O·R·I·G·I·N·A·L P·A·P·E·R

Indoor Environ 1992;1:88–102

Qingyan Chen[a]
Zheng Jiang[b]

[a] TNO Institute of Applied Physics, Delft, The Netherlands
[b] Department of Mechanical Engineering, Concordia University, Montreal, Que., Canada

Air Supply Method and Indoor Environment

Key Words
Diffuser
Simulation
Airflow
Air quality
Thermal comfort
Turbulent flow

Abstract
A ventilation jet diffuser is characterized by parameters such as diffuser effective area, diffuser dimension, diffuser position, air supply direction, flow rate, and air temperature. This paper studies the influence of the parameters of a jet diffuser on the airflow pattern, indoor air quality, and draft risk in an office with a jet diffuser on the rear wall near the ceiling. The presentation of furniture and occupants in the office is included in the numerical simulation. The structure of a jet diffuser used in practice is complicated. Therefore, a simplified method is introduced to simulate the diffuser. The method is compared with the experimental data. The agreement between the simulations and the measurements is reasonably good. The distributions of the air velocity, temperature, contaminant concentration, and percentage dissatisfied people due to draft risk with different parameters of the diffuser are numerically predicted by the k-ε model of turbulence. The effect of turbulence intensity is taken into account in the computation of draft risk. It has been found that the angle between the jet and the ceiling should be in the range from 20 to 60°C. The effective flow area has a strong impact on the indoor airflow pattern since it significantly affects the throw projection. The diffuser width has a larger influence on indoor air diffusion than the diffuser height. The distance between the inlet and ceiling has a remarkable influence on the total air movement near the ceiling, but has a minor impact on the air diffusion in the occupied zone. Air velocity distribution is sensitive to ventilation rate and supply air temperature. To achieve the same length of throw projection, the Reynolds number should be the same if the corresponding Archimedes number is close to each of them.

Introduction

The success of a ventilation system design is normally assessed by the air quality and thermal comfort level it provides in the occupied zone of a room. The method of distributing air is decisive for better air quality and thermal comfort in the room. It is common practice to install a jet diffuser in a wall near the ceiling of a room for distributing air. The region between the ceiling and the occupied zone serves as an entrainment region for the jet which causes a decay of main jet velocity as a result of the increase in the mass flow rate of the jet. Then the total air (the mixture of discharged air and entrained air) dilutes the contaminant in the occupied zone to achieve an

Accepted:
May 6, 1991

Dr. Qingyan Chen
TNO Institute of Applied Physics (TPD)
PO Box 155
NL-2600 AD Delft (The Netherlands)

1016–4901/92/0012–0088
$2.75/0

acceptable air quality. Since the total air is of low velocity and moderate temperature, the occurrence of draft is minimized.

The parameters of supply air, such as diffuser effective area, diffuser dimension, diffuser location, air supply direction, ventilation rate, and air temperature are very important in determining air distribution. Thus, it is necessary to provide means to investigate air distributions under various parameters of supply air. Then the nature and quality and thermal comfort may be assessed from the distributions. The assessment may help a designer to make an optimum design of jet diffuser and ventilation system. The main aim of the present paper is to study the influence of the supply air parameters of a jet diffuser on air distribution, indoor air quality, and thermal comfort in a room.

Research Approach

So far, the design of a jet diffuser and ventilation system is based on data obtained from experiments. The model is then modified until the desired conditions are achieved. It is costly and time consuming to construct a physical model at full scale. If a small-scale model is directly applied to the full scale, dynamic and thermal similarity must be achieved. However, it is impossible to achieve the equality of the Reynolds number and the Archimedes number in the scale model concurrently.

The increase in performance and affordability of high-speed computers, together with the continuous improvement of the mathematical models employed to describe the behavior of turbulent flows, have made the numerical treatment of the latter a practical option for predicting the air distribution in a room. Encouraging results have been achieved as reviewed by Whittle [1] and Nielsen [2]. With the numerical techniques, it is possible to study the influence of air supply parameters on the field distributions of air velocity, temperature, turbulence intensity, and contaminant concentration in a room, and consequently, on the thermal comfort and indoor air quality.

In the numerical techniques, approximations are often required in the conservation equations of motion in order to make them solvable. For example, the details of turbulent flow are difficult to calculate, and engineers are mainly interested in the mean values. Therefore, one turns to so-called turbulence models by which it is possible to compute the mean values. Recent reviews [1, 3] have come to the conclusion that the k-ε model of turbulence developed by Launder and Spalding [4], is still the most appropriate model for practical flow applications.

In the present study, a modified k-ε model of turbulence [5] is used. This model has been shown to be more suitable for predicting indoor airflow and heat transfer. A more detailed description of the model and a comparison between the computed results and experimental data are given by Chen et al. [5] and Borth [6].

The airflow program developed by Rosten and Spalding [7] has been employed to calculate air distribution. The computations involve the solution of three-dimensional equations for the conservation of mass, momentum (u, v, w), energy (H), contaminant concentrations (C), turbulence energy (k), and the dissipation rate of turbulence energy (ε). The governing equations of the model can be expressed in a standard form:

$$\mathrm{div}\,(\rho \vec{V}\varphi - \Gamma_\varphi \,\mathrm{grad}\,\varphi) = S_\varphi, \qquad (1)$$

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, H, or C. When φ = 1, the equation changes into the continuity equation.

The draft risk is evaluated by the comfort model developed by Fanger et al. [8] in which the turbulence intensity is taken into account. The model predicts the percentage of dissatisfied people due to draft (PD) in the following expression:

$$PD = (34 - T)\,(V - 0.05)^{0.62}\,(3.14 + 0.36\,VI)\ (\%) \qquad (2)$$

for $V < 0.05$ m/s insert $V = 0.05$ m/s, for $PD > 100\%$ use $PD = 100\%$, where T is the local air temperature (°C), V is the mean velocity (m/s), and I is the turbulence intensity (%). The turbulence intensity is defined as the velocity fluctuation over the mean velocity, and is calculated by:

$$I = 100\,(2k)^{0.5}/V\ (\%) \qquad (3)$$

The T, V and k in equations 2 and 3 can be obtained from the airflow computation.

Results

Simulation of a Jet Diffuser

The air diffusion in a room (distribution of air within a room by an inlet diffuser discharging supply air in various directions and planes) is dominated by the diffuser type and the air supply parameters of the diffuser. A diffuser as used in practice is often complicated. The study of the influence of air supply parameters on indoor air quality and thermal comfort must be based on a correct simulation of the diffuser used for distributing air to the room. In this section, a few methods used to simulate a jet diffuser are introduced.

The jet diffuser shown in figure 1a was selected for the present investigation. This diffuser has also been used as an example for the validation exercise in the International Energy Agency Annex 20 project (air flow patterns within buildings). It is a modern air terminal device and available on the market. It consists of 84 small round nozzles arranged in four rows in an area of 0.71 m × 0.17 m. The flow direction of each nozzle is adjustable, and a flow which has a complicated three-dimensional structure close to the 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 figure 1b, the 84 round nozzles can be simulated by 84 rectangular slots. In

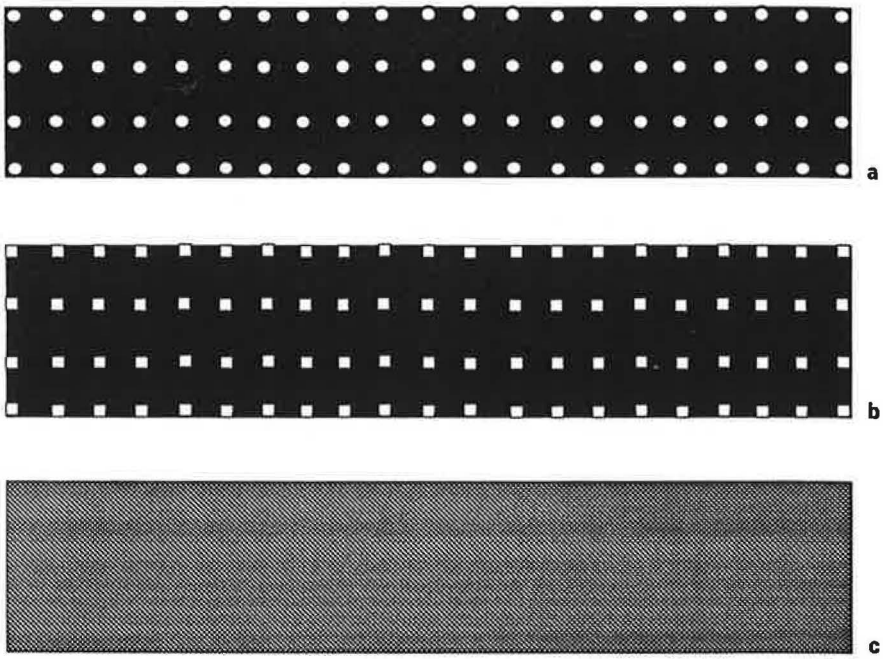

Fig. 1. The jet diffuser and its simulation methods. **a** The jet diffuser with 84 round nozzles. **b** Simulated by 84 slots. **c** Simulated by the momentum method (infinite slots).

the momentum method, the supply air momentum (mV_{in}) is set to be that of the 84 small round nozzles:

$$m\ V_{in} = m\ (\text{volume inflow rate / effective area}), \quad (4)$$

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

The simulation of the jet diffuser is coupled with the airflow simulation in a room, as shown in figure 2. The room is 4.2 m long, 3.6 m wide, and 2.5 m high, and is symmetrical in the mid-width section. The flow and thermal boundary conditions are summarized in table 1.

The computational results are compared with the experiments conducted by Blomqvist [unpubl. results], Fossdal [9], and Heikkinen [10]. Although the experiments are carried out on the same specification by IEA Annex 20, there are significant differences between these measurements [11]. The results of Fossdal [9] and Heikkinen [10] are similar, and therefore, they are selected for comparison with the computed ones as shown in figure 3. The comparison on the velocity profiles as indicated in

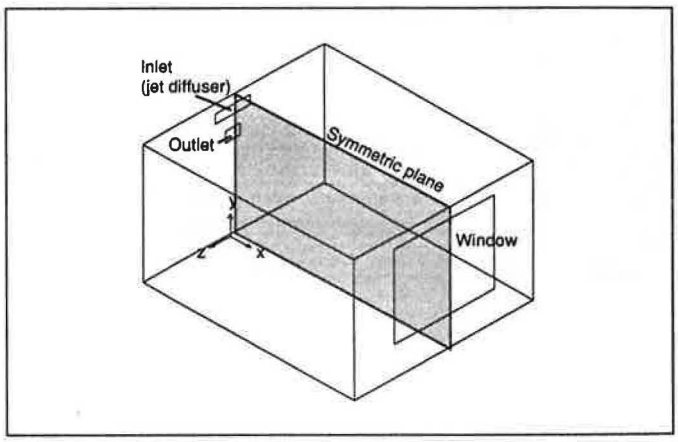

Fig. 2. The room with a jet diffuser.

Table 1. Thermal and flow boundary conditions in the experiments and computations

Case	Inflow ach	T_{inlet} °C	T_{window} °C	$T_{surface}$ °C	T_{outlet} °C
Fossdal [9]	3.0	14.0	30.3	21.4	20.8
Heikkinen [10]	3.0	14.94	29.87	21.26	20.72
84 slots	3.0	15.0	30.0	20.0	20.08
Momentum	3.0	15.0	30.0	20.0	20.04

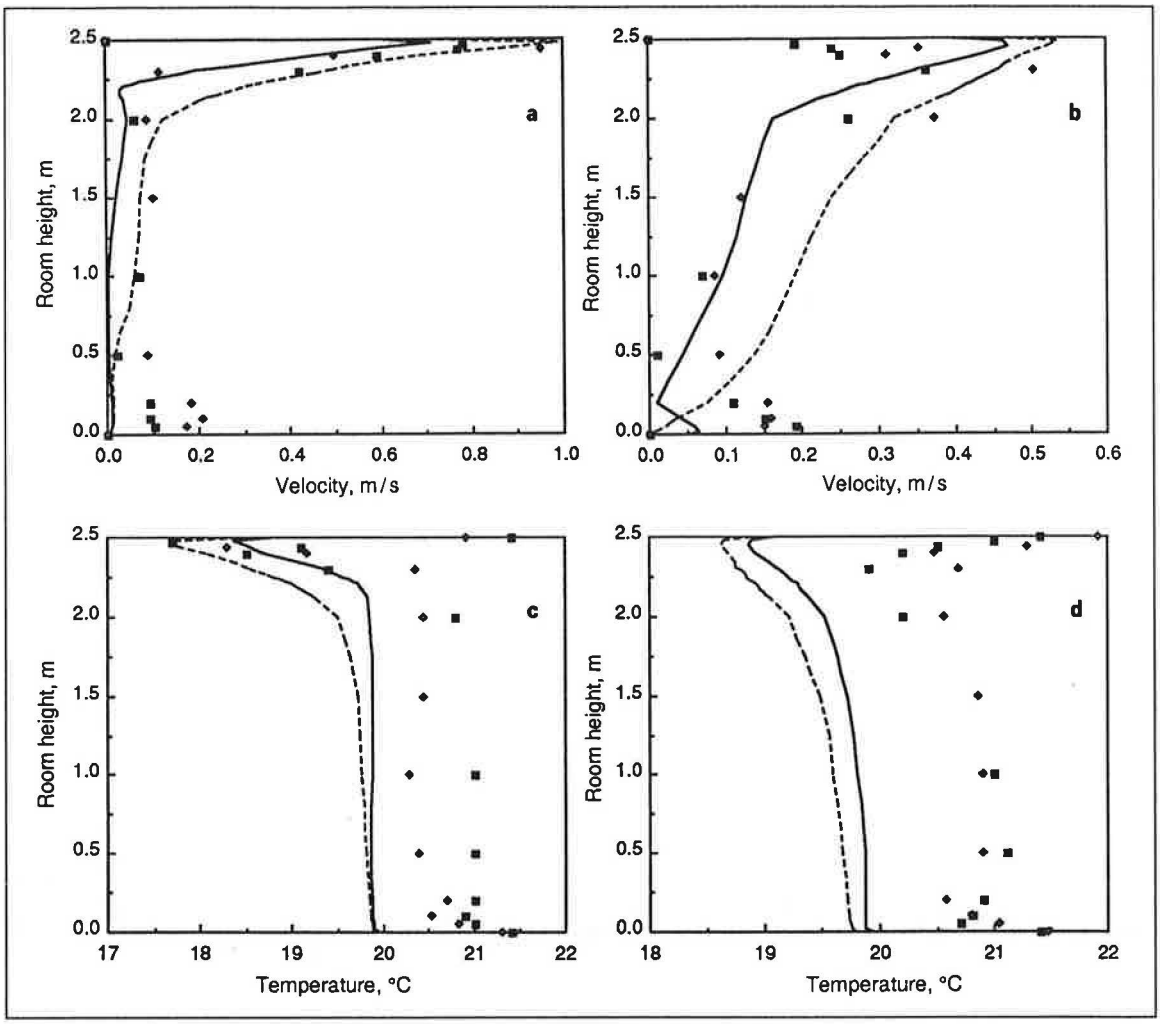

Fig. 3. Comparison between the computations and measurements in the symmetric plane. Velocity (m/s) at sections x =1.4 m (**a**) and x = 3.0 m (**b**), and temperatures (°C) at sections x = 1.4 m (**c**), and x = 3.0 m (**d**). —— = 84 slots; - - - = momentum; □ = Fossdal [9]; ◇ = Heikkinen [10].

figures 3a, b concludes that the computed results are in reasonable agreement with the measurements, although there are some discrepancies especially near the floor region. The computed velocities by the momentum method at section x = 3.0 m in the occupied zone seems slightly higher than the measured ones.

In general, the computed air temperature is about 1 °C lower than the measured one, as shown in figures 3c, d. There are differences in the thermal and flow boundary conditions given in table 1. 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 k-ε model, may be smaller since the grids used in the boundary are not sufficient. It will result in a lower temperature of room air. Nevertheless, the temperature profiles computed are similar to those measured.

It is costly to use the 84-slot method although the results are rather good, because a large number of grid nodes are needed for presenting the diffuser. In the present study, 42 × 12 grid nodes are used to simulate the diffuser. This amount of grid nodes is the minimum requirement in the 84-slot method. The more grid nodes are used, the more computing time is required to obtain a converged solution. In the momentum method, the results with 8 × 4 grid notes for simulation of the diffuser are in good agreement with the experimental data. There-

fore, the momentum method is recommended for practical applications.

In the following sections, the influence of the supply air parameters of a jet diffuser on indoor air diffusion is discussed. Table 2 gives a summary of the cases studied. The important nondimensional numbers that govern the flow, such as the Archimedes number and the Reynolds number, are presented for each case for easy comparison. In table 2, the global Archimedes number is defined as:

$$Ar = \frac{\beta g h \Delta T_0}{V_0^2} \tag{5}$$

and the Reynolds number as:

$$Re = \frac{\sqrt{A_{eff}} V_0}{\nu_l} \tag{6}$$

where β is the air expansion coefficient, g the gravity acceleration, h the room height, ΔT_0 the air temperature difference between room center and inlet, V_0 the air velocity through the inlet openings (i.e. the air inflow rate divided by the effective area), A_{eff} the effective area of the inlet, and ν_l the laminar viscosity of the air.

Standard Case

The studies of the parameters of a jet diffuser on air diffusion are conducted for an office that is 4.5 m in length, 4.5 m in width and 2.5 m in height as shown in figure 4. There are furniture, two computers, and two occupants in the room. The furniture and occupants are simulated by aerodynamic blockage. For grid economy, no aerodynamic blockage is used for the computers. The inside surface temperatures of the enclosures are 22.3 °C. To simulate a summer cooling situation, 150-Watt convective heat gain from the window due to solar radiation is assumed. The window is 3.1 m wide and 1.1 m high. The heat sources from each occupant and from each computer are 80 and 120 W, respectively.

A standard case is set up for comparison. In the standard case, the inlet size is 0.7 m in width and 0.15 m in height with an effective area ratio of 0.13, defined as the effective area over the gross area of the diffuser. The top boundary of the inlet is located 0.25 m from the ceiling. The air is supplied at 5 air changes per hour (ach) with an angle of 40° towards the ceiling. The temperature of the supply air is 17 °C. The turbulence intensity of the air supplied is 10%. The contaminant from the occupant *a* is normalized to be 0.01 ml/s, simulating smoking or CO_2.

Figure 5 shows the computed distributions of air velocities, temperature, smoke concentration, and the

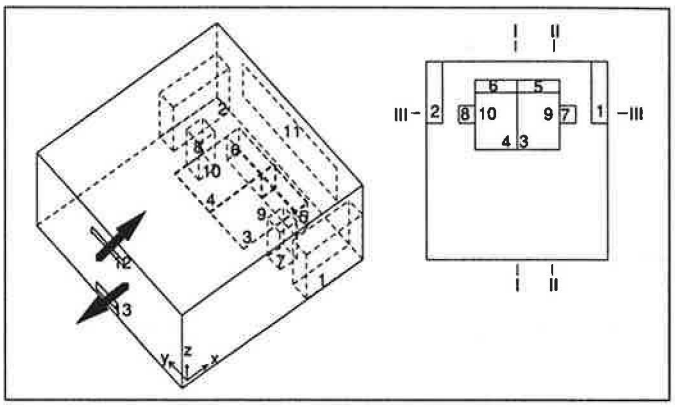

Fig. 4. The standard case. 1 = Bookshelf a; 2 = bookshelf b; 3 = table a; 4 = table b; 5 = filing cabinet a; 6 = filing cabinet b; 7 = occupant a; 8 = occupant b; 9 = computer a; 10 = computer b; 11 = window; 12 = inlet; 13 = outlet.

Table 2. Description of the supply air parameters of the jet diffuser

Sect.	Area ratio	Angle	Height m	Width m	d m	Ar	Re 10^4	Results figure
2	0.13	40°	0.15	0.70	0.25	0.020	3.92	5
3	0.13	20°	0.15	0.70	0.25	0.020	3.92	6
4	0.33	40°	0.15	0.70	0.25	0.125	2.48	7
5	0.19	40°	0.15	0.50	0.25	0.020	3.92	8
6a	0.13	40°	0.15	0.70	0.05	0.020	3.92	9
6b	0.13	40°	0.15	0.70	0.55	0.020	3.92	10
7	0.11	40°	0.15	0.70	0.25	0.023	3.33	11

Area ratio = effective area over the gross area of the diffuser; d = distance of the diffuser top boundary from the ceiling. Sections: 2 = Standard case; 3 = variation of air supply angle; 4 = variation of effective area; 5 = variation of diffuser dimension; 6a, b = variation of diffuser location; 7= variation of ventilation rate and air temperature.

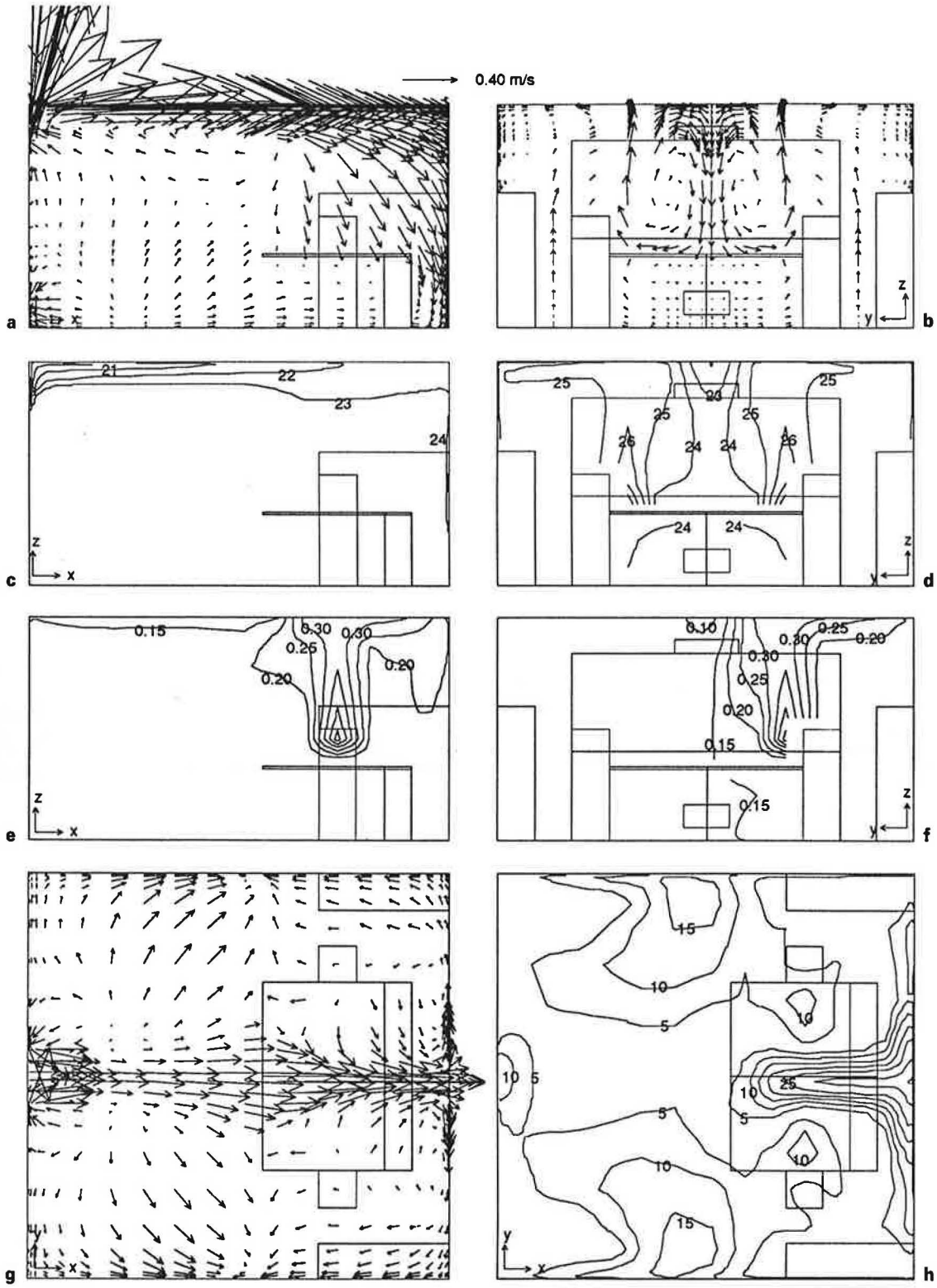

Fig. 5. Computed field distributions of the standard case. **a** Velocity in section I–I. **b** Velocity in section III–III. **c** Temperature (°C) in section I–I. **d** Temperature (°C) in section III–III. **e** Smoke concentration (ppm) in section II–II. **f** Smoke concentration (ppm) in section III–III. **g** Velocity in section 0.2 m from the ceiling. **h** Percentage dissatisfied people due to draft in section 1.65 m from the floor.

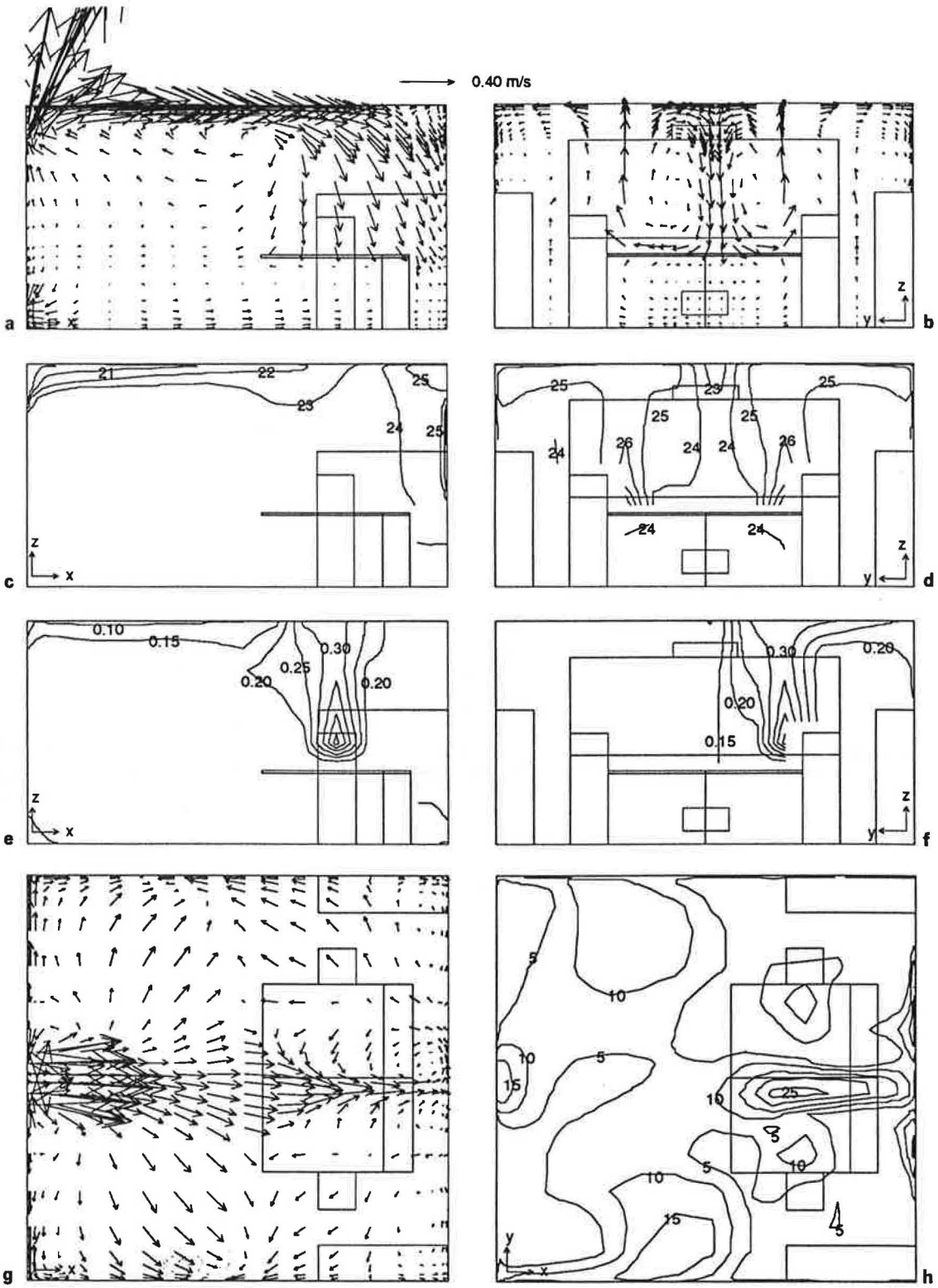

Fig. 6. Computed field distributions of the case with an air supply angle 20° toward the ceiling. **a** Velocity in section I–I. **b** Velocity in section III–III. **c** Temperature (°C) in section I–I. **d** Temperature (°C) in section III–III. **e** Smoke concentration (ppm) in section II–II. **f** Smoke concentration (ppm) in section III–III. **g** Velocity in section 0.2 m from the ceiling. **h** Percentage dissatisfied people due to draft in section 1.65 m from the floor (%).

percentage dissatisfied people due to draft risk in different sections of the room as defined in figure 4. This ventilation method is often preferable from the installation aspect, but may be inadvisable from the viewpoint of thermal comfort. It is because high velocities are found above the tables near the window (fig. 4a, b, g). As a result, the percentage dissatisfied people is high in that region as shown in figure 5h. The distribution of draft risk at the section 1.65 m above the floor is given since with the ventilation method down draft may occur in the upper part of the occupied zone. In general, the air temperature distribution in the occupied zone is uniform except in the area where heat sources are placed. The temperature difference between height 0.1 m and 1.8 m is less than 0.5 °C as illustrated in figure 5c. Although the ventilation rate for the office is considerably high (5 ach), the local smoke concentration in most areas is higher than that in a well-mixed ventilation room (0.142) because it is difficult to mix room air perfectly.

Variation of Air Supply Angle

The air supply angle of nozzle is often made to be adjustable so that users can adjust it to achieve a better air projection. Hence, it is necessary to investigate the influence of the air supply angle on indoor air diffusion. The investigation on the supply angle is performed for four different values, i.e. 20, 40 (the standard case), 60 and 80°. Hence, only the result for the case with 20° is shown in figure 6, because it is found that, except in the region near the diffuser, the results for the case with 60° are similar to those for the standard case and those for the case with 80° are similar to those with 20°.

A small counter flow in the upper right corner is seen in figure 6a. The same phenomenon can be observed in the case with an air supply angle of 80°. Hence, for a longer projection, the air supply angle should not be too large or too small.

The total air deflecting from the ceiling is earlier in the cases with an air supply angle of 20 and 80° than in the other two cases. It results in a higher air temperature distribution near the window. However, the draft risk distributions are similar in the 4 cases.

However, when air supply velocity is smaller and/or supplied air temperature is lower, the supply angle should be larger to accomodate the total air sticking on the ceiling. In other words, the strength of the Coanda effect depends on the Reynolds number and the Archimedes number of air inlet. For a jet diffuser, the air supply angle should be between 20 and 60°.

Variation of Effective Area

With the same gross area and mass inflow, varying the effective area of a diffuser implies changing supply air momentum. According to equation 4, the smaller the effective area, the higher the supply air momentum. Three effective area ratios, 0.13 (the standard case), 0.19, and 0.33, have been studied in this subsection.

The supply velocities through the opening for the three cases are 5.0, 3.5 and 2.0 m/s, respectively, for the angle of 40°. Figure 5 shows the computed results for the standard case (effective area ratio of 0.13). Figure 7 is for the case with an effective area ratio of 0.33. With a ratio of 0.19, the airflow pattern is very similar to that in the standard case (not shown here). The results obtained for the two cases (0.19 ratio and 0.13 ratio) imply that the total air projection does not become longer by only increasing the supply air momentum if the effective area ratio is small enough.

The velocity at the opening is 2 m/s when the effective area ratio is 0.33. The airflow is not symmetric as shown in figure 7g. Asymmetric flow has also been found in many other computations and experiments even if the boundary conditions and geometry are absolutely symmetrical. In the computations, the initial values and numerical scheme used generate a slight asymmetry.

In the case with an effective area ratio of 0.33, the supply air momentum is so low that the total air cannot reach the window. As a result, there is a counter flow near the hot window surface because of the thermal buoyancy. Since the airflow pattern is different from that in the standard case, it results in different distributions of air temperature, smoke concentration, and percentage dissatisfied people due to draft risk. The higher the supply momentum, the more uniform the room air temperature and the higher the level of discomfort. This can be seen from the distributions of air temperature and draft risk shown in figures 5 and 7. The smoke concentration is less sensitive to the effective area ratio.

The above results indicate that the total air projection may become longer by decreasing the effective area. However, there is a limit. The total air projection will remain the same if the effective area ratio is small enough. Besides, it should be noted that too high a supply air momentum will result in a higher noise level.

Variation of Diffuser Dimension

For the study concerning the variation of diffuser dimension, the air supply momentum through the openings and air inflow rate remain the same. Therefore, the variation of the diffuser dimension changes the gross area

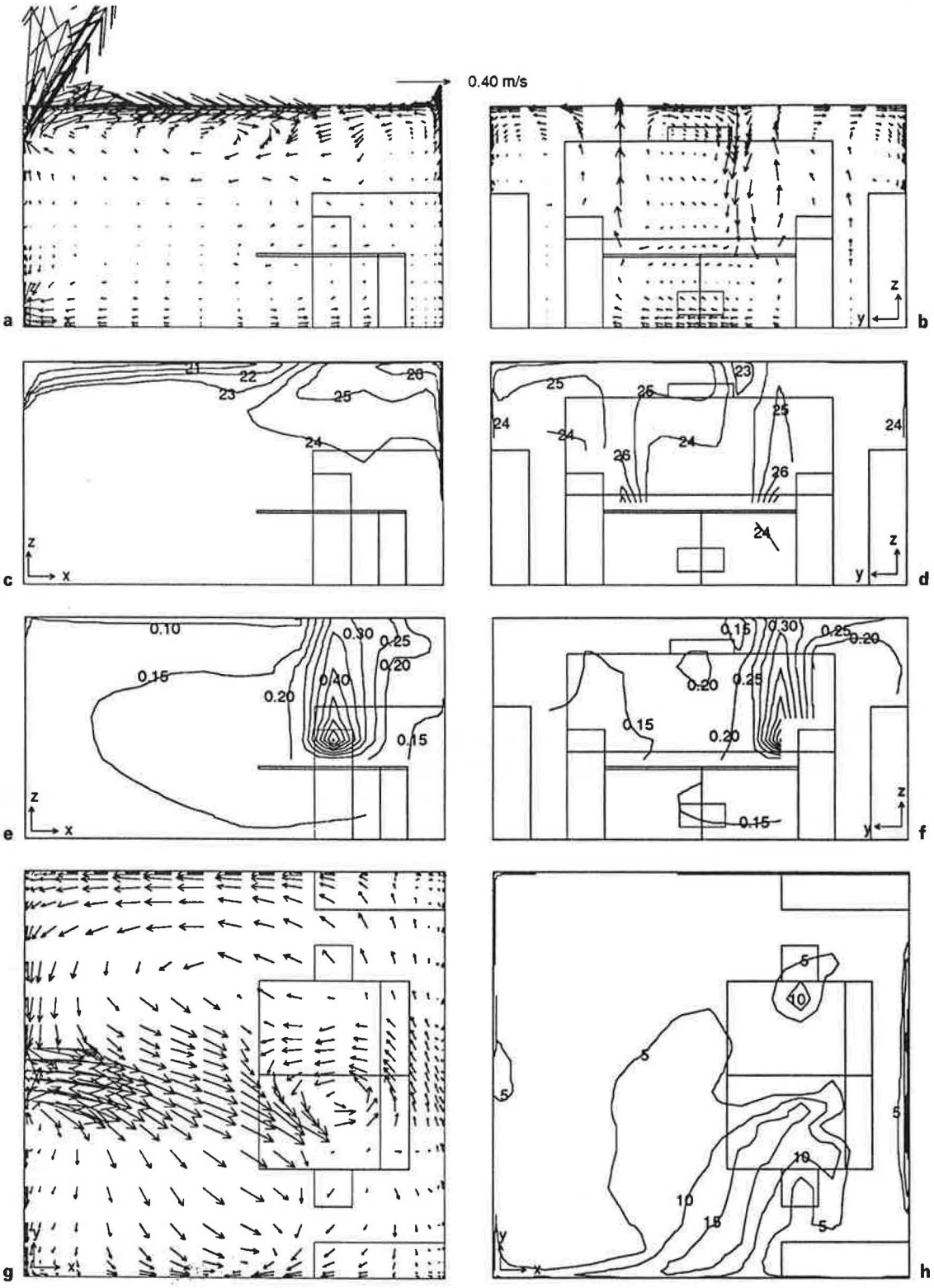

Fig. 7. Computed field distributions of the case with an effective area ratio of 0.33. **a** Velocity in section I–I. **b** Velocity in section III–III. **c** Temperature (°C) in section I–I. **d** Temperature (°C) in section III–III. **e** Smoke concentration (ppm) in section II–II. **f** Smoke concentration (ppm) in section III–III. **g** Velocity in section 0.2 m from the ceiling. **h** Dissatisfied people (%) due to draft in section 1.65 m from the floor.

but not the effective area. In the standard case as shown in figure 5, the inlet is 0.7 m (width) × 0.15 m (height). Four other cases with different inlet dimensions, 0.7 m × 0.25 m, 0.7 m × 0.35 m, 0.5 m × 0.15 m, and 0.9 m × 0.15 m, are used for comparison.

The airflow patterns are not sensitive to the inlet height except in the area close to the inlet. The computed results with an increased inlet height (h = 0.25 and 0.35 m) are similar to those for the standard case illustrated in figure 5. However, the inlet width is an important factor influencing indoor airflow patterns. If the inlet width is reduced, the induction of jet flow is weaker. As a result, the total air movement is somewhat counteracted by the rising natural convection currents on the heated window and, therefore, deflects from the ceiling before reaching the window as shown in figure 8 (inlet size 0.5 × 0.15 m). On the other hand, the total air reaches the inside walls and descends for some distance along them. Consequently, the draft risk near the side walls is higher. In the case with an inlet size of 0.9 × 0.15 m, the air momentum from the large gross area produces a large throw toward the window. The air velocities near the ceiling are higher. The higher velocities result in 40% dissatisfied at 1.65 m above the floor near the window.

The length of total air projection is related to the area of jet flow-contacting room air. The width of a jet diffuser is normally larger than the height. Hence, the change in the width makes a significant variation on the area that contacts surrounding room air. This may be the reason why the airflow patterns are more sensitive to diffuser width than diffuser height.

Variation of Diffuser Location

There is only one diffuser in the room in the present study. The jet diffuser is placed in the central plane. A room with multiple diffusers may be regarded as the combination of several single-diffuser rooms. Hence, the variation of the diffuser location along the room width was not investigated. This subsection is dedicated to the variation of diffuser location in room height. In the standard case, the top boundary of the inlet is 0.25 m from the ceiling. The results for the standard case is given in figure 5. Figures 9 and 10 are for the cases with the top boundary 0.05 and 0.55 m from the ceiling, respectively. The distributions of air velocity near the ceiling are quite different between those three cases. The higher the inlet location, the larger the air velocities. However, figures 5h, 9h, and 10h show a small discrepancy in the distributions of the percentage dissatisfied people due to draft in the upper part of the occupied zone. In most parts of the room, the difference in inlet location does not cause an evident change in the distributions of air temperature and smoke concentration. This indicates that the distance between the inlet and ceiling has little impact on indoor air diffusion in the zone of occupation.

Variation of Ventilation Rate and Air Temperature

If the ventilation system works with a variable-air-volume control strategy, the diffuser must work properly under different ventilation rates. Therefore, it is necessary to study the airflow pattern, indoor air quality, and thermal comfort for different ventilation rates.

The space load (Q) is fixed in this study, therefore, the reduction of the ventilation rate means a decrease of supply air temperature if the space load is unchanged. This is because ventilation rate and supply air temperature are related:

$$Q = m\, C_p\, (T_{out} - T_{in}), \tag{7}$$

where m is the mass inflow rate, C_p is specific heat, and T_{out} and T_{in} are the air tempertures at the outlet and inlet, respectively.

In the standard case, the ventilation rate is 5 ach, the supply air temperature 17 °C, and the effective area ratio of the diffuser 0.13. Figure 11 shows the results with a ventilation rate of 3 ach, a 13 °C supply air temperature and effective area ratio of 0.11. According to equation 7, the two cases shown in figures 5 and 11 have the ability to remove 500 W space load. Thus, the average room air temperature is the same as shown in figures 5c, d, 11c, d.

If the effective area ratio for the standard case is used for the latter case, the total air would deflect from the ceiling before reaching the window. Therefore, a smaller effective area ratio is used for the case with a 3-ach ventilation rate to increase the supply air momentum. The air velocity through the effective area is 5 m/s for the standard case and 6 m/s for the other case. This measure ensures good mixing in the occupied zone. Very similar airflow patterns are observed (fig. 5a, b, g, 11a, b, g).

In the case with a 3-ach ventilation rate, the Archimedes number as defined in equation 5 is close to that of the standard case. The total air has the same projection length in the two cases because the Reynolds numbers are nearly the same as shown in table 2. This is an important rule if a diffuser must work with a different ventilation rate. This rule may be used only for the case with the same diffuser configuration. When a diffuser works with a very small ventilation rate, a low Reynolds number effect may appear. An extensive study on the problem has been conducted experimentally by Skovgaard et al. [12].

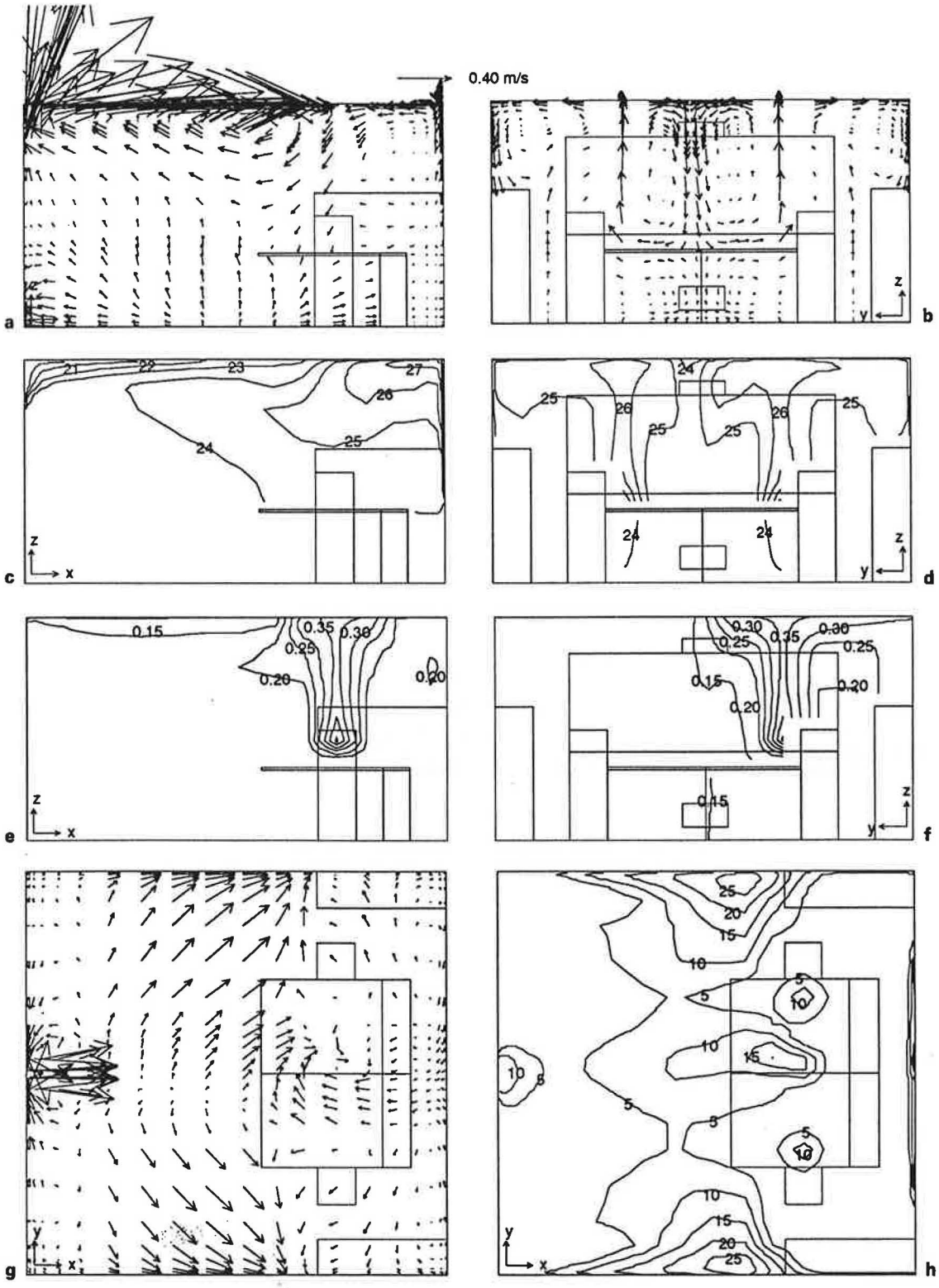

Fig. 8. Computed field distributions of the case with an inlet of 0.5 m in width and 0.15 m in height. **a** Velocity in section I–I. **b** Velocity in section III–III. **c** Temperature (°C) in section I–I. **d** Temperature (°C) in section III–III. **e** Smoke concentration (ppm) in section II–II. **f** Smoke concentration (ppm) in section III–III. **g** Velocity in section 0.2 m from the ceiling. **h** Dissatisfied people (%) due to draft in section 1.65 m from the floor.

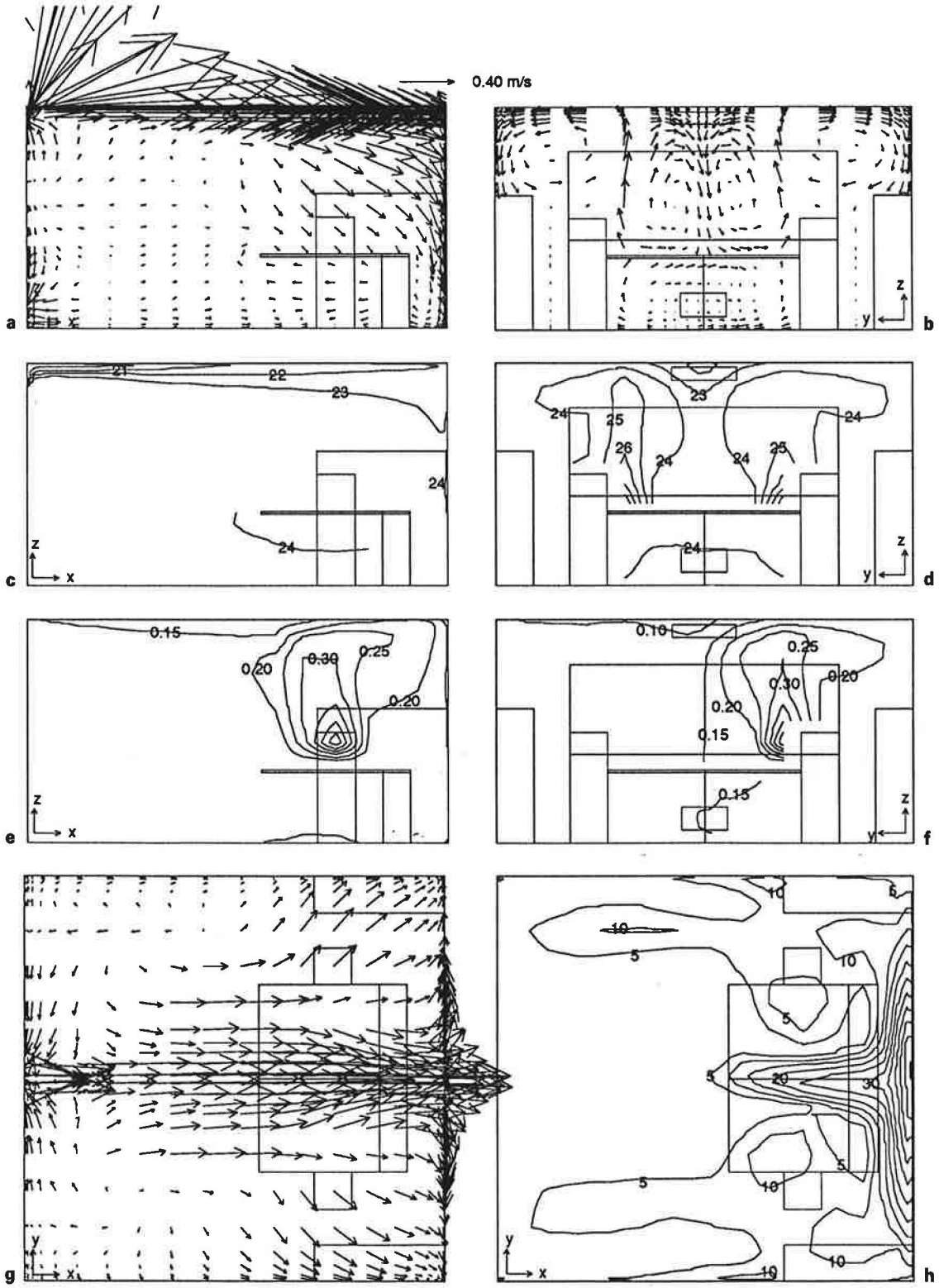

Fig. 9. Computed field distributions of the case with the top boundary of the inlet located 0.05 m from the ceiling. **a** Velocity in section I–I. **b** Velocity in section III–III. **c** Temperature (°C) in section I–I. **d** Temperature (°C) in section III–III. **e** Smoke concentration (ppm) in section II–II. **f** Smoke concentration (ppm) in section III–III. **g** Velocity in section 0.2 m from the ceiling. **h** Dissatisfied people (%) due to draft in section 1.65 m from the floor.

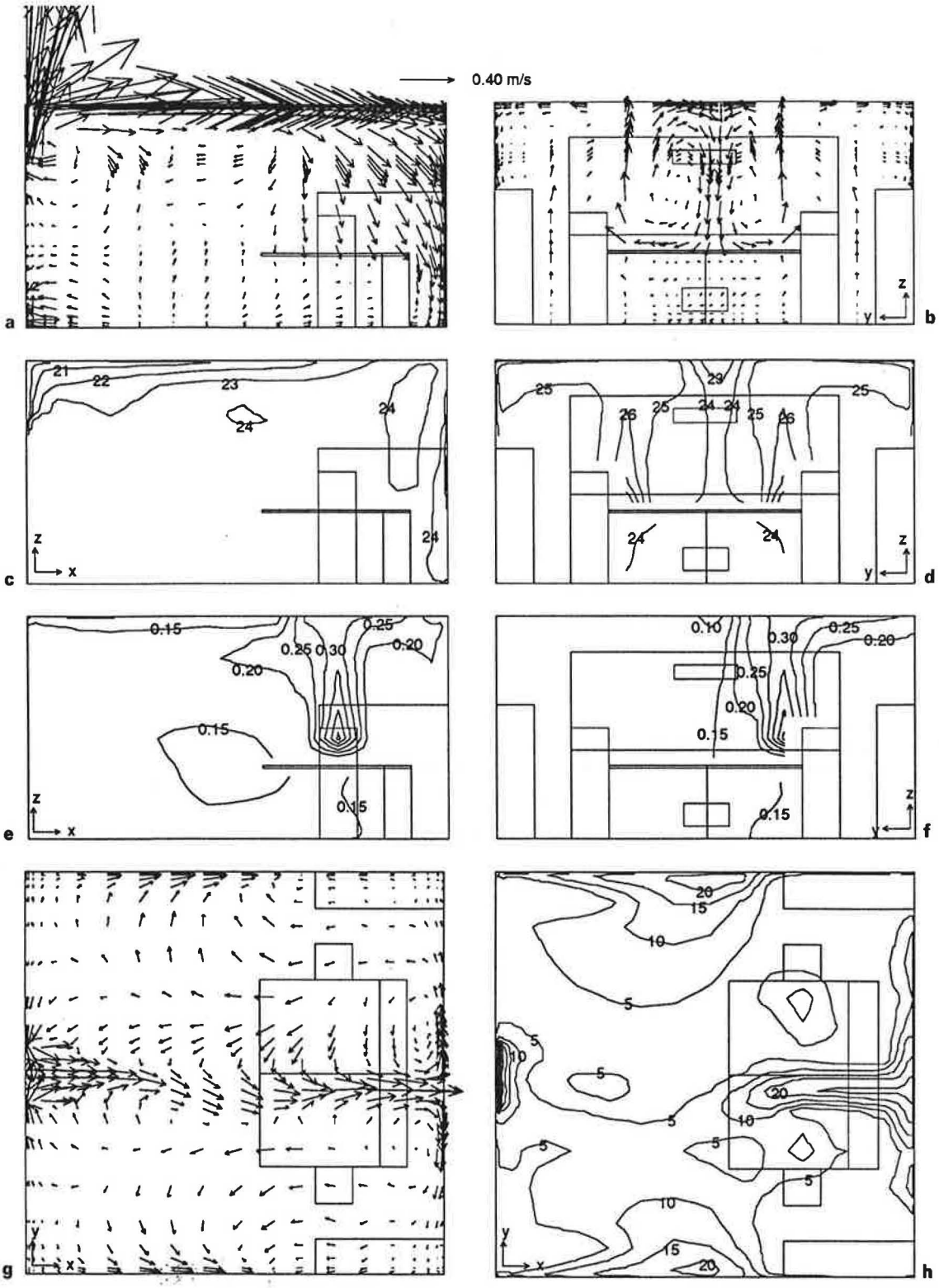

Fig. 10. Computed field distributions of the case with the top boundary of the inlet located 0.55 m to the ceiling. **a** Velocity in section I–I. **b** Velocity in section III–III. **c** Temperature (°C) in section I–I. **d** Temperature (°C) in section III–III. **e** Smoke concentration (ppm) in section II–II. **f** Smoke concentration (ppm) in section III–III. **g** Velocity in section 0.2 m from the ceiling. **h** Dissatisfied people (%) due to draft in section 1.65 m from the floor.

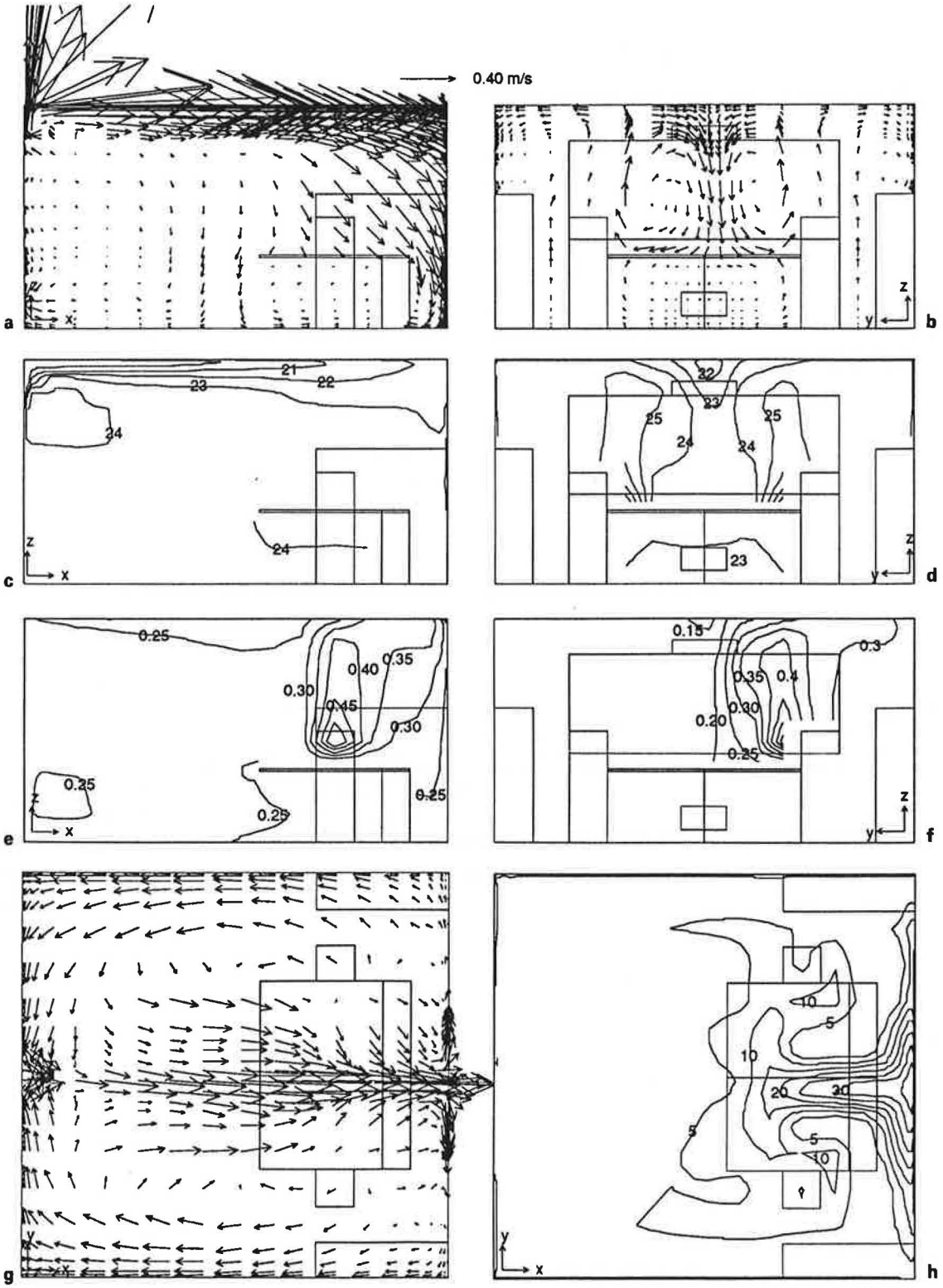

Fig. 11. Computed field distributions of the case with a 3-ach ventilation rate. **a** Velocity in section I–I. **b** Velocity in section III–III. **c** Temperature (°C) in section I–I. **d** Temperature (°C) in section III–III. **e** Smoke concentration (ppm) in section II–II. **f** Smoke concentration (ppm) in section III–III. **g** Velocity in section 0.2 m from the ceiling. **h** Dissatisfied people (%) due to draft in section 1.65 m from the floor.

Figures 5e, f, 11e, f show that the lower the ventilation rate, the higher the contaminant concentration. However, the distributions in the two cases look alike regardless of the difference in the absolute values.

With a smaller ventilation rate, the draft risk can be reduced even if the supply air momentum is higher. This can be seen by comparing figures 5h and 11h.

Conclusions

Two simulation methods, the 84-slot method and the momentum method, have been used to model a jet diffuser with 84 round nozzles. Corresponding experimental data are used for comparison. The agreement between the computations and the measurements is reasonably good. The computing cost with the 84-slot method is high. Hence, the momentum method is recommended to be used to simulate a complex diffuser in practice.

Computations have been conducted to study the influence of the parameters of a jet diffuser on indoor air diffusion. The parameters being examined include diffuser effective area, diffuser dimension, diffuser position, air supply direction, flow rate, and air temperature. The following conclusions may be drawn:

(1) The angle between the jet and the ceiling should not be too large or too small in order to obtain a longer projection of total air. For a better throw projection, an angle between 20 and 60° is desirable.

(2) The effective flow area has a significant impact on indoor airflow pattern since it significantly affects total air movement. A smaller effective area increases the supply air momentum. Consequently, a longer projection of total air may be expected and the room air temperature will be uniform. However, this may result in a higher discomfort level.

(3) The diffuser width has a stronger influence on indoor air diffusion than the diffuser height. This is because the variation of the inlet width implies a larger change in the inlet area that contacts the surrounding room air.

(4) The distance between inlet and ceiling has a remarkable influence on total air movement near the ceiling, but has a minor impact on air diffusion in the occupied zone.

(5) The air velocity distribution is sensitive to the ventilation rate and the supply air temperature. To achieve the same length of the throw projection, the Reynolds number should be the same if the corresponding Archimedes number is close to them. This is particularly important if the ventilation system works with a variable-air-volume control strategy.

References

1 Whittle GE: Computation of air movement and convective heat transfer within buildings. Int J Ambient Energy 1986;3:151–164.

2 Nielsen PV: Progress and trends in air infiltration and ventilation research; in Proc 10th AIVC Conf. Coventry, Air Infiltration and Ventilation Centre, 1989.

3 Rhodes N: Prediction of smoke movement: An overview of field models. ASHRAE Trans 1989;95(1).

4 Launder BE, Spalding DB: The numerical computation of turbulent flows. Comput Methods Appl Mech Eng 1974;3:269–289.

5 Chen Q, Moser A, Huber A: Prediction of buoyant, turbulent flow by a low-Reynolds-numer k-ε model. ASHRAE Trans 1990;96(1): 564–573.

6 Borth J: Numerical simulation of air flows in rooms. Eur Res Community Flow Turbulence Combustion Bull 1990;5:9–14.

7 Rosten HI, Spalding DB: The PHOENICS reference manual: Version 1.4, Report TR200. London, CHAM Ltd, 1987.

8 Fanger PO, Melikov AK, Hanzawa H, Ring J: Turbulence and draft: The turbulence of airflow has a significant impact on the sensation of draft. ASHRAE J 1989;31(7):18–23.

9 Fossdal S: Measurement of testcase E (mixed convection, summer cooling) (preliminary), Report of International Energy Agency Annex 20, 1990.

10 Heikkinen J: Measurements of case B2, B3, E2 and E3 (draft), Working Report of International Energy Agency Annex 20, 1990.

11 Chen Q, Moser A: Simulation of a multiple-nozzle diffuser; in Proc 12th AIVC Conf Air Mov Ventilation Control within Buildings. Coventry, Air Infiltration and Ventilation Centre, 1991.

12 Skovgaard M, Hyldgaard CE, Nielsen PV: High and low Reynolds number measurements in a room with an impinging isothermal jet; in Proc 2nd Int Conf Roomvent '90. Oslo, NORSK VVS, 1990, pp B1.1–17.