AIRFLOW PATTERN IN AN AIR-CONDITIONED SEMINAR ROOM

K W D CHEONG, S C SEKHAR, K W THAM AND E DJUNAEDY

School of Building and Real Estate
National University of Singapore
10 Kent Ridge Crescent
Singapore 119260

ABSTRACT

The pattern of airflow influences the propagation of airborne pollutants, the thermal environment and general comfort conditions. In designing a good HVAC system, it is ideal to determine the airflow distribution in the occupied zone to ensure good quality of air and comfort condition are provided to the occupants. In most instances, it may not be feasible to conduct such study experimentally. This paper presents an investigation on the predictions of air movement within a room and compared them with the physical measurements.

This study is carried out in a seminar room at a University. The 3D turbulent re-circulating airflow in the room was numerically simulated based on the site measurements, such as the air velocities at the air supply grilles. The accuracy of numerical prediction is influenced by several parameters. Some of these parameters include inlet airflow modelling, meshing procedure and turbulence model. In this paper, the inlet airflow modelling will be discussed in detail.

The predicted results of airflow pattern were validated against the measured data. Results show that the methods of modelling at the inlet diffuser has great impact on the air flow pattern within the room and therefore affect the accuracy of the air velocity prediction.

Keywords : airflow, air-conditioned, CFD, classroom, diffuser.

INTRODUCTION

An accurate understanding of indoor air distribution is crucial to the design of heating, ventilating, and air-conditioning (HVAC) systems in providing thermal comfort and indoor air quality. The effectiveness of an HVAC system with respect to comfort and contaminant control is subject to the manner in which supply air is circulated within the occupied zone and exhausted via the exhaust grille. The pattern of air circulation is influenced by locations of air inlets and outlets, room geometry, interior furnishings, HVAC systems in the building, and other factors.

Numerical prediction of air flow patterns in mechanically ventilated rooms has been a research direction for almost two decades. Since 1970s, Computational Fluid Dynamics (CFD) showed that it was capable to predict the flow field in large domains with relatively small openings [1-2]. In the recent years, the field of ventilation engineering has started to use CFD as a design and analytical tool since it offers a radical change in available analytical tools. The engineer can predict the impact of a certain design of an air-conditioning system on the indoor climate and the energy management of real buildings. CFD has for many years been applied in the simulation of room air movement in air-conditioned spaces. In the earlier applications, Chen and Kooi [3] and Chen, Meyers and Kooi [4] describe the analysis of 3-dimensional flow in an office type room with cooling using CFD analysis combined with experimental verification. In some recent applications of CFD, Smith et al [5] used CFD to predict airflow and pressure distribution in ducts. Predictions were compared with results obtained experimentally and CFD was found to be a useful method for the prediction of pressure loss coefficient (k-factors) of duct fittings. Schulte et al [6] has illustrated the usefulness of CFD as a design tool for ventilation system. It enables the velocity and temperature fields to be investigated in significantly greater detail than is possible with either analytical or experimental models.

The aim of the work is to compare the predicted results of the CFD modelling with that of the measurements. The collated results of the airflow pattern in the seminar room will be used to assess the thermal comfort experienced by the occupants.

METHODS

The Seminar Room

Experimental measurements and numerical simulation were carried out in the seminar room of size 7.5m x 5m x 2.6m in height. The room is served by a variable air volume box via four air supply square diffusers (four-way diffusers) and air is exhausted via two return grilles. The room has desks and chairs to accommodate up to 20 occupants. The activity in the room is usually sedentary with heat generation from the overhead projector during presentations from the students, fluorescent lamps and solar heat gain from the sun. The plan of the ceiling and the furniture arrangement in the seminar room are shown in Figures 1 and 2 respectively.

Measurements of airflow and temperature in the room

Airflow and temperature measurements were conducted at five vertical sections. Figure 3 shows the Sections A, B, C, D and E. Each section has 24 air velocity and temperature measuring points at 4 different heights (0.1 m, 0.6 m, 1.1 m, and 1.7 m from the floor level). These measurements were conducted for both isothermal and non-isothermal conditions. Under isothermal conditions, heat gained from artificial lighting, overhead projector, occupants will not be taken into consideration. The difference between the supply air temperature and the room air temperature is negligible (less than 1° C).

The room air velocity was measured using the omni-directional anemometer. The air temperatures were measured using thermocouples that were connected to a data-logger. In some critical locations, smoke visualisation tests were conducted to assist in the confirmation of the airflow pattern in the room. In addition, airflow rates at all the diffusers were measured using flow-hood and vane anemometer. These values were used in the numerical simulation of the model.

Numerical Simulation

Air velocity and temperature simulations under several conditions were studied using CFD techniques. However, only the isothermal case is presented in this paper. Three-dimensional air velocity distribution in the seminar room model was analysed using steady-state analysis technique. Effect of the airflow turbulence was assumed by adopting standard k-epsilon two-equation model using the finite volume method based on SIMPLE algorithm [6]. The grid was generated automatically using an unstructured grid generator software. The resulted models contained about 350,000 cells. The effects of different methods applied in the modelling of the air inlet conditions were explored. This may have great impact on the airflow pattern within the room and therefore affect the accuracy of the airflow predictions.

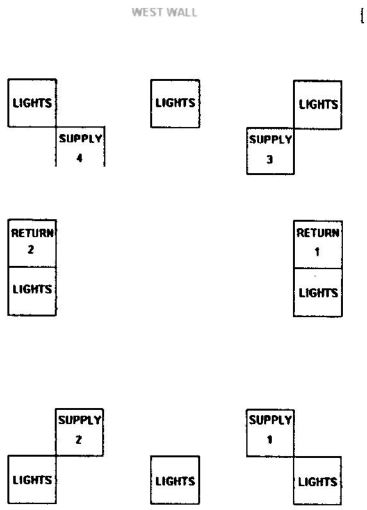


Fig. 1 Ceiling Plan

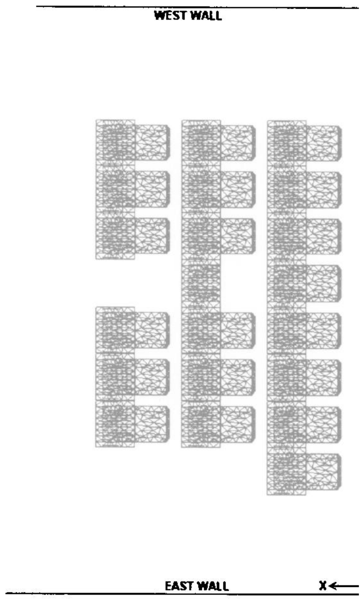


Fig. 2 Furniture Arrangement

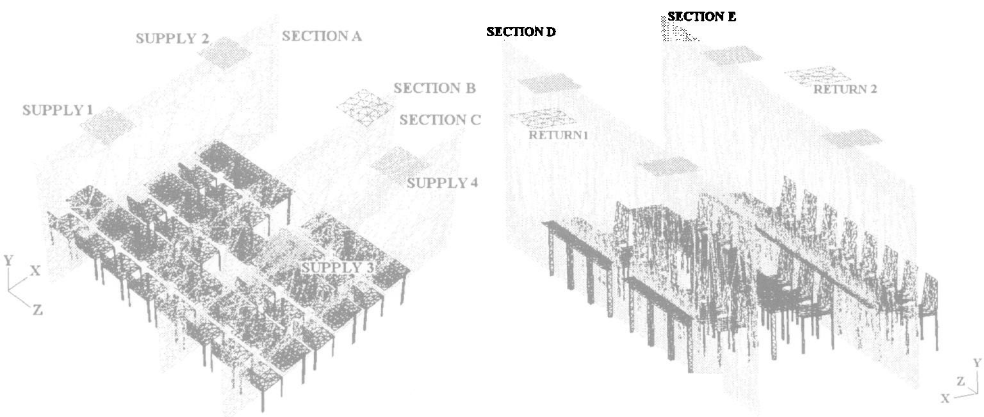


Fig. 3 Measurement Sections at A, B, C, D, and E

In the first model, the inlet supply was modelled as uniform air distribution across the area of the diffuser with air supplied vertically downward. In the second model, each diffuser was divided into four sections (see Figure 4).

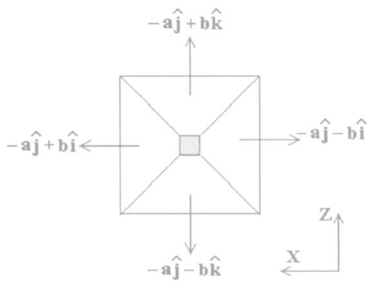


Fig. 4 Supply Diffuser (Second Model)

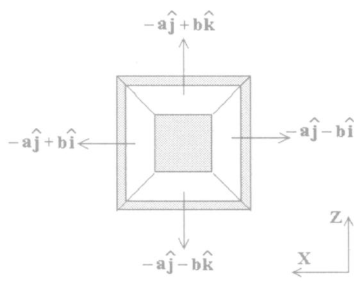
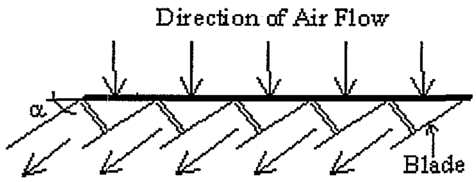


Fig. 5 Supply Diffuser (Third Model)



When the air flow is deflected at the blade, the effective area [double thin line] will be reduced by a factor of sin (α) compared to the original area [bold line]. Total effective area is the sum of the individual effective areas.

Fig. 6 Effective Area of Diffuser

The velocity remained uniform across the area of the diffusers but each section was assigned to supply air at different directions. The deflection angle of the diffuser's blade is taken to be 30° downward from the ceiling level. The third model is similar to the second except that the total area of the supplied air is reduced (Figure 5), taking only the effective area in the direction of the flow (Figure 6 shows the effective area of the diffuser). These three methods are basically known as the basic model [7]. The fourth model used the actual measured velocity and considered the maximum velocity as a constant uniform value across the effective area of the diffusers.

RESULTS AND DISCUSSION

In this paper, only Section C is used for discussion. Figure 7 shows the results of the air velocity measurements for the isothermal condition. It was observed that the air velocity is lower at the zone immediately below the diffusers. However, the air velocity is relatively higher at the zones around the diffuser due to the deflected airflow caused by the blade of the diffusers. It was observed that the air velocity is higher near to the floor. Figure 7 shows that occupants sitting in the first row may be thermally uncomfortable due to the draughty conditions (0.5m/s - 0.58 from head to the feet). This draughty condition was experienced by occupants sitting in the last row.

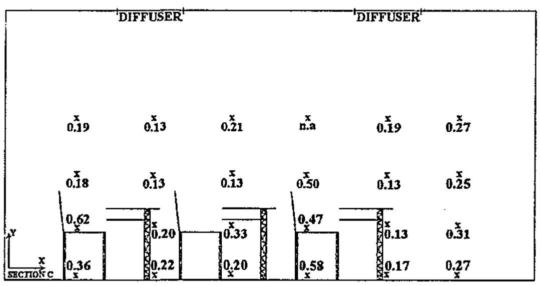


Fig. 7 Measured Velocity Profile at Section C

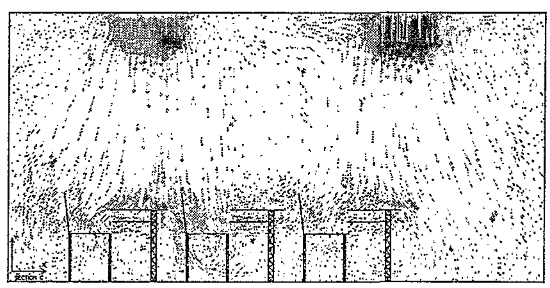


Fig. 8 Predicted Air Flow (First Model)

Figure 8 shows the numerical results for the supply inlet using the first method. This model did not consider the deflection angle of the diffusers at the inlet flow and the results predicted were obviously very different from the measured air flow pattern.

Figure 9 shows the result of the second model with provision made for the deflection angles of the diffuser. The flow was prescribed and the total face area of the diffuser was used and the resultant air velocity was low. In this case, the inlet momentum was significantly underestimated and the throw of the air-jet from the diffusers did not prevail. The provision of deflection effect by the blade did not produce actual air flow pattern in the seminar room.

In the third model, the total face area was reduced to a smaller effective area and would produce a higher inlet velocity at the diffuser. Figure 10 shows that the predicted results of the third model. The predicted results show an up-flow circulation of air at the top corner of the walls but this did not prevail in the smoke visualisation test (see the direction of the flow in Figure 10). This shows that the provision of the effective area had still resulted in an underestimation of the inlet momentum and the throw could not reach the side of the walls.

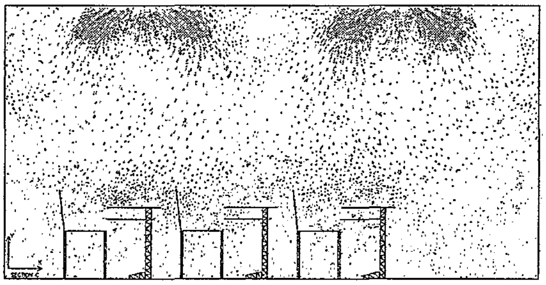


Fig. 9 Predicted Air Flow (Second Model)

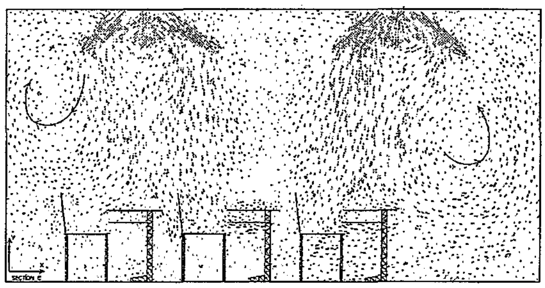


Fig. 10 Predicted Air Flow (Third Model)

The fourth model is an attempt to prescribe the measured the face velocity of the diffuser to the inlet model. The maximum measured velocity was used to overcome the underestimation of the momentum. Figure 11 shows that the predicted results were in close correlation to that of the measured results.

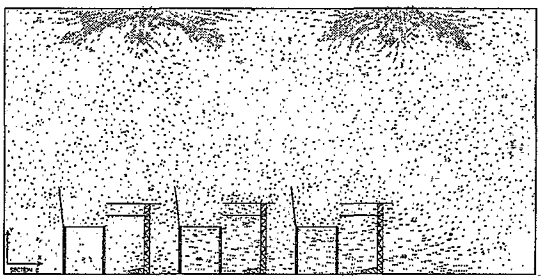


Fig. 11 Predicted Air Flow (Fourth Model)

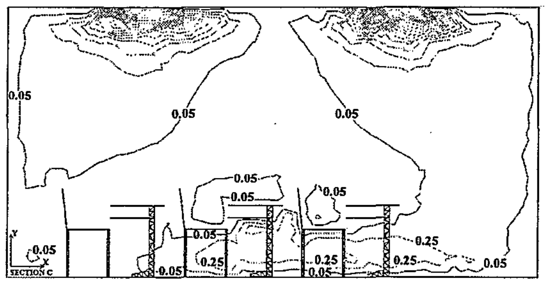


Fig. 12 Velocity Contour (Fourth Model)

The throw produced by the diffuser could reach the top corner of the side walls and moves downward towards the floor and this is similar to that of the air velocity measurements. In addition, several high velocity regions were predicted in the area near the floor level.

However, the quantitative values of the air velocity predicted (Figure 12) does not yield the same measured values (Figure 4) in absolute terms. The high velocity region near the floor level in the predicted results has lower velocity compared to the measured results although this model has the highest initial momentum as compared to the earlier model. Further increase of velocity at the inlet was not possible without reducing the model area, since the total flow rate will not be the same as the measured flow rate.

One alternative solution to overcome this problem is to introduce the velocity distribution in the inlet model instead of using the uniform velocity across the air supply diffuser. The velocity distribution can give us the higher momentum without sacrificing the flow rate.

CONCLUSION

The measured data has been used to validate some numerical models. This validation is important to ensure that CFD methods can be used to simulate the actual condition. This paper has shown that this method can be used with confidence to predict the airflow qualitatively.

The measured results show that high air velocity region was observed near the chairs and floor level. This could cause discomfort to the occupants sitting in the first and last row due to the draughty condition (0.36 - 0.58 m/s).

Several models of inlet supply have been used to achieve a better air velocity prediction with the numerical method. This is to emphasize the importance of detailed inlet supply specification (as part of the boundary conditions) in the accuracy of the prediction. It was shown that the provision of effective area in the supply model (3rd model) is not sufficient to improve the predicted result. It was also shown that the uniform air velocity distribution across the area of the inlet model gave reasonable results (4th model). This can be further improved by adopting another inlet model to introduce the varying velocity distribution across the face of the air supply diffuser instead of using the uniform velocity. In addition, questionnaire survey will be conducted to evaluate on the occupants' perception of the thermal comfort in this seminar room.

ACKNOWLEDGEMENTS

This work is funded by the National University of Singapore under research project RP972054.

REFERENCES

1. Nielsen, P V, Restivo, A and Whitelaw, J H. 1978. The velocity characteristics of ventilated rooms, *ASME Journal of Fluid Eng.*, Vol 100, pp 291.
2. Gosman, A D, Nielsen, P V, Restivo, A and Whitelaw, J H. 1980. The flow properties of rooms with small ventilation openings, *ASME Journal of Fluid Eng.*, Vol. 102.
3. Chen, Q and Van der Kooi, J. 1988. Accuracy of a program for combined problems of energy analysis indoor air flow and air quality. *ASHRAE (American Society of Heating, Refrigerating and Air-conditioning Engineers) Transactions*, Vol. 94(2).
4. Chen, Q, Meyers, C A and Van der Kooi, J. 1989. Convective heat transfer in rooms with mixed convection. *International Seminar on Indoor Air Flow Patterns*, University of Liege, Belgium.
5. Schulte, T E, Bergstrom D J and Bugg J D. 1998. Numerical simulation of three-dimensional airflow in unfurbished rooms. *ASHRAE Transactions*, Vol. 104(2), pp 68-76.
6. Patankar, S V. 1980. *Numerical heat transfer and fluid flow*. McGraw-Hill, New York.
7. Nielsen, P.V. 1991. Models for the Prediction of Room Air Distribution. *12th AIVC Conference.*