Figure 1. Flow characteristic for small and large opening.

**ABSTRACT**

Conventional method to predict ventilation rate induced by wind is based on the orifice equation associated with the discharge coefficient and wind pressure coefficient. In the cross-ventilation phenomena, however, this method has a problem due to the difficulty to predict resistance of the building related with total pressure loss. In this paper, therefore, the stream tube caught by the inlet opening is analyzed to investigate the pressure loss due to the transformation (and possibly convergence and divergence) of the stream tube. Two types of model, simple-shaped rectangular model and detached house model, were analyzed with three cases of porosity by using CFD prediction. Flow fields inside and outside of the model are to be compared between two types of model. Besides, based on the stream tube analysis, cross-sectional area and average pressure inside the stream tube will be shown and compared between two types of model. For the detached house model, finally, static pressure inside the stream tube will be compared with that on the floor.

**KEYWORDS**

Cross-Ventilation, CFD, Wind Tunnel, Stream Tube, Total pressure

**INTRODUCTION**

Wind induced cross-ventilation is considered to be beneficial to get thermal comfort in a hot summer. In predicting flow rate through cross-ventilated room, the discharge coefficient obtained from the chamber method and pressure difference predicted from the wind pressure coefficient are usually applied to orifice equation as below:

\[ Q = C_d A_{opening} \sqrt{\frac{g}{\rho}} (P_w - P_l) \]  

(1)

where, \( C_d \) is the discharge coefficient of the room, \( P_w \) and \( P_l \) are the wind pressures on the windward and leeward side of the model obtained from the wind pressure coefficient of a sealed building. It is often said that this model can be reliable in the case of small opening, while it is not for large opening. Many research mentioned the problem of this equation for long time. Kobayashi et al. (2007) have also shown the problem of this equation in conventional method to predict the discharge coefficient by chamber method and it was shown that chamber method conducted under the windless condition with fan cannot evaluate the total pressure loss in cross-ventilated flow field. It was also shown that using the wind pressure coefficient obtained from a sealed building could cause excessive evaluation of driving pressure in the case of large opening. Flow characteristic inside the building provided with large openings is different from that with small openings as shown in Figure 1 (Kotani et al. (2006)).

† Corresponding Author: Tel: + 81-6-6879-7645, Fax: + 81-6-6879-7646  
E-mail address: kobayashi_tomohiro@arch.eng.osaka-u.ac.jp

![Flow characteristic for small and large opening](image-url)
Murakami et al. (1991), Kato (2004) defined “crack” as a small opening where kinetic energy mostly dissipates in the room, and a large opening where, mostly preserved. In these papers, it is suggested total kinetic energy preservation equation derived from Navier-Stokes equation be considered adding to mass conservation, which is called Power Balance Model illustrated by Guffy et al. (1989). As a recent study, Axley (2005) also suggested power balance approach for the analysis of multi zone airflow. Although these methods considering power balance in building system can evaluate airflow in building, investigation of energy dissipation (i.e. total pressure loss) is needed. In the previous paper, therefore, Kobayashi et al. (2005) have determined stream tube caught by the inlet opening for a simple-shaped rectangular model exposed to a free flow field, and calculated the average total, static, and dynamic pressure inside the stream tube by CFD, which can investigate energy dissipation with some cases of the opening size. In this paper, a detached house model set on the floor, considered as closer to actual condition of cross-ventilation, will be analyzed adding to a rectangular model. Then, properties of the stream (e.g. cross-sectional area and total, static, and dynamic pressure) are compared with each other.

For a detached house model, furthermore, static pressure inside stream tube (considered to be difficult to measure) is to be compared with that on the floor (which is easy to be measured) in order to investigate the possibility of alternation in measurement.

**METHOD**

**Models**

Based on the previous wind tunnel experiment, two types of model were analyzed by CFD, the simple-shaped rectangular model and the detached house model. For the rectangular model, details are shown by Kobayashi et al. (2006), and for the house, shown by Sandberg (2004). Figure 2 shows the geometry of models. The side walls of rectangular model has the thickness of 6.0 mm, and end wall, 0.8 mm in order to obtain sharp edge. Values in figure indicate external length (Internal length for X, Y, Z axis, is 180, 120, 120 mm). This model is provided with openings of same size on both windward and opposite sides. Here, side length of opening is written as L. As for the detached house model, thickness of the side walls and roof is 4.0 mm, and thickness of the gable walls is 9.0 mm. This model is also provided with two openings of the same size on both sides whose width is 80 mm, and height is h mm.

![Figure 2. Geometry of analyzed model](image)

**Cases**

Parameter for the analysis was opening size. Three cases of opening size were set for each type of model. Figure 3 shows the geometry of each case with corresponding porosity $\Phi$ defined as opening area / façade area. The opening size on the windward and leeward side was the same for all cases.

![Figure 3. Cases of opening porosity $\Phi$ for each type of model](image)
Summary of CFD prediction
Analysis was conducted by CFD simulating previous wind tunnel experiment, because it is supposed to be beneficial to explore the properties of stream tube. For all calculations, Fluent 6.1 was used as the CFD code (Kobayashi et al. (2006)). All analyses were conducted at steady state with SIMPLEC as a pressure-velocity-coupling algorithm and Reynolds stress model as a turbulence model.

For the rectangular model, model was located at the center of the wind tunnel, and exposed to a free flow field of 10 m/s with relative turbulent intensity of 1.0%. Figure 4 shows the calculated area of CFD, a quarter of the wind tunnel cross-section was calculated with two symmetry planes.

For the detached house model, on the other hand, model was set on the floor of wind tunnel with the opening oriented as perpendicular to the wind direction, as shown in Figure 5. Figure 6 shows the profile of the approach flow regarding mean velocity and relative turbulent intensity given as inlet boundary conditions. To be able to measure the pressure on the floor the wind tunnel was run without roughness elements. For both types, models were oriented as wind direction was perpendicular to the end walls.

![Figure 4](image)
**Figure 4. Calculated area and mesh layout for rectangular model**

![Figure 5](image)
**Figure 5. Calculated area for detached house model**

<table>
<thead>
<tr>
<th>Table 1. Summary of CFD analysis</th>
</tr>
</thead>
<tbody>
<tr>
<td><strong>Program</strong></td>
</tr>
<tr>
<td><strong>Differentiation</strong></td>
</tr>
<tr>
<td><strong>Algorithms</strong></td>
</tr>
<tr>
<td><strong>Turbulence model</strong></td>
</tr>
<tr>
<td><strong>Model type</strong></td>
</tr>
<tr>
<td><strong>Boundary condition</strong></td>
</tr>
<tr>
<td><strong>Initial velocity</strong></td>
</tr>
<tr>
<td><strong>Initial turbulence intensity</strong></td>
</tr>
<tr>
<td><strong>Length scale</strong></td>
</tr>
<tr>
<td><strong>Reynolds number</strong></td>
</tr>
<tr>
<td><strong>Inlets</strong></td>
</tr>
<tr>
<td><strong>Outlet</strong></td>
</tr>
<tr>
<td><strong>The number of grids</strong></td>
</tr>
</tbody>
</table>

![Figure 6](image)
**Figure 6. Inlet boundary condition for detached house**
RESULTS AND DISCUSSION

Flow field inside and around model

In order to investigate the difference in airflow through and passing around the model between both models qualitatively, flow field is compared. Figure 7 shows the distribution of velocity divided by the velocity of approach flow (10m/s for rectangular, and 19m/s for house). Velocity inside the detached house model does not reach to 1.0 because of the influence of floor. As a well-known tendency, velocity (i.e. kinetic energy) behind the model becomes small in the case of small openings, while it is significantly preserved when openings are large. Consequently, the wake zone of flow recirculation behind the model becomes larger for small openings.

In all cases, velocity becomes larger around the windward corner of the model because of contraction due to flow separation. For the rectangular model with large openings, the separation area becomes smaller because of less vertical flow along the windward wall. On the other hand, opposite tendencies can be seen in the detached house type. This is supposed due to the effect of eaves.

Figure 7. Distribution of non-dimensional velocity by CFD

Figure 8 shows the contour line of turbulent kinetic energy. As for the rectangular model, value of turbulent kinetic energy is not so large around the outside corner of the model. This is supposed due to inadequate mesh layout of CFD outside of model, focusing on stream tube only inside. Of course appropriate mesh layout should be applied when the flow outside of model is analyzed, and Takeda et
al. (2007) have studied it in another study. However, tendencies in these results obtained here are obvious. In the case of large openings, much less turbulent kinetic energy is produced outside of the model, while larger production can be seen inside the model, compared with small openings. As indicated in the result of velocity distribution, the detached house model shows different tendencies from the rectangular model around the windward eaves. As for inside of the detached house model with large opening, larger turbulent kinetic energy is distributed than the case of the small openings.

**Determination of the stream tube**
Stream tube caught by the inlet opening could be determined by setting out particles from the edge of the opening. For the stream tube on the windward side and inside of the rectangular model, particles were set out from the edges of inlet opening. Inside of the model, however, small part of these particles kept being expanded and did not flow out from the model. Consequently, for the stream tube on the leeward side, particles were set out from the outlet opening and particles flowing directly to downward (not circulate behind the model) were chosen as the outline. In determining the stream tube through detached house model, a large part of particles set out from the inlet opening showed circulation inside the model as shown in Figure 9. In order to evaluate the stream tube quantitatively, a particle flowing to negative direction of X-axis was stopped being traced. Rigorously this method cannot
determine the stream tube in a real sense, violating mass conservation. In order to evaluate basic properties of stream tube, particles flowing to only the positive direction of X-axis were chosen as shown in Figure 10 and the stream tube determined in this way was defined as “dominant stream tube”, distinguished from the “whole stream tube” defined as the stream tube caught by the inlet opening. Figure 11 shows dominant stream tube (a) on the windward side and inside of model, and (b) on the leeward side of model.

Figure 10. Determination of dominant stream tube

Figure 11. Determined dominant stream tube

Cross-sectional area of the stream tube

As the first analysis of the stream tube, cross-sectional area was calculated based on determined stream tube. This analysis is important to associate transformation of the stream tube with total pressure loss as a future prospect. Calculated area is shown in Figure 12 for each case. Dominant stream tube far downstream of detached house could not be determined as its shape was disturbed. Results are evaluated in non-dimensional value divided by the opening area.

In all results, change in shape becomes smaller as the opening becomes larger. For far upstream of the rectangular model, non-dimensional cross-sectional area becomes approximately 0.7 to 0.8. This area is also written as catchment area / opening area (Sandberg (2002)). As for detached house model, cross-sectional area of the dominant stream tube is approximately 0.6 times of the whole. Cross-sectional area far upstream of the model becomes larger as the opening becomes smaller. This tendency is opposite to rectangular model. Comparing those results between whole stream tube of detached house and that of rectangular of close value of opening porosity (e.g. 11.6% of rectangular and 11.54% of detached house, 20.7% of rectangular and 18.46% of detached house), good agreement can be seen on the windward side and inside of the model in spite of the difference in approach flow profile.

Figure 12. Non-dimensional cross-sectional area (Shaded area indicates model)
**Average pressure inside stream tube**

Figures 13 shows the area-weighted average pressure inside the stream tube. Averaged pressure is divided by the total pressure inside the stream tube of far upstream of the model in order to ignore the effect of velocity profile of approach flow and to facilitate the comparison. In all results, the total pressure loss cannot be seen before the inflow. As a common tendency between two types, static pressure shows relatively uniform distribution inside the model with small openings. In the case of the rectangular model, static pressure drop mostly occurs at the inlet opening, while it is also seen at the outlet opening in detached house model. Dynamic pressure of the rectangular model tends to be larger inside the model due to the exposure to a free flow. Nevertheless, total pressure distribution is almost the same between two types of model of the close value of porosity except the leeward side.

**Static pressure on the floor and inside the detached house model**

In investigating the properties of stream tube, it can be useful in a sense if static pressure inside the stream tube can be altered by pressure on the floor, because it can be easily measured in the wind tunnel experiment. In this research, it is also supposed to investigate the stream tube and its pressure loss by means of wind tunnel experiment, and this analysis explores to the possibility of alternation. Figure 14 shows the static pressure along the centerline on the floor obtained by CFD analysis and wind tunnel experiment, and inside stream tube obtained only by CFD analysis. Static pressure is divided by reference dynamic pressure of approach flow (228 Pa).

**CFD result on the floor shows good agreement with that of experiment except windward side of the model where eddy is generated. It also shows good agreement with pressure inside stream tube. From**

---

*Figure 13. Average pressure inside the stream tube (Shaded area indicates model)*

*Figure 14. Comparison of static pressure on the floor and inside stream tube*
these results, it can be said that it is reasonable to evaluate static pressure inside stream tube as pressure on the floor at least inside and leeward side of the model.

CONCLUSIONS
In predicting cross-ventilation rate, the total pressure loss through the room needs to be evaluated accurately. From the viewpoint of stream tube analysis, in this paper, cross-sectional area and average pressure were investigated inside the stream tube through the rectangular / detached house models. Although static pressure and dynamic pressure showed different tendencies between two types of model, total pressure was almost the same when compared in close value of opening porosity of the model. For detached house model, only the dominant part of the stream tube was determined. To consider whole flow rate, however, another part of stream tube circulating inside model must also be evaluated as a problems to be solved. As a final analysis of this paper, the possibility of the alternation of static pressure inside stream tube by pressure on the floor that could be helpful in wind tunnel analysis was explored and it was successfully shown.

As future prospects, total and static pressure needs to be investigated with a number of parameters, (e.g. opening porosity, model length, wind directions, internal partitions and so on.) by CFD analysis and wind tunnel experiment in order to evaluate the flow resistance of flow accurately.

ACKNOWLEDGEMENTS
Authors would like to appreciate Mr. Leif Claesson for conducting wind tunnel experiment. Besides, authors also appreciate Grant-in Aid for Young Scientists (B) of the Ministry of Education, Culture, Sports, Science and Technology, Japan, 2004-No.16760474 (Representative, H. Kotani) that supported a part of this research.

REFERENCES