#4165

NUMERICAL SIMULATION OF THERMAL CONVECTION IN A ROOM WITH NATURAL VENTILATION CAUSED BY BUOYANCY

J. Tsutsumi
Tohwa University
Fukuoka, 815 Japan

ABSTRACT

The main purpose of this paper is to simulate air flow distribution in a room with natural ventilation caused by buoyancy. The boundary conditions of openings are proposed to express natural ventilation when a simulation area is limited to an indoor space. The simulation model is one room with two openings on a wall and a heater on a floor. Objective air flow is the thermal convection produced by the heater and natural ventilation induced by the thermal convection. The Large Eddy Simulation(LES) is adopted for the turbulence. The results of the numerical simulation are compared with those of a model test to examine their accuracy. The model test is consisted from air flow visualization and air temperature measurement. The stream line by the numerical simulation corresponds to the visualized flow, while there is a little difference between the temperature distribution by the numerical simulation and that by the model test.

1. INTRODUCTION

The distribution of air flow speed and temperature in a room influence the thermal sensation of occupants. Air distributions have to be simulated to design comfortable and healthy dwelling space. Nowadays, numerical simulation is one of the most useful methods to predict them. Most of numerical simulations for indoor air flow have solved forced ventilation and thermal convection in a room, e.g.(1),(2).
Though natural ventilation has almost always influences on indoor thermal environment, there are few numerical simulations for natural ventilation(3). The indoor air distribution with natural ventilation has to be simulated. The most difficult problem to solve natural ventilation is the boundary condition of openings, since natural ventilation is decided by both indoor and outdoor air flows.
In this paper, the boundary conditions of openings are proposed. They expresses natural ventilation in the case that the simulation area is limited to an indoor space. The indoor air distribution with natural ventilation is simulated. The results of the numerical simulation are compared with those of a model test to examine the accuracy.

2. SIMULATION MODEL

The simulation model is shown in Fig.1. The experiment model mentioned after has the same geometry as this model. The vertical section of x-y

plane is a constant shape of square. The length of a side of the plane, L_0, is the representative length used for the non-dimensionalization. The horizontal axes are x and z, and the vertical one is y. The model has two openings of which widths are $L_0/10$ at the upper and the lower end of a wall. The model has a heater of which breadth is $L_0/2$ at the centre of the floor. The heater produces thermal convection in the model and the temperature difference between inside and outside of the model. Then, the natural convection induces natural ventilation caused by buoyancy. This is the objective air flow to simulate here.

3. NUMERICAL SIMULATION

3.1 Basic Equations

The indoor air flow in the model is not always regarded as full turbulent flow, since the air flow speed of natural ventilation induced by thermal convection is slow and the opening scale is small. The LES is adopted to simulate such a flow. The LES used here is the Deardorff-Smagorinsky model(4),(5). The basic equations are as follows:

$$\frac{\partial u_i}{\partial x_i}=0 \tag{1}$$

$$\frac{\partial u_i}{\partial t}+\frac{\partial}{\partial x_j}(u_i u_j)=-\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left(\nu_{SGS}\cdot e_{ij}+\frac{1}{\sqrt{Gr}}\frac{\partial u_i}{\partial x_j}\right)+\delta_{i2}\theta \tag{2}$$

$$\frac{\partial \theta}{\partial t}+\frac{\partial}{\partial x_i}(\theta u_i)=\frac{\partial}{\partial x_i}\left(\chi_{SGS}\frac{\partial \theta}{\partial x_i}+\frac{1}{Pr\sqrt{Gr}}\frac{\partial \theta}{\partial x_i}\right) \tag{3}$$

$$e_{ij}=\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i} \tag{4}$$

$$\nu_{SGS}=\chi_{SGS}=(Cs\cdot\Delta)^2\left(\frac{(e_{ij})^2}{2}\right) \tag{5}$$

$$\Delta=(\Delta x_1\cdot\Delta x_2\cdot\Delta x_3)^{1/3} \tag{6}$$

where, x_i=coordinate axis, t=time, u_i=velocity component in x_i direction, p=pressure, θ=temperature, δ_{ij}=Kronecker's delta, ν_{SGS}=subgrid scale(SGS) eddy viscosity coefficient, χ_{SGS}=SGS eddy diffusion coefficient, Δx_i=grid interval in x_i direction, Cs=Smagorinsky constant, Gr=Grashof number, Pr=Plandtl number. All these equations are non-dimensionalized by the representative values. The representative length is mentioned before. The representative speed is $(g\beta\Delta\theta L_0)^{1/2}$ which is called the buoyancy speed. The representative temperature difference is the difference between the heater surface temperature and outdoor air temperature. Gr is found from these representative values.

3.2 Boundary Conditions

Boundary conditions on openings To define the boundary conditions on openings, the air flow through the openings is presumed as follows: 1)The air flow through the openings is laminar flow. 2)The velocity component perpendicular to the opening surfaces dose not change spatially in this direction. 3)The velocity components parallel to the opening surface are naughts. These presumptions make the momentum equations as follows:

$$\frac{\partial u}{\partial t} = -\frac{\partial p}{\partial x} + \frac{1}{\sqrt{Gr}}\left(\frac{\partial^2 u}{\partial y^2} + \frac{\partial^2 u}{\partial z^2}\right) \tag{7}$$

here, the pressure gradient term is found from the assumption that the pressure gradient is linear in the ducts which are assumed on the outside of the openings. The viscosity terms are calculated as shown in Fig.2. Since the outdoor pressure and temperature are defined as the standard for non-dimensionalization, non-dimensional pressure and temperature outside of the ducts are naughts. The temperature at the opening is regarded as the same temperature as the upstream one.

Boundary conditions on walls The velocity component perpendicular to the wall is naught. The boundary layer of the velocity components parallel to the wall is assumed to fit the power law. The heat transmission through walls is thought as shown in Fig.3. The outside heat transfer coefficient is set constant. The inside heat transfer is assumed to be analogous to the velocity boundary layer and to fit the power law with the exponent of $1/n$. From these heat flux, following three equations are found:

$$c_w \rho_w \frac{l_w}{4}(\theta_{w1}^{N+1} - \theta_{w1}^{N}) = (q_1 - q_2)\Delta t \tag{8}$$

$$c_w \rho_w \frac{l_w}{2}(\theta_{w2}^{N+1} - \theta_{w2}^{N}) = (q_2 - q_3)\Delta t \tag{9}$$

$$c_w \rho_w \frac{l_w}{4}(\theta_{w3}^{N+1} - \theta_{w3}^{N}) = (q_3 - q_4)\Delta t \tag{10}$$

where, Δt=time step interval, c_w, ρ_w=specific heat, specific gravity of the walls. The superscript means the time step. The new time step temperatures are solved from these equations. The outside heat transfer coefficient is non-dimensionalized as the Nusselt number, $Nu = \alpha_o L_0 / \lambda_a$.

3.3 Calculation Conditions

The calculation algorism is the MAC method. The calculation points are set on the staggered grid system which divides the simulation area into 20 equally in each direction. The Smagorinsky constant is set 0.1. There are several other parameters in this model. The thermal convection is decided by the Grashof number and the Prandtl number. The boundary conditions of the openings need the length of the ducts which is assumed to be set outside the openings. The exponent of the power law have to be set for the boundary layer of the wall surface. The Nusselt number is necessary for the heat transmission through the walls. These parameters are decided by the model test mentioned after.

4. MODEL TEST

The shapes of the experiment model is shown in Fig.1. The representative length, L_0, is 60cm. The experiment model is made of acrylic plastic of 5mm in thickness. Nichrome wire is used for the heater which is covered

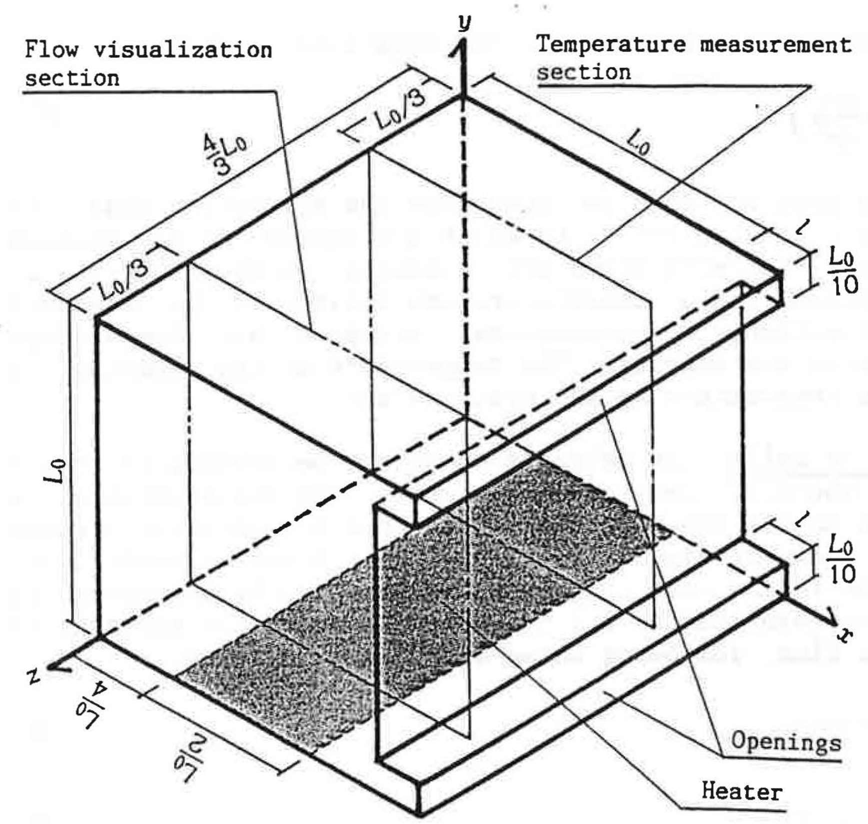


Fig.1 Simulation model

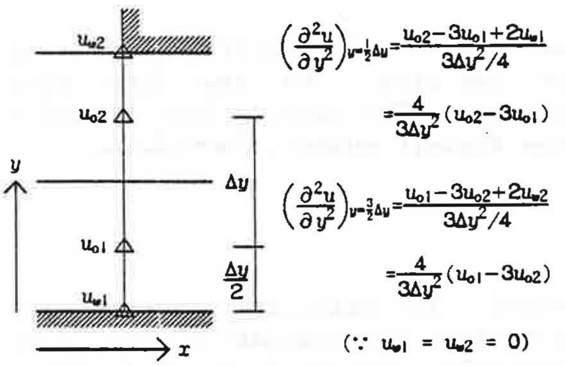


Fig.2 Boundary condition on openings

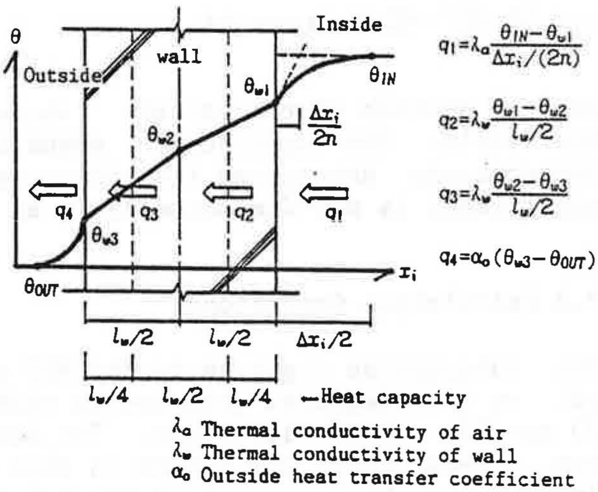


Fig.3 Boundary condition of temperature on walls

Table 1.Thermal Constants for Boundary Conditions of Temperature

Thermal constants	Air	Acryl
Thermal conductivity(J/msK)	0.256×10^{-1}	0.240×10^{0}
Specific heat(J/kgK)	0.100×10^{4}	0.147×10^{4}
Specific gravity(kg/m^3)	0.120×10^{1}	0.120×10^{4}

by copper plate. The surface temperature of the heater was recognized to be uniform before the real experiment. Inside and outside air is free through two openings. The objective natural ventilation is only caused by buoyancy. The whole experiment system is covered by outer booth to avoid the influence of the air movement around the model.

The experiment consists of the visualization of air flow in the model and the measurement of temperature distribution. The air flow is visualized by smoke tracer which is supplied at the speed of almost naught. The visualized space is a vertical section as shown in Fig.1. The section is cut out by a light sheet which is made by a slide projector.

The vertical section of temperature measurement is also shown in Fig.1. The measurement points are set on a grid system which divide the section into 10 equally in each axis. The temperatures are measured by C-C thermo couples of 0.1mm in diameter.

The Plandtl number of the experiment fluid is round 0.7. The opening duct length of the experiment model is $0.1L_0$. The exponent of the power law of the boundary layer is assumed 1/4. The Nusselt number of the outside surface of the walls is assumed 200. It is equivalent to the heat transfer coefficient of about $8W/m^2K$. The Grashof number is controlled by the surface temperature of the heater. The experiment is carried out with the Grashof number of 1×10^8. Thermal constants of air and wall material are shown in Table 1.

5. SIMULATION RESULTS

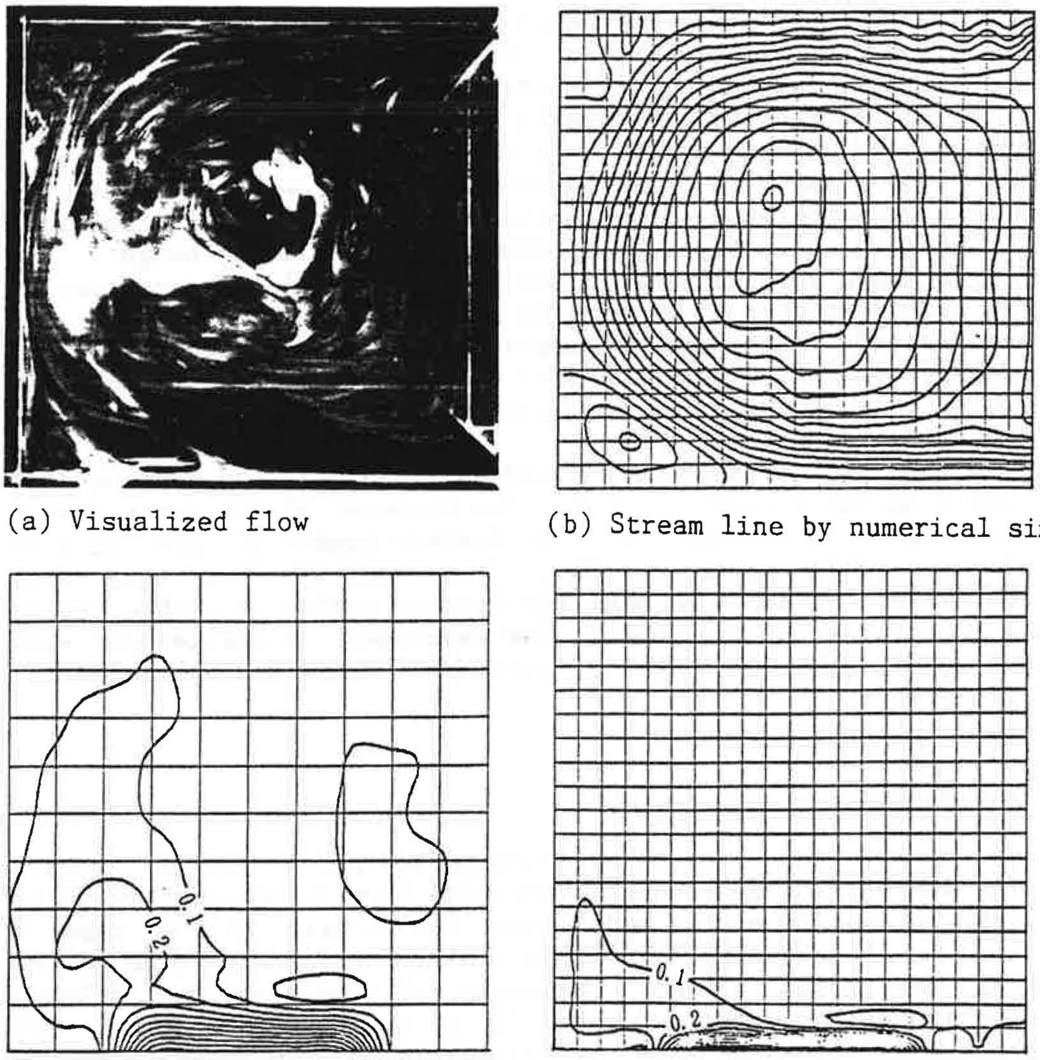
The LES is the time dependent calculation method and it gives instant values of the variables. The simulation results have to be averaged to be compared with the results of the model test. The interval of time step is 1×10^{-4}. It is equal to 2.4×10^{-4} in model scale. The exposure time of the visualization photograph is $2sec$. The number of time steps for average should be equivalent to more than $2sec$. It is 1×10^4 in this case. The time average starts after the air flow is regarded as the steady flow.

The results of the numerical simulation and the model test are shown in Fig.4. The stream lines by the numerical simulation are compared with the visualization photograph. The temperature distributions are compared each other, though the number of the measurement points is different from that of the calculation points.

The stream lines have close resemblance to the visualized flow. The air temperature distributions show a little difference between the numerical simulation and the model test. The difference is thought to be caused by the difference between the measurement and the calculation points, by the averaging time and by the boundary condition of the numerical simulation.

6. CONCLUSIONS

The results of the numerical simulation show good correspondence with those of the model test. It means that the boundary conditions of the openings proposed in this paper is reasonable. However, it is a great problem that the thermal convection parameters simulated here are not so large enough as the indoor air flow in full scale. Even if the whole simulation space is larger and the Grashof number becomes greater, the air flow at the openings is not regarded as full turbulence. The LES is the appropriate method for this case.



(a) Visualized flow (b) Stream line by numerical simulation

(c) temperature by model test (d) Temperature by numerical simulation

Fig.4 Comparison between the model test and the numerical simulation

REFERENCES

(1) Murakami,S., Kato,S.: Numerical and Experimental Study on Room Air Flow - 3-D Predictions Using the $k-\varepsilon$ Turbulence Model, Building and Environment, Vol.24, No.1, pp.85-97, 1989

(2) Zohrabian,A.S., Mokhtarzadeh-Dehghan,M.R., Reynolds,A.J.: A Numerical Study of Buoyancy-Driven Flows of Mass and Energy in a Stairwell, Proceedings of the 9th AIVC Conference, Vol.2, pp.183-204, 1989

(3) Tsutsumi,J-I., Katayama,T., Hayashi,T., Zhang,Q., Yoshimizu,H.: Numerical Simulation of Indoor Turbulent Air Flows Caused by Cross-ventilation and its model experiments, Proceedings of the 9th AIVC Conference, Vol.2, pp.141-156, 1989

(4) Deardorff,J.W.: A Numerical Study o f Three-dimensional Turbulent Channel Flow at Large Reynolds Numbers, J. of Fluid Mechanics, Vol.41, part 2, pp.453-480, 1969

(5) Smagorinsky,J.S.: General Circulation Experiments with the Primitive Equations; Part 1 Basic Experiments, Monthly Weather Review, Vol.91, pp.99-164, 1963