CFD simulation of airflow characteristics of swirling floor diffusers

H.T. Xu*, J.L. Niu

Department of Building Service Engineering, The Hong Kong Polytechinc University, Hong Kong, P. R. 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 a swirling out-flow jet, so that more rapid mixing with ambient air can be achieved. In this paper, we use the latest CFD technique to investigate the impact of these different designs on thermal comfort in the near nozzle region, in view of the increasing application of floor-level air supply systems. The preliminary simulation results indicate that raising air supply temperature from 18 to 22°C or reducing the air supply velocity from 1.5 to 1.0 m/s can provide better thermal comfort in terms of thermal field uniformity.The results will be further validated with experiments, and the method is expected to be used to help optimize diffuser designs.

INDEX TERMS

Air distribution; CFD simulation; Underfloor air supply

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 modelling 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 (Chen and Jiang, 1996; Emvin and Davison, 1996; Srebric and Chen, 2001b) investigated systematically several simplified modelling 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

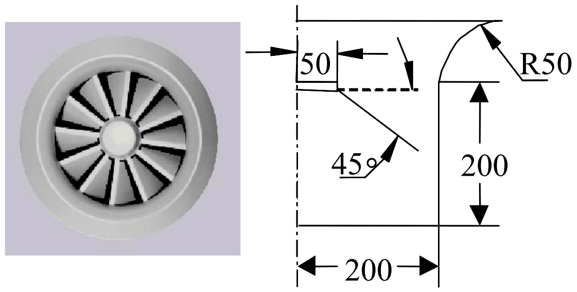
* Corresponding author.

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 take a 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 a 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 need not be concerned about. More exactly known is the floor-level airflow and more clearly analysed is the thermal comfort in the occupied zone. The merging 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 of each was about 5 mm; and in total 67 291 grids were used in this simulation.



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

Figure 1. The complex swirling air

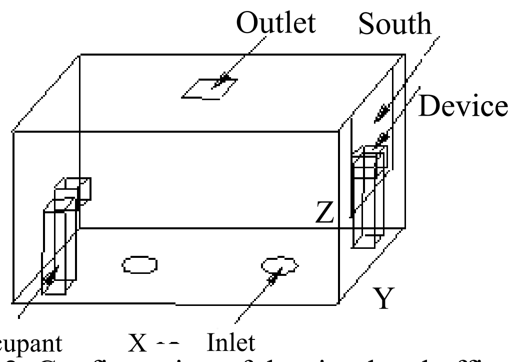


Figure 2. Configuration of the simulated office

For the airflow calculation, the standard k–ε turbulence model (Launder and Spalding, 1974) is used and the 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 5.1 m in length, 3.6 m in width and 2.6 m

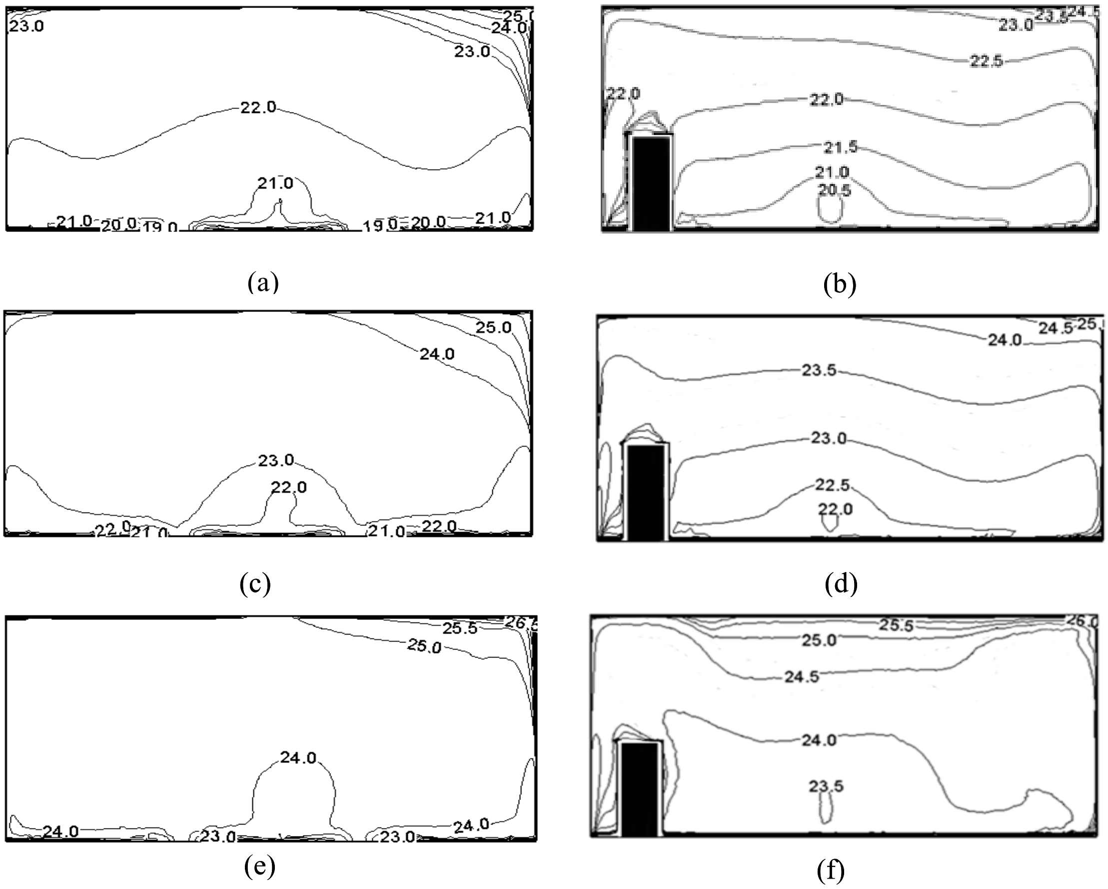


Figure 3. Temperature contour inside the simulation room
(a) t_{inlet}=18°C and Y=0.0m; (b) t_{inlet}=18 °C and Y= - 1.3m;
(c) t_{inlet}=20 °C and Y=0.0m; (d) t_{inlet}=20 °C and Y=- 1.3m;
(e) t_{inlet}=22 °C and Y=0.0m; (f) t_{inlet}=22 °C and Y=- 1.3m.

in height. There are two occupants (each occupant generating a convective heat of 50 W and radiant heat of 25 W). Heat gains also come from other internal heat sources in convective form. The convective heat is assumed to be 448 W by two large electrical appliances. Figure 2 shows the configuration of the 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 1.5 m/s and the air supply temperatures were set at 18, 20 and 22°C and the boundary conditions were kept constant. The simulation conditions are an air temperature of 27 ±1 °C, a relative humidity of 50 ± 20% and a mean radiation temperature of 27 ± 2°C. These conditions provide a comfortable underfloor air-conditioning environment, according to previous research.

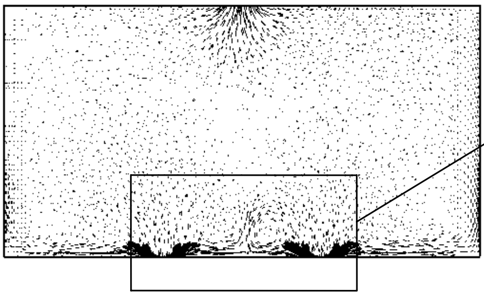


Figure 4a. Velocity vector (Y=0.0m)

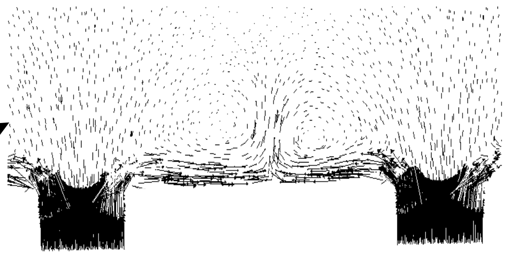


Figure 4b. Partial velocity vector (Y=0.0m)

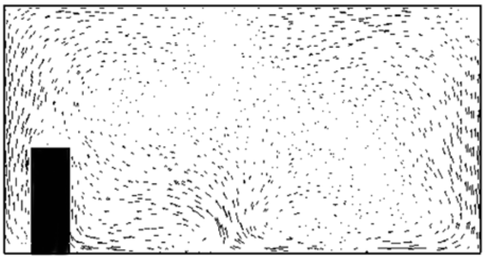


Figure 4c. Velocity vector (Y=-1.3m)

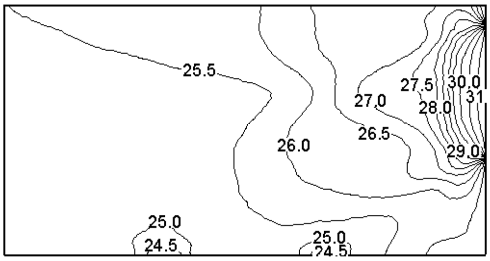


Figure 5. Radiant temperature (Y=0.0m)

RESULTS AND DISCUSSION

As a 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).

The simulation velocity distribution is typically characteristic by convection flow and buoyancy due to the temperature difference (Figure 4a and c). In more detail, Figure 4(b) 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 indicate both the momentum method and the box method would fail to represent this flow feature. Two air streams are mixed in the middle place of the diffusers and accordingly form eddies.

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°C 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 environments 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 a certain distance away from the nozzle, the draft risks will normally diminish. Raising the supply air temperature from 18 to 20 and 22°C, the PD is reduced and is very uniform in the internal zone (Figure 6a–c). In the middle floor, the PD is

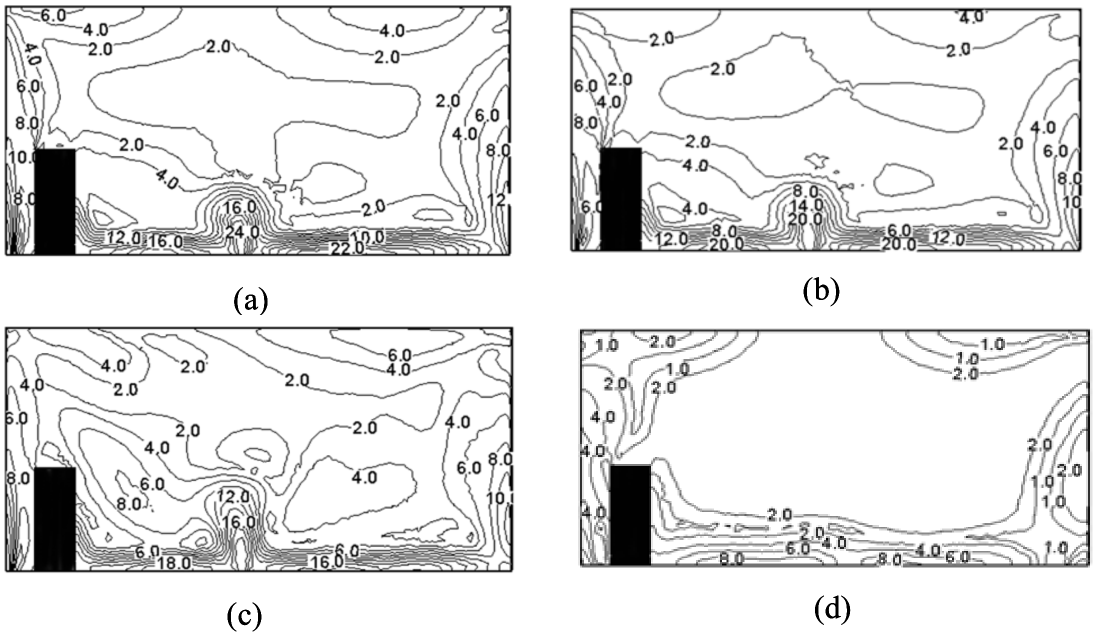


Figure 6. PD contour due to draft

(a) t_{inlet}=18 °C, V_{inlet} = 1.5m/s; (b) t_{inlet}=20°C, V_{inlet} = 1.5m/s;

(c) t_{inlet}=22°C, V_{inlet} = 1.5m/s; (d) t_{inlet}=20°C, V_{inlet}=1.0m/s .

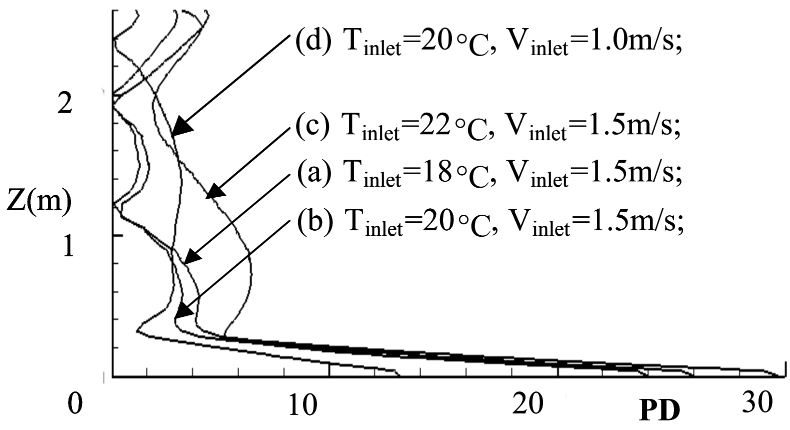


Figure 7. PD due to draft (X= -0.8m, Y=-0.9m)

very high due to the mixed airflows. Also, from Figures 6(d) and 7, we can see that PD reduces sharply near the floor level if we set the inlet velocity at 1.0 m/s and keep the inlet temperature at 20°C.

CONCLUSIONS

The main objective of this article was to describe the method for characterizing swirling air diffusers for CFD simulation of the room airflows. The swirling diffusers used in the underfloor air distribution system can be simulated by unstructured grids including the in-duct airflow instead of the box and momentum methods. From the simulated results we can see that the velocity distribution pattern around the diffuser zone (Figure 3b) was very complicated, which was combined with the vertical flow downward and the mixture of the two diffuser airflows. Obviously, this cannot be exactly illuminated by the box and momentum methods. The simulation temperature results also showed that the temperature difference exists in the room and the influence of buoyancy force is very strong. Near the floor level, PD is reduced with the temperature increase and the influence of the inlet velocity is also another important factor for PD. A series of swirling diffusers' simulation will be carried out in the next steps and will also include experiments.

ACKOWLEDGEMENTS

This project is financially supported by the Research Grant Committee 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. *Proceedings of Roomvent'96*, Vol. 1, pp. 227–234.

Emvin, P. and Davidson, L. (1996). A numerical comparison of the diffuse in case EI ANNEX20. *Proceedings of Roomvent'96*, Vol. 1, pp. 219–226.

Jacobsen, T.V. and Nilelsen, P.V. (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. *Proceedings of the Institution of Mechanical Engineers. Part G: Journal of Aerospace Engineering* 212, 379–391.

Launder, B.E. and Spalding, D.B. (1974). The numerical computation of turbulent flows. *Computer Methods in Applied Mechanical Engineering* 3, 269–289.

Niu, J.L. and Kooi, J.V.D. (1992). Grid optimisation of k–ε turbulence modelling of the natural convection in rooms. *Proceedings of ROOMVENT-92: Air Distribution in Rooms—Third International Conference*, Vol. 1, pp. 207–223.

Niu, J.L., Zuo, H.G.. and Burnett, J. (2001). Simulation methodology of radiant cooling with elevated air movement. *7th International IBPSA Conference*, pp. 265–272.

Srebric, J. and Chen, Q. (2001a). A method of test to obtain diffuser data for CFD modeling of room airflow. *ASHRAE Transaction* 107 (Part 2), 108–116.

Srebric, J. and Chen, Q. (2001b). 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* 8 (3), 277–294.

Suhas, V.P. (1989). *Numerical Heat Transfer and Fluid Flow*.

Tritton, D.J. (1988). *Convection, Physical Fluid Dynamics*, 2nd edn, Chapter 14, pp. 163–165.