

EXPERIMENTAL DATA AND EVALUATION OF NUMERICAL METHODS FOR NATURAL CONVECTION IN A RECTANGULAR ENCLOSURE WITH A UNIFORMLY HEATED BOTTOM SURFACE

Junji Onishi¹, Hiroshi Ikegami², Nobuyoshi Otsuka³, Seiichi Ito² and Minoru Mizuno⁴

¹ Department of Intelligent Machine Engineering, Osaka Electro-Communication University, 18-8 Hatsu-Cho, Neyagawa, Osaka Japan, ² Osaka Gas Co. Ltd., ³ Mitsubishi Electric Corp.,

⁴ Department of Environmental Engineering, Osaka University

ABSTRACT

Flow and temperature distributions were measured for natural convection in a rectangular enclosure filled with water. In order to express a scale model of a residential room with a floor heating system, one of the vertical walls of the test space was cooled to keep the surface temperature almost constant and the bottom wall surface was uniformly heated. Experiments were performed at the Rayleigh number $1.9 \times 10^{10} \sim 3.8 \times 10^{10}$. PTV (Particle Tracking Velocimetry) method was applied to measure very slow flow fields, and thermocouple device was employed for temperature field measurements. At near wall regions, detailed data were obtained mainly in a center vertical plane under several heating conditions.

Measured data are compared with the calculated results. In the calculations, several turbulence models were compared such as standard k- model with wall functions, two layer model and low Reynolds number k- models. None of the models give satisfactory results, however, the results of two-layer model are more acceptable than those of other models.

INTRODUCTION

Natural convections play important roles in the indoor environment formations such as in panel heating systems or in various passive solar houses etc. Recently CFD simulations have been utilized widely for the investigations of indoor air environments. However, their quantitative validations are not yet satisfactory, especially for natural convections in enclosures. Main reasons why the reliability of CFD simulations is not yet satisfactory are as follows. Firstly, natural convection flows in large, three dimensional (3-D) spaces usually show complicated flow and turbulence properties, so available turbulence models based upon high Reynolds number flows may not work well for the quantitatively precise predictions. Secondly, although many researchers reported detailed experiments of natural convections, they were limited to two dimensional cases [1] or treated the cases in which only vertical walls were heated or cooled [2]. However, natural convection of 3-D rooms with a heated bottom wall (floor) are important from the practical viewpoint and reliable experimental data for those cases are not available or may be very few [3].

In the present study, flow and temperature fields were measured for natural convection caused in 3-D test space filled with water. In order to realize a residential room with a floor panel heating system, the bottom wall surface of the test room was uniformly heated and one of the vertical walls was cooled to keep the surface temperature almost constant. In order to ensure turbulent flow fields, all experiments were performed at sufficiently high Rayleigh number, however, turbulent fluctuations were not measured and only time averaged quantity was measured both in flow and temperature fields in this study.

Flow fields of natural convection in 3-D rooms are complicated ones and have some peculiar turbulent properties. Although the flow fields are mainly induced by a temperature difference between wall surface and room air, the temperature difference is at most 20~40 °C, then very low air movements occur throughout the room. As for the results of a scale model experiments performed in this study, thin boundary layers are formed along the heated bottom wall and

the cold vertical wall, and greater part of the space is occupied with a core region in which the temperature is almost uniform and no flow occurs. However, the core region is not in perfectly stationary state and some small and gentle motions are observed. In short, the region of fully developed turbulent flow (isotropic turbulence) does not exist except the thin area in the boundary layers and greater part of it is occupied with the core region showing unclear properties.

Available turbulence models based upon high Reynolds number flow seem to be inappropriate to predict such unique flow fields numerically, that is, large eddy simulation (LES) or direct numerical simulation of Navier-Stokes equation (DNS) should be preferable. However, since room thermal environments are essentially transient phenomena, long term unsteady calculations are often required to evaluate, for instance, the efficiency of heating or cooling systems. LES may not be applicable in such calculations, for unrealistic computer efforts are required. Consequently, many researchers or engineers are obliged to apply standard k- model or, at most, its modified versions from the practical point of view.

Some numerical studies have been presented on the simulations of natural convection in enclosures [4],[5], however, most of them were treating 2-D calculations and comparing with the experimental results performed by Cheeseright [1] in which detailed temperature and flow fields were measured for 2-D space with a vertical hot surface and a cold surface facing each other. In those researches, standard k- model and some types of low Reynolds number models were used. Chen[5] compared some turbulence models including two-layer model [6] and reported that two layer model is most preferable in the calculation of heat transfer. These turbulent models have been investigated by many researchers, however, examples applied to 3-D spaces are very few especially to spaces with a heated bottom surface. Ozoe et al. [7] calculated 3-D natural convection in a cubic enclosure with the bottom surface heated and obtained interesting results, however, the quantitative validation was not sufficient.

In this study, some turbulence models were assessed through comparisons of calculated results with

experimental data. The models were limited to the k-two-equation model and its modified versions, because the aim of the present study is to evaluate the accuracy of practically applicable models to long term unsteady simulations [8]. A CFD code 'SCIENCE' developed by authors [9] was used for all calculations.

EXPERIMENT

Experimental setup

Schematic of experimental apparatus is shown in Fig.1. As a test space, $L \times L \times L$ cubic chamber was used, where $L=400\text{mm}$. It is filled with pure water up to the top of a slit which is set up at the center ($y=0.5L$) of the upper wall (No.6). The bottom wall of the test space (wall No.5) is made of 15mm thick resin board (bakelite). At the inner surface of it, thin stainless steel sheets, the thickness of which is 50 μm , are pasted. The sheets are connected each other in a series. Then, uniform heat flux conditions are realized by supplying electric power to the stainless sheets directly. A 50mm thick insulating material is inserted under the resin board.

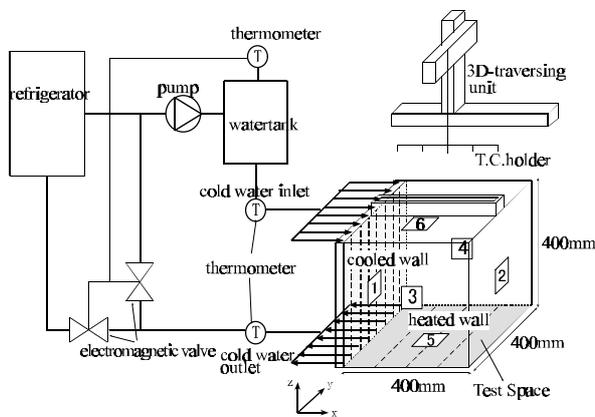


Fig.1 Experimental setup.

The cold wall (wall No.1) is a kind of water jacket made of thick copper plate having eight vertical channels in the back side of it. The cooled water is supplied to the each channel separately through a brass tube which has a small valve to adjust the flow rate of the supply water to keep the inner surface temperature uniform. In order to lower the large temperature fluctuation of discharged water from the refrigerator, a water tank (capacity tank) and two electromagnetic valves are installed in the cold water circuit. This realized the temperature fluctuation of the inlet water within $\pm 0.1\text{C}^\circ$. The other walls (wall No. 2,3,4 and 6) are made of 15mm thick acrylic boards.

Measurements of temperature fields

The surface temperature of the bottom wall was measured inserting 0.1mm diameter thermocouples (T.C.) on the back of the stainless heater. To measure the cold surface temperature, small holes were dug from the back side surface of the cold wall to near the inner surface, then, 0.1mm diameter thermocouples were put into the holes. Fig. 2 gives examples of measured data of these surface temperatures.

As a reference, temperature data at the vicinity of both the hot surface (about 1mm from the hot surface) and the cold surface (about 0.5mm from the cold surface) are also shown. All data were measured using a data logger the sampling interval of which was set two second. The temperature fluctuation near the bottom surface is fairly large, while that of near the cold surface is relatively small and almost disappears at the cold surface. Because of the small heat

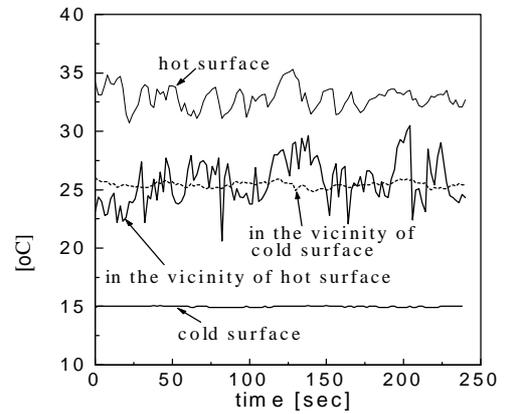


Fig.2 Surface temperatures and water temperatures of their vicinity.

capacity of the stainless heater, the surface temperature is affected by the water temperature fluctuation. This large temperature fluctuation near the bottom surface may be a unique characteristic of the natural convection caused by a horizontal heated plate in a closed space.

Since the distributions of the inner surface temperature of other walls (wall No. 2,3,4 and 6) were almost uniform, they were measured at only one point, i.e. at the center of each wall using a surface type T.C. together with the outer surface temperature at the corresponding positions. For the temperature field measurement, two different methods were used in this study. A preliminary experiment revealed that a large recirculating flow occurred in the test space, however, the boundary layer along walls were very thin and a greater part of the space was occupied by a nearly stagnant core region. A T.C. holder that has eight T.C. junctions was used to measure temperature distributions of the core region with a 3-D traversing unit that is capable of setting positions with the accuracy of $\pm 0.1\text{mm}$. For the measurement of near wall regions, a device shown in Fig.3 was used to determine the position of T.C. junction exactly. The device is moved slowly downward to the surface. At first, the metal detector made of brass reaches the surface, then, the metal cylinder is moved down by a precise positioning device like a micrometer. When the tip of the metal cylinder touches the metal detector, the position of T.C. junction is determined. This method enabled to determine the distance of T.C. from the wall surface with the accuracy of $\pm 0.03\text{mm}$.

Measurements of Flow Fields

An image processing technique was applied to measure the flow fields. In the test space shown in Fig.1, large counter clockwise recirculation occurred in the x - z plane and flows in the central vertical plane ($y=0.5L$) may be regarded as two dimension. In this study, only flow field in the central plane was measured. It was visualized with tracer particles and a laser light sheet. 5W laser light sources were used to light up the vertical cross section. The diameter of the tracer particles is 75~150 μm and its specific weight is 1.01. The visualized flow pattern was recorded on a VTR with a CCD video camera and was analyzed with an image

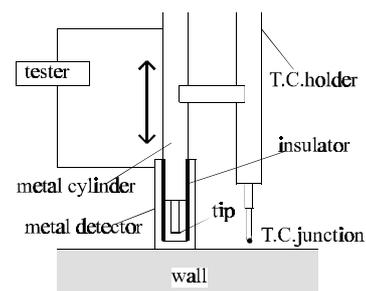


Fig.3 Device to determine the position of wall surface

processing technique, PTV (Particle Tracking Velocimetry). The algorithm of the PTV is based upon a method of tracing the gravity center of several tracer particles on four flames sequentially recorded at the same time intervals of 1/30 sec. Since the boundary layer along walls are very thin, only a few tracer particles were observed in it. Therefore, in order to increase the screen image resolution, regions along the hot or cold surface were divided into four areas in the image taking respectively. However, only 20~50 velocity vectors were determined per one data sampling process in each area. Therefore, the sampling process were repeated about fifty times for the same image taking area, then, the determined velocity vectors were superimposed in it. Consequently, about 1000~2000 instantaneous velocity vectors were obtained in the image taking area.

Experimental conditions

Experiments are classified according as the heated area of the hot wall surface. In this study, experiments for five different conditions as is shown in Table 1 are carried out. In case1-1~case1-3, the whole area of the bottom surface is heated, while a half area ($0 < x < L/2$) is heated in case2-1 and the rest area ($L/2 < x < L$) is heated in case2-2. Q_H is electric power input supplied from an apparatus which is able to keep supplying voltage constant. Then, the power input is kept constant exactly throughout the experiment. It required about 24 hours that the thermal conditions became steady in all cases. The uniformity of the local heat flux q_H at the heater surface was confirmed through the preliminary experiment.

As the experiments were carried out in an air-conditioned room, air temperature surrounding the test space were kept in the range of 20 to 25 °C. This guaranteed the small temperature difference between the test space and the surroundings although the thermal insulation of walls is not sufficient. For all experimental cases, the ratio of the heat loss through each wall to the power input Q_H was less than 1%, namely, the amount of heat supplied at the bottom wall surface is nearly equal to that removed at the cold surface.

The major parameter that determine the structure of natural convection in enclosures is the Rayleigh number and is defined

$$Ra = \frac{g b \Delta q L^3}{\nu \alpha} \quad (1)$$

Here, g is gravitational acceleration, β is coefficient of volume expansion, ν is kinematic viscosity, and α is thermal diffusivity. The reference temperature T_c is defined as

$$\Delta q = q_{H,avg} - q_{c,avg}$$

where, $q_{H,avg}$ is the averaged value of heated surface

constant temperature is established fairly well at the cold surface.

is derived using $T_{c,avg}$ and $T_{H,avg}$ for all cases, then, Rayleigh numbers in the present study are determined. For case 1-1, case 1-2, case 1-3, Rayleigh numbers were 1.9×10^{10} ($T_{c,avg}=13.0^\circ\text{C}$, $T_{H,avg}=26.4^\circ\text{C}$, $\Delta T=13.4^\circ\text{C}$), 3.0×10^{10} ($T_{c,avg}=15.2^\circ\text{C}$, $T_{H,avg}=36.7^\circ\text{C}$, $\Delta T=21.5^\circ\text{C}$) and 3.8×10^{10} ($T_{c,avg}=17.3^\circ\text{C}$, $T_{H,avg}=44.4^\circ\text{C}$, $\Delta T=27.1^\circ\text{C}$), respectively. In all cases, Ra numbers may be large enough to realize an actual room conditions using the test space shown in Fig.1.

CALCULATION

Turbulence model

In this study, three turbulence models, () standard type k- model, () low Reynolds number k- model and () two layer k- model, were compared with measured data obtained in this study. They are introduced briefly below.

Standard k- Model

The standard type k- model (ST model) is widely used in the calculations of airflows and thermal environments in rooms. Usually it is applied with a wall function (WF) bridging the laminar sub-layer in near wall regions. However, as is shown later, the boundary layers are very thin, therefore, calculating grids should be fine enough to be able to resolve the thin layer including the laminar sub-layer. Therefore, a wall function applicable to the laminar sub-layer is employed in the present study. The eddy viscosity ν_t is defined as

$$\nu_t = C_m \frac{r k^2}{e} \quad (2)$$

The turbulent energy k and its dissipation rate e are calculated from transport equations of k and e respectively.

The wall function used with ST model is as follows [10];

For velocity boundary layer,

$$\left. \begin{aligned} u^+ &= y^+ & (0 \leq y^+ \leq 11.63) \\ u^+ &= \frac{1}{k} \ln(9.78 y^+) & (11.63 \leq y^+) \end{aligned} \right\} \quad (3)$$

For temperature boundary layer,

$$8.75 \frac{\nu}{y^+} \frac{d\theta^+}{dy^+} = 1 \quad (1/4)$$

temperatures T_H measured at 16 points and $T_{c,avg}$ is that of the cold surface temperature T_c . As is mentioned in §2.2, all instantaneous temperatures are measured at the sampling intervals of 2 sec. In order to obtain T_c and T_H , 900 instantaneous values (corresponding to the data during 1800 sec) are averaged at all measuring points. T_c was measured at 28 points and $|T_{c,avg} - T_c|$ was within 0.3°C through all experimental cases, namely, the condition of

f in eq.(9) is a function of turbulent Reynolds number R_t and expresses the damping effect in the near wall viscosity-affected regions. f_1, f_2 are also functions considering viscosity effect in near wall regions. Although various models have been proposed about these functions, the advantage or disadvantage among them is not confirmed well, then, in the following calculations, Launder-Sharma model [11] is employed in which E of eq.(6) is set 0 because of its unclear physical basis. In the model, $f_1=1.0$, and f_2, f are expressed by

$$f_2 = 1 - 0.3 \exp(-R_t^2) \quad (7)$$

$$f_m = \exp\left(\frac{-3.4}{1 + R_t/50}\right) \quad (8)$$

where, R_t is defined as $R_t = k^2 / (\nu \epsilon)$.

Two Layer Model

In order to obtain reasonable solutions, LRN model usually requires high numerical resolution near wall regions, therefore, computer efforts often increase to be beyond the performance of available computers. In order to save necessary computing grids, 2-Layer $k-\epsilon$ model (TL model) was introduced by Rodi [6]. In the TL model, near wall region is divided into two layers according as the turbulence level, one is inner layer (very near wall regions) where viscosity effect is dominant and the other is outer layer (regions away from the wall) in which high Reynolds number flow is dominant and ST model is applied. In the inner layer, transport equation of the dissipation rate ϵ is not solved and the eddy viscosity ν_t and ν_m is calculated with following simple relations.

$$\nu_t = C_m k^{1/2} l_m \quad (9)$$

$$\epsilon = \frac{k^{3/2}}{l_e} \quad (10)$$

As for the length scale l, l_m, l_e , some variations are available. In this study, a model given by Norris and Reynolds [12] and by Wolfstein [13] are investigated. The Norris and Reynolds model is expressed as

$$l_m = C_l y_n \left[1 - \exp\left(-\frac{R_k}{A_m}\right) \right] \quad (11)$$

$$l_e = \frac{C_l y_n}{1 + 5.3/R_k} \quad (12)$$

and $A = 50.5$ is used. In the Wolfstein model, l_m and l_e are expressed in the same form as

$$l_m = C_l y_n \left[1 - \exp\left(-\frac{R_k}{A_m}\right) \right] \quad (13)$$

$$l_e = C_l y_n \left[1 - \exp\left(-\frac{R_k}{A_e}\right) \right] \quad (14)$$

A_m and A_e are entirely empirical constants and their optimum values may be different in different phenomena. Patel et al. adopted $A_m = 70, A_e = 5.08$ and Launder et al. adopted $A_m = 62.5, A_e = 3.8$ respectively[6]. Turbulent Reynolds number R_k and a constant C_l is given as

$$R_k = \frac{k^{1/2} y_n}{\nu_t}, \quad C_l = k C_m^{-3/4} \quad (15)$$

where, ν_t is kinetic viscosity and y_n is a distance form the nearest wall, $C_m = 0.09$ and $C_m = 0.42$.

Calculating Procedure

In order to resolve thin boundary layers, minimum mesh size near wall grid point should be small enough, however, in the core regions much coarser grids may be applicable, therefore, non-uniform mesh systems are used in all calculations. The minimum mesh size is different from each

turbulent model and is taken to be 0.05mm ($x/L, z/L=1.25 \times 10^{-4}$) ~ 0.2 mm ($x/L, z/L=5 \times 10^{-4}$) both for the cold wall and for the hot wall. The maximum mesh size is set as 20mm ($x/L, z/L=0.05$). Basic equations are discretized and solved using SIMPLE algorithm with power law scheme. As the boundary conditions of temperature, a constant temperature T_c is given at the cold surface ($x=0$) and uniform heat flux $q_H = Q_H/L^2$ is given at the bottom surface ($z=0$). To determine the surface temperature T_H at the bottom surface, and to evaluate heat flux $q_{w,c}$ at the cold surface, temperature gradients

$$\left. \frac{\partial q}{\partial z} \right|_{z=0}, \quad \left. \frac{\partial q}{\partial x} \right|_{x=0} \quad (22)$$

are calculated in the accuracy of 3rd order. Although steady calculations are performed in all cases, numerical instability often occurs in converging process and convergence criteria are not satisfied automatically because of the fluctuation of the residuals. Then, calculation is terminated after 1000 more iterations if following conditions are satisfied,

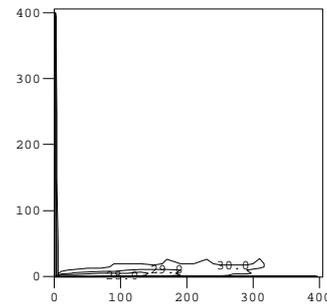
() Residuals of all variables are small enough.

() Over all heat balance is $0.99 < Q_C/Q_H < 1.01$.

here, Q_C and Q_H are the sum of q_C and q_H at each surface respectively. Calculated values of the last 1000 iterations are averaged for all variables in order to avoid the difference arise from the timing to terminate calculations.

RESULTS AND DISCUSSIONS

Some measured results were compared with calculated results under the conditions that Q_H is 1000 W and $T_{c,avg}$ is 15.2 respectively which correspond to the conditions of case 1-2. Although all other cases were also compared with calculated results and both quantitatively and qualitatively similar results were obtained, they were not introduced in this



paper because of the limitation of the paper length.

ST model

At first, the effect of the minimum mesh size was examined varying it from 1.25 mm to 0.2mm. From the results, it was proved that the dependence of calculated results on the mesh sizes was diminished in case of the minimum mesh size smaller than 0.4mm. Then, all calculations with ST model were performed with the minimum mesh size of 0.2mm.

(a) Experiment .

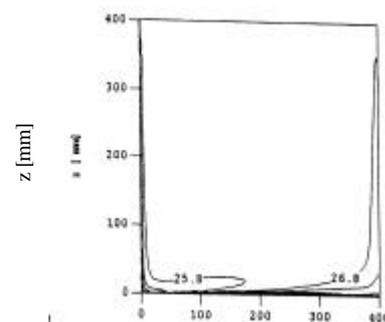


Fig.5 Comparison of temperature contours at

Comparisons of temperature and flow fields in the central vertical plane are shown in Fig.5 and Fig.6. As is shown in Fig.5, calculated results give qualitatively similar tendency in the temperature field except the corner regions, however, temperature level is lower in the calculation than in the experiment. This is because temperature gradient at $x/L=0$ surface may be overestimated in the calculation. Fig.6 is results for flow fields in the center vertical plane. Fairly

along the cold surface agreed well with measured profiles, however, we did not have good predictions for other cases except the results for velocity profiles at the upper regions ($z/L = 0.875$) in the cold wall boundary layer (This data is not included in this report).

TL model

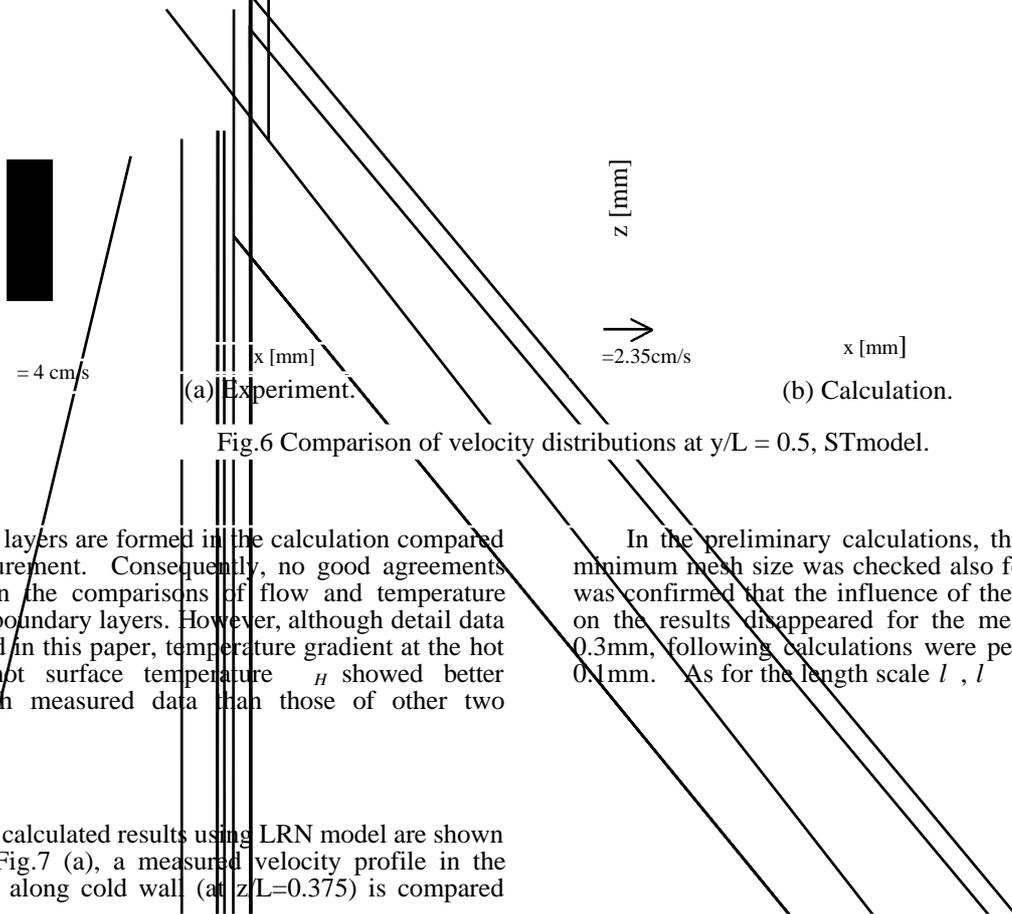


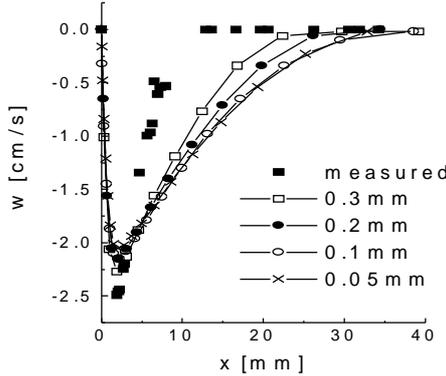
Fig.6 Comparison of velocity distributions at $y/L = 0.5$, STmodel.

thick boundary layers are formed in the calculation compared with the measurement. Consequently, no good agreements are obtained in the comparisons of flow and temperature profiles in the boundary layers. However, although detail data is not presented in this paper, temperature gradient at the hot surface and hot surface temperature T_H showed better agreement with measured data than those of other two models.

In the preliminary calculations, the dependence on the minimum mesh size was checked also for the TL model. It was confirmed that the influence of the minimum mesh size on the results disappeared for the mesh size smaller than 0.3mm, following calculations were performed setting it as 0.1mm. As for the length scale l, l' , TL mode24

LRN model

Examples of calculated results using LRN model are shown in Fig.7. In Fig.7 (a), a measured velocity profile in the boundary layer along cold wall (at $z/L=0.375$) is compared



with the calculated data and in Fig.7 (b), a measured temperature profile is compared with the calculate data in the boundary layer along hot wall ($x/L=0.375$). As a parameter, minimum mesh size is chosen in these calculations. The affect of the minimum mesh size on calculated results seems to disappear for the mesh size smaller than 0.1mm, in other words, the velocity profile near the cold wall should be 0.1mm or less. This means that at least 5 to 7 meshes should be placed in a near wall layer since non-uniform mesh system is adopted in this study. In the LRN model calculations, only temperature profiles in the boundary layer

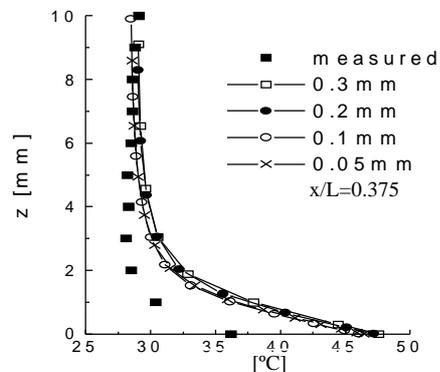
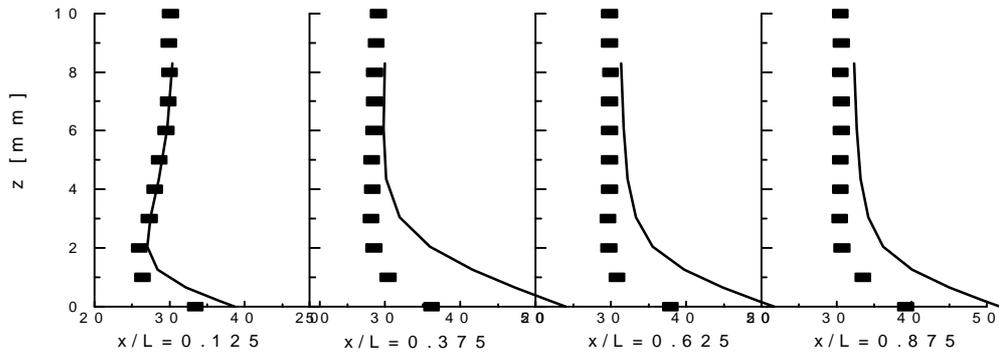


Fig.7 (b) Temperature profile near the hot wall, LRN model.



along the cold wall, while the accuracy of temperature profiles near the hot wall is poor especially at very near wall regions as is shown in Fig.8

Evaluations of each model are summarized in Table 2 separately according as variables and as locations. The symbols imply the levels of the accuracy of calculations, namely, \square is very good, \square is good, \square is a little poor and \times is poor. ST+TL implies a test case in which ST model is used for the calculation near the hot wall region and TL model

is used for that near the cold wall region. Either LRN model or TL model worked well for the calculations near the cold wall region. This may be explained that the turbulence structure near the hot wall essentially differ from that near the cold wall, therefore, Van Driest's mixing length hypothesis which both LRN model and TL model are based upon does not hold there. The reason why ST+TL model gives best results may be related to the fact that is described in 'ST model', however, physical basis for the reason is not clear.

Table 2 Evaluation of each model.

	cold-	cold-v	hot-	hot-v
ST	\times			
LRN				
TL				
ST+TL				

Fig.8 Comparison of temperature distributions near the hot wall, TL model.

CONCLUDING REMARKS

Flow and temperature distributions were measured for natural convection in a rectangular enclosure filled with water. To realize a scale model of a residential room with a floor heating system, one of the vertical walls of the test space was cooled and the bottom wall surface was uniformly heated. Experiments were performed at the Rayleigh number $1.9 \times 10^{10} \sim 3.8 \times 10^{10}$. Detailed data were obtained mainly in a center vertical plane under several heating conditions. Although turbulent components were not measured in this study, some reliable data were obtained and the data may be useful to validate available numerical methods.

In the evaluation of calculation methods, three types of modified k- models were examined. In order to determine the required minimum mesh sizes near wall layers, the dependence of calculated results on them were examined through a parameter study. All calculations were performed using the determined minimum mesh sizes with which practically applicable minimum mesh systems were realized. Both LRN model and TL model showed good results for the flow and temperature fields along the cold wall, however, they didn't work well for those along the hot wall. This is explained that the turbulence structure near the hot wall essentially differ from that near the cold wall, therefore, Van

Driest's mixing length hypothesis which both LRN model and TL model are based upon does not hold there. In order to develop a turbulent model applicable to calculating such unique flow field, detailed data including turbulence quantities should be prepared. This may be the area of our future study.

NOMENCLATURE

Q_H : total heat input of hot wall	[W]
q_H : local heat flux at hot wall surface	[W/m ²]
u : x-directional flow velocity	[cm/s]
w : z- directional flow velocity	[cm/s]
c_c : local surface temperature of cold surface	[°C]
H_c : local surface temperature of hot surface	[°C]
$c_{c,avg}$: wall averaged value of measured c_c	[°C]
$H_{c,avg}$: wall averaged value of measured H_c	[°C]

REFERENCES

- Cheesewright R. King K.J. and Zial S., Experimental Data for the Validation of Computer Codes for the Prediction of Two-Dimensional Buoyant Cavity Flows, in 'Significant Questions in Buoyancy Affected Enclosure or Cavity Flows, ed. J.A.C.Humphrey et al., ASME (1986), pp.75-81,1986.
- Hiller W. J., Koch S. and Kowalewski T. A., Tree-Dimensional Structures in Laminar Natural Convection in a Cube enclosure, Vol.2 (1989), pp.34-
- Cooper L., Air movement and stratification in Rooms with Localized Heating of a Wall or Floor, Proc. 5 th Int. Conf. on Air Distribution in Rooms, Vo.2 (1996), pp.373-380.
- Henkes,R.A.W.M. and Hoogendoorn,C.J., Natural- Convection Flow in a Square Cavity Calculated with Low-Reynolds-Number Turbulence Models, Int. J. of Heat and Mass Transfer, Vol.34, No.2, (1991),pp.377-388.
- Chen, Q., Comparison of Different k- Models for Indoor Air Flow Computations, Numerical Heat Transfer, Vol.28, Part B, (1995), pp.353-369.
- Rodi,W., Experience with two-layer models combining the k- model with a one equation model near the wall., 29th Aerospace Sciences Meeting, AIAA-91-0216",(1991), Reno, Nevada.
- Ozoe H., Mouri A. et al., Numerical Calculutions of Three-Dimensional Turbulent Natural Convection in a Cubical Enclosure Using a Two-Equation Model for turbulence, Trans. of the ASME, Vol. 108, Journal of Heat Transfer, 1986, pp.806-813.
- Onishi J., Soeda H. and Mizuno M., Numerical Study on a Low Energy Architecture Based upon Distributed Heat Storage System, Proc. World Renewable Energy Congress'99 , (1999),pp.83-86, Perth, Australia,
- Onishi, J.,Takeya, N. and Mizuno, M., Study on Numerical Prediction Methods for Indoor Air-Flow and Thermal

Environments(Part 1). Transactions of SHASEJ, No.55 (1995), pp.23-34 (in Japanese).

10. Patankar,S.V. and Spalding,D.B., Heat and Mass Transfer in Boundary Layers, (1970), Intertext Books,London.
- 11 Launder,B.E. and Sharma,B.I., Application of the Energy-Dissipation Model of Turbulence to the Calculation of Flow Near a Spinning Disc, Lett. Heat Mass Transfer, Vol.1, (1974),pp.131-138.

12. Norris L.H. and Reynolds W.C., Turbulent channel flow with a moving wavy boundary, Rept. No. FM-10, Stanford University, Dept. Mech. Eng., 1975.
13. Chen H.C. and Patel V.C., Near-Wall Turbulence Models for Complex Flows Including Separation, AIAA Journal, Vol.26, 1988, pp.641-648.

