AIR MOVEMENT & VENTILATION CONTROL WITHIN BUILDINGS
12th AIVC Conference, Ottawa, Canada
24-27 September, 1991

POSTER 1

SIMULATION OF A MULTIPLE-NOZZLE DIFFUSER

(IEA Annex 20, Subtask 1, Research Item 1.20)

Qingyan Chen* and Alfred Moser

Swiss Federal Institute of Technology Zurich
Energy Systems Laboratory
ETH-Zentrum, CH-8092 Zurich, Switzerland

---

* Present address:
TNO Institute of Applied Physics (TPD)
P. O. Box 155, 2600 AD Delft, The Netherlands

SYNOPSIS

The air diffusion in a room is dominated by the diffuser type and the air supply parameters of the diffuser. The design of a diffuser used in practice is often complex. Reliable information from simulations can be useful in improving diffuser design. And the correct airflow pattern computed can help us to understand the flow phenomenon in order to obtain a comfortable indoor environment. Accurate flow information around the diffuser is therefore required.

A "HESCO"-type diffuser was selected as an example for the validation exercise in the IEA Annex 20 project (Air flow pattern within buildings). It consists of 84 small round nozzles that are arranged in four rows in an area of 0.71 m x 0.17 m. With the same effective area, the diffuser is simulated by 1, 12, and 84 simple rectangular slots and by the momentum method. In the momentum method, the supply air momentum is set to be that of the 84 small round nozzles. The simulation of the diffuser is incorporated in the airflow computation in a room. The three dimensional flow is computed by a flow program with a low-Reynolds-number k-ε model of turbulence.

Corresponding measurements from other researchers are used for comparison. It may be concluded that the momentum method and the method which uses 84 simple rectangular slots predict air velocity and temperature distributions in the room similar to those from the experiments. The computing cost with the 84 slots method is extremely high. Hence, the momentum method is suggested to be used to simulate a complex diffuser in practice.

INTRODUCTION

The air diffusion in a room is dominated by diffuser type and the air supply parameters of the diffuser. The difficulty in analysis of airflow around a jet diffuser is that the geometry configuration of a diffuser used in practice is complex, and the flow is three-dimensional, turbulent, and with a considerably high Reynolds number. It is necessary to acquire accurate flow information around the diffuser and to simulate the diffuser correctly in airflow computations.

There are several methods applicable for simulating a complex diffuser. The inlet box model used by Nielsen *et al* (1979) is one of the earliest models. In principle, the model can be used for predicting room air motion with any kind of inlet diffuser. But the data used in the inlet box model must be obtained from experiment or a more detailed computation. However, the data needed in the model may not be always available.

Recently, Nielsen (1990) imposed a formula for a jet diffuser based the results from experiment. Lemaire (1990) applied the formula to prescribe velocity profile of a diffuser so that room airflow can be computed. The results are promising. Since the formula varies with the diffuser type, this method requires a large amount of time and effort.

Heikkinen (1990a) simulated a complex diffuser by a so-called basic model. In the basic model, the diffuser is simulated by a rectangular slot with the same effective area as the complex diffuser. The method was used to predict the airflow in a room with a complex diffuser under iso-thermal condition. It gave a reasonable indication of the airflow pattern in the room although there are discrepancies between the computations and experiment. But the method is not suitable for non-isothermal flow as will be discussed in this paper.

The aim of the paper is to find a suitable tool to simulate the air velocity and temperature distribution in a room with a complex jet diffuser. The tool could be used to improve diffuser design. And the correct airflow pattern computed can give a better understanding of the flow phenomena which may be useful in obtaining a comfortable indoor environment.

SIMULATION OF A MULTIPLE-NOZZLE DIFFUSER

A "HESCO"-type diffuser as shown in Figure 1a was selected as an example for the validation exercise in the IEA Annex 20 project. The diffuser is a modern air terminal device and available on the market. It consists of 84 small round nozzles that are arranged in four rows in an area of 0.71 m x 0.17 m. The flow direction of each nozzle can be adjusted, and a flow which has a complicated three-dimensional structure close to opening with a high entrainment of room air may result.

With the same effective area, the diffuser is simulated by two methods: the simple-rectangular-slot method and the momentum method. As shown in Figures

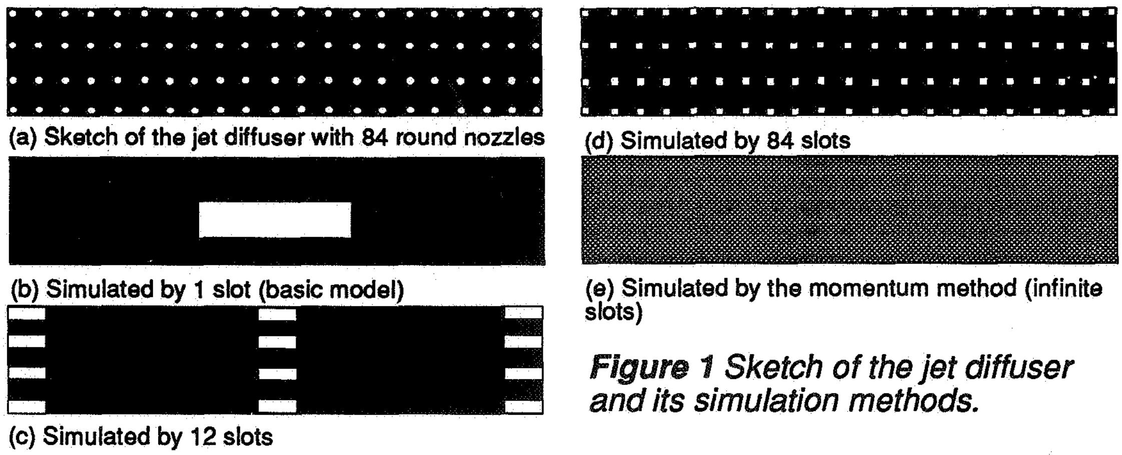

***Figure 1** Sketch of the jet diffuser and its simulation methods.*

1b-d, the 84 round nozzles can be simulated by 1, 12, or 84 rectangular slots, respectively. The total effective area of the 1, 12, or 84 slots is the same as that of the 84 round nozzles. The one slot method is defined as the basic model in the IEA Annex 20 project. In the momentum method, the supply air momentum, mV_{in}, is set to be that of the 84 small round nozzles:

$$m\, V_{in} = m \text{ (volume inflow rate / effective area)} \tag{1}$$

where m is the mass inflow rate. This method can be regarded as setting infinite nozzles/slots as shown in Figure 1e. In the numerical approach, it is performed by characterizing the flow rate in the inlet with a fraction of the effective area over gross area of the diffuser. The fraction indicates the area of the grid cells within the inlet area available for the supply air. Different diffusers can be simulated by giving different supply momentum and its initial directions, .

In the present study, the low-Reynolds-number k-ε model of turbulence is used (Chen *et al.* 1990) to predict turbulent diffusion in airflow computation. This model has been verified to be more suitable for predicting indoor airflow and heat transfer since the agreement between the computed and measured results is very good. A more detailed description of the model and the comparison between the computed results and experimental data are given by Chen *et al.* (1990) and Borth (1990). The turbulence model was implemented in the airflow program PHOENICS-84 (Chen 1990) that is developed by Rosten and Spalding (1987). The computations involve the solution of three-dimensional equations for the conservation of mass, momentum (u, v, w), energy (H), turbulence energy (k), and the dissipation rate of turbulence energy (ε). The governing equations of the model can be expressed in a standard form:

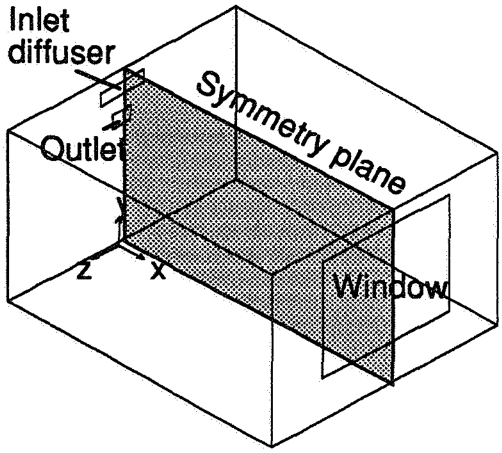

Figure 2 *Sketch of the room with a jet diffuser.*

$$\text{div}\,(\rho\,\vec{V}\,\phi - \Gamma_\phi\,\text{grad}\,\phi) = S_\phi \tag{2}$$

where ρ is the air density, $\vec{V}$ is the air velocity vector, Γ_ϕ is the diffusive coefficient, S_ϕ is the source term of the general fluid property, and ϕ can be any one of 1, u, v, w, k, ε, or H. When ϕ = 1, the equation changes into the continuity equation.

RESULTS

For more reliable information, the simulation of the HESCO diffuser is incorporated in the airflow computation in a room as shown in Figure 2. The room is 4.2 m long, 3.6 m wide, and 2.5 m high with the diffuser placed in the rear wall near the ceiling. Since the room is symmetrical in plane z = 0, the computations are carried out for half of the room. The front wall has a hot window surface for modeling a summer cooling situation. The window size is 2.0 m in width and 1.6 m in height with a surface temperature of 30°C and the rest enclosure surfaces 20°C. All 84 nozzles are adjusted with an angle 40° upwards. The actual inlet area (effective area) is 0.008 m^2. The air change rate of the room is 3 ach which corresponds to a mass inflow of 0.0315 m^3/s. This implies that the mean velocity of the supply air is 3.94 m/s. The turbulence intensity of the supply air is estimated to be 10%. The supply air temperature specified by IEA Annex 20 is 15.0°C. An outlet is placed below the inlet with a dimension of 0.3 m in width and 0.2 m in height. Corresponding measurements, which were carried out in different IEA Annex 20 participating countries (Fossdal 1990; and Heikkinen 1990), are used for comparison.

The same grid distribution (22 x 34 x 34) was used for all the computations for half of the room to eliminate the error caused by grid diffusion. In the one-slot method (basic model), the slot was simulated by 7 x 4 cells. Each slot has 4 x 2 cells in the 12-slot method. However, only one cell is used for each slot in the 84-slot method because of the huge computing cost. The round nozzles are simulated by rectangular ones as shown in Figures 1b, 1c, and 1d. More than five hours of CPU time is used for each computation in a CRAY X/MP super computer. The sum of the absolute mass residuals of each cell is about 10% of the mass inflow and the continuity error between the inlet and outlet is 0.5% of the mass inflow.

The computational results are illustrated in Figures 3 to 6. Figures 3a, 4a, 5a, and 6a are the velocity vectors in plane z =0.0 m (the symmetrical plane); Figures 3b, 4b, 5b, and 6b the velocity vectors in plane x = 4.10 m (near the window); Figures 3c, 4c, 5c, and 6c the velocity vectors in plane y = 2.425 m (near the ceiling via the inlet); Figures 3d, 4d, 5d, and 6d the velocity vectors in plane y = 0.10 m (near the floor); and Figures 3e, 4e, 5e, and 6e and 3f, 4f, 5f, and 6f the velocity scalar and air temperature in the symmetrical plane respectively. The results predicted by the one-slot and 12-slot methods are quite different from those by the 84-slot and momentum methods. The later two methods give similar results.

The computed results have been compared with the experiments carried out by Fossdal (1990), and Heikkinen (1990). Although the experiments are conducted on the same specification by IEA Annex 20, there are differences between the measurements as shown in Figures 7 and 8. The comparisons are only indicated for the symmetrical plane because it is the most important plane. We have noted that there is an asymmetry in the experiments but the asymmetry is not significant. The comparison for the velocity profiles concludes that the 84-slot and momentum methods are in reasonable agreement with the measurements. In general, the computed air temperature is about 1 K lower than the measured (Figure 8). There are differences in the thermal and flow boundary conditions as given in Table 1.

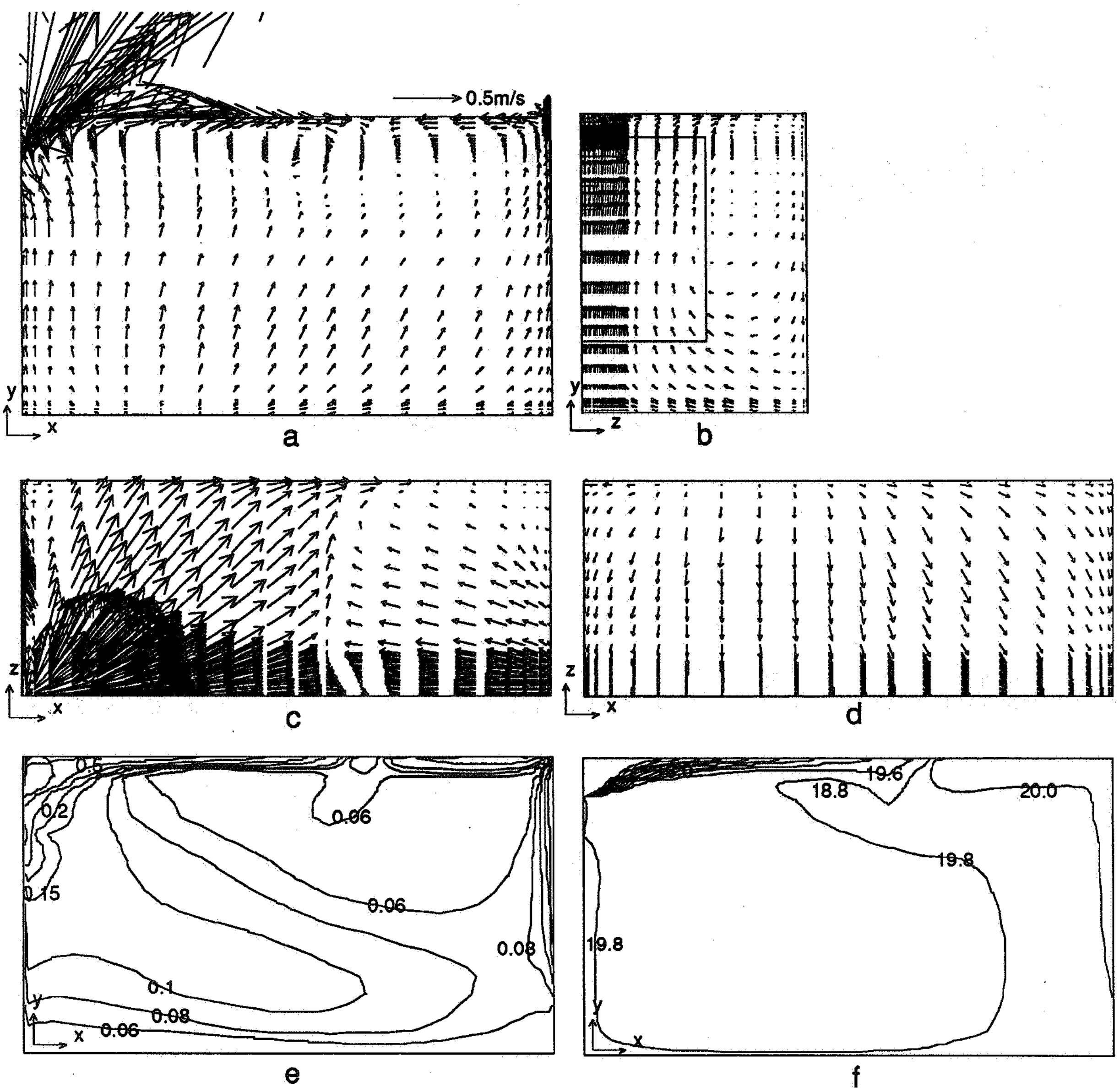

Figure 3 *Velocity and temperature distributions simulated by the one-slot method. (a) velocity vectors in plane z = 0.0 m (symmetrical plane); (b) velocity vectors in plane x = 4.10 m; (c) velocity vectors in plane y = 2.425 m; (d) velocity vectors in plane y = 0.10 m; (e) velocity scalar in plane z = 0.0 m; (f) temperature in plane z = 0.0 m (°C).*

Higher surface temperatures in the experiments will result in a higher temperature of room air. However, the heat exchange between the window surface and room air, which is computed by the low-Reynolds-number model, may be a little bit smaller since the grids used in the boundary are not sufficient. This will give a lower temperature of room air. Figure 7 shows that the temperature profiles by the 84-slot and momentum methods are very similar to

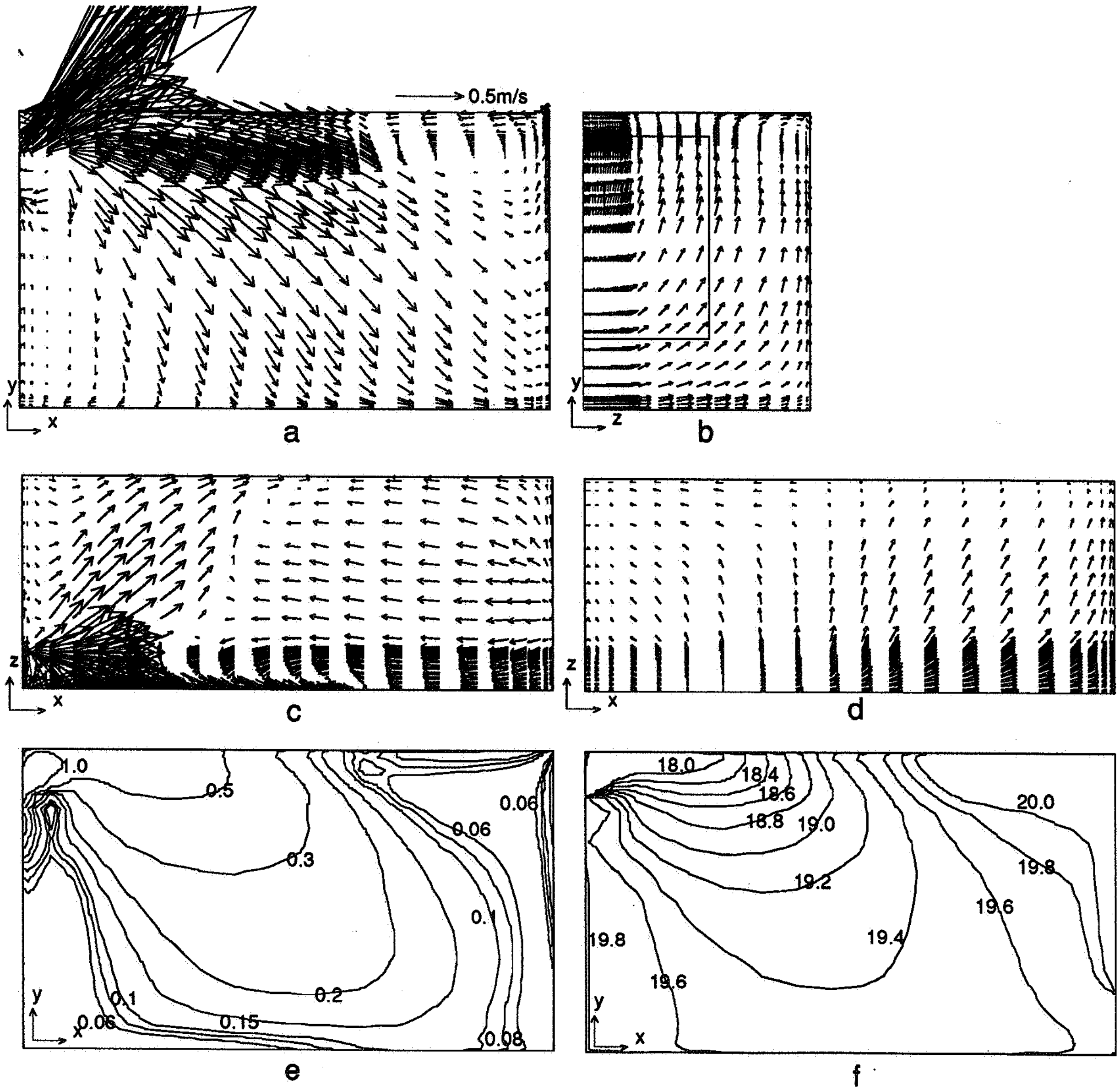

Figure 4 *Velocity and temperature distributions simulated by the 12-slot method. (a) velocity vectors in plane z = 0.0 m (symmetrical plane); (b) velocity vectors in plane x = 4.10 m; (c) velocity vectors in plane y = 2.425 m; (d) velocity vectors in plane y = 0.10 m; (e) velocity scalar in plane z = 0.0 m; (f) temperature in plane z = 0.0 m (°C).*

those measured regardless the average temperature of room air.

The simulated velocity distrbution by the one-slot method (Figures 3a and 3c) shows that the jet deflects from the ceiling in the mid-length of the room. The simulation by the 12-slot method presents similar results. This implies that the air

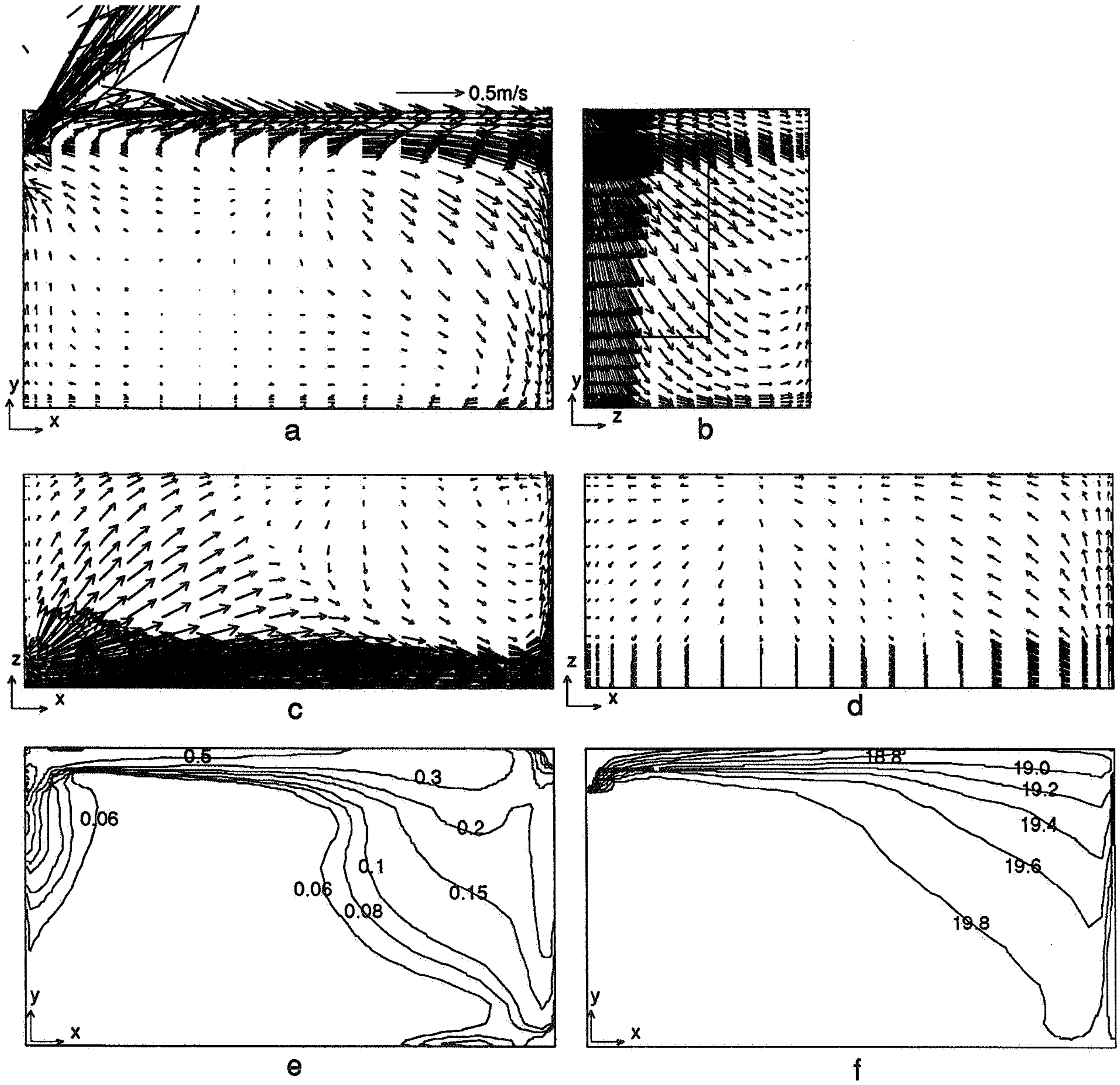

***Figure 5** Velocity and temperature distributions simulated by the 84-slot method. (a) velocity vectors in plane z = 0.0 m (symmetrical plane); (b) velocity vectors in plane x = 4.10 m; (c) velocity vectors in plane y = 2.425 m; (d) velocity vectors in plane y = 0.10 m; (e) velocity scalar in plane z = 0.0 m; (f) temperature in plane z = 0.0 m (°C).*

velocity decay from the jet is too fast by comparing the corresponding experimental data as shown in Figure 7. It is difficult to identify which computation, by the 84-slot method or by the momentum method, is in better agreement with the measurements because there are some differences in the two sets of experimental data.

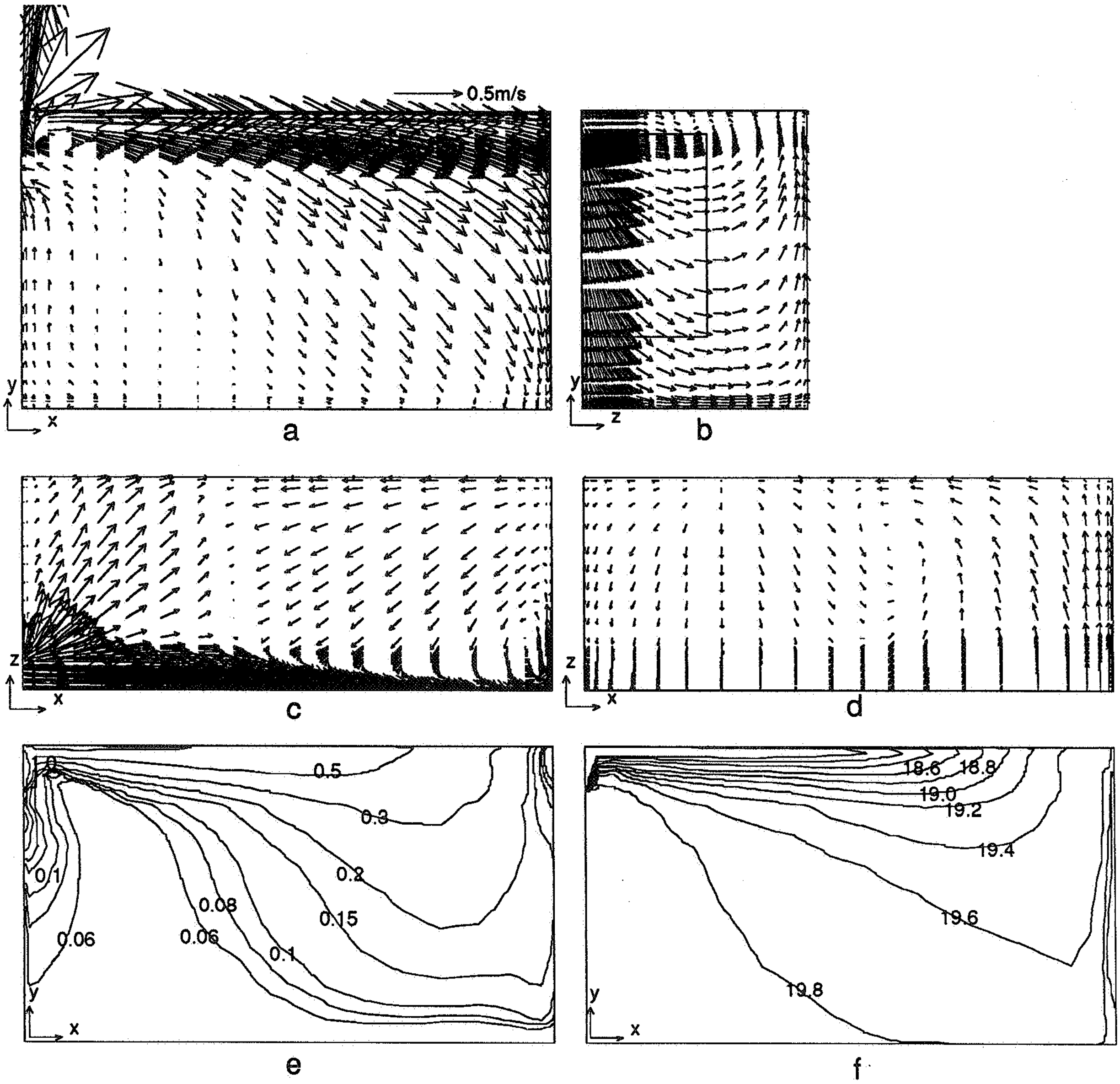

Figure 6 *Velocity and temperature distributions simulated by the momentum method. (a) velocity vectors in plane z = 0.0 m (symmetrical plane); (b) velocity vectors in plane x = 4.10 m; (c) velocity vectors in plane y = 2.425 m; (d) velocity vectors in plane y = 0.10 m; (e) velocity scalar in plane z = 0.0 m; (f) temperature in plane z = 0.0 m (°C).*

The computing cost with such a fine grid is too high to be used at present in practical applications. It is found that the inlet simulated by 28 slots gives very close results to those by 84 slots. If less slots are used, the grid employed for simulating the inlet can be reduced. As a result, computing cost will be lower considerably. The momentum method could be the most economical one among

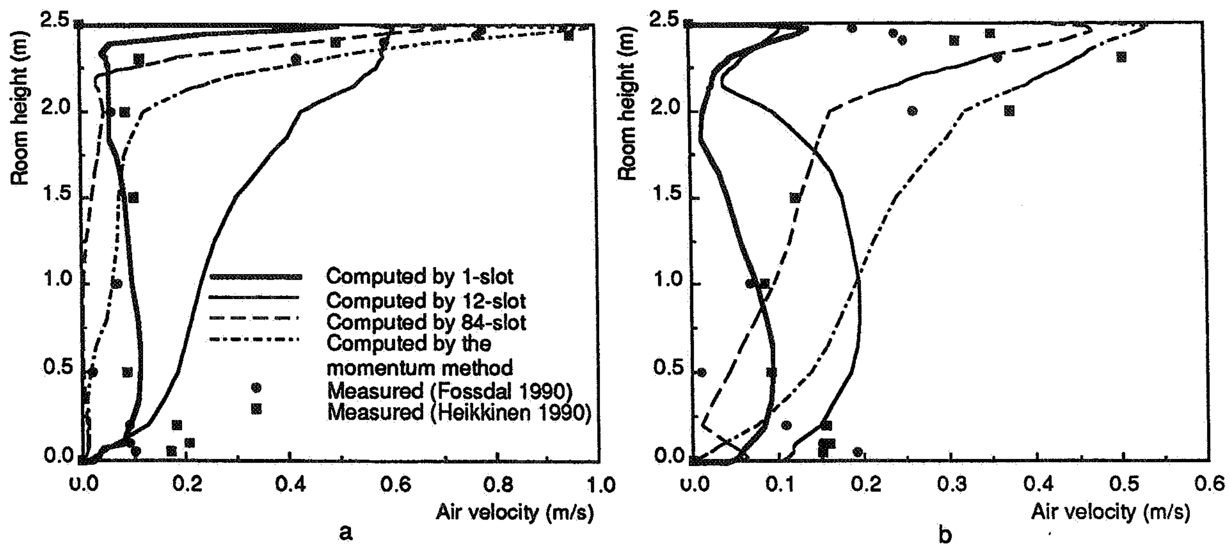

Figure 7 Comparison between the computations and measurements on velocity in z = 0.0 m plane. (a) at x = 1.4 m section and (b) at x = 3.0 m section.

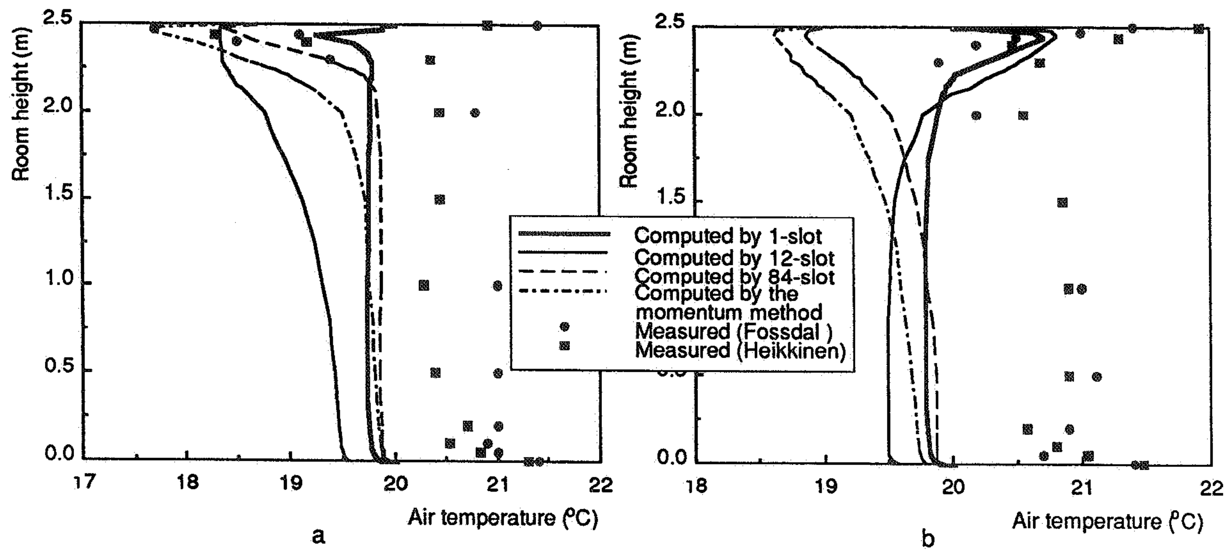

Figure 8 Comparison between the computations and measurements on air temperature in z = 0.0 m plane. (a) at x = 1.4 m section and (b) at x = 3.0 m section.

those method because only a few grids are required for describing an inlet. Another computation by the momentum method is done with a grid distribution of 22 x 29 x 15. The results are nearly the same as those by 22 x 34 x 34. Hence,the momentum method is recommended to be used to simulate a complex diffuser in practice.

Table 1 Boundary conditions in the experiments and computations.

Case	Inflow ach	T_{inlet} °C	T_{window} °C	$T_{surface}$ °C	T_{outlet} °C
Fossdal	3.0	14.0	30.3	21.4	20.8
Heikkinen	3.0	14.94	29.87	21.26	20.72
1 slot	3.0	15.0	30.0	20.0	19.70
12 slots	3.0	15.0	30.0	20.0	19.87
84 slots	3.0	15.0	30.0	20.0	20.08
Momentum	3.0	15.0	30.0	20.0	20.04

CONCLUSION

A number of approximated methods have been used to simulate a high-momentum jet diffuser with 84 round nozzles. Corresponding measurements from other researchers are used for comparison. The momentum method and the method using 84 simple rectangular slots predict air velocity and temperature distributions in the room similar to those from the experiments. The computing cost with the 84 slots method is extremely high. Hence, the momentum method is suggested to be used to simulate a complex diffuser in practice.

ACKNOWLEDGEMENTS

This investigation was financially supported by the Swiss Federal Office of Energy (BEW).

REFERENCES

Borth, J. 1990.
"Numerical simulation of air flows in rooms"
European Research Community on Flow Turbulence and Combustion Bulletin, Vol. 5, pp. 9-14.

Chen, Q. 1990.
"Construction of a low Reynolds number k-ε model"

The PHOENICS Journal of Computational Fluid Dynamics and Its Applications, Vol. 3, No. 3, pp. 288-329.

Chen, Q.; Moser, A.; and Huber, A. 1990.
"Prediction of buoyant, turbulent flow by a low-Reynolds-number k-ε model"
ASHRAE Transactions, Vol. 96, Part 1, pp. 564-573.

Fossdal, S. 1990.
"Measurement of Testcase E (Mixed Convection, Summer Cooling)"
Preliminary Report of International Energy Agency Annex 20.

Heikkinen, J. 1990.
"Measurements of Case B2, B3, E2 and E3"
Working Report of International Energy Agency Annex 20.

Heikkinen, J. 1990a.
"Modelling of the air inlet device"
Working Report of International Energy Agency Annex 20.

Lemaire, A.D. 1990.
"Simulation of test case b"
Presented in the 6th IEA Annex 20 Expert Meeting, Nice, France.

Nielsen, P.V.; Restivo, A.; and Whitelaw, J.H. 1979.
"Buoyancy-affected flows in ventilated rooms"
Numerical Heat Transfer, Vol. 2, pp. 115-127.

Nielsen, P.V. 1990.
"Selection of air terminal device"
Working Report of International Energy Agency Annex 20

Rosten, H.I. and Spalding, D.B. 1987.
"The PHOENICS Reference Manual (Version 1.4)"
Report No. TR200, CHAM Ltd., London.