Modelling of Supply Air Jet from Diffusers of Four-way packaged Air-conditioner for CFD Analysis on Unsteady Airflow in Room

Norikazu Yasuda*[1], Toshio Yamanaka[1], Tomohiro Kobayashi[1], Jihui Yuan[1] and Choi Narae[1]

1 Osaka University
Suita 565-0871
Osaka, Japan
**: yasuda_norikazu@arch.eng.osaka-u.ac.jp*

ABSTRACT

Recently in Japan, many buildings introduce packaged air-conditioner (hereinafter, this is called "PAC") as an air conditioning equipment. Most of the PACs can change the direction of supply air blown from the unit and the "Swinging Mode" can change airflow direction periodically as one of the airflow direction setting of PAC. Before constructing the building, appropriately predicting the indoor environment (e.g. temperature distribution or air velocity distribution and so on) is important for indoor environment design. CFD analysis is one of the simulation technique based on the fluid dynamics and have been used to predict the building environment. However, it will take long time to finish non-steady analysis while analysing swinging airflow with non-steady analysis in CFD, it may cause trouble in design situation.

In order to shorten the analysis time and acquire more precise result, the purpose of this research is to establish a CFD model to possibly analyse the swinging airflow in non-steady condition, using dynamic Prescribed Velocity (P.V.) method. In this paper, the experiment to understand the characteristics of the airflow blown from the PAC was carried out. The results of airflow direction measured by an anemometer, airflow volume measured by a cardboard duct, airflow velocity near air supply opening measured by hot-wire anemometer with X-type probe are obtained to establish the CFD model of analysing the swinging airflow with unsteady condition in this study.

KEYWORDS

Packaged Air Conditioner, Hot-wire Anemometer, CFD Analysis, Dynamic P.V. Method

1 INTRODUCTION

Currently, packaged air conditioner (hereinafter referred as PAC) as air conditioning equipment has been introduced to many buildings in Japan, i.e. office building. In indoor environment design phase, predicting the indoor environment is considered as one of the most important things to do. To predict the indoor environment, for example, temperature distribution, air velocity distribution, pollutant distribution, CFD analysis is often used. However, while analysing CFD model in non-steady situation, the time to finish CFD analysis is sometimes too long. One of the way to shorten the analysing time of CFD is to establish a simpler model to calculate the equally large meshes while compared to the original model. In the past, a CFD model of various kind of diffuser or supply opening was conducted (Nielsen,1976, Skovgaard,1991, Kondo,2002, Niwa,2016). Many operation modes i.e. cooling mode and warming mode with different airflow direction and airflow volume, can be set in PAC. Therefore, various kinds of airflow blown from PAC need to be predicted by CFD.

The purpose of this study is to make a simpler airflow model of the PAC based on dynamic Prescribed Velocity (P.V.) method, which can be applied to the all airflows blown from PAC. Thus, the time of analysing CFD could be shortened, and the accuracy of the CFD analysis would be better by using the simpler model.
In addition, airflow direction, airflow volume and airflow velocity were measured by experiment for the purpose of reproducing the airflow by CFD analysing using P.V. method in this study.

2 PROFILE OF EXPERIMENTAL ROOM

The cross section and reflecting ceiling plan of the experimental room are detailed and shown in Figure 1.

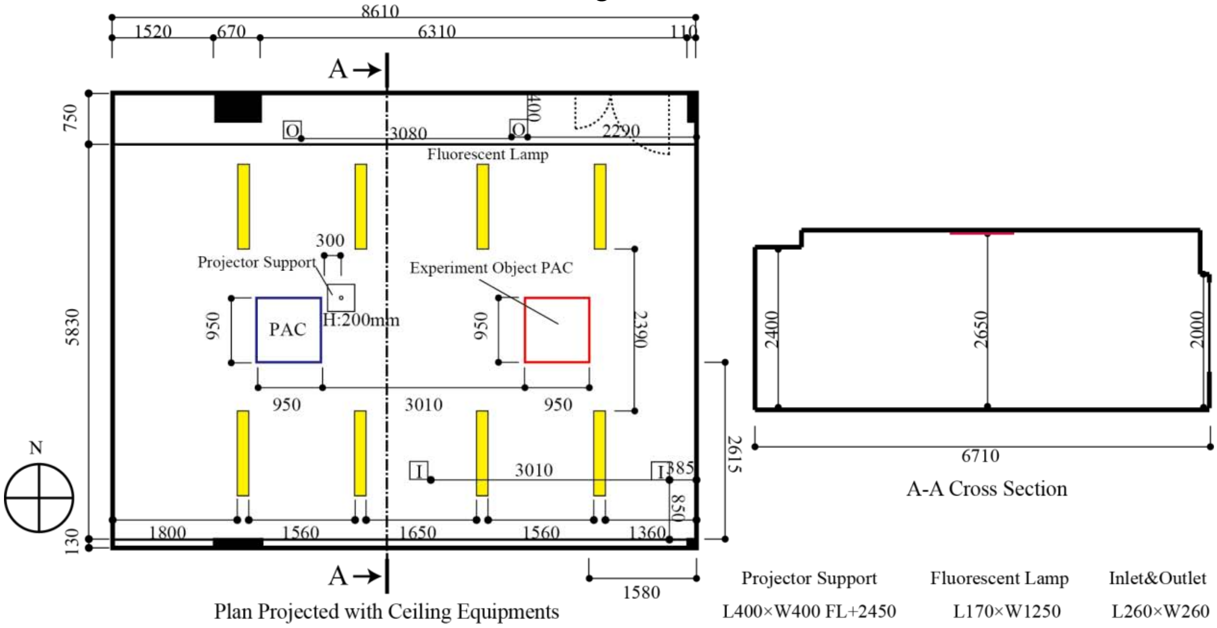


Figure 1: Cross section and reflecting ceiling plan of the experimental room

The experimental room is on the seventh floor (top level) of the seven stories building in Suita Campus, Osaka University. Although two PACs are set on the ceiling of the experimental room, PAC on the east side was selected as the experimental target. It is because there is a projector support which was used to install a projector near the PAC on the west side and it was assumed that the projector support might affect the airflow blown from the PAC on the west side. The experimental PAC is shown in Figure 2.

Figure 2: Experimental PAC

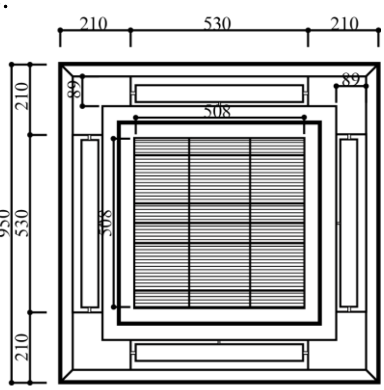


Figure 3: Experimental PAC

The PAC is a four-way ceiling cassette type air conditioner. The plan of the PAC and setting conditions of the PAC are respectively shown in Figure 3 and Table 1.

Table 1: Kinds of setting of target PAC used for experiment

Operating Mode	Fan only, Cooling, Heating, Automatic
Airflow Volume Mode	High, Middle, Low, Calm
Airflow Direction Mode	Mode1, Mode2, Mode3, Mode4, Mode5, Swing mode

Airflow directions assumed as the angle from the ceiling, were named as Mode1 to Mode5 respectively from shallow angle to deep angle to the ceiling. Swing mode is a model that the direction of airflow can be changed every second between the angle of Mode1 to Mode5.
We conducted some experiments using this PAC to obtain the experimental results of airflow direction, airflow volume and airflow velocity.

3 AIRFLOW DIRECTION MEASUREMENT USING THERMAL ANEMOMETER

3.1 Purpose of the experiment

This experiment was done using thermal anemometer to clarify the airflow direction of five modes of the PAC. The purpose of this experiment is to grasp the direction of the airflow blows from PAC and to determine the proper angle of X-type probe used in airflow velocity measurement which will follow this experiment. The model of thermal anemometer is shown in Table 2.

3.2 Experimental method

The method of airflow direction measured by experiment is explained as following: Firstly, thermal anemometer's probe was set to the airflow velocity measurement points as shown in Figure 4, and measure the velocity at each measurement points. The measurement point was set on the central plane vertically cut through the centre of the PAC. Next, the measurement point where the maximum velocity was observed in the measurement cross section will be determined. Lastly, draw a regression line based on measurement points where maximum velocity was observed using least squares method. The origin of the regression line was fixed on the origin in the centre of supply opening of PAC. Measuring scene is shown in Figure 5.

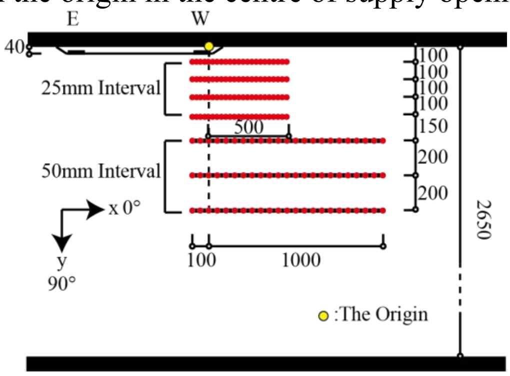


Figure 4: Measurement point

Figure 5: Measuring scene

Table 2: Model of thermal anemometer

Model of Thermal Anemometer	KANOMAX Model 6501
Range of the Velocity able to Measure[m/s]	0.01~30
Measurement Accuracy	±2[%] or 0.02[m/s]

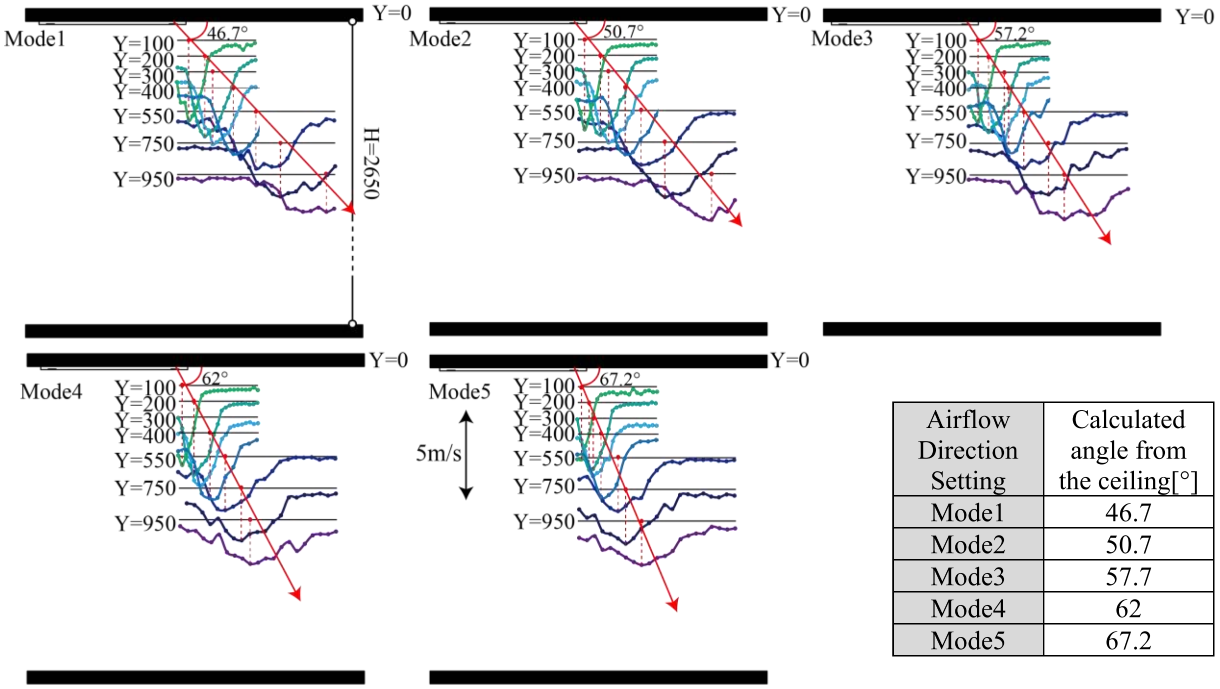
Experimental condition of this measurement is shown in Table 3. The thermal anemometer has uniform sensitivity to flow angle so the position of the probe was considered to be able to accurately measure the velocity. To move the thermal anemometer vertically and horizontally, a traverser that can automatically move in horizontal direction, and a steel pipe standing which can moved in vertical direction, was used.

Table 3: Experimental condition

Measurement Time [s]	30
Measurement Frequency [Hz]	1
Airflow Volume Setting	High
Room Temperature [°C]	13

3.3 Experimental results

The results of airflow velocity and the airflow direction of each airflow direction mode calculated from the airflow velocity are shown in Figure 6. The red points in Figure 6 are the measurement points where the maximum airflow velocity was measured in each measurement cross section.



Airflow Direction Setting	Calculated angle from the ceiling[°]
Mode1	46.7
Mode2	50.7
Mode3	57.7
Mode4	62
Mode5	67.2

Figure 6: Airflow velocity measurement results

It is clarified that the angles between the angles of the airflow blown from the target PAC from the ceiling varies from about 47° to 67°. In addition, from the experimental results, it is shown that the airflow velocity declines as the distance from the centre of an axis of the airflow. The maximum velocity in each measurement cross section becomes smaller with the distance from the ceiling to the measurement cross section. The results acquired from this experiment are used to the determination of the airflow direction of the target PAC.
The airflow direction data is also used for CFD analysis.

4 AIRFLOW VOLUME MEASUREMENT USING CARDBOARD DUCT AND THERMAL ANEMOMETER

4.1 Purpose of the experiment

This experiment was done using thermal anemometer which is also used in airflow direction measurement experiment and cardboard ducts which is made of plastic cardboard. The purpose is to acquire the amount of the airflow volume rate blown from the supply opening and return opening. The data of the airflow volume is intended to be used as the boundary condition of the supply opening in CFD analysis.

4.2 Experimental method

The airflow volume was calculated by multiplying the airflow velocity measured by the thermal anemometer at each measurement point shown in Figure 7 by the area divided for each measurement point. 36 measurement points and 16 measurements points were distributed at the end of the cardboard duct for four supply openings and one outlet opening respectively. The area for each measurement point are 0.001335 m^2 at supply opening and 0.01756 m^2 at outlet opening, respectively.

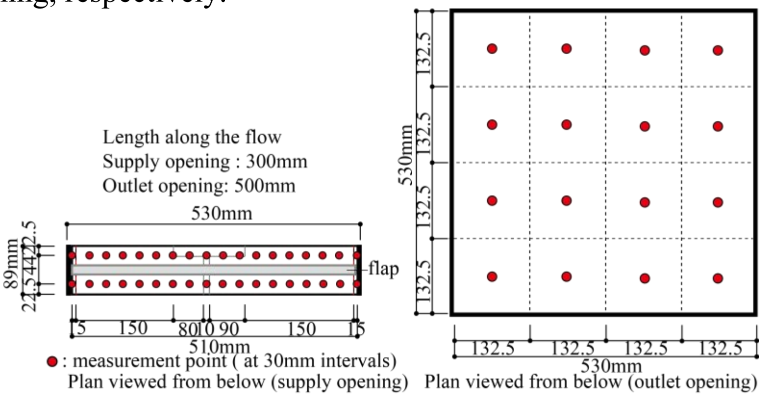


Figure 7: Measurement points

Figure 8: Measurement scene

The experimental condition is almost the same as that of airflow direction measurement. The measurement was conducted when the airflow direction mode was fixed at mode3. The measuring scene is shown in Figure 8. The length of each cardboard duct along the flow is respectively 300mm for supply opening and 500mm for outlet opening.

4.3 Experimental results

The experimental results of airflow volume rate of each supply opening and outlet opening are shown in Figure 9.

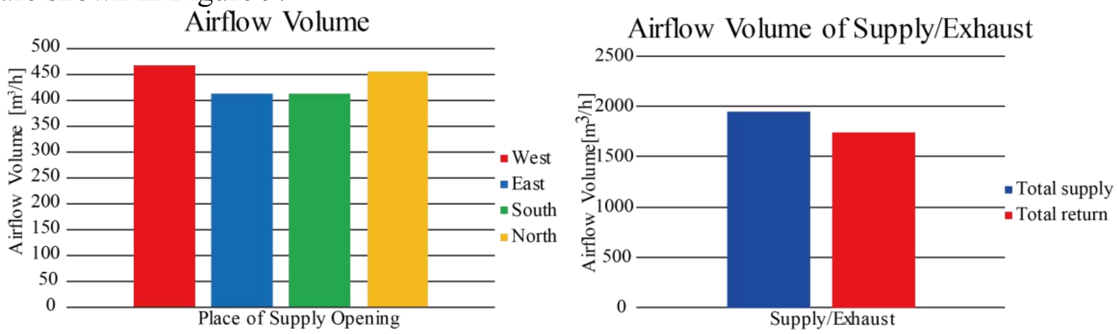


Figure 9: Airflow volume measurement results

The results (Fig.9) showed that the calculated airflow volume of the west supply opening is larger than the other three supply openings. Thus, the experiment was carried out at the supply opening in the west. Furthermore, it is shown that the calculated total supply airflow volume is larger than the return airflow volume. The reason is considered that the probe of thermal anemometer was set on a surface of the entrance of the cardboard duct and the velocity around the entrance of cardboard duct is unstable especially in the measurement of returning airflow rate. The calculated airflow volume is used for CFD analysis as boundary condition given to supply opening.

5 AIRFLOW VELOCITY MEASUREMENT USING HOTWIRE ANENOMETER

5.1 Purpose of the experiment

This experiment was done using hotwire anemometer with X-type probe (measuring time:60[s], measurement frequency:1000[Hz], airflow volume setting: high, operating mode: fan only, airflow direction mode: mode3). The purpose of this experiment is to grasp the characteristics of the airflow blown from the target PAC, and to apply these measured velocities to CFD analysis as boundary condition using P.V. method.

5.2 Experimental method

Experimental equipment of this measurement is shown in Figure 10. As previously mentioned, this experiment was conducted at the west supply opening of PAC.

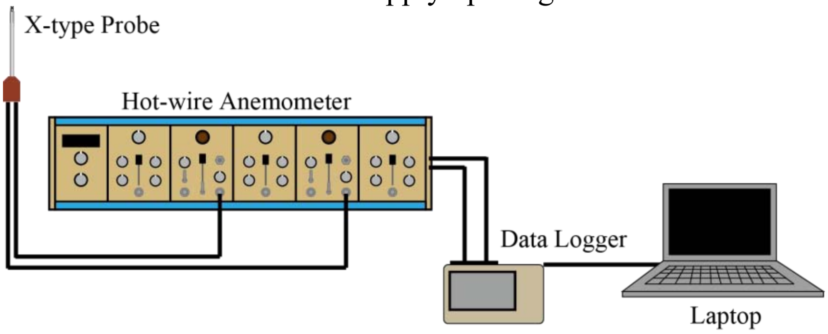


Figure 10: Airflow velocity measurement equipment

According to the results of preliminary measurement about angle characteristics which was done previously, the direction of X-type probe was set at an angle of 45° incline against the supply opening. Two hotwires were named “Channel 1” and “Channel 2” respectively. Channel 1 is the hotwire which is stretched vertically and Channe2 is the hotwire stretched horizontally. Airflow velocity was calculated by equation (1) to (3). (refer to Nomenclature)

$$E_1 = \alpha U sin(45° + \theta) \tag{1}$$

$$E_2 = \alpha U sin(45° - \theta) \tag{2}$$

$$\alpha = \frac{10}{U_{max}} \tag{3}$$

From equation (1) and (2), following equation (4) and (5) are derived.

$$U = \frac{U_{max}}{10\sqrt{2}} \sqrt{2({E_1}^2 + {E_2}^2)} \tag{4}$$

$$\theta = tan^{-1} \frac{E_1 - E_2}{E_1 + E_2} \tag{5}$$

In equation (3), “10” is the output voltage when the airflow velocity is U_{max}. The measurement points are shown in Figure 6 and Figure 11. By using airflow velocity at supply opening, turbulence characteristic, turbulence kinetic energy k and turbulence dissipation rate ε, were also calculated by the following equation (4) to (10).(refer to Nomenclature)

$$k = \frac{3}{8}(\overline{u'u'} + \overline{v'v'}) \tag{4}$$

$$\varepsilon = C_D \frac{k^{\frac{3}{2}}}{l} \tag{5}$$

$$\overline{u'_\iota(t)u'_\iota(t+\tau)} = \lim_{T\to\infty} \frac{1}{T} \int_1^\infty u'_i(t)u'_i(t+\tau)\, dt \tag{6}$$

$$\rho_i(\tau) = \frac{\overline{u'_\iota(t)u'_\iota(t+\tau)}}{\overline{(u'_i(t))^2}} \tag{7}$$

$$T_i = \int_0^{\tau_0} \rho_i d\tau \tag{8}$$

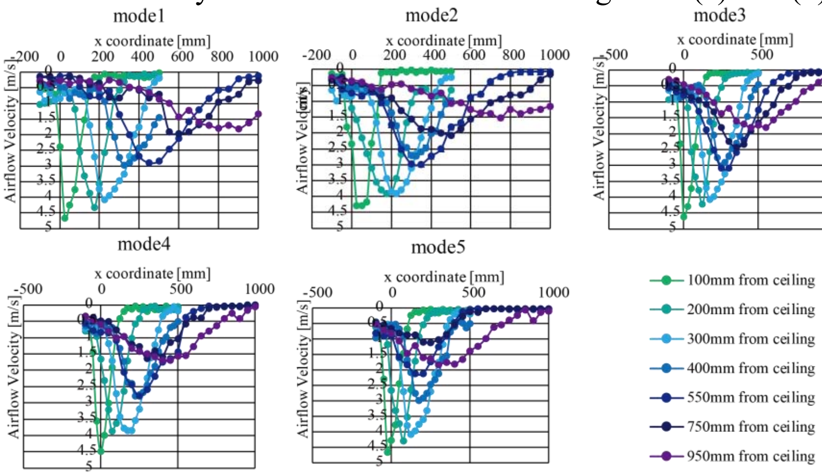
$$\Lambda = \bar{u} T_i \tag{9}$$

$$l = C_D^{\frac{1}{4}} \Lambda \tag{10}$$

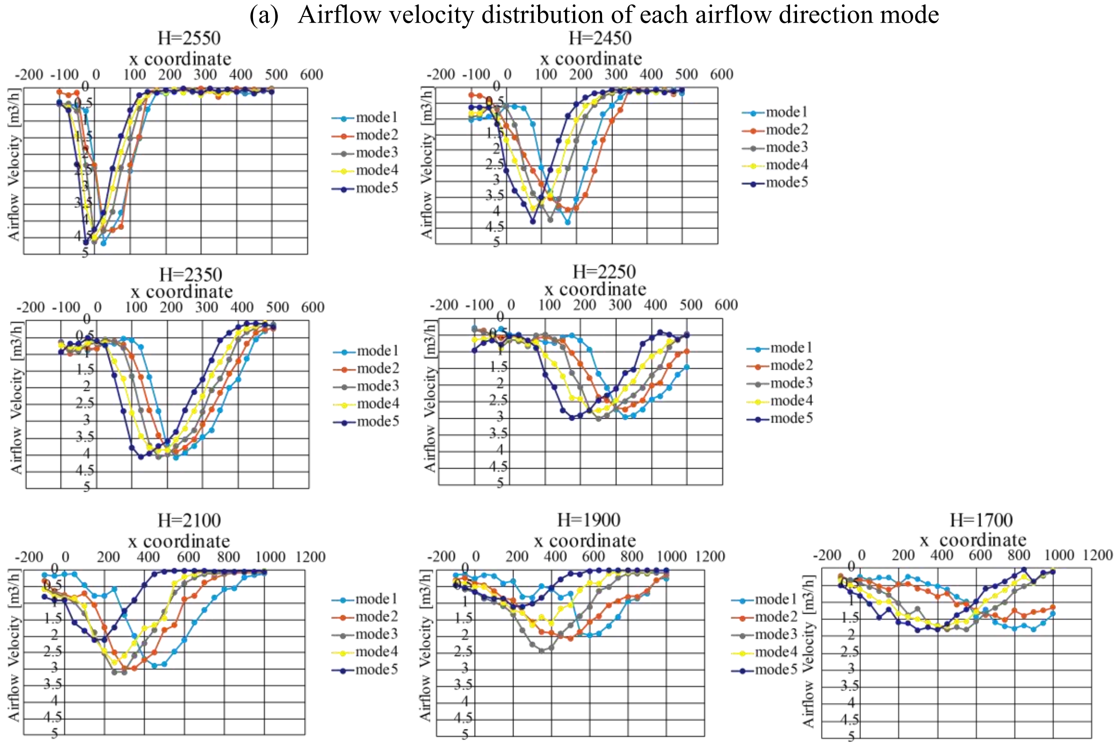
It must be noticed that X-type probe can't measure airflow velocity from the third dimension, so that the u' and v' which were calculated from x direction velocity and y direction velocity include the standard deviation of the third dimension. Therefore, the calculated u' and v' are larger than actual figure. It is assumed that the standard deviation of the third dimension is the same as the average value of u' and v' and that is what leads equation (4) to calculate turbulence kinetic energy.

5.3 Experimental results

The results of airflow velocity measurement are shown in Figure 11(a) and (b).



(a) Airflow velocity distribution of each airflow direction mode



(b) Airflow velocity distribution of each cross section

Figure 11: Airflow velocity measurement results

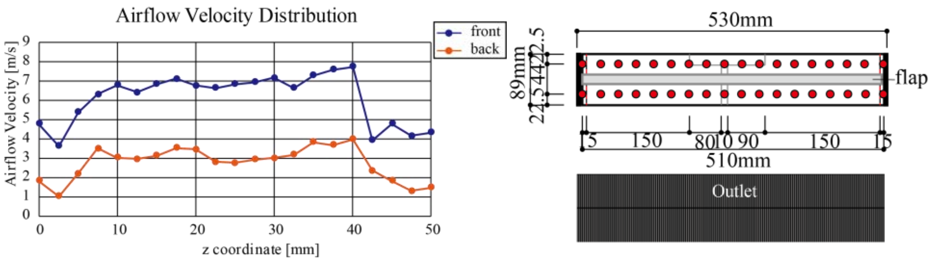


Figure 12: Airflow velocity at supply opening

The same as the airflow direction measurement using thermal anemometer, the larger the distance from the supply opening to the measurement point, the smaller the airflow velocity. From the results of H=2550, airflow velocity at over x=200mm is under 0.25 m/s in all airflow modes, thus the x coordinate at 300mm is enough for the length of velocity fixed surface. The maximum measurement point of each measurement cross section changes as the mode changes. In addition, turbulence statistics are calculated and shown in Figure 13.

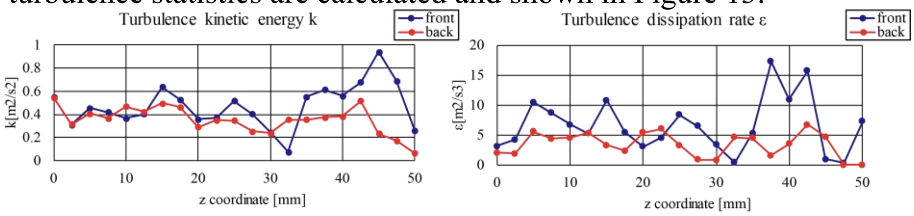


Figure 13: Turbulence statistics

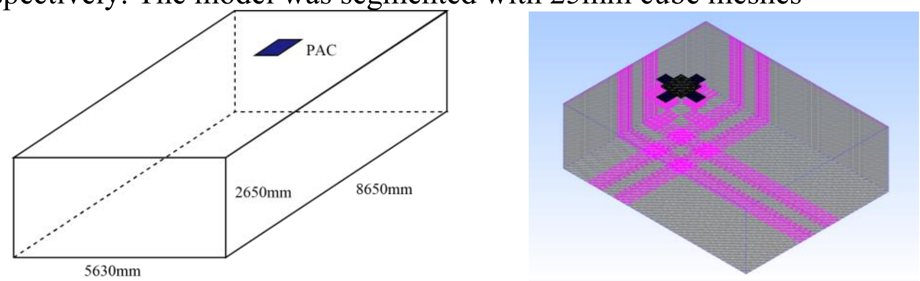
6 CFD ANALYSIS USING P.V. METHOD

6.1 Purpose of this CFD analysis

The purpose of this CFD analysis is to reproduce the airflow of the PAC by using P.V. method (Nielsen,1992) and to check whether P.V. method is appropriate to analyse the airflow blown from PAC. It is assumed that more accurate result obtained by P.V. method will be gained.

6.2 CFD model

The CFD model is shown in Figure 14. The dimensions of simulated room is length of 8610mm, width of 5630mm and height of 2650mm. A PAC is set the ceiling. At supply opening of the PAC, the velocity is uniformly distributed and the air volume blow from each supply opening is set to be same. The value of the airflow volume is 467 [m^3/s] which was acquired from the measurement. Analysis and boundary conditions are shown in Table 4 and Table5, respectively. The model was segmented with 25mm cube meshes



(a) Isometric drawing (b) Meshes

Figure 14: CFD analysis model

Table 4: CFD analysis condition

CFD Code	STREAM V14
Turbulence Model	Standard k-ε
Analysis Domain	8650×5630×2650
Number of Meshes	10,328,175(275×351×107)
Algorithm	SIMPLE
Discretization Scheme	QUICK

Table 5: Boundary conditions

(a) Boundary surface

Boundary Surface	Wall Function
Wall	Non-slip
Floor	Non-slip
Ceiling	Non-slip

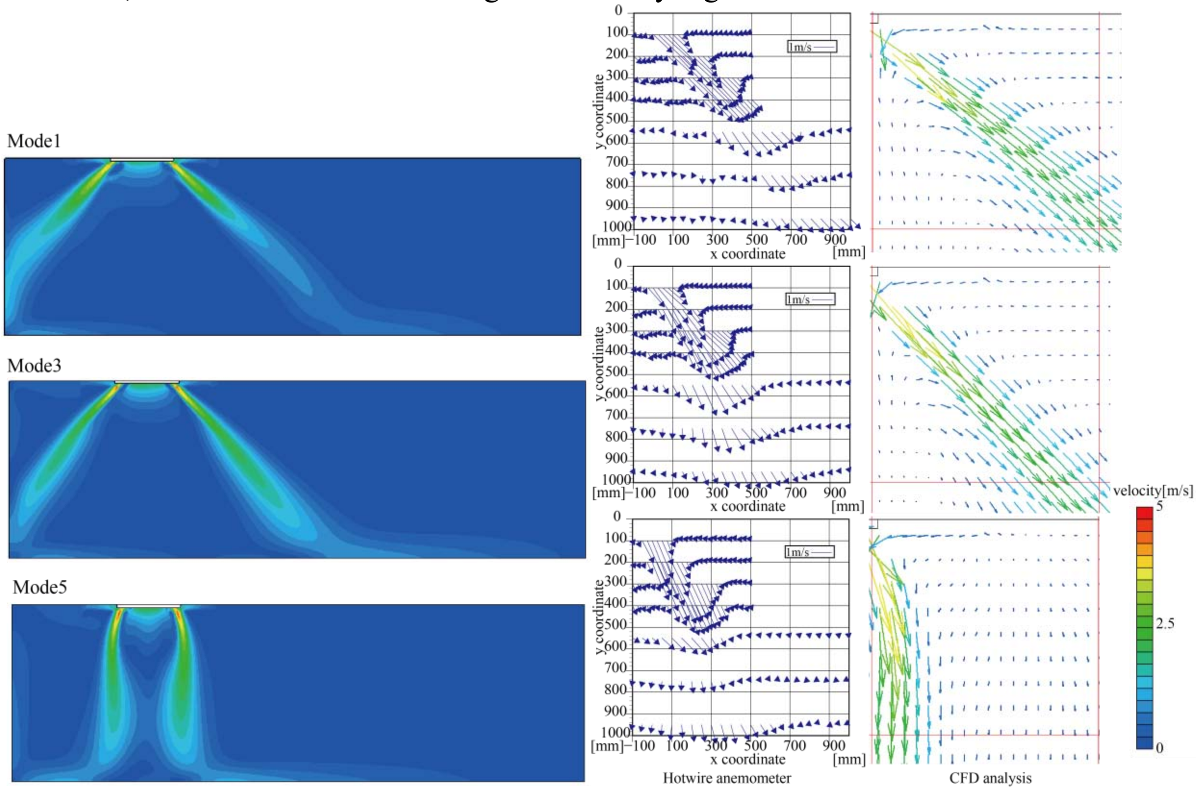
(b) Flow

PAC	Airflow Rate [m^3/h]	Angle from the Ceiling [°]
Inlet	467 × 4	46.7/57.2/67.2
Outlet	Natural Outlet Boundary	-

P.V. method was used and velocity fixed surfaces was set at y=100mm cross section where the airflow velocity data was acquired by the experiment. The velocity given to the velocity regulation surfaces are different depending on the air direction mode.

6.3 CFD analysis results

The results of CFD analysis is shown in Figure 15. The analysis results of Mode5 showed that the airflow direction goes downwards at the point far from ceiling. In addition, it is also shown that the velocity measured by hotwire anemometer is lower than the velocity from CFD analysis. Therefore, we must reconsider the height of velocity regulation surface.



(a) Velocity contour (CFD) (b) Velocity vector (Experiment) (c) Velocity vector (CFD)

Figure 15: CFD analysis results

7 CONCLUSION

In this research, the experiments were conducted in the room with four way packaged air conditioner. From the experiment, the characteristics (airflow direction, volume, velocity and turbulence statistics) of the airflow blown from the PAC were acquired. By using experimental results, CFD analysis was carried out to reproduce the airflow. It is realized that several points must be investigated to make the analysis more accurate.

ACKNOWLEDGEMENT

This research is under joint research with Mitsubishi Electric Corporation. The authors are sincerely grateful to the Mitsubishi Electric Corporation and gratefully acknowledge the fund and the cooperation of people who related to this research.

Nomenclature	
E_1, E_2 [V]	Output voltage of Channel 1 or Channel 2 at hotwire anemometer
α [-]	Caribrating coeficient
U[m/s]	Velocity of the airflow hit to hot wires from the front
θ [°]	Angle of the direction X-type probe and the line x=y
U_{max} [m/s]	Velocity which was used when caribrating hotwire anemometer (=10 m/s)
k[m2/s2]	Turbulence kineticenergy
u',v' [m/s]	Stadard deviation of the velocity of x-direction or y-direction
ε [m^3/s^2]	Turbulence dissipation rate
C_D[-]	Model constant (=0.09)
l[m]	Turbulence length scale
t, τ [s]	Time
τ_0 [s]	The time when τ_i goes 0
ρ [-]	Autocorrelation coefficient
T_i [s]	Turbulence time scale
Λ [m]	Integral length scale

REFERENCE

Nielsen, P.V. (1976) FLOW IN AIR CONDITIONED ROOMS: *Ph. D. thesis from the Technical University of Denmark*

Skovgaard,M(1991) MODELLING COMPLEX INLET GEOMETRIES IN CFD: applied to air flow in ventilated rooms, *12th AIVC Conference. pp. 184~199, 1991*

Kondo,Y(2002). MODELING OF AIR DIFFUSER INSTALLED AT CEILING FOR CFD SIMULATION : Experiments and numerical simulation on complex air diffuser at isothermal condition (Part 1), *Architectural Institute of Japan, The Summaries of Designing Papers of Architectural Institute of Japan*

Niwa,K(2016). A CFD Analysis on Airflow in Room with Line Slot Diffuser for Modeling of CFD parts Effect of Turbulence Intensity Settings of Inlet Airflow, *The Society of Heating, Air-Conditioning and SanitaryEngineers of Japan, The summaries of Technical Papers of Kagoshima,201-204*

Nielsen, P.V. (1992) DESCRIPTION OF SUPPLY OPENINGS IN NUMERICAL MODELS FOR ROOM AIR DISTRIBUTION: *Instituttet for Bygningsteknik, Aalborg Universitetscenter*