#6081

International Symposium on Room Air Convection and Ventilation Effectiveness
University of Tokyo, July 22 – 24, 1992

Numerical Simulation of Cross-Ventilation in a Single Unit House

Jun-ichiro TSUTSUMI
Assoc. Professor, Tohwa University
Minami-ku, Fukuoka 815 Japan

Tadahisa KATAYAMA
Professor, Kyushu University
Kasuga-shi, Fukuoka 816 Japan

Tetsuo HAYASHI
Assoc. Professor, Kyushu University
Kasuga-shi, Fukuoka 816 Japan

Ping HE
Graduate Student, Kyushu University
Kasuga-shi, Fukuoka 816 Japan

ABSTRACT
This report presents the results of the numerical simulation of cross-ventilation and the examination of the flow rate of cross-ventilation. The numerical simulation model is a rectangular prism area around a single unit house. The model house has two relatively large windows on opposite walls which face the wind vertically. The air flow in the indoor and the outdoor space is calculated continuously. Two kinds of partially fined mesh system are applied to the numerical simulation. One is the multi mesh method, and the other is the partial fine area method. The latter means the re-calculation of the air flow in the partial space on finer mesh system with the calculation results on coarse mesh system. The indoor air flow distributions calculated by these two methods agree well. The flow rate obtained by the numerical simulation is larger than that calculated by the equation for ordinary ventilation rate.

KEYWORDS Cross-Ventilation, Numerical Simulation, Flow Rate
Partial Fine Mesh, Indoor and Outdoor Air Flow

1. INTRODUCTION

Cross-ventilation usually means natural ventilation with large flow rate caused by the wind through open windows or doors. The wind directly blows into the indoor space with a small loss of its dynamic energy. Cross-ventilation has great effect on the thermal sensation of occupants(Ishii et al. 1990). This is the important purpose of cross-ventilation in summer. The thermal effect of cross-ventilation depends upon the distribution of indoor air flow speed which is greatly influenced by the wind around the building. It is almost impossible to calculate the flow rate of cross-ventilation exactly by the usual equation for ventilation rate.

Cross-ventilation should be simulated in order to predict the indoor air flow distribution to utilize the cross-ventilation better as the natural cooling device, and to examine the flow rate of cross-ventilation to model the equations for cross-ventilation rate finally. In this paper, 3-dimensional numerical simulation of cross-ventilation is discussed. The indoor air flow and the air flow around a model house should be simulated continuously to consider the interaction of them(Tsutsumi et al. 1990). Since the indoor and outdoor simultaneous simulation needs a large simulation area and a large number of calculation points, special calculation methods for in and around a model house are provided to reduce the calculation points and the calculation time to simulate the changable air flow.

2. NUMERICAL SIMULATION METHOD

2.1 Simulation Model

The whole area for numerical simulation is shown in Figure 1. It is a 3-dimensional space around a model house. The model house is a cube with two square openings, which is used as a one-room model and a two-room model with a partition as shown in Figure 2. The length of a side of the model house is the representative length of the simulation model. It is placed on the ground level, facing its windows vertically to the wind.

The calculation points are basically fixed on the staggered grid system which divides the representative length 9 equally. This grid system called "basic mesh". The air flow in and near the model house should be simulated on finer grid system, because the air flow remarkably changes in this area. Therefore, two kinds of mesh are applied for finer grid system. One is "multi mesh"(Kurabuchi et al. 1991), and the other is "partial fine area". The former is the combination of the basic mesh and finer mesh of which the area is shown in Figure 1. The latter means that the calculation on finer mesh is independently carried out in a limited area which is shown in Figure 3 from the results of the basic mesh. The finer mesh scale is the halp of the basic mesh in all the directions.

2.2 Calculation Methods

The basic equations for the numerical simulation are shown in Table 1. The k-ε 2-equation medel is used as the turbulence model. Forward difference is applied to the differential terms with respect to time. Basically, central difference is applied to the differential terms with respect to space. Only the advective terms of k and ε transport equations are changed into up-wind difference. Vertical distribution of the normal velocity component on the inflow boundary is given as the power law with the exponent of 1/4. Tangential velocity components are naught on the inflow and outflow boundary. Normal velocity component on the outflow boundary is found from the continuity of flux. The value of k and ε are found from the wind tunnel tests. They are shown in Figure 4. The boundary conditions on the upper and the side boundaries are free-slip. The boundary condition on solid boundaries for the tangential velocity components is the power law with the exponent of 1/4. The boundary condition for the other valiables are free-slip.

The numerical simulation of cross-ventilation in the one-room model is carried out on the basic mesh and on the multi mesh, and that in the two-room model is carried out on the multi mesh. The numerical simulation of cross-ventilation in the one-room model and that in the two-room model are carried out on the partial fine area, using the results of the one-room model on the basic mesh as the boundary conditions. Moreover, the numerical simulation of the air flow around the model house without cross-ventilation is carried out on the basic mesh to find the wind pressure on the wall surface to examine the ventilation rate.

Table 1. Basic equations for the numerical simulation

$$\frac{\partial U_i}{\partial t} + \frac{\partial}{\partial x_j}(U_i\, U_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j}\left(\nu t\, E_{ij} + \frac{1}{Re}\frac{\partial U_i}{\partial x_j}\right) \quad (1) \qquad E_{ij} = \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \quad (2)$$

$$\frac{\partial k}{\partial t} + \frac{\partial}{\partial x_i}(k\, U_i) = \frac{\partial}{\partial x_i}\left\{\left(\frac{\nu t}{\sigma_1} + \frac{1}{Re}\right)\frac{\partial k}{\partial x_i}\right\} + \nu t\, E_{ij}\frac{\partial U_i}{\partial x_j} - \varepsilon \quad (3) \qquad \nu t = C_D\frac{k^2}{\varepsilon} \quad (4)$$

$$\frac{\partial \varepsilon}{\partial t} + \frac{\partial}{\partial x_i}(\varepsilon\, U_i) = \frac{\partial}{\partial x_i}\left\{\left(\frac{\nu t}{\sigma_2} + \frac{1}{Re}\right)\frac{\partial \varepsilon}{\partial x_i}\right\} + C_1\frac{\varepsilon}{k}\nu t\, E_{ij}\frac{\partial U_i}{\partial x_j} - C_2\frac{k\varepsilon}{\nu t} \quad (5) \qquad \frac{\partial U_i}{\partial x_i} = 0 \quad (6)$$

xi:axis, t:time, Ui:velocity component of xi direction, P:pressure νt:eddy viscosity coefficient
k:turbulent kinetic energy, ε:energy dissipation rate, C_D=0.09, C_1=1.59, C_2=0.18, σ_1=1.0, σ_2=1.3

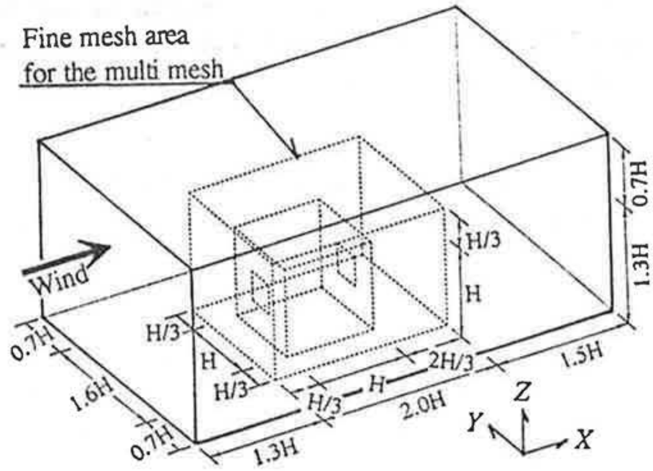


Figure 1. Whole calculation area of numerical simulation.

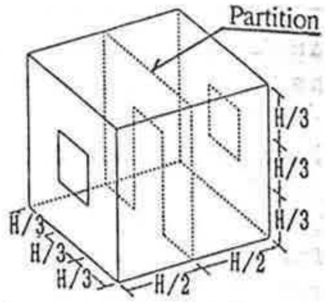


Figure 2. Model house.

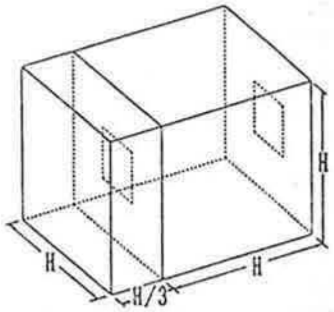
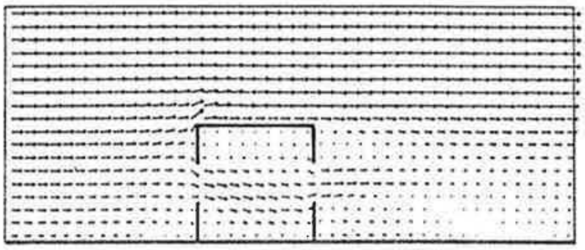
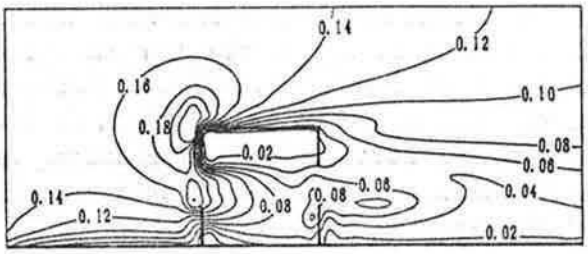


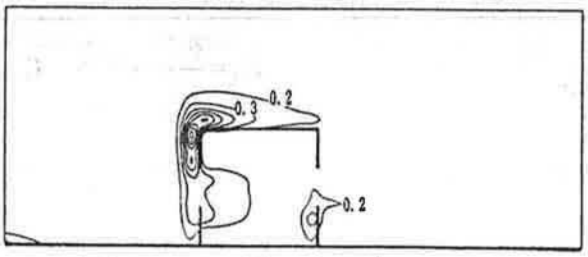
Figure 3. Calculation area for the partial fine area.



(a) Wind vector

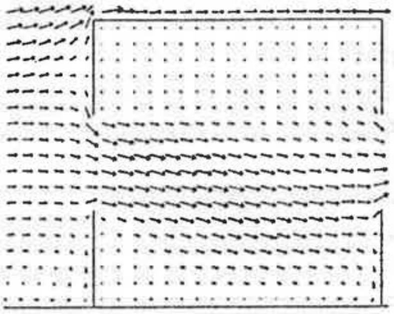


(b) Turbulent kinetic energy

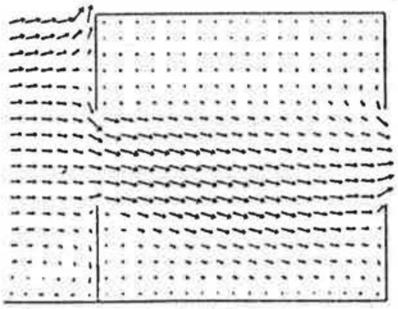


(c) Energy dissipation rate

Figure 4. Numerical simulation results on the central vertical section by the basic mesh.



(a) Multi mesh



(b) Partial fine area

Figure 5. Wind vector on the central vertical section in and near the one-room model.

3.RESULTS AND DISCUSSION

3.1 Numerical Simulation Results

The results of numerical simulation on the basic mesh are shown in Figure 4. These are the distributions of wind vector, turbulent kinetic energy and energy dissipation rate on the vertical section at the centre of *Y* direction. The air flow through the model house is well simulated. *k* and ε increase near the model house.

The distributions of wind vector in and near the model house by the multi mesh and the partial fine area are shown in Figure 5. They are of the same section as that of Figure 4. The detailed air flow in the model house which does not appear in Figure 4 is clearly presented in Figure 5. The results of two kinds of mesh agree well.

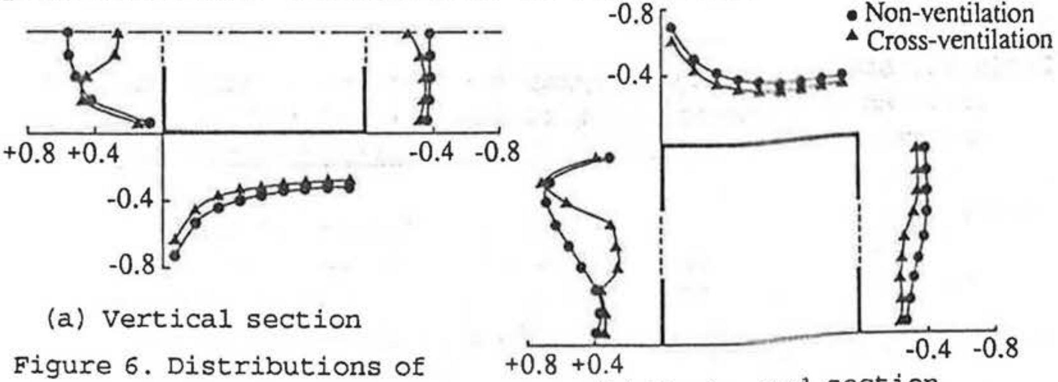
3.2 Ventilation rate

Ordinary ventilation rate is found by the following equation, if the air flow goes through simple openings straightly.

$$q = \frac{1}{\sqrt{\sum_i \left(\frac{1}{Ai\alpha i}\right)^2}} \sqrt{2\Delta P} \qquad (i = 1,2,\cdots) \tag{7}$$

where, q:ventilation rate, i:opening number, Ai: opening area, αi:flow coefficient (0.65)
ΔP:wind pressure difference between windward and leeward

The wind pressure used in this equation is usually measured without ventilation. The wind pressure distributions on the wall surfaces by the numerical simulation on the basic mesh with and without cross-ventilation are shown in Figure 6. The average wind pressure differences on the opening area of these two cases are shown in Table 2. The wind pressure on the opening with cross-ventilation is smaller than that without ventilation. The wind pressure without ventilation is used in the following discussion



(a) Vertical section

(b) Horizontal section

Figure 6. Distributions of wind pressure on the wall surfaces of the central vertical section and horizontal section.

Table 2. Average wind pressure differences between inlet and outlet opening

Ventilation	ΔP <Inlet>-<Outlet>
Cross-ventilation	0.5965
Non-ventilation	0.9191

Table 3. Flow rates found by the equation (1) in the case of non-ventilation

Model	Flow rate by eq.(7)
One-room model	0.06942
Two-room model	0.06528

Table 4. Flow rates found by the numerical simulation and the comparisons of them to those by the equation (1)

Simulation condition		Inlet flow rate	Outlet flow rate	Ratio to eq.(7)
Multi mesh	One-room model	0.08464	0.08464	1.2192
	Two-room model	0.08644	0.08644	1.3241
Partial fine area	One-room model	0.08569	0.08566	1.2342
	Two-room model	0.08448	0.08445	1.2939

The flow rate of the cross-ventilation in the one- and two-room model by eq.(7) is shown in Table 3. The flow rate by the numerical simulation is shown in Table 4 and compared with those by eq.(7). There are small differences between the inlet and outlet flow rate in the case of the partial fine area. The flow rate of the two-room model is larger than that of the one-room model in the case of the multi mesh in spite of the increase of resistance.

4.CONCLUSIONS

Cross-ventilation is well simulated if the indoor and the outdoor air flow are continuously calculated. Both the multi mesh and the partial fine area method can provide the detailed indoor air flow. The ventilation rate calculated by the usual equation is smaller than that of the numerical simulation by 20-30%, because the air flow keep the dynamic energy from the inlet to the outlet.

REFERENCES

Ishii,A.; Sakakibara,N.; Katayama,T.; Tsutsumi,J.;and Nishida,M. 1990. An experimental study on human sensation to air flow in naturally ventilated rooms. Environmental International, Vol.16, pp.61-70.

Kurabuchi,T; and Kamata,M. 1991. A local refinement method of the MAC mesh system Part 1: The Mullti-Mesh method and its applications for two dimensional indoor airflow problems. J. of Archit. Plann.Environ. Engng, AIJ, No.425, pp.1-9.

Tsutsumi,J; Katayama,T.; Hayashi,T.; and Watanabe,T. 1990. Numerical simulation of turbulent air flow in a house induced by cross-ventilation, INDOOR AIR '90, Vol.4, pp.323-328.