are based on actual flow field and no assumption is included. As Kato [9] noted, major problem to establish the power balance model is how to predict lost power inside the stream tube.

The microscopic detail of cross-ventilation flow is a quite complicated phenomenon, and even the flow mechanics have not been sufficiently clarified. The objective of this work is to analyze the stream tube of cross-ventilated room, and evaluate the lost power for prediction of the flow rate. Before that, details of the cross-ventilation phenomenon must be clarified. In this paper, therefore, two kinds of wind tunnel tests are considered. One is based on pressure measurement on the ground plane, which may serve to classify the flow characteristics. The other is based on detailed velocity field measurements using particle image velocimetry (PIV) measurements on the windward and leeward sides of the model. Considering the challenge to analyze cross-ventilation airflows in future work, CFD analyses simulating those wind tunnel tests are conducted and their accuracy are also to be shown.

**METHODS**

**Model and Cases**

Figure 1 gives the basic configuration of the studied model. Two openings of the same size are provided for the windward and leeward walls. In order to understand the difference in flow pattern due to opening size, three cases of porosity $\phi$ (opening area / façade area) [12] were studied (See Table 1). Figure 2 shows the geometry of the studied cases.

![Figure 1. Geometry of studied model](image1)

![Figure 2. Studied cases](image2)

Table 1. Studied cases of opening size

<table>
<thead>
<tr>
<th></th>
<th>Small Opening</th>
<th>Middle Opening</th>
<th>Large Opening</th>
</tr>
</thead>
<tbody>
<tr>
<td>$w$ [mm]</td>
<td>17</td>
<td>80</td>
<td>80</td>
</tr>
<tr>
<td>$h$ [mm]</td>
<td>17</td>
<td>40</td>
<td>80</td>
</tr>
<tr>
<td>Porosity [%]</td>
<td>0.83</td>
<td>9.23</td>
<td>18.46</td>
</tr>
<tr>
<td>$t/\sqrt{A_{opening}}$</td>
<td>0.53</td>
<td>0.16</td>
<td>0.11</td>
</tr>
</tbody>
</table>

(1) Y-Z direction  (2) X-Y direction  (1) $\phi=0.83$ %  (2) $\phi=9.23$ %  (3) $\phi=18.46$ %
Pressure Measurement on Ground

In order to understand the cross-ventilation phenomenon by wind tunnel test, the static pressure on the ground plane was measured using a pressure plate with 300 pressure taps. The experiment was conducted in the closed-circuit type wind tunnel of University of Gävle, which has a cross-section of 3.0 m width and 1.5 m height. The detached house model was located on the pressure plate installed in the floor of the wind tunnel. The wind direction was fixed to be perpendicular to the opening. Figure 3 shows the layout of the pressure taps and the model on pressure plate. To keep the Reynolds number sufficiently high, the test model was exposed to a free flow of 19 m/s. The wind tunnel was run without any roughness elements to obtain sufficiently high pressures. This naturally makes a difference in the approach velocity profile from that found in typical field conditions. However, this discrepancy is not important for this context to understand the basic flow characteristics. Figure 4 shows the profiles of velocity and turbulent intensity of the approach flow.

PIV Measurement around Detached House Model

The details of the airflow patterns is also one of the flow characteristics that can be different between cross-ventilation through large openings and infiltration through cracks. To investigate differences in these flow pattern due to the opening size, velocity distributions on the windward and leeward side were measured by using a PIV measurement system. A smoke generator was located on the leeward side of the model as shown in Figure 5. The smoke could approach the model through the closed-circuit wind tunnel. A CCD camera was located on the side of the test model. A double pulsed Nd:YAG laser to generate light sheet was located on the windward or leeward side depending on the measured region. The laser sheet was oriented along the center plane of the model. The time interval of the laser double pulse was set as 30 μs. The sampling frequency was 5.0 Hz, and the sampling time was 24 seconds; i.e. 120 pairs of photos were obtained for one measurement.
The Fast Fourier Transform (FFT) cross-correlation method was used for the analysis. By averaging the cross-correlation within each interrogation window and detecting the correlation peak, a 2-D velocity distribution was obtained. Figure 6 shows the measured region for the windward and leeward side. For the windward measurement, the size of measured area was 171.8 mm long by 128.8 mm high. For leeward measurement, on the other hand, it was 282.2 mm long 211.7 mm high. Although the pressure measurements were conducted under a free flow of 19 m/s, the PIV measurements were done under a flow of 10 m/s to facilitate the measurement, assuming the similarity of flow due to the high Reynolds numbers used. Table 2 gives the summary of the PIV measurement.

Table 2. Summary of PIV measurement

<table>
<thead>
<tr>
<th>Camera Frame Size</th>
<th>1,344 pixel x 1,024 pixel</th>
</tr>
</thead>
<tbody>
<tr>
<td>Interrogation Area</td>
<td>64 pixel x 64 pixel</td>
</tr>
<tr>
<td>Overlap</td>
<td>75%</td>
</tr>
<tr>
<td>Total Number of Vectors</td>
<td>4,941</td>
</tr>
<tr>
<td>Time Interval of Pulse</td>
<td>30ms</td>
</tr>
<tr>
<td>Sampling Frequency</td>
<td>5.0 Hz</td>
</tr>
<tr>
<td>Sampling Time</td>
<td>24 seconds</td>
</tr>
</tbody>
</table>

CFD Analysis

Since more detailed analyses of the airflow are needed to investigate the energy loss in the future work, the wind tunnel test was simulated by CFD analysis. A commercial program Fluent 6.2 was used. Figure 7 illustrates the calculated domain and mesh layout. To reduce the computational load, only half of the domain was modeled assuming a symmetry plane. Total number of grids points used was 802,874 for the small opening case, 1,069,544 for the middle opening case, and 877,644 for the large opening case. The inlet boundary conditions were based on the measured profiles of velocity and turbulent intensity shown in Figure 4, and turbulent length scale was set as 0.07 times of hydraulic diameter of the wind tunnel (140 mm). At the outlet boundary, the gauge pressure was fixed as 0 Pa. Steady state calculations were done using the SIMPLEC algorithm with a QUICK discretization scheme for advection term of the governing equations. As for the turbulence model, a Reynolds stress model was used after obtaining sufficient convergence of calculation using the standard k-ε model.

RESULTS AND DISCUSSIONS

Pressure distribution of ground

Figure 8 shows the static pressure along the centre line of the model on the floor obtained from the measurement and CFD analysis. Obtained results are normalized by dynamic
pressure of 19 m/s. CFD results show relatively good agreement with measurement except within the inside of case (2) and on the windward side in the cases of relatively small openings, where a static pressure drop can be seen in measurement. It is believed that this pressure drop is caused by a vortex generated on the windward side, which is to be discussed later. From these results, it might be possible to classify the flow tendency roughly into three types. When the openings are small, a large pressure drop occurs at the windward and leeward openings and an almost uniform pressure can be seen inside the room. For larger openings, the pressure loss is mainly caused by the inlet opening, and internal pressures become lower than that observed for small openings. When the openings are much larger, the pressure gradient at the openings become smaller and internal pressures increase again.

Figure 8. Static pressure along the centre line on the ground plane

Flow pattern around detached house model

Figure 9 shows the windward airflow patterns obtained from the PIV measurement and CFD analysis. The length of velocity vector for the reference velocity is the same for all plots.

Figure 9. Vector field on the windward of model
The airflow pattern obtained from CFD analysis is qualitatively well simulated. For small openings, it can be seen that a large vortex is generated in front of the windward opening because of the large resistance. It is believed that this causes the pressure drop on the ground plane. For the porosity 9.23%, a small vortex can still be seen in PIV result, though it cannot be seen in CFD result. For large openings, the vortex cannot be seen any more, because the lower resistance allows more flow to enter into the model.

Figure 10 shows the flow patterns on the leeward side of the model obtained from PIV measurement and CFD analysis. Although the outflow from the leeward opening of CFD result seems to have larger component in the mainstream direction, the flow pattern is qualitatively well simulated. The flow from the small opening is diffused in the large wake and its dynamic pressure seems to be dissipated. For middle opening, the dynamic pressure remains to some extent, and the stream seems to be diverged into two parts; i.e. an upper part that flows in backward and a lower part that flows in the leeward direction directly. In the case of large openings, almost all of the stream tube from the opening flows directly in leeward direction with its dynamic pressure remaining.

To evaluate the accuracy of CFD and how the dynamic pressure is dissipated/remaining more quantitatively, detailed distributions of X-component of velocity on the leeward side are shown in Figure 11 for both PIV measurement and CFD analysis. Here, velocities along three vertical lines of X=300, 400, and 500 mm are shown. In the measurement result of the largest opening case, unnatural velocities can be seen along the line of X=500 mm at the height around 50 mm. This seems to be an error of PIV measurement caused by the correlation peak detected at incorrect location. As for the accuracy of simulation, CFD results overestimate the velocity around the ground plane, though tendencies are simulated well. In addition, the velocity simulated by CFD is not as decreased as that of PIV. Although CFD results could simulate the flow pattern, more efforts to improve the accuracy seem to be needed.
Comparing the differences due to opening size, it is obvious that velocity in mainstream direction still remains even at the leeward region (X=500 mm). This is an essential difference in flow between infiltration and cross-ventilation.

These differences in flow present the difference in flow condition. The first one is the typical tendency of the infiltration through cracks as Murakami et al [8] showed. In this case, the flow is diffused sufficiently after flowing through openings. In addition, it is believed that the static pressure behind the leeward openings becomes almost the same as that of a sealed model. As a result, the conventional method to predict the flow rate shown in Eq. (1) works well. The third flow pattern, large openings, is obviously different. The flow is not immediately diffused in the wake and the static pressure on the leeward side is not same as wind pressure of a sealed building either as the authors [2] showed. This means that both of the two assumptions in the conventional method are not correct. Consequently, the flow rate cannot be predicted precisely. The second flow pattern, the case of middle opening size, can be interpreted to be between these two extreme conditions. According to the opening size and probably room depth also, flow condition seems to vary continuously.

By these analyses, the essential differences in flow condition due to the opening size were clarified and two extreme flow conditions and one example between them were shown. To investigate the stream tube and the transported power, CFD analysis is needed. Since the accuracy was not good enough quantitatively, some settings (e.g. turbulence model, grid system and so on) must be changed to improve the accuracy, though the tendencies in flow pattern could be well simulated.

**CONCLUSIONS**

The cross-ventilation is quite a complicated phenomenon and it is a “flow” problem, which varies depending on many factors like opening size. Therefore, it cannot be possible for conventional prediction method of flow rate to work well for the flow through large openings. In this paper, differences in flow condition have been analyzed by means of wind tunnel test and CFD analysis, and the accuracy of CFD was also studied.

Based on the pressure distribution on the ground plane, tendency of the airflow may be roughly classified into three types. When the openings are small relative to the facade, the pressure drops are caused principally by both windward and leeward openings, and almost uniform internal pressure can be seen. For openings of moderate porosities, a large pressure drop occurs at the inlet opening for those cases when inlet and outlet openings are of the same size and internal pressure is still uniform. For openings of large porosity, two equal sized openings and the room within work as if they were one resistance; the pressure drops
gradually inside the model. By PIV measurement the flow patterns could also be qualitatively distinguished as follows. In the case of small openings, a vortex is generated in front of the model, and outflow from the model is diffused immediately in a wake on the leeward side. For large openings, the windward vortex disappears, and the flow from the leeward openings flows into downward directly with its exit kinetic energy preserved. This obviously violates the assumption of the conventional prediction method for cross-ventilation of the flow rates. According to opening size, flow condition continuously varies between the two types.

The authors aim to establish a new prediction method based on a mechanical energy balance inside the stream tube passing through/around the building. Then, the CFD study seems to be necessary to investigate the stream tube. By comparing the results obtained from the wind tunnel test and CFD analysis, qualitatively good agreement could be seen for flow pattern. As a future prospect, the stream tube passing through the detached house model is to be analyzed based on the CFD analyses with higher accuracy. Then, the transported power and the lost power inside the stream tube are to be analyzed.

ACKNOWLEDGEMENT

Authors would like to thank Ms. Elisabet Liden, research engineer in University of Gävle, for kindly helping in PIV measurement. Valuable comments provided by Professor James W. Axley (Yale University) are gratefully acknowledged. A part of this work was supported by Grant-in-Aid for JSPS Fellows (20 912, representative Tomohiro Kobayashi).

REFERENCES