CFD SIMULATION OF AIRFLOW IN ROOM WITH MULTI-CONE CEILING DIFFUSER USING MEASURED VELOCITY AND TURBULENT PARAMETERS IN LARGE SPACE

H. Kotani, T. Yamanaka, Y. Momoi

Dept. of Architectural Eng., Graduate School of Eng., Osaka University
2-1 Yamadaoka, Suita Osaka 565-0871, Japan, Tel: +81-6-6879-7645, Fax: +81-6-6879-7646
kotani@arch.eng.osaka-u.ac.jp, http://www.arch.eng.osaka-u.ac.jp/~labo4/

Summary

A multi-cone diffuser is difficult to treat in CFD simulations of the airflow in rooms, because its complicated shape tends to make complicated flow near the diffuser and needs many meshes. The velocity in the x, y, z-direction, the turbulent kinetic energy and the length scale around the multi-cone ceiling diffuser were measured in a large space. CFD simulations by standard k-ε model with measured velocities and turbulent parameters as the boundary conditions were conducted. The model experiments to know the airflow in a room with this diffuser were also made, and the velocities and turbulent kinetic energies were obtained to compare with the CFD results. The results of CFD and the model experiments were practically in good agreement.

Introduction

A ceiling set multi-cone diffuser is often used in commercial buildings as a supply device. This diffuser has cone-shaped inner core and this cone can be adjusted two positions to make the flow along the ceiling for the cooling and the vertically downward flow for the heating. This diffuser can generate both the radial flow and the axial flow as shown in Figure 1.

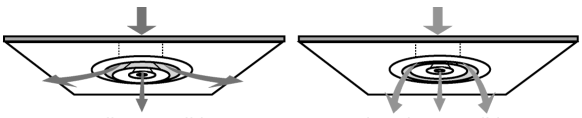


Figure 1: Two types of flow generated by multi-cone diffuser

There are studies of simple representation of these complex shaped diffusers to reduce the calculation load in the simulation by the computational fluid dynamics (CFD). Nielsen et al. (1978) and Gosman et al. (1980) succeeded to use the box method and the prescribed velocity method. Nielsen (1992) completed these studies in IEA Annex 20 works. Srebric and Chen (2001) used the box method and the momentum method, and also proposed a method of test for ASHRAE Standard. These methods were developed mainly for the wall-mounted supply devices that generate a wall jet under the ceiling and the ceiling-mounted devices that generate a radial jet.

The multi-cone diffuser generates two types of airflow and the verification is needed when these method are applied. Okaichi et al. (2000) made CFD simulation of the room flow with the circular multi-cone diffuser in case of the heating condition (see Figure 1). The supply boundary conditions were measured except ε and the box method using all surface of the box was applied. ε was tested by four length scale of turbulent as the simulation parameters. The results were compared with the mean velocities and turbulent energies in the room obtained by the experiment. The results were well, but the optimum length scale was not found. Kondo et al. (2001) applied the box method and the prescribed velocity method to the rectangular multi-cone diffuser in case of the cooling condition (see Figure 1). The boundary conditions were calculated by CFD with unstructured-grid that can describe detailed shape of the diffuser. The effectiveness of this CFD was guaranteed by comparison with the measured mean velocity in the room flow. They finally made CFD with structured grid using the box method and the prescribed method, and the

results were agreed with the experiment.
This paper presents the measurement results of the velocity in the x, y, z-direction, the turbulent kinetic energy and the length scale around the circular multi-cone diffuser in a large space by means of the constant-temperature hot-film anemometer. The model experiments to investigate the airflow in a room with this diffuser are also made and the velocities and turbulent kinetic energies are obtained by ultra sonic anemometer. CFD simulations with measured velocities and turbulent parameters as boundary conditions by standard k-ε model are conducted. The results of the simulation were practically in good agreement with the experiment.

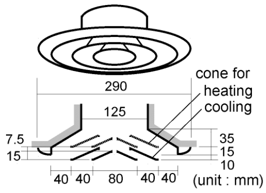
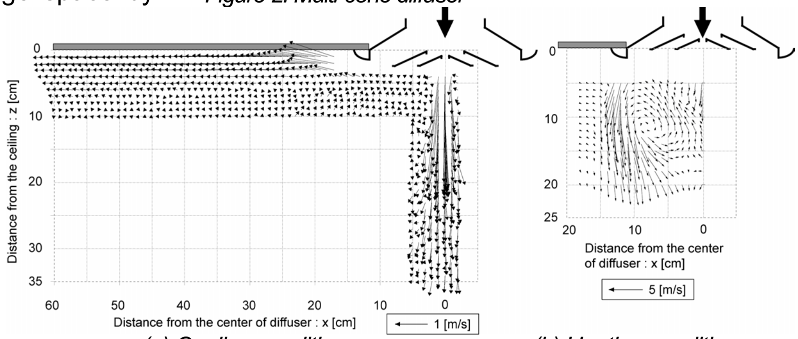


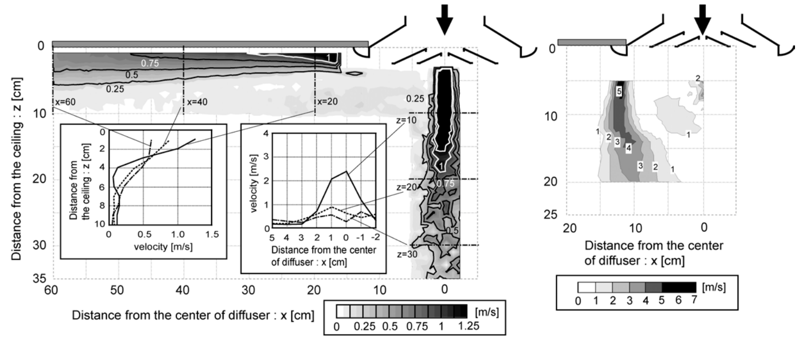
Figure 2: Multi-cone diffuser

Table 1: Measurement conditions

cone position	for cooling, heating
supply temparature	same as the ambient (isothermal)
supply velocity	2 m/s at the neck of diffuser
supply flow rate	88.4 m^3/h



(a) Cooling condition *(b) Heating condition*
Figure 3: Measured velocity fields around the multi-cone diffuser



(a) Cooling condition *(b) Heating condition*
Figure 4: Measured velocity distributions around the multi-cone

Airflow Pattern around Multi-cone Diffuser

The airflow around the multi-cone diffuser has been measured by authors' research group. The diffuser's shape and dimension is shown in Figure 2. Table 1 shows measurement conditions. Velocities in x, y and z direction were measured in a large space by the constant-temperature hot-film anemometer which can measure the velocity and direction simultaneously because of its probe with split film. Data was collected during 30 seconds with its sampling frequency of 250 Hz at one measuring point. The length scale of turbulence and the dissipation rate of the kinetic energies; ε was also calculated by the autocorrelation of measured velocities.

Figure 3 shows the velocity fields around diffuser. In the cooling condition, the vertical downward flow and horizontal radial flow along the ceiling are observed. There is weak entrained flow vertically toward the ceiling flow. In the heating condition, the large vortex by entrained flow is obviously existed. The supply air flows along the outmost surface of a diffuser with high velocity. The down flow from the center of the diffuser is weaker than main flow. The velocity distributions are shown in Figure 4. Profiles of horizontal ceiling flow are similar to the previous studies using wall-mounted diffuser by Nielsen (1992), and Srebric and Chen (2001).

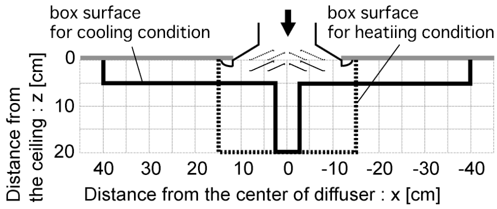


Figure 5: Boundary surface for multi-cone diffuser

Measurement of Boundary Condition

The box size is the most important thing for using the box method. In this paper, it depends on the velocity distribution and the box size is decided as shown in Figure 5 where the jet can be sufficiently developed and the large vortex is included in the box. The box method used by Nielsen (1992) provides the condition of free-slip for the parallel surface to the main flow (surface of z=5, x=2.5cm in cooling condition and x=15cm in heating condition in Figure 5). However, the variables are provided at all surfaces because the entrained flow is needed to describe. The measurement of velocities and the calculation of

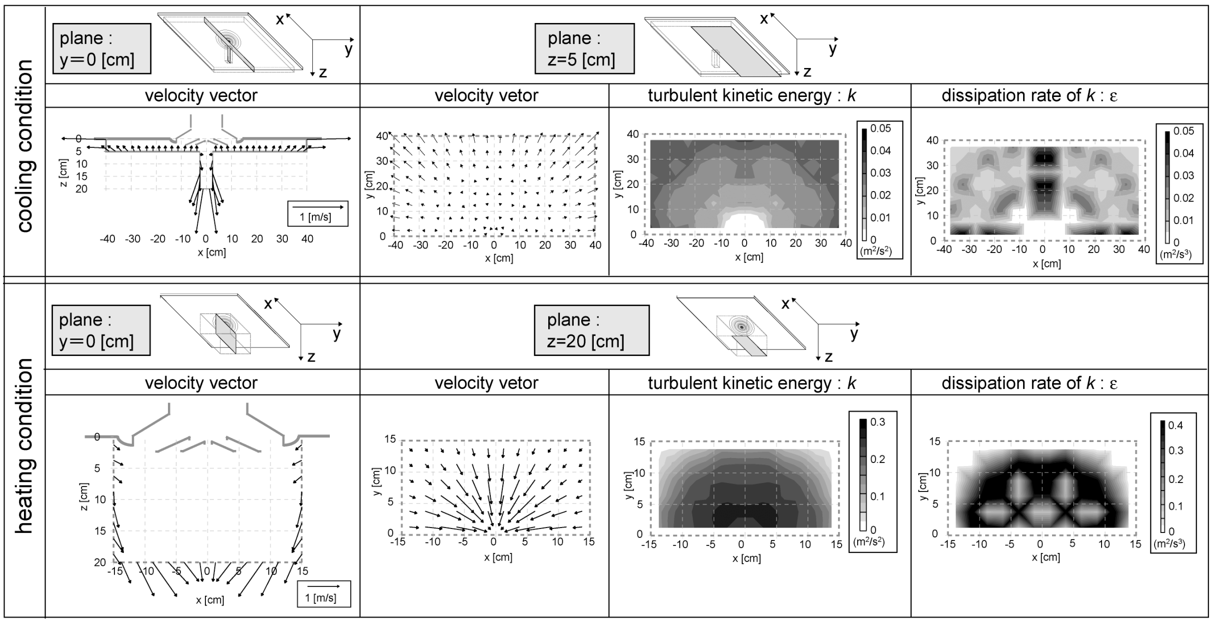


Figure 6: Measured values as the boundary conditions for CFD

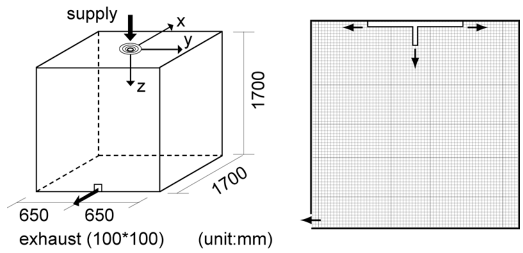


Figure 7: CFD model of room and mesh system for cooling condition

turbulent parameters were conducted at the boundary surface in a large space by the same method as the previous chapter. Figure 6 shows the measurement results. The entrained flow at the parallel surface to the main flow can be described as expected.

CFD and Experimental Setup

3-D CFD simulations of the room airflow are conducted by standard k-ε model using the commercial code, STREAM ver.4 (Software Cradle Co. Ltd, 2000). The dimension of the room is shown in Figure 7 and the room is divided into 25mm meshes. The third-order QUICK scheme is used for the convection. Supply boundary conditions are given by the measurement (see Figure 6).

The model experiment of the same sized room is also conducted for CFD verification. Velocities of x, y and z direction in the room are measured by means of the ultrasonic anemometer at 100mm grid. Data is collected during 30 seconds with its sampling frequency of 10 Hz at one point. Both CFD and experiment are conducted in the isothermal condition.

Results and Discussion

Figure 8 shows the comparison between measured values and CFD results. In the cooling condition, the supply air separates vertical down flow and horizontal radial flow. The horizontal radial flow faces to a wall and downward flow along the wall is generated. The air in the center of the room is entrained to the ceiling radial flow. These tendencies and the peculiar vortex at the upper-right corner in the figure are well reproduced by CFD.

In the heating condition, the supply air of high velocity forms the axial vertically jet. This jet reaches to the floor and generates a large circulation flow. The obvious vortex is seen at the lower-right side in the figure, but its position of CFD is rather higher than that obtained by measurement. The simulated velocity decay of the jet is quicker than the measurement. This is the most important subject for a future study.

In both conditions, the turbulent kinetic energies are different in one figure. This may be a possible reason for difference in sampling frequency and space resolution of the anemometer. The ultrasonic anemometer used in the measurement cannot measure the detailed fluctuation and averages the fine eddy. On the other hand, CFD uses the boundary condition of high frequency and resolution measured by hot-film anemometer.

Conclusions

The airflow pattern around the multi-cone diffuser is complicated because it can generate different two types of flow, therefore the box method is useful for using in CFD simulations.

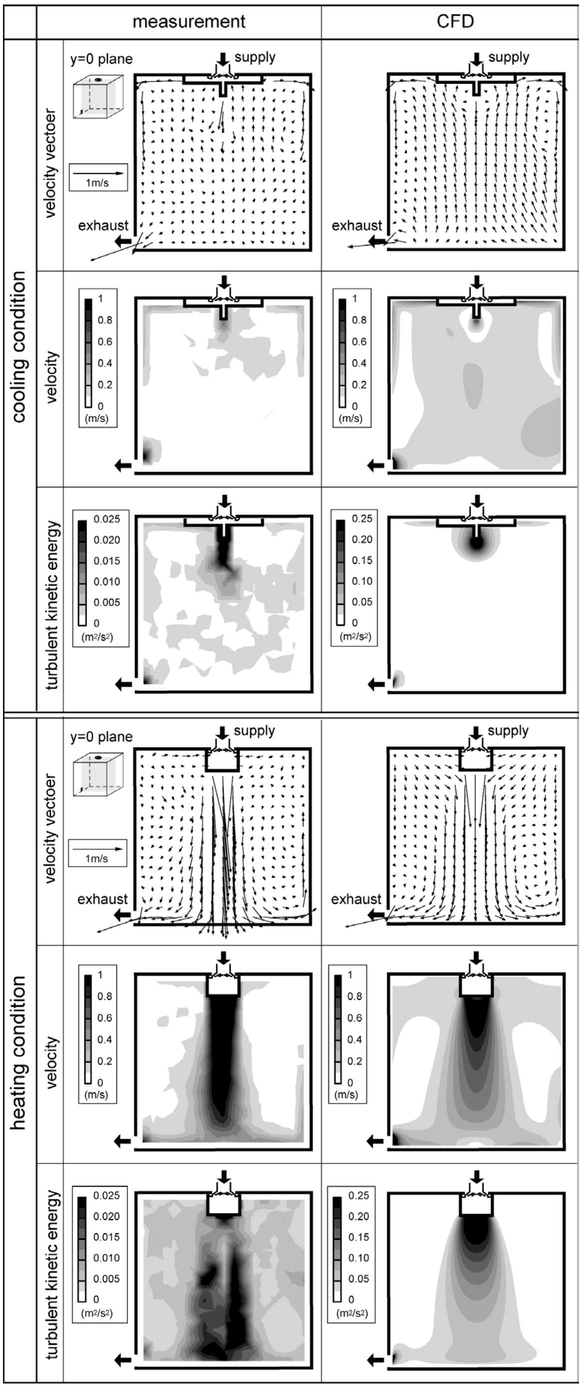


Figure 8: Comparison between measurement and CFD results

The provision of velocities and turbulent parameters at all surfaces of the box is useful for describing the entrainment flow to the supply jet. CFD simulation using the measured values as the supply boundary condition can predict the airflow pattern inside the room in practical accuracy, except for the decay of axial jet velocity in the heating condition.

Acknowledgements

The authors would like to thank Mr. Takeshi Hirano, Mr. Atsuo Okaichi and Mr. Masahiro Kato for their valuable support.

References

Gosman, A.D., P.V. Nielsen, A. Restivo, and J.H. Whitelaw, 1980. The flow properties of rooms with small ventilation openings, *Transactions of ASME*, Vol.102: 316-323.

Hirano, T., R. Sato, T. Yamanaka, H. Kotani, and K. Miyamoto. 1997. Measurement of the airflow velocity and turbulent energy around anemostat type diffuser to patch onto CFD. *Proc. Technical Meeting of SHASE-Japan*: 509-512 (*in Japanese*).

Kondo Y., Y. Nagasawa, T. Moriya, M. Sekiguchi, and K. Harimoto. 2001. Modeling of complex ceiling diffuser in CFD - part 1 and part 2. *Proc. Technical Meeting of SHASE-Japan*: 717-724 (*in Japanese*).

Nielsen, P.V., A. Restivo and J.H. Whitelaw. 1978. The velocity characteristics of ventilated rooms, *Journal of Fluid Engineering*, Vol.100: 291-298.

Nielsen, P.V. 1992. Description of supply openings in numerical models for room air distribution, *ASHRAE Transactions*, Vol.98(1): 963-971.

Okaichi A., T. Yamanaka, H. Kotani, and M. Kato. 2000. Study on CFD of rooms with anemostat type diffuser- part 2. *Proc. Technical Meeting of kinki branch of AIJ-Japan*: 245-248 (*in Japanese*).

Okaichi A., T. Yamanaka, H. Kotani, and Y. Momoi. 2001. CFD of airflow in room with complex shaped diffuser - part2. *Proc. Technical Meeting of kinki branch of SHASE-Japan*: 133-136 (*in Japanese*).

Software Cradle Co. Ltd. 2000. STREAM for Windows Version 4 User Guide, Osaka.

Srebric, J., Q. Chen.2001.A method of test to obtain diffuser data for CFD modeling of room airflow, *ASHRAE Trans.*,vol.107(2):108-116.