Building Simulation 2003

Eighth International IBPSA Conference
Eindhoven, Netherlands
August 11-14, 2003

A NEW METHOD OF CFD SIMULATION OF AIRFLOW CHARACTERISTICS OF SWIRLING FLOOR DIFFUSERS

H.T. Xu, and J.L. Niu
Department of Building Services Engineering
The Hong Kong Polytechnic University, Hong Kong
China

ABSTRACT

Air supply diffusers used in air-conditioning systems can be classified as ceiling diffusers, side-wall diffusers, floor diffusers, jet nozzles, and low velocity displacement diffusers. Fixed or adjustable slats are usually used to control airflow directions. Recently, swirling vanes are used in floor diffusers to create swirling outflow jet, so that more rapid mixing with ambient air can be achieved. In this paper we used the latest CFD technique to investigate the airflow pattern and the impact on thermal comfort in the near nozzle region of a floor-level swirl-type diffuser. The preliminary simulation results indicate that re-circulation region in the near nozzle can only be realistically predicted by including the swirling devices in the calculation domain. The results will be further validated with experiments, and the method is expected to be used to help optimize diffuser designs.

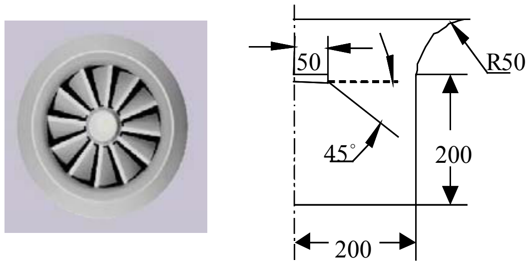
INTRODUCTION

Air diffusers are used widely in air-conditioning systems and the air diffusion is very much influenced by the characteristics of different diffuser designs. For floor-level air supply systems, swirling diffusers are most popular. The method of modeling the diffuser is critical as it has an important impact on the accuracy of the predicted airflow pattern in the room. Computational Fluid Dynamics (CFD) simulation is one of the most useful techniques for predicting the air distribution in the air-conditioned room. Some researchers (Emvinand and Davison 1996; Chen and Jiang 1996; Srebric and Chen 2001) investigated systematically several simplified modeling methods for complex air diffusers. They have identified two simplified methods, the box and momentum methods, to be most appropriate for use in CFD simulations of indoor airflows. When the box method is used in the CFD simulation, it needs the distributions of air velocity, air temperature, and contaminant concentrations around the diffuser. The method on how to determine the box size has been given by some researchers (Srebric and Chen, 2001a). Similarly, the momentum method requires the airflow rate, discharge jet velocity or effective diffuser area, supply air turbulence properties, supply air turbulence properties, supply air temperature, and contaminant concentrations.

Unfortunately, the box method is not suitable for low Reynolds number flows, such as floor-level air supply system. For this system, the buoyancy force strongly influences the jet development from its discharge. Like the displacement diffuser system, the jet changes its profile shape and position very rapidly in front of the diffuser (Jacobsen and Nielsen 1993). The other disadvantage about the box method is that it needs measured data, which may cost long time and may need some expensive equipment. The momentum method is another simplified way to impose boundary conditions for diffusers in CFD simulations. For floor-level air supply diffuser, the air motion near the nozzle region is very important because of its impact on the floor-level's airflow and thermal conditions, not just like near the ceiling diffuser zone that we needn't concern. More exactly known the floor-level airflow, more clearly analyzed the thermal comfort in the occupied zone. The merge of the small jets is accompanied with momentum loss (Lai and Naser 1998). Hence, the momentum is not conserved. Therefore, in this paper we used a drastically new method to simulate the airflow from the swirling-type air diffuser. In this method, the CFD simulation domain is extended into the supply air duct, and the detailed airflows within the diffuser are included using the unstructured-grid technique.

SIMULATION METHODS

In this paper, we simulated a complex swirling diffuser with CFD technique and unstructured grids were employed to represent the complex geometries. As shown in Figure 1, the simulated diffuser contains 12 swirling vanes at the swirling angle of 45°. To simulate the swirling functions, each swirling vane was included in the calculation domain as a solid surface. This obviously required fine grids in the diffuser region, and the grid size was about 5 mm, and in totally, 67291 grids were used in this simulation.



(a) Photo (from TROX)   (b) Section (mm)

Figure 1. The complex swirling air diffuser

For the air flow calculation, standard κ– ε turbulence model (Launder and Spalding 1974) is used and Boussinesq assumption (Tritton 1988) is used to account for the buoyancy effects due to the temperature difference. Also caution is taken in locating the discretization grids in the near wall region to compensate for the deficiencies of the standard k- ε model. A hypothetical office room is simulated. The room is 5.1m in length, 3.6m in width and 2.6m in height. There are two occupants (each occupant generating a convective heat of 50 watts and radiant heat of 25 watts). Heat gains also come from other internal heat sources in convective form. The convective heat is assumed to be 448 watts by 2 large electrical appliances. Figure 2 shows the configuration of the office room.

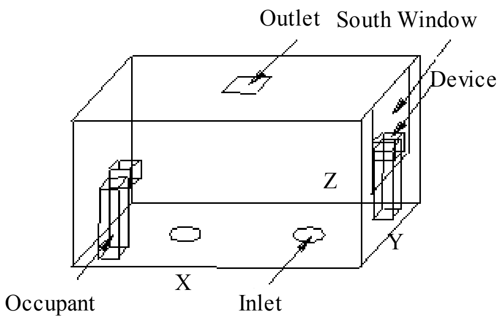
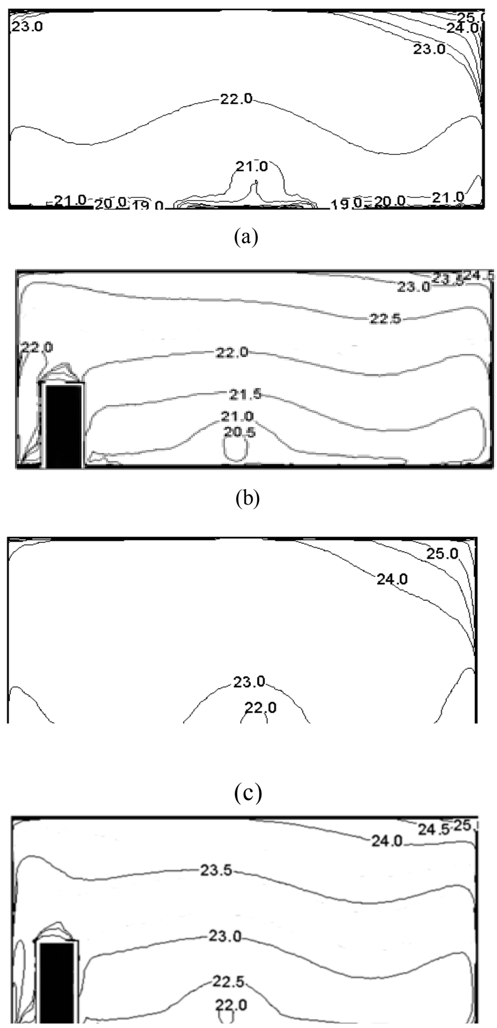


Figure 2. Configuration of the simulated office room

Three room thermal conditions were preset based upon a thermal dynamic simulation of the whole room (Niu et al 2001). Two swirling floor diffusers were used. For this case, the air supply velocity was firstly set at 1.5m/s and the air supply temperatures were set at 18℃, 20℃ and 22℃ respectively and the boundary conditions were kept constant. The simulation conditions are an air temperature of 27±1℃, a relative humidity of 50±20%, and a mean radiation temperature of 27±2℃. These conditions provide a comfortable underfloor air-conditioning environment, according to previous research.

RESULTS AND DISCUSSIONS

As typical underfloor air distribution system, the air temperature near the floor level tends to be low, and there is a vertical air temperature difference between the feet/ankle level and the head level, which is determined by boundary conditions and the inlet conditions. In comparison with low velocity displacement diffuser (Niu et al 2001), this mixing between the air jet and room air is much more rapid, and therefore the vertical temperature gradient is much lower. If the boundary conditions are kept constant, the temperature difference reduces with the raised inlet temperature (Figure 3).

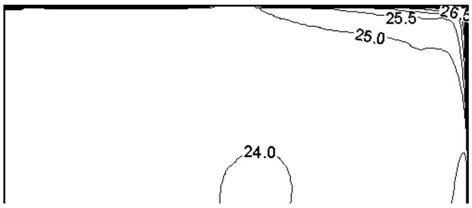


(a)

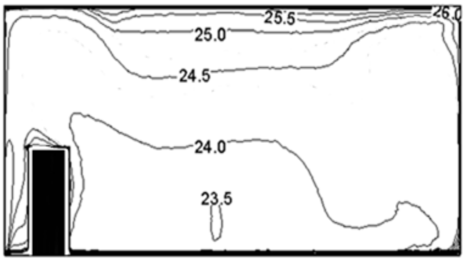
(b)

(c)

(d)



(e)



(f)

Figure 3. Temperature contour

(a)t_{inlet}=18℃,Y=0.0m;(b) t_{inlet}=18℃,Y=-1.3m;

(c)t_{inlet}=20℃,Y=0.0m;(d) t_{inlet}=20℃,Y=-1.3m;

(e)t_{inlet}=22℃,Y=0.0m;(f) t_{inlet}=22℃,Y=-1.3m;

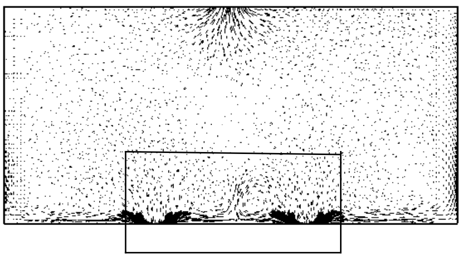


Figure 4a. Velocity vector (Y=0.0m)

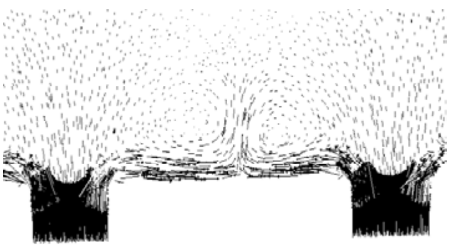


Figure 4b. Partial velocity vector (Y=0.0m)

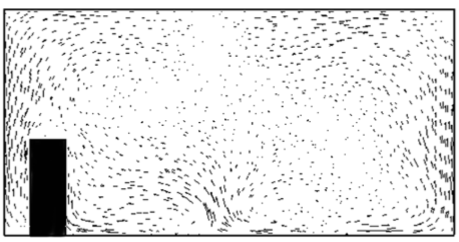


Figure 4c. Velocity vector (Y=-1.3m)

The simulation velocity distribution is typically characteristic by convection flow and buoyancy due to the temperature difference (Figure 4a and 4c). In more detail, Figure 4b shows the velocity near and in the two diffusers. It can be seen that the swirling jet flows along the floor, and right above the diffuser flow re-circulation occurs. This may indicates both the momentum method and the box method would fail to represent this flow feature. Two airstreams are mixed in the middle place of the diffusers and accordingly form eddies.

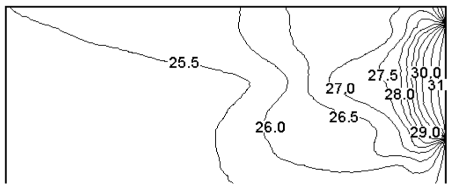
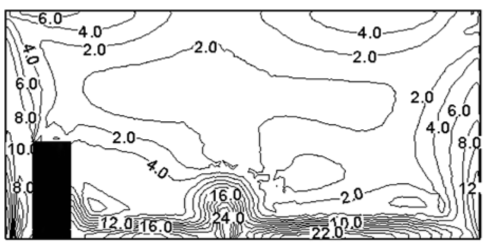


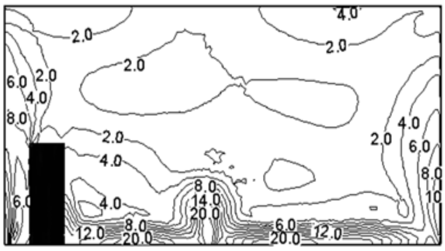
Figure 5. Radiant temperature (Y=0.0m)

The mean radiant temperature distribution pattern (Figure 5) is dominated by the warm window surface. In general, the radiant temperatures near the window are around 5℃ higher than those in the internal areas. It should be noted that in our simulation, only the radiant temperature effects due to internal temperature differences are calculated, but the direct radiant heat from the electrical appliances to people are not calculated. In office environment this is most likely the case.

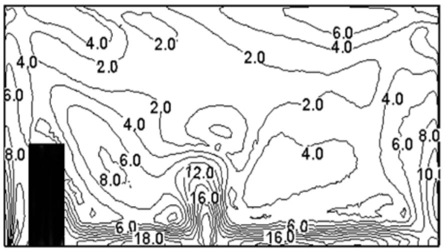
Around the supply nozzle, the local percentage dissatisfied due to draft (PD) tends to be high, due to the combined effects of low temperature, relatively high velocity, and the existence of turbulence. At certain distance away from the nozzle, the draft risks will normally diminish. Raising the supply air temperature from 18℃ to 20 ℃ and 22℃, the PD is reduced and is very uniform in the internal zone (Figure 6a, 6b and 6c). In the middle floor, the PD is very high due to the mixed airflows. Also, from the figure 6d and figure 7, we can see that PD reduces sharply near the floor level if we set the inlet velocity at 1.0m/s and keep the inlet temperature at 20℃.



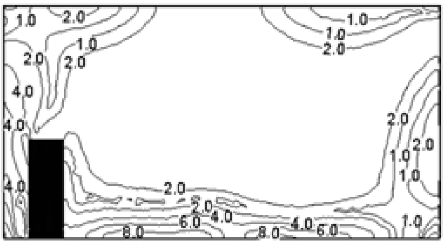
(a)



(b)



(c)



(d)

Figure 6. PD contour due to draft

(a) t_{inlet}=18℃, v_{inlet} =1.5 m/s; (b) t_{inlet}=20℃, v_{inlet} =1.5 m/s;

(c) t_{inlet}=22℃, v_{inlet} =1.5 m/s; (d) t_{inlet}=20℃, v_{inlet} =1.0m/s

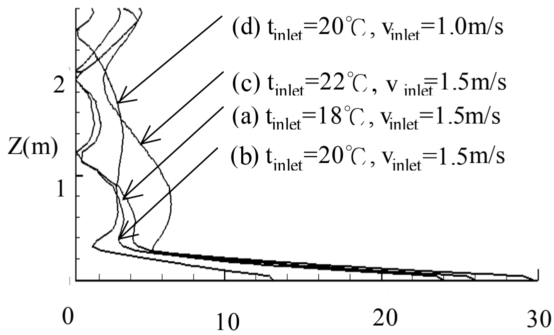


Figure 7. PD due to draft

CONCLUSIONS

The main objective of this article is to describe the method for characterizing swirling air diffusers for CFD simulation of the room airflows. The swirling diffusers used in the under-floor air distribution system can be simulated by unstructured grids including the in-duct airflow. From the simulated results we can see that the velocity distribution pattern around the diffuser zone (Figure 3b) was very complicated. Re-circulation was seen in the core region very close to the nozzle, which cannot be accurately represented by the box and momentum methods. The simulation temperature results also showed the vertical temperature difference exists in the room under the influence of buoyancy force, but the vertical stratification is much weaker than that in the case of low velocity displacement ventilation. Near the floor level, PD is reduced with the supply temperature increased, and the influence of inlet velocity is also another important factor to PD. Simulations of a series of swirling diffusers will be carried-out in the next step and experimental validations will also be performed.

ACKOWLEDGEMENTS

This project is financially supported by the Research Grant Council of the Hong Kong SAR government and the project number is PolyU 5028/00E (B-Q369).

REFERENCES

Chen, Q. and Jiang, Zh. 1996. Simulation of a Complex Air Diffuser with CFD Techniques. *Proc of Roomvent'96.* Vol.1, pp.227-234.

Emvin, P. and L. Davidsen. 1996. A Numerical Comparison of the Diffuse in Case EI ANNEX20. *Proc. of Roomvent'96.* Vol.1, pp.219-226.

Jacobsen, T.V. and P.V. Nilelsen. 1993. Numerical Modeling of Thermal Environment in a Displacement-Ventilated Room. *Proceedings of Indoor Air'93:5.*

Lai, J.C. and Naser, A. 1998. Two Parallel Plane Jets: Comparison of the Performance of Three Turbulence Models. *Proc Instn Mech Engrs. Part G J. Aerospace Engineering.* Vol.212, pp.379-391.

Launder, B.E., and Spalding, D.B. 1974. The Numerical Computation of Turbulent Flows. *Computer Methods Appl. Mech. Engr.*. Vol.3, pp.269-289.

Niu, J.L, Zuo, H.G. and J. Burnett. 2001. Simulation Methodology of Radiant Cooling with Elevated Air

Movement. 7^{th} *International IBPSA Conference.* pp 265-272.

Niu, J.L., and Kooi, J.V.D. 1992. Grid Optimisation of k-ε Turbulence Modelling of the Natural Convection in Rooms. *Proceedings ROOMVENT-92: Air Distribution in Rooms - Third International Conference*. Vol.1, pp.207-pp.223.

Srebric, J. and Chen, Q. 2001. A Method of Test to Obtain Diffuser Data for CFD Modeling of Room Airflow. *ASHRAE Transaction.* Vol.107, Part2, pp.108-116.

Srebric, J. and Chen, Q. 2001. Boundary Conditions for Diffusers in Room Air Distribution Calculations. *Clima 2000/Napoli 2001 World Congress.*

Srebric, J. and Chen, Q. 2002. Simplified Numerical Models for Complex Air Supply Diffusers. *HVAC&R Research.* Vol.8, No.3, pp.277-294.

Suhas V. Patankar. *Numerical Heat Transfer and Fluid Flow*. 1989.

Tritton, D.J. 1988. Convection, Physical fluid dynamics. 2^{nd} *edition, Chapter 14.* pp.163-165.