NUMERICAL SIMULATION OF TURBULENT AIR FLOW IN A HOUSE INDUCED BY CROSS-VENTILATION

J.Tsutsumi
Towa University
Chikushi-gaoka Minami-ku, Fukuoka
Japan 815

T.Katayama and T.Hayashi
Kyushu University
Kasuga-koen Kasuga-shi, Fukuoka
Japan 816

T.Watanabe
Chubu Electric Company
Higashi-shinmachi Higashi-ku, Nagoya
Japan 461-91

Indoor air flow which is induced by cross-ventilation is greatly influenced by the wind around the building. Numerical simulations of turbulent air flow outside and inside a house with open windows are carried out as models of cross-ventilation. The simulation models are 2-dimensional spaces which are vertical and horizontal sections. The k-e 2-equation model is used as a turbulence model. The air flow on the inlet opening of the house is given to inflow boundaries of multi-room models of which simulation area is limited to indoor space. The validity of the numerical simulations is examined in comparison with the results of wind tunnel tests.

INTRODUCTION

Cross-ventilation, which means a large air flow rate of natural ventilation through open windows or doors, is a simple, old-fashioned, familiar and effective passive cooling method(1,2,3,4). The cooling effect of cross-ventilation is formed from the heat flux on the skin of occupants carried by the sensible air flow. It greatly depends upon the distribution of air flow speed in a room. Indoor air flow have to be predicted in order to design a house to utilize cross-ventilation better.

Two major methods for the prediction of air flow are model tests and numerical simulations. Numerical simulations are more flexible for various shapes of houses than model tests. However, the accuracy of the results are not always identified, and it is difficult to fix boundary conditions which influence the results greatly. Since cross-ventilation is the wind which directly blows into a room through openings, the air flow around a building have great influence on indoor air flow. When a simulation area is limited to an indoor space, inflow boundary conditions have to be given to the inlet openings like a forced ventilation model(5), and the simulation result is a little difference from the actual phenomenon.

The main purpose of this paper is to present a numerical simulation method which can express cross-ventilation adequately. One of the most reasonable ways for this purpose is to simulate the wind around a building and indoor air flow simultaneously. Then, the air flow distributions on the inlet openings are given to the inflow boundary of multi-room models of which calculation areas are limited to indoor spaces. The validity of the numerical simulations is examined in comparison with model tests in a wind tunnel.

SIMULATION MODELS

The models of wind tunnel tests and numerical simulations are shown in Fig. 1. The model of wind tunnel tests is a rectangular cubic box. It is separated into two rooms by a partition. Each room has two openings on opposite walls. Both wind tunnel tests and numerical simulation are done on condition that the wind blows into the openings perpendicularly. The streamwise direction is x, the horizontal lateral direction is y and the vertical direction is z.

The plan of a room is a rectangular of which sides ratio of x:y is 1:2. There are some cases that another partition with an opening is set at the center of x-direction in each room. The partitions makes the model four rooms of which plans are square. Numerical simulation models are the vertical and the horizontal sections of the wind tunnel test model. The vertical section is an x-z plane at the center of the opening width. The horizontal section is an x-y plane at the center of the model height.

WIND TUNNEL TESTS

The wind tunnel used for the model tests is a close circuit boundary layer type. It has a working section of 8m long, 1.5m wide and 1m high. The model is fixed at 5.5m downstream on the working section floor. The approaching wind for the experiments is made to be similar to the natural wind. The profiles of mean wind speed and turbulence intensity are formed by spires, a screen and roughness elements. The screen which is made of wooden square bars is fixed at 3.3m upstream from the model. 9 spires are set in 2 rows upstream of the screen. 2 kinds of roughness elements are scattered on the working section floor from the screen to 2.2m downstream.

The profiles of the mean wind speed and the turbulence intensity at the point where the model would be fixed are shown in Fig. 2. These are profiles measured by a hot-wire anemometer without the model. The mean wind speed profile is approximately regarded as the power law profile with the exponent of 1/5. Turbulence intensity in the range of the model height is almost constantly 17%.

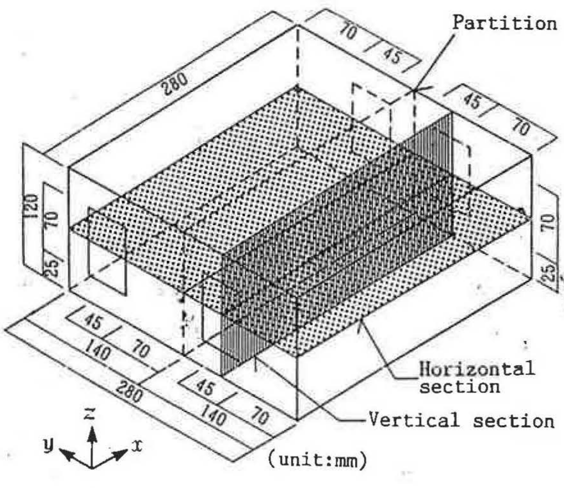


Fig. 1 Simulation model

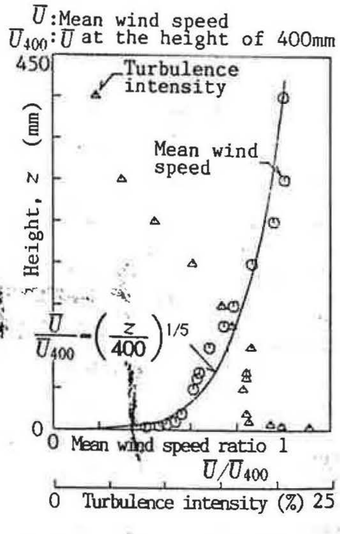


Fig. 2 Approaching wind profile

There are two purposes in the wind tunnel tests. One is to get the data, especially the kinetic energy and the length scale of turbulence, for inflow boundary conditions of the numerical simulations. They are calculated from the data measured by a hot-wire anemometer. The other is to obtain air flow speed distributions in two sections of the model which are the numerical simulation models. A non-directivity thermistor anemometer is used to measure them. They are non-dimensionalized by the representative speed, and compared with the numerical simulation results. The representative speed is the wind speed of the approaching wind at the height of the model. The wind tunnel tests are carried out on condition that the representative speed is 5m/s.

NUMERICAL SIMULATIONS

Indoor air flow and the wind around buildings can be always regarded as turbulent flows. It is necessary to use a mathematical model of turbulence for numerical simulations. The k-ε 2-equation model is adopted in this paper(5). The governing equations are as follows:

$$\frac{\partial U_i}{\partial x_i}=0 \qquad (1)$$

$$\frac{\partial U_i}{\partial t}+\frac{\partial}{\partial x_j}(U_iU_j)=-\frac{\partial \Pi}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\nu_t\cdot E_{ij}+\frac{1}{Re}\frac{\partial U_i}{\partial x_j}\right) \qquad (2)$$

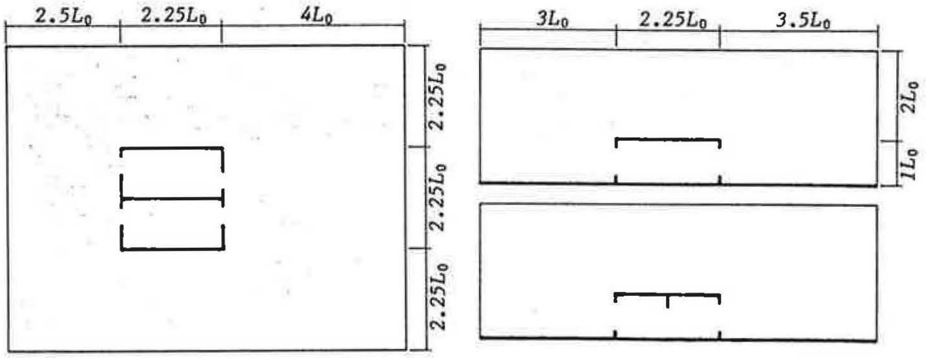
$$\frac{\partial k}{\partial t}+\frac{\partial}{\partial x_i}(kU_i)=\frac{\partial}{\partial x_i}\left\{\left(\frac{\nu_t}{\sigma_1}+\frac{1}{Re}\right)\frac{\partial k}{\partial x_i}\right\}+\nu_t\cdot E_{ij}\frac{\partial U_i}{\partial x_j}-\varepsilon \qquad (3)$$

$$\frac{\partial \varepsilon}{\partial t}+\frac{\partial}{\partial x_i}(\varepsilon U_i)=\frac{\partial}{\partial x_i}\left\{\left(\frac{\nu_t}{\sigma_2}+\frac{1}{Re}\right)\frac{\partial \varepsilon}{\partial x_i}\right\}+C_1\frac{\varepsilon}{k}\nu_t\cdot E_{ij}\frac{\partial U_i}{\partial x_j}-C_2\frac{k\varepsilon}{\nu_t} \qquad (4)$$

$$\nu_t=C_D\frac{k^2}{\varepsilon} \qquad (5)$$

$$E_{ij}=\frac{\partial U_i}{\partial x_j}+\frac{\partial U_j}{\partial x_i} \qquad (6)$$

where, x_i=coordinate axes, t=time, U_i=mean velocity component in x_i direction, Π=mean pressure with $(2/3)k$, ν_t=turbulent eddy viscosity, k=kinetic energy of turbulence, ε=kinematic dissipation rate of turbulence energy, Re=Reynolds number. C_D, C_1, C_2, σ_1 and σ_2 are empirical and experimental constants. They are fixed as 0.09, 1.59, 0.18, 1.0 and 1.3, respectively. All the variables are non-dimensionalized by the representative length and the representative speed. The former is the model height and the latter is the approaching wind speed at the model height. Re is found about 5×10^4 from the wind tunnel tests.



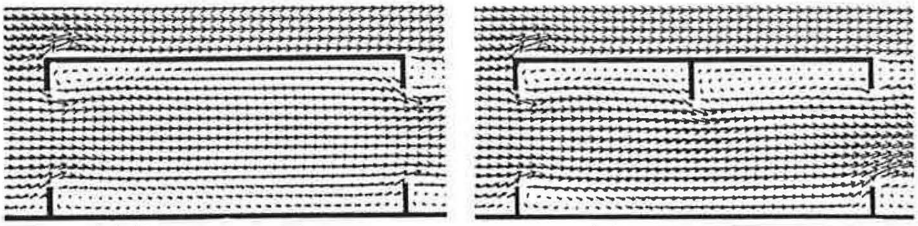
(a) Horizontal section model (b) Vertical section model

Fig. 3 Numerical simulation models (L_0: Representative length)

There are two types of numerical simulations. One is indoor and outdoor air flow simultaneous simulation, and the other is indoor air flow limited simulation. They are called "indoor-outdoor simulation" and "indoor-limited simulation". The indoor-outdoor simulation models are shown in Fig. 3. (a) is vertical section models, (b) is a horizontal one.

The finite difference method is used to discrete the equations. The forward difference and the central difference are basically adopted for time and space, respectively. As for the advective terms in the transport equations of k and ε, the upstream difference is applied. The calculation points are set on the grid system which divides the representative length into 16 equally in all the directions.

All the variables except inflow boundary are set naught at the start of calculation. As for the inflow boundary, the initial values of the variables are given from the wind tunnel tests for the indoor-outdoor simulation. Then, they are given from the results of the indoor-outdoor simulation for the



(a) Without partition (b) With partition

Fig. 4 Numerical simulation results of vertical section models

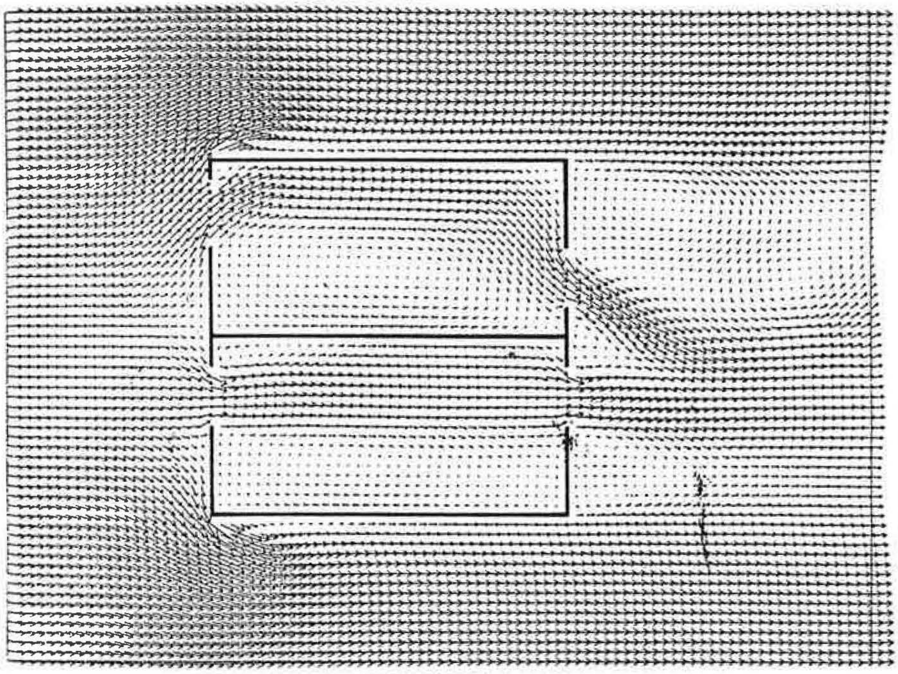


Fig. 5 Numerical simulation results of horizontal section model

indoor-limited simulation. The mean velocity components perpendicular to the walls are naught, and those parallel with the walls are found from the power law profile with the exponent of 1/4. The boundary condition of k is free-slip and that of ε is found from the wall function as follows:

$$\varepsilon = C_D^{3/4} \cdot k^{3/2} \Big/ \left\{ \kappa \left(\frac{\Delta n}{2} \right) \right\} \qquad (7)$$

where, κ=Karman constant, Δn=grid interval in the normal direction. On the outlet boundary, the normal velocity component is found by the flux balance in a cell and the tangential velocity components is free-slip.

NUMERICAL SIMULATION RESULTS

The distributions of air flow vectors as the results of the indoor-outdoor simulation of the vertical section models are shown in Fig. 4. They are not whole simulation areas but parts near the model building. (a) is a one-room model and (b) is a model with a partition. The difference between the air flow distributions on the inlet openings of the two models is hardly found. There is little influence of the partition on the air flow of the inlet opening.

The results of the indoor-outdoor simulation of the horizontal section model is shown in Fig. 5. It is quite clear that the positions of inlet openings have influence on the distributions of air flow near the openings. Unless outside and inside air flow of the model building are simultaneously simulated, such difference between two openings can not appear.

The air flow distributions on the inlet openings by indoor-outdoor simulation are given to the inlet boundary condition of the indoor-limited simulations. These simulation models are horizontal sections with three types of partitions. The results of these simulations are compared with those of wind tunnel tests. The distributions of non-dimensional scalar speed of indoor air flow are shown in Fig. 6. The patterns of air flow speed by the numerical simulations have resemblance to those by the wind tunnel tests in all types. The numbers of the scalar speed by the numerical simulations differ a little from those by the wind tunnel tests. The prime reason of the difference is thought that the numerical simulations are 2-dimensional air flow, while the wind tunnel tests are 3-dimensional one.

CONCLUSIONS

It is realized that the simultaneous simulation for indoor and outdoor air flow is reasonable and suitable method for the prediction of indoor air flow distribution induced by cross-ventilation. When the flow distributions on the inlet openings are given to the simulation models which are limited to indoor spaces as the inflow boundary conditions, the indoor-limited simulations can also express the indoor air flow induced by cross-ventilation, and they reduce calculation time about 1/20 in comparison with the indoor-outdoor simulation.

REFERENCES

1.Tsutsumi J, Katayama T, Ishii A (1984) Measurements of Indoor Air Flow by Cross-ventilation and Its Effect on Thermal Comfort in Apartment Houses. 10th Int'l Congress Biometeorology: 283.

2.Ishii A, Katayama T, Tsutsumi J, Nishida M, Sakakibara N (1987) Estimation of Thermal Environment in a Naturally Ventilated Room with Windows Opened by Using Data from Full-scale Measurements. Indoor Air'87 3: 409-413.

3.Olgyay V (1963) Design with Climate. Princeton University Press, Princeton. p.94-113.

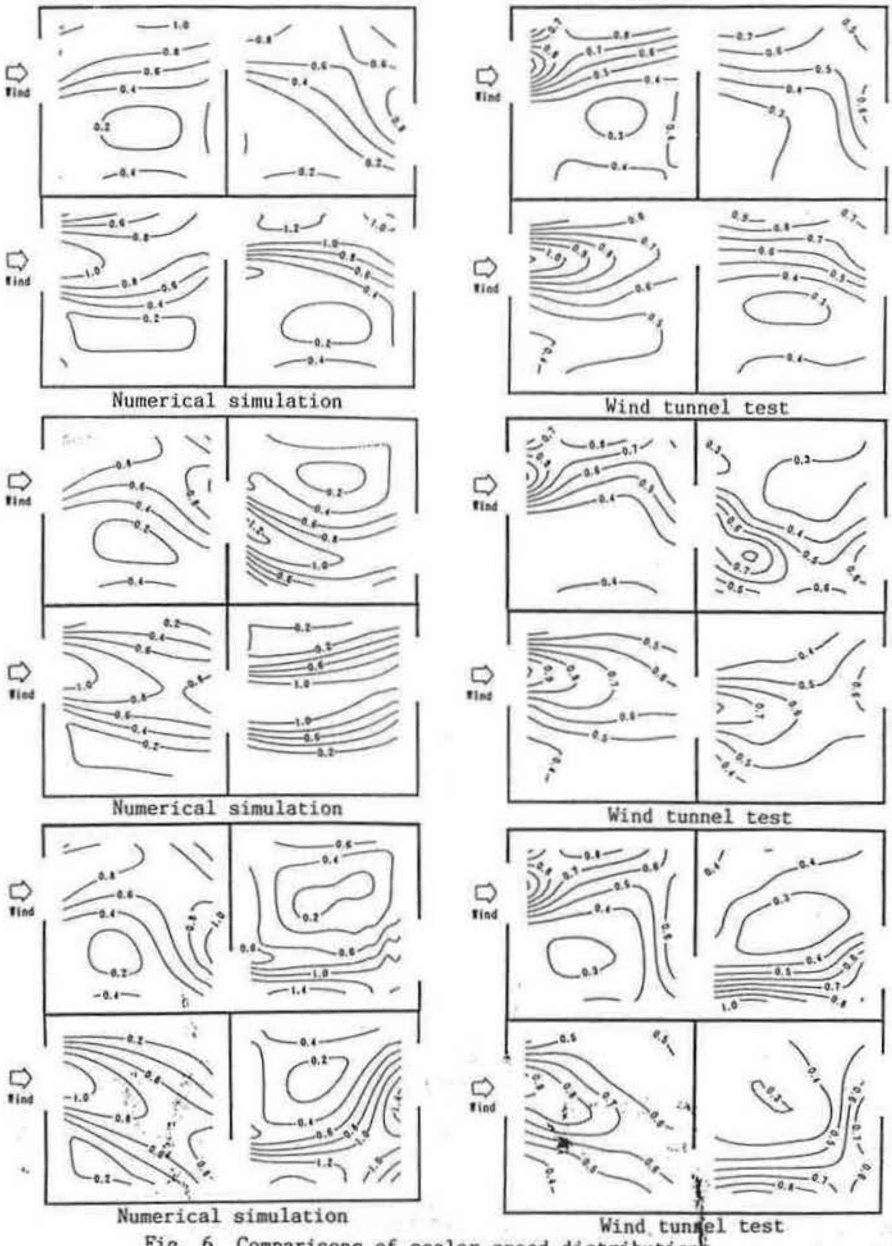


Fig. 6 Comparisons of scalar speed distributions

4.Fathy H (1986) Natural Energy and Vernacular Architecture. University of Chicago Press, Chicago. p.52-60.

5.Tsutsumi J, Katayama T, Hayashi T, Zhang Q, Yoshimizu H (1989) Numerical Simulation of Indoor Turbulent Air Flows Caused by Cross-ventilation and Its Model Experiments. Proceedings 9th AIVC Conference 2: 141-156.