Prediction of Heat Emission Effect at Small Single-sided Openings in Apartment Houses

Ayako Nishimura[1], Masashi Imano[2], Kei Ogino[3],
Keigo Nomura[3],Yuzo Sakamoto[4]

[1] The Tokyo Electric Power Company, M. Eng.
[2] Assist. Prof., Grad. School of Eng., The Univ. of Tokyo, Dr. Eng.
[3] Graduate Student, Grad. School of Eng., The Univ. of Tokyo.
[4] Prof., Grad. School of Eng., The Univ. of Tokyo, Dr. Eng.

Abstract
In this study, we propose a method for forecasting heat emission effect in a small single-sided slit opening. In order to make a heat flux forecasting model, we carried out field measurement, wind tunnel experiment, and CFD analysis. In the field measurement, we measured heat flux and grasped the relationship of heat flux of the opening and airflow around the building surface. In the wind tunnel experiment, we measured airflow around the opening and grasped turbulence characteristics. In the CFD analysis, we used RANS model of high Reynolds numbers type and considered optimum turbulence model by comparing with the results of wind tunnel experiment. Finally, we used standard k-ε model under non-isothermal conditions and calculated the coefficients of the heat flux forecasting model from the relationship of heat emission quantity at the opening and airflow around the building surface by CFD analysis.

Keywords: apartment houses, single-sided openings, natural ventilation, heat emission, heat flux, field measurement, wind tunnel experiment, CFD, RANS model

1. Introduction

Recently, an expectation of heat emission effect by natural ventilation has risen in terms of energy conservation and comfortableness. However, openings in apartment houses can be installed only on one side wall, and ventilation system using the back and forth differential pressure cannot often be used. Nevertheless, the quantitative findings to estimate the effect under adverse circumstances are few. Warren et al insisted that the ventilation in the room with a single-sided opening mainly resulted in turbulent mixing layer between the indoor stationary air and the wind flowing parallel to the wall, and the ventilation quantity was proportional to the wind velocity around the building's surface. In addition, Kono et al proposed the method to estimate ventilation flow from wind velocity parallel to the wall. It is assumed that turbulent heat flux much contributes to heat transport of a single-sided opening. Therefore, it is necessary to grasp heat emission quantity including turbulent heat flux and solve the heat emission mechanism in order to estimate heat emission effect in the room with a single-sided opening. The purpose of this study is to grasp heat emission quantity of a small single-sided opening especially a slit opening (about 20 cm width) considering crime prevention, and to construct a method for forecasting heat emission effect.

2. Forecasting process of heat emission effect in a single-sided opening

We propose a forecasting process shown in Figure 1 by applying ventilation quantity forecasting method of Kono et al. To practice this method, we examined two matters; the arrangement of CFD analysis condition and making heat flux forecasting model.

The former have been already reported. We report the result of examination about heat flux forecasting model and trial calculation of heat emission effect forecasting process in this report.

3. Measurement of heat flux by field measurement

3.1 Definition of heat flux

Heat flux divided by the heat capacity of air is defined as follows.

Effective heat flux $\overline{u\,\theta}$ (Equation 1): heat transportation moving from inside to outside through an opening It is equal to heat loss per unit area from an opening.

Non-turbulent heat flux $U\Theta$ (Equation 2): heat transportation by mean flow, which is separated into inflow and outflow.

Turbulent heat flux $\overline{u'\theta'}$(Equation 3): It is calculated by subtracting non-turbulent heat flux from effective heat flux. It is heat transportation by turbulent correlation of wind and temperature.

$$\overline{u\theta} = \sum (u_t \cdot \theta_t) \cdot \Delta t / t \qquad \text{(Eq. 1)}$$

$$U\Theta = U_{in}\Theta_{in} + U_{out}\Theta_{out} \qquad \text{(Eq. 2)}$$

$$\overline{u'\theta'} = \overline{u\theta} - U\Theta \qquad \text{(Eq. 3)}$$

3.2 Outline of field measurement

We measured in a room of the third floor of a four-story RC manufacturing apartment houses in Shinjuku, Tokyo after sunset on 6 December 2009. Beforehand, we heated and stirred the air in the room with a heater so that inside temperature is about 1.5 degree higher than outside, for we assumed late summer. We started to measure an hour after stopping the heater. The opening is a slit opening considering crime prevention. It slides outside lengthwise and works as a wind catcher as well. We placed three-dimensional sonic anemometers 50 cm apart from the wall to measure wind velocity around the opening (Figure 4). We measured temperature and wind velocity at opening at the same time by the anemometer and thermocouples in order to gain heat flux. We placed the anemometers to six points, and thermocouples to 5 cm inside and outside from center point of measurement of anemometers. When the air flowed in, we used the temperature of outside thermocouples. When the air flew out, we used the temperature of inside thermocouples (Figure 5, 6). Measurement interval of wind velocity was 0.1 seconds, and that of temperature was 2 seconds. We analyzed wind velocity and temperature using linear interpolation per 0.1 seconds.

3.3 Results of field measurement and heat flux model

While measuring, 60 percent of the rooftop wind flowed from N, 40 percent flowed from NNW. Average wind speed was 1.7 m/s. The wind of the surface blew down at 45° along the wall from N. The wind velocity was 0.6 to 1.4 m/s. Figure 7 shows mean airflow quantity that flowed in and out, and wind velocity around the building surface of 10 minutes interval. Ventilation quantity was calculated by the product of each wind velocity at 6 points and window area which was equally divided into 6 parts. Although this study assumes wind-driven heat transfer, buoyancy ventilation also occurs in real experiment. So, we need to confirm the order of buoyancy ventilation. The ratio (C_b) of buoyancy ventilation quantity (Q_b) to total ventilation quantity (Q_t) was 20 to 35 percent. Buoyancy ventilation quantity was calculated as follows.

$$Q_b = \frac{1}{3} A C_d \sqrt{\frac{9.8 \Delta \theta h}{\theta}} \qquad C_b = Q_b / Q_t$$

We calculated heat flux by Equation (1)-(3). For these, we divided heat flux by indoor and outdoor temperature difference ($\Delta \theta$) (Equation (4), (5)). Then, we multiplied (1-Cb) In order to eliminate the influence of buoyancy, ($1-C_b$) was multiplied. Equation (4) is the standard heat flux.

$$K_{mean} = (1 - C_b) \cdot U\Theta / \Delta\theta \quad \text{(Eq. 4)}$$

$$K_{turb} = \overline{u'\theta'} / \Delta\theta \quad \text{(Eq. 5)}$$

Figure 8 shows the mean standard heat flux of 10 minutes interval gained by field measurement. The turbulent heat flux is 20 to 80 percent of the non-turbulent hear flux. However, the data with weak wind velocity of building surface and unstable wind direction were removed.

Figure 9 shows the relationship between the standardized non-turbulent heat flux (K_{mean}) and scalar wind velocity of building surface (U_R). Figure 10 shows the relationship between the ratio of turbulent heat flux to non-turbulent heat flux (K_{turb} / K_{mean}) and the turbulence intensity at the building surface (I_R). They are roughly proportional.

Thus, we gained equations of forecasting model (Equation (6), (7)) from the slope of straight line in Figure 9-10. From these equations, we led Equation (8) which shows the standardized effective heat flux by U_R and I_R. We measured 50 cm apart from the wall. In order to convert to the condition of 1m apart from the wall, we multiplied wind velocity by conversion factor 1.07 which was led by Kono et al.

4. Wind tunnel experiment

4.1 Outline of wind tunnel experiment

The experiment was done in the wind environment simulator laboratory, Faculty of Engineering The University of Tokyo. The model was in 1/4 scale. We set up the opening on the wind tunnel floor, and the chamber below (Figure 11). The size of opening was 50×330, and 45 mm thick. The size of the chamber was 700×600×700 mm (the room of 4 and a half mat was assumed). Figure 12 shows two types of openings, a simple opening and a lengthwise sliding opening which was the reproduction of the shape of sash. We disturbed the airflow in the wind tunnel by using turbulent lattice. Figure 13 shows the profile of wind velocity and turbulence intensity. In case of the lengthwise sliding opening, we changed inflow wind direction to the opening by 0°, 45°, 90°. We measured 3 direction wind velocity elements with a split film (low-pass frequency was 1Hz).

4.2 Results of wind tunnel experiment

Comparison in different shape of opening

Figure 14 shows wind velocity vector at the center section of simple opening and the lengthwise sliding opening in the wind direction 0°. In case of simple opening, a whirlpool occurred in the opening. We observed that the wind was drawn and flew into the chamber at the edge of leeward. The standard airflow quantity at the opening was 0.0076. It is about 1/3 of the result of experiment of Kono et al, 0.024. Standard airflow quantity can be calculated by dividing ventilation quantity by area of opening and inflow wind velocity. Ventilation quantity is the total sum of product of wind velocity toward normal direction and the area at each measure point.

We think this is because thicker thickness of opening is; lesser the turbulent mixture layer grows. In case of lengthwise sliding opening, windward airflow that collided with sliding plate was led to inside. However, most of inflow air cut corners because drawn airflow by leeward circulating airflow was strong. The standard airflow quantity at the inside of the opening was 0.032. It was approximately 1/7 of that at the outside of the opening. The standard airflow quantity of lengthwise sliding opening was approximately 4.2 times of that of simple opening.

Comparison in different inflow wind direction

We measured the wind velocity in the plane of outside with lengthwise sliding opening, changing inflow wind direction at 0°, 45°, 90°. Figure 15 shows the anemometry point of wind velocity, wind velocity of normal direction, and the turbulence energy. In case of 0°, we measured only the upper half, since it's symmetrical. In case of 0° and 45°, the wind flowed in from the windward opening, and flowed out from the leeward opening remarkably. The wind velocity was large, 1 to 1.5 m/s. On the other hand, in case of 90°, the wind flowed in from most of the area of the opening. The wind velocity was small, 0.3 m/s at its most. The turbulent mixture layer grew as the wind went to leeward. In case of 45°, it is assumed that shortcut of the wind will decrease, and substantial ventilation quantity will increase, for flow field isn't symmetrical, different from the case of 0°. Calculation of substantial ventilation quantity by CFD is reported in the next report.

5. CFD analysis

5.1 Outline of CFD analysis

We will calculate the heat flux of opening by CFD analysis in future. Therefore, we inspected the accuracy of calculation of airflow around the opening and compared turbulent models by CFD analysis under non-isothermal condition. Table 2 shows the outline of analysis. We reproduced an area which is centered at the center of the opening of wind tunnel experiment. Its sides of horizontal plane were 1.2m each. We interpolated measurements in wind tunnel experiment and used them as vertical profile of inflow wind. We inspected about three kinds of turbulent models, standard k-ε, RNG k-ε, and Realizable k-ε.

5.2 Results of CFD analysis

Figure 16 shows correlations of results of CFD analysis and wind tunnel experiment of wind velocity toward normal direction in the outside of the opening at each wind direction. Measuring points were shown in Figure 15. The smallest RMSE was at the standard k-ε model. Standard k-ε model accurately reproduced the airflow which collided with sliding plate vertically as in the case of 0°. However, it didn't accurately reproduce the airflow which collided with sliding plate slantingly as in the case of 45°, or in the turbulent mixture layer as the case of 90°. Figure 17 shows correlations of turbulent energy. It shows similar tendency.

6. Heat flux model by CFD analysis

6.1 Outline of CFD analysis

Table 3 shows conditions of temperature field. We assumed uniform heat generation of 20 W/m3 in the chamber. The wall of chamber was assumed to be complete insulation. The inflow wind direction was 0°, 22.5°, 45°, 67.5°, 90° (Figure 18). We reproduced external part (no openings) of the object building and surrounding buildings, and calculated airflow around the building to 16 directions by CFD. Then we gained the wind velocity profile around the building surface. Figure 19 shows this. This profile was weighted by occurrence rate and average was taken.

6.2 Results of CFD analysis

Figure 20 shows airflow quantity that passed through the opening and effective ventilation quantity that was calculated from temperature difference between inside and outside. The airflow quantity that passed through the opening was the largest at 0°, and decreases according to cosine curve. This is assumed that it depends on the vertical projection area of vane. However, the effective ventilation quantity was the largest at 67.5°. This is assumed that the air didn't take a shortcut easily, so flowed inside, and the room was ventilated effectively.

6.3 Calculation of coefficients of heat flux forecasting model

Figure 21 shows the method of calculating coefficients of heat flux forecasting model. Total heat emission quantity is equal to total heat generation in the chamber. Thus, we divided the heat emission quantity by the area of the opening, and calculated effective heat flux. We multiplied mean airflow velocity and mean temperature, and calculated non-turbulent heat flux. We subtracted non-isothermal heat flux from effective heat flux, and gained turbulent heat flux. We calculated coefficients of heat flux forecasting model from the relationship of effective heat flux and U_R, I_R (the height was 25 cm, actual scale was 1 m) (Table 4). These coefficients are used according to wind direction of vertical plane.

7. Conclusions

We proposed the forecasting process of heat emission effect and finally calculated the heat flux forecasting model. We found that standard k- model was the most accurate. In the future, we need to simulate with more accurate model such as LES model, or inspect with more turbulent inflow wind. We also need to examine hereafter the cause that the effective ventilation quantity is the largest at 67.5°

Nomenclature

u instantaneous wind velocity [m/s]

U mean wind velocity [m/s]

θ instantaneous temperature [℃]

Θ mean temperature [℃]

Q airflow quantity [m^3/s]

C_b buoyancy ventilation quantity / all airflow quantity [-]

$\Delta\theta$ indoor and outdoor temperature difference [℃]

I turbulence intensity [m^2/s^2]

A area of the opening [m^2]

k turbulence energy [m^2/s^2]

t time [s]

F heat flux [W/m^2]

K standardized heat flux [m/s]

α, β coefficient of the model [-]

Subscript

t instantaneous value per measuring interval (0.1 seconds)

in inflow

out outflow

b by buoyancy

t total

mean non-turbulence component

turb turbbulence component

eff effective component

R around the building surface

Acknowledgement

The Tokyo Electric Power Company R＆D Center and Hidekazu Tanaka, a postgraduate of Sakamoto laboratory of the University of Tokyo supported the field measurement.

References

1. P.R.Warren, “Ventilation through Openings on One Wall Only, Energy Conservation in Heating, Cooling, and Ventilating Buildings,” Vol.1, 1978, pp.189~206.

2. KONO Ryohei, et al, “Prediction of Airflow Rate of Single-sided Room in Cross Wind: The study on characteristics of ventilation with single-sided opening Part 3,” Journal of Environmental Engineering (Transactions of AIJ), 2008.

3. IMANO Masashi, “Development of a CFD Software for Predicting Cross Ventilation Flow in a House Using Adaptive Mesh Refinement and Super-Parallel Calculation,” Transactions of the Society of Heating, Air-conditioning and Sanitary Engineers of Japan, 2005, pp.1097~1100.

① 【Calculation of airflow around the building surface by CFD analysis】

We reproduce surroundings and externals part of the object building, and calculate airflow around the building surface

Examination 1: Arrangement of CFD analysis condition in order to get appropriate solutions

② 【calculation of standard heat emission quantity】

We substitute U_{CFD} and k_{CFD} that was calculated in 16 wind directions for the heat flux forecasting model that is described later, and calculate standard heat emission quantity per standard wind veloc ity and temperature difference between inside and outside of the room.

Examination 2: Make of heat flux forecasting model

Fig. 1 Forecasting process of heat emission effect

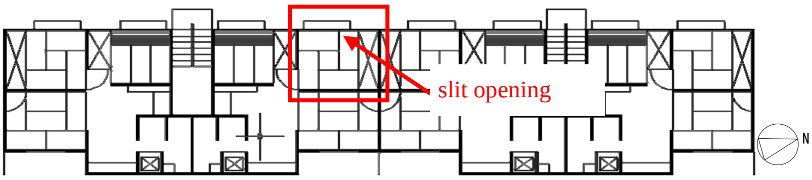


Fig. 2 Object unit (third floor)

Fig. 3 Object building

Fig. 4 Anemometer at the the building surface

Fig. 5 Anemometer and thermocouples at the opening

Fig. 6 Slit opening

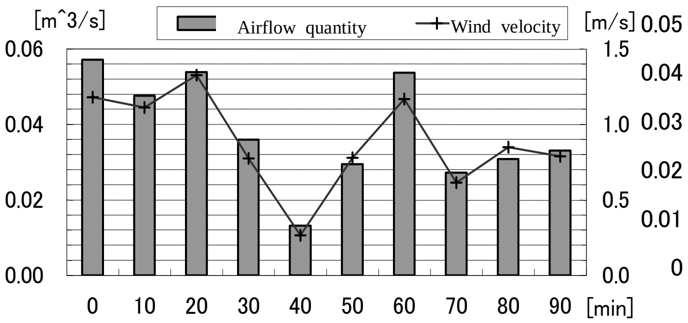


Fig. 7 Wind velocity around the building surface

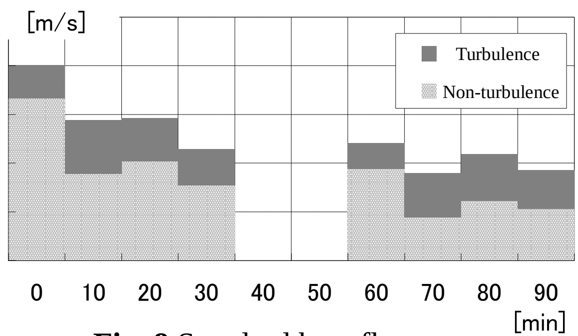


Fig. 8 Standard heat flux

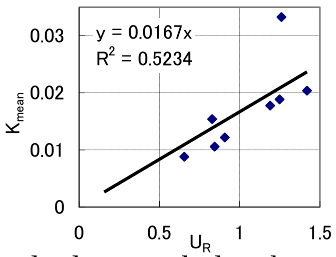


Fig. 9 Standard non-turbulent heat flux and scholar wind velocity around the building surface

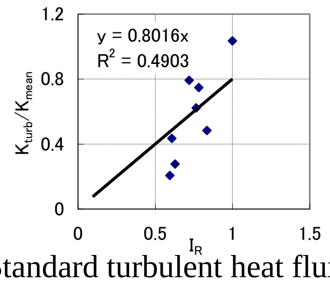


Fig. 10 Standard turbulent heat flux and turbulent energy around the building surface

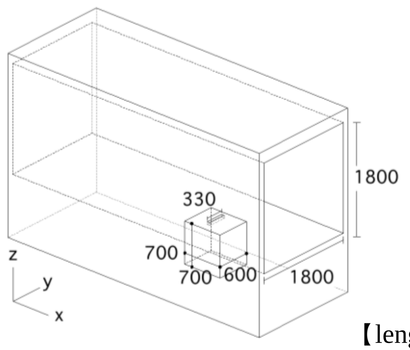


Fig. 11 Wind tunnel model (Unit of length : mm)

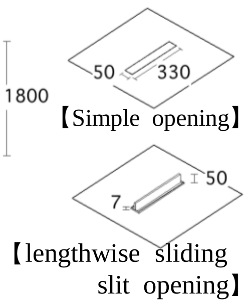


Fig. 12 Opening model (Unit of length : mm)

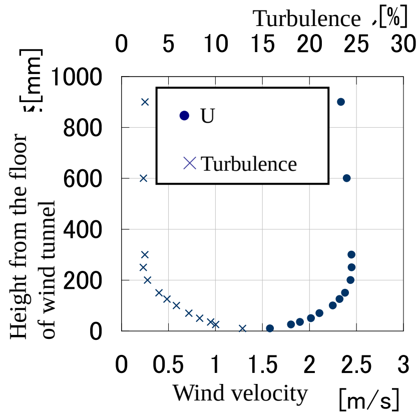


Fig. 13 Wind velocity profile

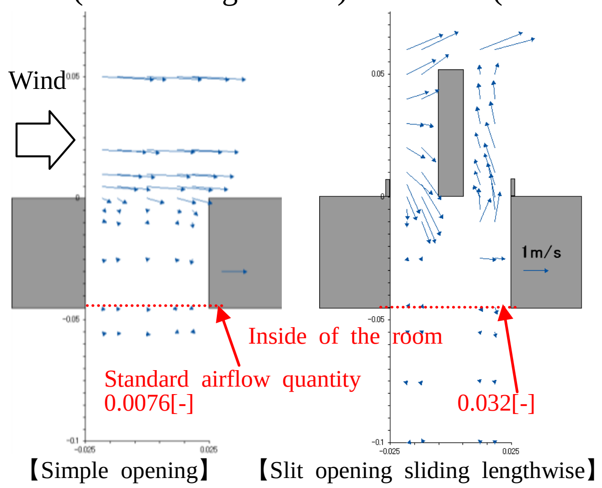


Fig. 14 Wind velocity vector of main wind direction (Inflow was positive at wind velocity.)

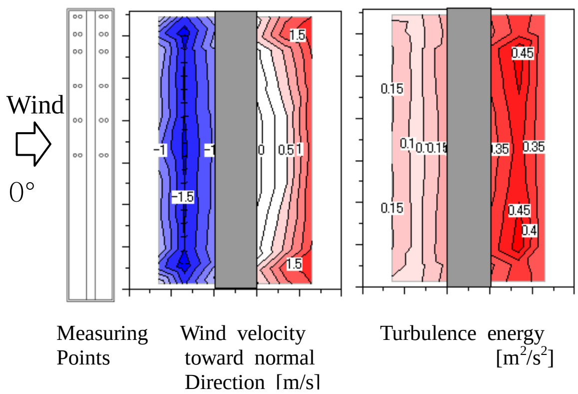
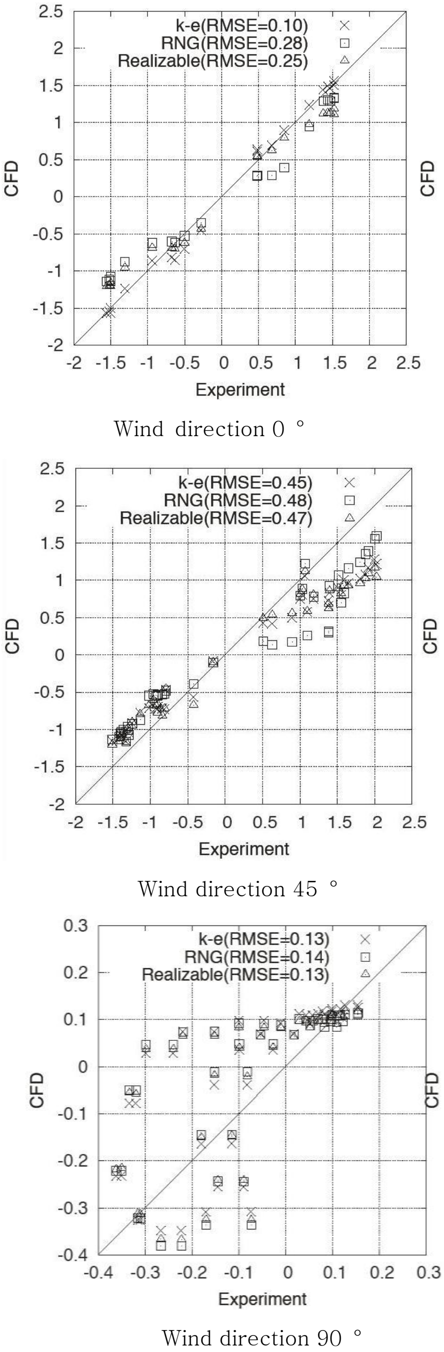


Fig. 15 Plane distribution in the outside of the opening (Inflow was positive at wind velocity.)



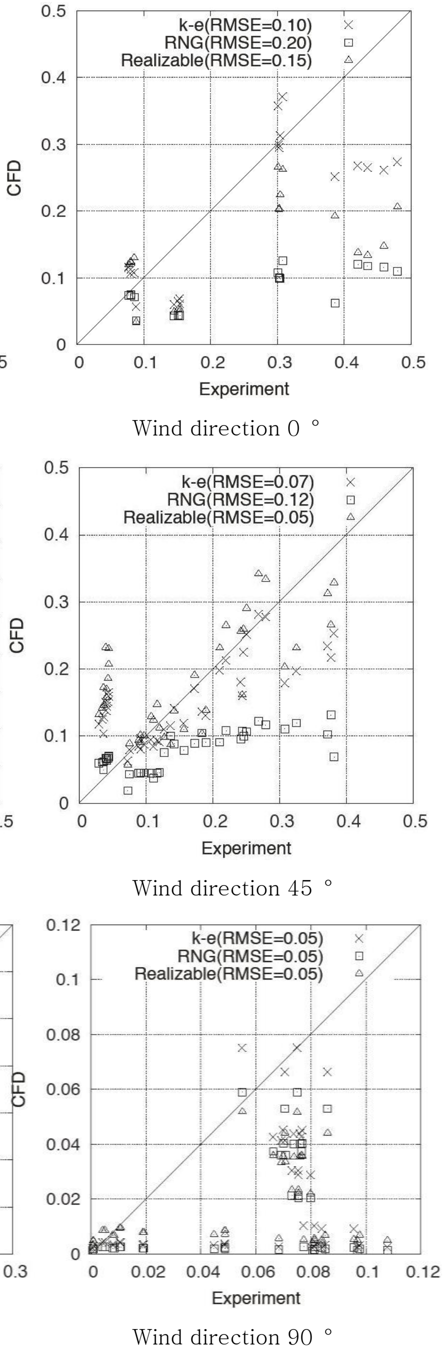


Fig. 16 Correlations of results of CFD analysis and wind tunnel experiment of wind velocity in the outside of the opening

Fig. 17 Correlations of results of CFD analysis and wind tunnel experiment of turbulent energy in the outside of the opening

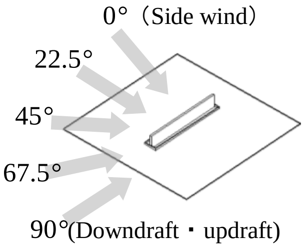


Fig. 18 Inflow wind direction

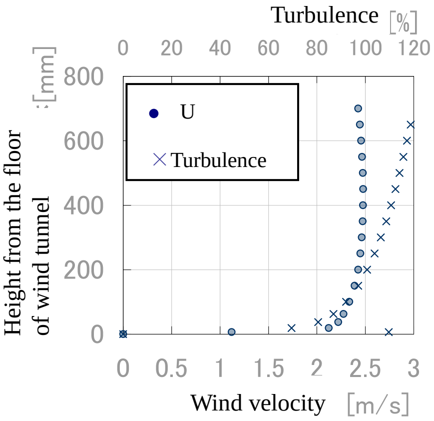


Fig. 19 Wind velocity profile

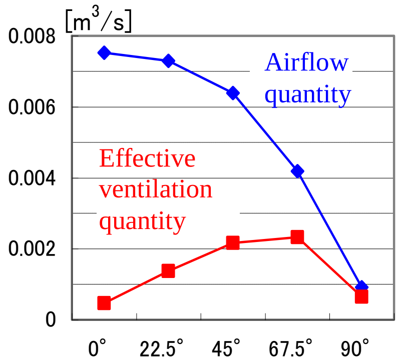


Fig. 20 Airflow quantity that passed through the opening and effective ventilation quantity

①【Separation of non-turbulent heat flux and turbulent heat flux】

All heat generation quantity = All heat emission quantity (steady) = Effective heat flux

Average wind velocity of the opening × Average temperature = Non-turbulent heat flux

Effective heat flux – Non-turbulent heat flux = Turbulent heat flux

②【Calculation of the coefficients of heat flux forecasting model for each direction】

Heat flux forecasting model : $K_{eff} = \alpha(1+\beta I_R) \cdot U_R$

→We divide heat flux by the area of the opening, temperature difference, and scholar wind velocity, and calculate α. $\alpha = F_{mean}/A/\Delta\theta/U_R$

→We divide the ratio of non-turbulent heat flux to turbulent heat flux by turbulent intensity, and calculate turbulence correcting coefficient β. $\beta = F_{turb}/F_{mean}/I_R$

Fig. 21 Method of calculating coefficient of heat flux forecasting model

Table 1 Outline of field measurement

Date of measurement	2009/12/6 19:12～20:52	The late summer was assumed. The room was heated before measurement. (Outside temperature standed at about 11℃.)	
Item of measurement	Measuring instrument	Interval	Remarks
Wind velocity on the roof	Three-dimensional sonic anemometers	0.1 seconds	on the staircase in the south (height 16.24m)
Wind velocity around the building surface	Three-dimensional sonic anemometers	2 seconds	50 cm apart from the wall
Wind velocity, Temperature at the opening	Three-dimensional sonic anemometers, thermocouples	0.1 seconds	Temperature : measured at 2 points for each side of the room, linear interpolation per 2 seconds
Room temperature, Outside temperature	Thermocouples	2 seconds	Room temperature : weighted average of 5 points volume Outside temperatue : the same measuring position as wind velocity around the building surface

Table 2 Outline of CFD analysis

Analysis code	Open source CFD tool : OpenFOAM
Area of calculation	1.2m(x)×1.2m(y)×1.36m(z)
Grid of calculation	Number of grid : 2.5 million, width of grid : 2.5mm～4cm
Solution of calculation	Steady solution, SIMPLE
Turbulence model	Standard k-ε, RNG k-ε
Advection term scheme	TVD (van Leer limiter)
Influx boundary	Profile by wind tunnel experiment
Wall boundary condition	Generarized logarithm rules for slickenside
Sky boundary	Slip boundary condition

Table 3 Outline of CFD analysis

Turbulence model	Standard k-ε model
Condition of non-isothermal flow field	
Buoyancy term	Disregard of influence(=0)
Uniform heat genaration in the room	20W/m^3
Boundary condition	Outside wall of chamber : 0 inclination (insulation)
Constant	Prandtl number=0.7
	Turbulence Prandtl number=0.9

Table 4 Coefficient of model

	α	β
0°	0.014	0.042
22.5°	0.039	0.080
45°	0.061	0.095
67.5°	0.068	0.051
90°	0.014	0.426