No. 3173

2976

NUMERICAL AND EXPERIMENTAL STUDY ON TURBULENT DIFFUSION FIELDS IN CONVENTIONAL FLOW TYPE CLEAN ROOMS

S. Murakami, D. Eng.   **S. Kato, D. Eng.**   **Y. Suyama**

ABSTRACT

Turbulent flow fields of velocity and diffusion in several types of conventional clean rooms are precisely analyzed both by model experiment and by numerical simulation based on the k-ε two-equation turbulence model. The detailed analyses of contaminant diffusion by simulation make it possible to comprehend clearly the structures of velocity and diffusion fields in clean rooms.

The flow fields in such rooms, as analyzed here, are mainly characterized by the inflow jet and the rising streams around it. The combination of one jet and rising streams forms a "flow unit." The total velocity field and the resulting diffusion field of contaminant in a room are well modeled as serial combinations of these "flow units."

INTRODUCTION

In designing an effective contamination control in a conventional (turbulent) flow type of clean room, an understanding of the flow field itself and also how to control the resulting diffusion field of contaminant is most important. Therefore, it is essential that clean room engineers and designers comprehend the entire flow field and its diffusion field of contaminant, not only qualitatively, but also quantitatively (Kato and Murakami 1986).

The flow fields in conventional clean rooms may be expected to be fully turbulent because the air exchange rates are always very high. Numerical simulation of the flow fields based on a turbulence model has become one of the most powerful and effective tools for analyzing the flow fields and the diffusion fields of contaminant in such rooms (Murakami et al. 1987). However, as the exact degree of accuracy of numerical simulation of turbulent flows is still somewhat unclear, the results of simulation had best be confirmed by model experiments.

The airflow pattern in a conventional clean room is mainly determined by the shape of the room and the number of supply outlets. Therefore, in order to accurately design the airflow for such a clean room, one should analyze each room independently. However, it is also wellknown that the flow fields of such clean rooms share many common characteristics, especially when the supply outlets are on the ceiling. In this study, the flow fields and the resulting diffusion fields of contaminant in conventional clean rooms, whose supply outlets are located on the ceiling, are precisely analyzed.

Shuzo Murakami, Dr. Eng., is Professor, Institute of Industrial Science, University of Tokyo, Japan;
Shinsuke Kato, Dr. Eng., is Associate Professor, Institute of Industrial Science, University of Tokyo, Japan;
Yoshimi Suyama, formerly Joint Researcher, Institute of Industrial Science, University of Tokyo, is now Researcher, Institute of Technology, Hazama-Gumi Co.Ltd.,Japan.

The distribution of contaminant diffusion is a very useful means by which to comprehend the diffusion field. On the other hand, the diffusion field alone cannot give effective information for evaluating ventilation efficiency because, when given two patterns of contaminant diffusion, it is often difficult to judge which one is better. For this purpose, we need a simple index that can express the characteristics of the diffusion pattern as a quantitative value. Murakami and Kato (1986) proposed the new concept of **ventilation efficiency** for the diffusion fields of contaminant and presented a method by which to express the different distributions of contaminant concentration as a whole and to evaluate the difference of ventilation efficiency. We will here briefly summarize the new concept of **ventilation efficiency** and apply it to the diffusion fields in the clean rooms under discussion.

DIMENSIONLESS STUDY

In this study, physical quantities are made dimensionless by dividing them by representative quantities. Those quantities are selected as the width of supply outlet, L_0, its bulk velocity, U_0, and the mean contaminant concentration, C_0, over all exhaust outlets. The value of C_0 is necessarily equal to the ratio of the contaminant generation rate to the air supply volume rate.

MODEL CLEAN ROOMS ANALYZED

Four types of clean room models are used for analysis in this study. The specifications of each room are presented in Table 1. Figure 1 shows the geometry of each room model. Generally, the representative length, the width of supply outlet L_0 in a conventional flow type of clean room, is about 0.6 m. The height of the ceiling of the clean room models in full scale thus corresponds to 4.5×0.6=2.7 m. The source points of contaminant are located under the supply outlet, near the wall, and at the center of the room, respectively. Their heights from the floor are set equally at 1.25 in dimensionless value (the height of the ceiling is 4.5 in dimensionless value). Another source point of contaminant is located in front of the exhaust inlet, where its height from the floor is 0.5. As contaminant is assumed to be of passive scalar quantity in this study, which has no effect on the momentum equations, its transportation or diffusion is fully controlled by the flow. Flow fields and resulting diffusion fields are assumed to be in steady states. Contaminant generation is also assumed to be constant.

MODEL EXPERIMENTS

Each model experiment was conducted using a 1/6 scale model. The representative length, the width of supply outlet L_0, was set as 0.1 m in the room model. The velocity of the jet from supply outlet U_0 was set at about 6 m/s. The Reynolds number of the inflow jet $U_0 L_0/\nu$ is about 4.2×10^4. The velocity of the jet from the supply outlet in full-scale conventional clean rooms is usually set at about 1 m/s. Therefore, the Reynolds number for the model experiment is the same as that of a full-scale clean room.

Air velocity is measured by means of a tandem type, parallel hot-wire anemometer, which can discern the vector components of turbulent flow (Murakami et al. 1980).

The distribution of contaminant concentration is investigated by means of a tracer gas diffusion experiment. Since ethene (C_2H_4), whose density is nearly the same as that of air, is used as the tracer, the buoyancy effect of the tracer can be disregarded. Its concentration is measured by means of F.I.D. gas chromatography.

NUMERICAL SIMULATION

Model equations (k-ε two-equation turbulence model) are given in Table 2. The boundary conditions are tabulated in Table 3. The flow fields in rooms divided into the mesh systems shown in Figure 2 are solved by the finite difference method. The numerical simulation method follows that given in Murakami et al. (1987). After the room flow fields are obtained, the contaminant diffusion fields are calculated using such flow field properties as the distributions of velocity vectors and of eddy viscosity.

The simulated flow fields are not entirely steady and symmetrical due to numerical instability. However, asymmetry of the flow fields is very slight and can be disregarded. The calculated contaminant diffusion fields are thus also slightly asymmetric in correspondence to the flow fields.

EXPRESSION METHOD OF CONTAMINANT DIFFUSION FIELD

In this study, contaminant diffusion fields are expressed by four methods. The latter three in particular correspond exactly to the **ventilation efficiency** scales proposed by Murakami and Kato (1986). The details of the definition of these new scales are described in the companion paper by Kato and Murakami (1988).

Distribution of Contaminant Concentration in Case of Point Source

A distribution of contaminant concentration where contaminant is generated as a point source in a room is first to be presented. It should be noted that the concentration is made dimensionless by dividing it by the representative value. This distribution gives suggestive information, which allows intuitive comprehension of the contaminant diffusion field in a clean room. From this distribution, we can surmise where the contaminant is likely to remain or where the contaminant is likely to be exhausted smoothly. These distributions, mainly given by the numerical simulation, make up the basic data for calculating the three new ventilation efficiency scales.

Spatial Average Concentration

A spatial average concentration corresponds exactly with the **First Scale of Ventilation Efficiency (SVE1)**. It is derived by calculating the spatial average of the distribution of the contaminant concentration over the entire space. In a situation where the contaminant generation and exhaust are stationary, the spatial average concentration is proportional to the averaged time the contaminant is present in the room. This condition may be easily explained as follows. When the generated contaminant takes more time to be convected to the exhaust inlet, it is certain that there exists more contaminant within the room in spite of the constant generation and constant exhaust of contaminant. Therefore, the spatial average concentration indicates how quickly the contaminant generated in the room is exhausted by the flow field.

It should be noted again that the concentration of contaminant is made dimensionless by dividing it by the representative value. In a situation where the contaminant is mixed with air prior to reaching its supply outlet, the spatial average concentration is equal to 1. Therefore, if the value goes below 1, it means that the **ventilation efficiency** in the room is relatively good and that the contaminant is likely to be easily exhausted. If the value exceeds 1, it means that the **ventilation efficiency** is not so effective and that the contaminant is likely to stay longer in the room.

Mean Radius of Diffusion

The square root of the second moment of the distribution of the contaminant concentration is the **Second Scale of Ventilation Efficiency (SVE2)**. It is reasonably termed the "Mean Radius of Diffusion." In the calculation of the second moment of the distribution of the contaminant concentration, the center of gravity of the concentration distribution is set at origin in the coordinates. Because the concentration distribution is three-dimensional, six components of second moments are obtained; three are normal moments and three are cross-moments. In this study, the resultant radius, which is the square root of the sum of the three normal second moments, is used as the mean radius of diffusion.

Concentration in Case of Contaminant Generated Uniformly throughout Room

A concentration in the case of contaminant generated uniformly throughout a room is the **Third Scale of Ventilation Efficiency (SVE3)**. The air mass from a supply outlet travels through the room to the exhaust inlet. In a situation where the contaminant is uniformly and continuously generated throughout a room, the convected air mass from a supply outlet is gradually contaminated by mixing with the generated contaminant. Its concentration, that is, the degree to which it becomes contaminated, seems to be proportional to the elapsed time from when the air mass leaves the supply outlet. Thus, the concentration at a given point in the case of uniform contaminant generation throughout a room surely corresponds to the mean traveling time of supply air to that point.

CORRESPONDENCE BETWEEN EXPERIMENT AND SIMULATION

Flow Field

As is shown in Figure 3 (a) and (c) and Figure 7 (a) and (c), the results of the simulation of the flow field correspond well with those of the experiment. Figure 3 (a) and (c) show a comparison of the simulated with the experimental results of the distribution of the velocity vectors in the sectional plane, including the supply outlet for the Type 1 clean room. Figure 7 (a) and (c) show a comparison in the case of Type 2. Detailed comparisons are given in Murakami et al. 1987.

Diffusion Field

As is shown in Figure 3 (b) and (d) and Figure 7 (b) and (d), the results of the simulation of the contaminant diffusion field, in the case where the contaminant is generated at a point near the floor in the supply jet, correspond well to those of the experiment. Although the contour lines of concentration are not exactly the same, the main characteristics of the contaminant diffusion are well reproduced, that is, the shape of the high concentration region under the tracer source, the low concentration region under the supply outlet, and so on. However, the results of the simulation tend to be more diffusive than those obtained by the experiment, and the value of the contaminant concentration tends to be smaller than that given by the experiment for areas where the concentration is high and to be larger for areas where the concentration is low.

FLOW AND DIFFUSION FIELD FOR TYPE 1 (ONE SUPPLY OUTLET)

In the Case of Contaminant Generated under Supply Outlet

The flow field is shown in Figure 3 (c) and (e). The jet from the supply outlet collides with the floor and diverges toward the wall. The diverged streams reach the sidewalls and turn up toward the ceiling. The distribution of concentration in the case where the contaminant is generated in the supply jet is shown in Figure 3 (d) and (f). The contaminant source point is marked as Ⓐ. The concentration is very high in the area between the source of contaminant and the floor. However, the value of the concentration is rather uniform throughout the room and is more than 0.5, except for the area just beneath the supply outlet where it is very clean (Figure 3 (d)). The spatial average concentration is 0.89 and the mean radius of diffusion is 2.75, which is 29% of the relevant length of the room, 8.4. The relevant length of the room is defined as the square root of the sum of the square of each of the three dimensions of the room.

In the Case of Contaminant Generated between Supply Jet and Wall

Figure 4 shows the distributions of concentration in the case of the contaminant being generated between the supply jet and the wall at points B, C, and D, respectively. The generated contaminant is convected and diffused by the diverged flow near the floor and by the rising stream along the wall (Figure 4 (a), (b), and (c)). When the air velocity is relatively weak at the source of the contaminant, it diffuses in all directions (Figure 4 (b)). The spatial average concentrations are 0.98 (in the case of point B), 1.28 (in the case of point C), and 1.55 (in the case of point D). These values become larger as the source points are located closer to the wall. The mean radii of diffusion are 2.4 (in the case of point B), 2.3 (in the case of point C), and 2.1 (in the case of point D). These values become smaller as the sources are located nearer to the wall.

In the Case of Contaminant Generated Near Exhaust Inlet

The contaminants generated near the exhaust inlet are convected and diffused, some by the flow toward the exhaust inlet and others by the rising stream along the wall, as shown in Figure 5. The spatial average concentration is 0.96, which is greater than the value in the case of contaminant generated in the supply jet. The mean radius of diffusion is 2.2.

In the Case of Contaminant Generated Uniformly throughout Room

Figure 6 shows the distribution of the concentration in the case of the contaminant being generated uniformly throughout the room. The concentration becomes higher as the travel time of supplied air increases. The concentration is low under the supply outlet and is slightly higher at the floor. The concentration takes its highest value around the supply outlet, as shown in Figure 6 (c). Thus, in terms of air mass movement, because the area around the supply outlet is farthest from the supply outlet and the air mass takes longest to reach the area around the supply outlet, the probability of the air around the supply outlet being contaminated is the highest.

FLOW AND DIFFUSION FIELD FOR TYPE 2 (FOUR SUPPLY OUTLETS)

Characteristics of Flow Field

The distributions of velocity vectors in the several sectional planes are shown in Figure 7. Many characteristics of the flow pattern of Type 1 often appear in Type 2. It may be reasonable modeling to regard the flow pattern of Type 2 as a combination of four flow patterns of Type 1. The flow pattern of Type 1, which is characterized by a vertical down jet from the supply outlet and the rising streams around it, might be called a "flow unit," each of which occupies a quarter space of Type 2.

In the Case of Contaminant Generated under Supply Outlet

The supply jet hits the floor and diverges in all directions. Rising streams are formed between the area of supply outlets and the area near the side walls (Figure 7 (c)).The contaminant that is generated in the supply jet spreads in accordance with this flow field. The concentration is highest in the area from just below the source of the contaminant to the floor (Figure 7 (d)). The value of concentration is more than 0.5 only in the quarter part of the room that corresponds to the single "flow unit" in which the contaminant is generated (Figure 7 (h)). In the remaining space of the room, concentration is very low (Figure 7 (d) and (h)). The spatial average concentration is 0.76 and is less than the value in the same case of Type 1. The mean radius of diffusion is 3, which is 25% of the relevant length of the room, 12.1, and is relatively less than the value in the same case for Type 1. These results are caused by the fact that the spreading area of the contaminant is confined to one flow unit.

In the Case of Contaminant Generated Near Wall

Figure 8 shows the flow and diffusion field when the contaminant is generated near the wall. The contaminant is diffused by the rising streams along the wall (Figure 8 (a) and (b)). As converging flows along the ceiling toward the supply outlet convect the contaminant, the value of concentration is high near the supply outlet (Figure 8 (c) and (d)). The value of concentration exceeds 0.5 in about one-half the room, which corresponds to two flow units. The contaminant is generated at the exact boundary of the two flow units. The spatial average concentration is rather high at 1.05. However, the mean radius of diffusion is 2.8, which is smaller than that in the former case where the contaminant is generated in the supply jet.

In the Case of Contaminant Generated at Center of Room

As the contaminant is convected by the rising streams in the center of the room, as is shown in Figure 9, it spreads throughout the room. Such a situation is caused when the contaminant is generated at the crossing point of the boundaries of the four flow units. The spatial average concentration is 1.5 and the mean radius of diffusion is 3.2, both the highest in the case of Type 2.

In the Case of Contaminant Generated Near Exhaust Inlet

The main part of the contaminant generated near the exhaust inlet is discharged by the flow toward the exhaust inlet. It does not spread to the room, as shown in Figure 10. The spatial average concentration is 0.06, which is extremely low. The mean radius of diffusion is 1.7 and is also very small.

In the Case of Contaminant Generated Uniformly throughout Room

Figure 11 shows the distribution of concentration in the case of the contaminant being generated uniformly throughout the room. As stated already, the concentration becomes higher as the air travels a greater distance. The major characteristics of the distribution pattern of the concentration are almost the same as in the case of Type 1. However, at the corner of the ceiling, the concentration becomes higher, which differs from the distribution pattern in the case of Type 1.

FLOW AND DIFFUSION FIELD FOR TYPE 3 (SIX SUPPLY OUTLETS)

In the Case of Contaminant Generated under Supply Outlet

The flow field of Type 3 is shown in Figure 12 (a) and (b). As in the case of Type 2, the flow pattern of Type 3 may be regarded as a serial combination of flow units, in this case, six units. When the contaminant is generated in the flow unit facing the exhaust inlet, which corresponds to contaminant generation at point A (Figure 12 (c) and (d)), the contaminant is directly exhausted and hardly diffused to other flow units. The contaminant spreads very well in this 'flow unit' with relatively high concentration. In this case, the spatial average concentration is 0.53, which is a very low value, and the mean radius of diffusion is 2.7, 19% of the relevant length of the room of 14.3. However, when the contaminant is generated in the flow units that do not contain the exhaust inlet, which is the case of contaminant generation at point B (Figure 12 (e) and (f)), the contaminant spreads not only into the flow unit in which the contaminant is generated, but also into the flow units that are adjacent to the contaminated flow unit and on the way to the exhaust inlet. In this case, the concentration is very high in the contaminant-generating flow unit. The spatial average concentration is 1.51 and the mean radius of diffusion is 3.7, 26% of the relevant length of the room. Both scales of **ventilation efficiency** take large values.

In the Case of Contaminant Generated Near Wall

Figure 13 shows the flow and diffusion field when the contaminant is generated near the wall. The contaminant is convected and diffused by the rising stream along the wall (Figure 13 (b) and (d)). The contaminant spreads only into the three flow units on one side of the room, while the other side of the room is not contaminated at all and remains very clean. The spatial average concentration is 1.82, the highest in the case of Type 3. The mean radius of diffusion is 3.3.

In the Case of Contaminant Generated at Center of Room

Figure 14 shows the distribution of concentration in the case of the contaminant being generated at the center of the room. The contaminant is convected and diffused into the whole room. The value of concentration in the room is more than 1.0 except for the area just under the supply outlet. In this case, the spatial average concentration is 1.72 and the mean radius of diffusion is 3.6. Both **ventilation efficiency** scales take larger values.

In the Case of Contaminant Generated Near Exhaust Inlet

Because the contaminant generated near the exhaust is discharged very smoothly, it is not diffused into the room, as shown in Figure 15. The spatial average concentration is only 0.03, and the mean radius of diffusion is 0.13. The latter value is much smaller than that in the same case of Type 2.

In the Case of Contaminant Generated Uniformly throughout Room

Figure 16 shows the distribution of concentration in the case of the contaminant being generated uniformly. The concentration becomes higher as the supplied air travels a greater distance. The major characteristics of the concentration distribution pattern are almost the same as in the case of Type 2.

FLOW AND DIFFUSION FIELD FOR TYPE 4 (NINE SUPPLY OUTLETS)

In the Case of Contaminant Generated under Supply Outlet

The flow field of Type 4 is shown in Figure 17 (a) and (b). As with Type 2 or Type 3, it is logical to regard the pattern of Type 4 as a serial combination of flow units, in this case nine units. When the contaminant is generated in the flow unit that faces the exhaust inlet corresponding to point A (Figure 17 (c) and (d)), the contaminant hardly diffuses into the other flow units, although the concentration is very high in that single flow unit. The spatial average concentration in this case is only 0.26, and the mean radius of diffusion is 2.26, 14% of the relevant room length of 16.

When the contaminant is generated in the center flow unit adjacent to the wall, which source point corresponds to point C (Figure 17 (e) and (f)), the contaminant spreads, not only within that center flow unit, but also into the adjacent flow units that are located on the way to the exhaust inlet. That one-third of the room is contaminated, but the remaining two-thirds of the room is very clean. The spatial average concentration is 1.15, and the mean radius of diffusion is 3.3. The latter value is considerably greater than that of the contaminant being generated at point A.

When the contaminant is generated in the center of the room at point E, all of the space is contaminated. Because this flow unit in which the contaminant is generated does not face the exhaust outlet but is adjacent to all the other flow units, the contaminant is convected by the flow toward the exhaust through all the other flow units. The spatial average concentration is 1.37, and the mean radius of diffusion is 4.25, 26% of the relevant length of the room.

In the Case of Contaminant Generated Near Wall

Figure 18 shows the diffusion field when the contaminant is generated at point B, at point C, at point D, and at point E. These source points move from the area neighboring the wall to the center of the room. The spatial average concentrations are 1.56, 1.15, 1.43, and 1.37, respectively. When the contaminant is generated at the boundary of the flow unit, where strong rising streams are formed, the spatial average concentration takes a higher value. The mean radii of diffusion are 3.1, 3.3, 3.6, and 4.25. Therefore, the mean radius of diffusion becomes greater as the contaminant source is placed farther from the wall.

In the Case of Contaminant Generated Uniformly throughout Room

Figure 19 shows the distribution of concentration. The major characteristics of the concentration distribution pattern are almost the same as in the cases of Type 2 or Type 3. The highest value is observed near the ceiling around the supply outlet and at the corners of the ceiling.

DISCUSSION

From the results of the simulations, it may be concluded that the mean flow structure in a conventional clean room that has supply outlets on the ceiling is composed of series of flow units that consist of one supply jet and the rising streams around it. Such a flow unit is useful in comprehending the complicated flow pattern in a clean room in which the supply outlets are on the ceiling. Furthermore, this concept of a flow unit seems to be helpful in comprehending the contaminant diffusion in a conventional clean room. It is a well-known fact that the exhaust flow has little influence on the whole flow pattern. Therefore, it is not detrimental that this model of a flow unit does not incorporate the function of the exhaust inlet.

When the contaminant is generated in a flow unit, the contaminant diffusion is confined within that unit. If the flow unit faces the exhaust inlet, the contaminant is not convected to the other flow units and only a small amount of the contaminant spreads to them by turbulent diffusion. If the contaminant is generated in the flow unit that does not face any exhaust inlets, the contaminant is convected to the flow units that face the exhaust outlet and the remaining flow units are not contaminated. If they do become contaminated, it is only to a small degree, because such contamination is caused solely by turbulent diffusion, which has much less ability to transport the contaminant than does mean flow convection. The turbulent Reynolds number (Peclet number), $U_0 L_0/\nu_t$, in these cases is on the order of 100, which means in general that the ability to transport the contaminant by convection is a hundred times greater than that of turbulence diffusion.

When the contaminant is generated at the boundary of two flow units, where strong rising streams are usually formed, the contaminant diffuses into both and passes through the other flow units that are located on the path of the flow to the exhaust inlet.

The characteristics of the structure of the diffusion field described above are quantitatively assessed very well by means of the new **scales of ventilation efficiency.**

CONCLUSIONS

It is confirmed that numerical simulation of the velocity and diffusion fields in a conventional flow type of clean room is very useful in comprehending flow and diffusion patterns. The characteristics of the airflow and contaminant diffusion in a conventional clean room with ceiling supply outlets are summed up as follows.

1. Mean flow structures of the airflow are modeled very well as serial combinations of flow units, which consist of one supply jet and the rising streams around it.
2. The resulting diffusion field is mainly caused by the convection of the mean airflow. The structure of the diffusion fields also becomes very clear by introducing the concept of flow units.
3. The new **scales of ventilation efficiency,** which are the spatial average concentration, the mean radius of diffusion, and the concentration in case of contaminant generated uniformly throughout a room, are very useful measures for comparing the different diffusion fields and for quantitatively comprehending diffusion properties. These scales are strong tools that summarize in clear fashion very complex information on room diffusion fields, which is hard to characterize clearly by any other means.

At the next stage, we will analyze the effects of flow obstacles on the flow, the effects of varying the arrangement of outlets and inlets, and the effects of varying the volume of supply air to each supply outlet.

NOMENCLATURE

C_D, C_1, C_2 = empirical constants in turbulence model (*cf.* Table 2)
C = mean contaminant concentration
C_0 = representative concentration defined by that of exhaust outlet
k = turbulence kinetic energy
k_0 = boundary value for k of inflow
l = length scale of turbulence
l_0 = boundary value for l of inflow
L_0 = representative length defined by width of supply outlet
P = mean pressure
R_e = Reynolds number, $R_e = U_0 L_0/\nu$
R_{et} = turbulence Reynolds number, $R_{et} = U_0 L_0/\nu_t$
U, V, W = X, Y, Z components of velocity vector
U_i, U_j = components of velocity vector
U_0 = representative velocity defined by inflow air velocity
κ = von Karman constant, 0.4
ρ = fluid density
ν = molecular kinematic viscosity
ν_t = eddy kinematic viscosity
$\sigma_1, \sigma_2, \sigma_3$ = turbulence Prandtl/Schmidt number of k, ε, C (*cf.* Table 2)

REFERENCES

Kato, S., and Murakami, S. 1986. "Study on air flow in conventional flow type clean room by means of numerical simulation and model test." 8th International Symposium on Contamination Control, Milan, Sept.9~12, pp. 781~791.

Kato, S., and Murakami, S. 1988. "New ventilation efficiency scales based on spatial distribution of contaminant concentration aided by numerical simulation." ASHRAE Transactions 1988, Vol.94, Part 2.

Murakami, S., and Komine, H. 1980. "Measurement of three components of turbulent flow with tandem hot-wire probe." Transactions of Architectural Institute of Japan, No. 297, pp. 21~28.

Murakami, S., and Kato, S. 1986. "New scales for ventilation efficiency and calculation method by means of 3-dimensional numerical simulation for turbulent flow." Transactions of the SHASE of Japan, vol.32, pp. 91~102.

Murakami, S.; Kato, S.; and Suyama, Y. 1987. "Three-dimensional numerical simulation of turbulent airflow in ventilated room by means of a two-equation model." ASHRAE Transactions, vol.93, Part2.

ACKNOWLEDGMENTS

The authors are grateful for the generous cooperation of S.Nagano, joint researcher at the Institute of Industrial Science, University of Tokyo, and Fujita Corp. Ltd. This study has been partially supported by a Grant-in-Aid for Scientific Research from the Japanese Ministry of Education, Culture and Science.

TABLE 1

Specifications of Model Clean Rooms

Model Clean Room TYPES	Dimension of Plan (mm×mm)	Height of Ceiling (mm)	Number of Supply Outlets	Number of Exhaust Inlets	Supply Air Velocity (m/s)	Number of Air Changes (1/h)
TYPE 1	3000×3000	2700	1	4	1.0	53.3
TYPE 2	4800×4800	2700	4	4	1.0	83.3
TYPE 3	6600×4800	2700	6	4	1.0	90.9
TYPE 4	6600×6600	2700	9	4	1.0	99.2

TABLE 2

Two-Equations Model (Three-Dimensional)

$\frac{\partial U_i}{\partial X_i} = 0$	(1) Continuity equation
$\frac{\partial U_i}{\partial t} + \frac{\partial U_i U_j}{\partial X_j} = -\frac{\partial}{\partial X_i}\left(\frac{P}{\rho} + \frac{2}{3}k\right) + \frac{\partial}{\partial X_j}\left(\nu_t\left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i}\right)\right)$	(2) Momentum equation
$\frac{\partial k}{\partial t} + \frac{\partial k U_j}{\partial X_j} = \frac{\partial}{\partial X_j}\left(\frac{\nu_t}{\sigma_1}\frac{\partial k}{\partial X_j}\right) + \nu_t S - \varepsilon$	(3) Transport equation for k
$\frac{\partial \varepsilon}{\partial t} + \frac{\partial \varepsilon U_j}{\partial X_j} = \frac{\partial}{\partial X_j}\left(\frac{\nu_t}{\sigma_2}\frac{\partial \varepsilon}{\partial X_j}\right) + C_1\frac{\varepsilon}{k}\nu_t S - C_2\frac{\varepsilon^2}{k}$	(4) Transport equation for ε
$\nu_t = k^{1/2}\, l = \left(C_D\frac{k^2}{\varepsilon}\right)$	(5) Equation for deciding ν_t
$\frac{\partial C}{\partial t} + \frac{\partial C U_j}{\partial X_j} = \frac{\partial}{\partial X_j}\left(\frac{\nu_t}{\sigma_3}\frac{\partial C}{\partial X_j}\right)$	(6) Concentration equation

here $S = \left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i}\right)\frac{\partial U_i}{\partial X_j}$, $\sigma_1 = 1.0$, $\sigma_2 = 1.3$, $\sigma_3 = 1.0$, $C_D = 0.09$, $C_1 = 1.44$, $C_2 = 1.92$

TABLE 3

Boundary Conditions for Numerical Simulation

(1) Supply Outlet boundary :	$U_t = 0.0$, $U_n = U_{out}$, $k = 0.005$, $l = 0.33$, $C = 0.0$ suffix t : tangential component, n : normal component U_{out} : Supply outlet velocity, $U_{out} = 1.0$
(2) Exhaust Inlet boundary :	$U_t = 0.0$, $U_n = U_{in}$, $\partial k/\partial Z = 0.0$, $\partial \varepsilon/\partial Z = 0.0$, $\partial C/\partial Z = 0.0$ U_{in} : Exhaust inlet velocity, in case of Type2 : $U_{in} = 1.0$
(3) Wall boundary :	$\partial U/\partial Z_{z=0} = m U_{t\,z=h}/h$, $U_n = 0.0$, $\partial k/\partial Z = 0.0$, $\partial C/\partial Z = 0.0$ $\varepsilon_{z=h} = [C_D k_{z=h}^{3/2}]/[C_D^{1/4}\kappa h]$ h : Length from the wall surface to the center of the adjacent cell m : 1/7, Power law of profile $U_t \propto Z^m$ is assumed here. κ : 0.4, von Karman constant

TABLE 1

Specifications of Model Clean Rooms

Model Clean Room TYPES	Dimension of Plan (mm×mm)	Height of Ceiling (mm)	Number of Supply Outlets	Number of Exhaust Inlets	Supply Air Velocity (m/s)	Number of Air Changes (1/h)
TYPE 1	3000×3000	2700	1	4	1.0	53.3
TYPE 2	4800×4800	2700	4	4	1.0	83.3
TYPE 3	6600×4800	2700	6	4	1.0	90.9
TYPE 4	6600×6600	2700	9	4	1.0	99.2

TABLE 2

Two-Equations Model (Three-Dimensional)

$\frac{\partial U_i}{\partial X_i} = 0$	(1) Continuity equation
$\frac{\partial U_i}{\partial t} + \frac{\partial U_i U_j}{\partial X_j} = -\frac{\partial}{\partial X_i}\left(\frac{P}{\rho} + \frac{2}{3}k\right) + \frac{\partial}{\partial X_j}\left\{\nu_t\left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i}\right)\right\}$	(2) Momentum equation
$\frac{\partial k}{\partial t} + \frac{\partial k U_j}{\partial X_j} = \frac{\partial}{\partial X_j}\left(\frac{\nu_t}{\sigma_1}\frac{\partial k}{\partial X_j}\right) + \nu_t S - \varepsilon$	(3) Transport equation for k
$\frac{\partial \varepsilon}{\partial t} + \frac{\partial \varepsilon U_j}{\partial X_j} = \frac{\partial}{\partial X_j}\left(\frac{\nu_t}{\sigma_2}\frac{\partial \varepsilon}{\partial X_j}\right) + C_1\frac{\varepsilon}{k}\nu_t S - C_2\frac{\varepsilon^2}{k}$	(4) Transport equation for ε
$\nu_t = k^{1/2} l = \left(C_0 \frac{k^2}{\varepsilon}\right)$	(5) Equation for deciding ν_t
$\frac{\partial C}{\partial t} + \frac{\partial C U_j}{\partial X_j} = \frac{\partial}{\partial X_j}\left(\frac{\nu_t}{\sigma_3}\frac{\partial C}{\partial X_j}\right)$	(6) Concentration equation
here $S = \left(\frac{\partial U_i}{\partial X_j} + \frac{\partial U_j}{\partial X_i}\right)\frac{\partial U_i}{\partial X_j}$, $C_0 = 0.09$, $\sigma_1 = 1.0$, $\sigma_2 = 1.3$, $\sigma_3 = 1.0$, $C_1 = 1.44$, $C_2 = 1.92$	

TABLE 3

Boundary Conditions for Numerical Simulation

(1) Supply Outlet boundary:	$U_t = 0.0$, $U_n = U_{out}$, $k = 0.005$, $l = 0.33$, $C = 0.0$ suffix t: tangential component, n: normal component U_{out}: Supply outlet velocity, $U_{out} = 1.0$
(2) Exhaust Inlet boundary:	$U_t = 0.0$, $U_n = U_{in}$, $\partial k/\partial Z = 0.0$, $\partial \varepsilon/\partial Z = 0.0$, $\partial C/\partial Z = 0.0$ U_{in}: Exhaust inlet velocity, in case of Type2: $U_{in} = 1.0$
(3) Wall boundary:	$\partial U/\partial Z_{\,z=0} = m U_{t\,z=h}/h$, $U_n = 0.0$, $\partial k/\partial Z = 0.0$, $\partial C/\partial Z = 0.0$ $\varepsilon_{z=h} = [C_0 k^{3/2}_{z=h}]/[C_0^{1/4} \kappa h]$ h : Length from the wall surface to the center of the adjacent cell m : 1/7, Power law of profile $U_t \propto Z^m$ is assumed here. κ : 0.4, von Karman constant

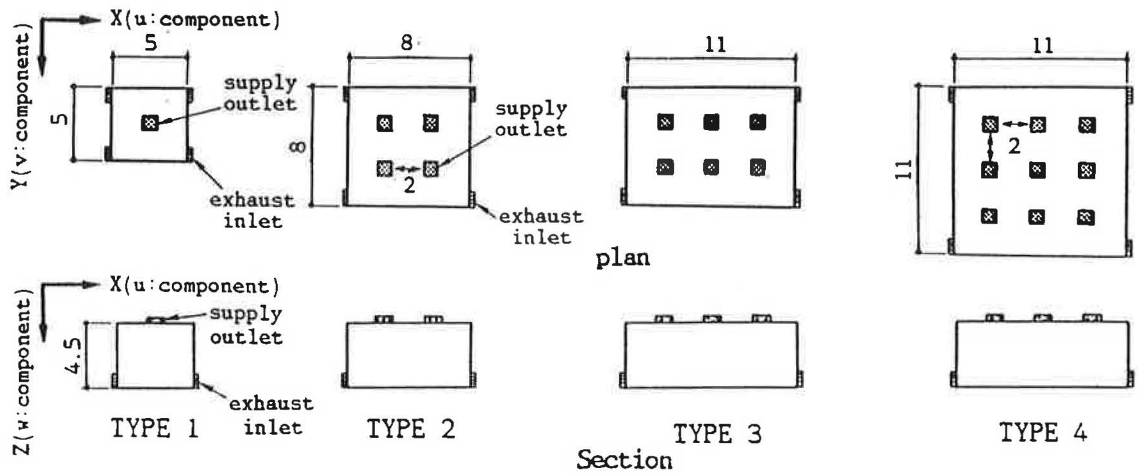


Figure 1 Plans and sections of model clean rooms (Length scale in this figure is nondimensionalized by the width of supply outlet L_O)

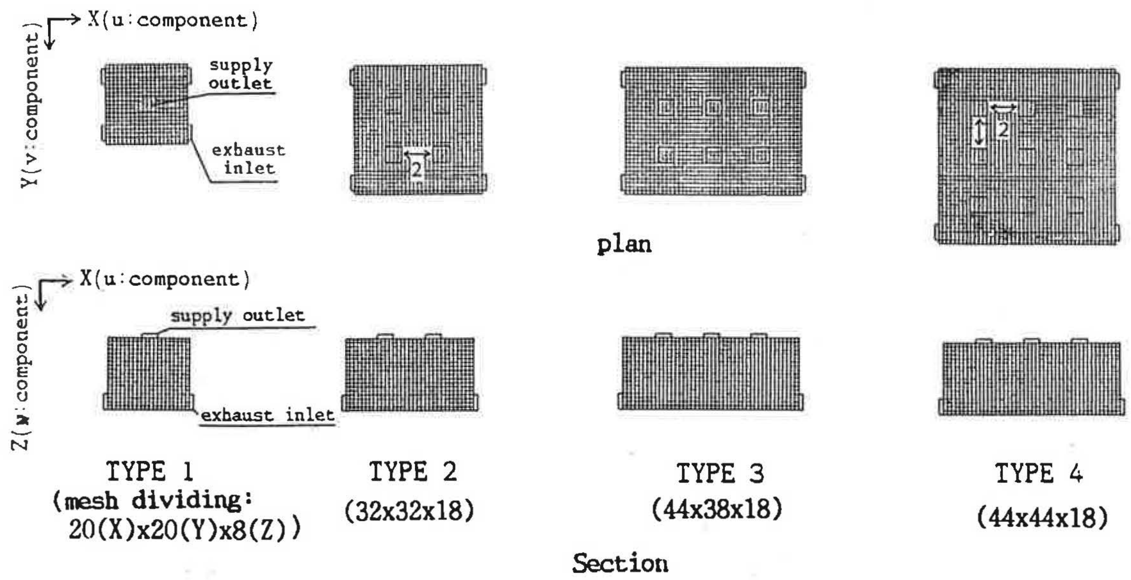


Figure 2 Mesh systems

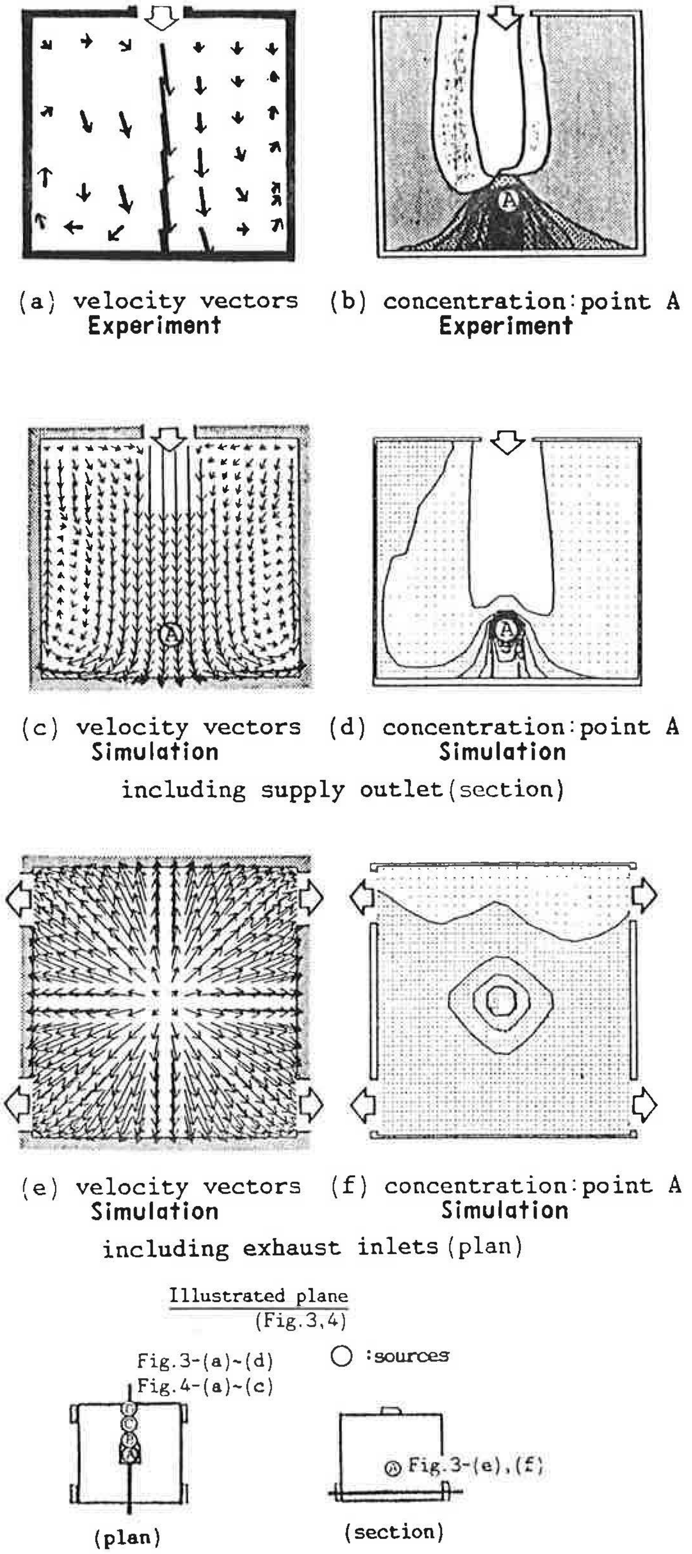


Figure 3 Velocity vectors and contaminant distribution in TYPE 1 clean room model (1 outlet & 4 inlets, source : point A)

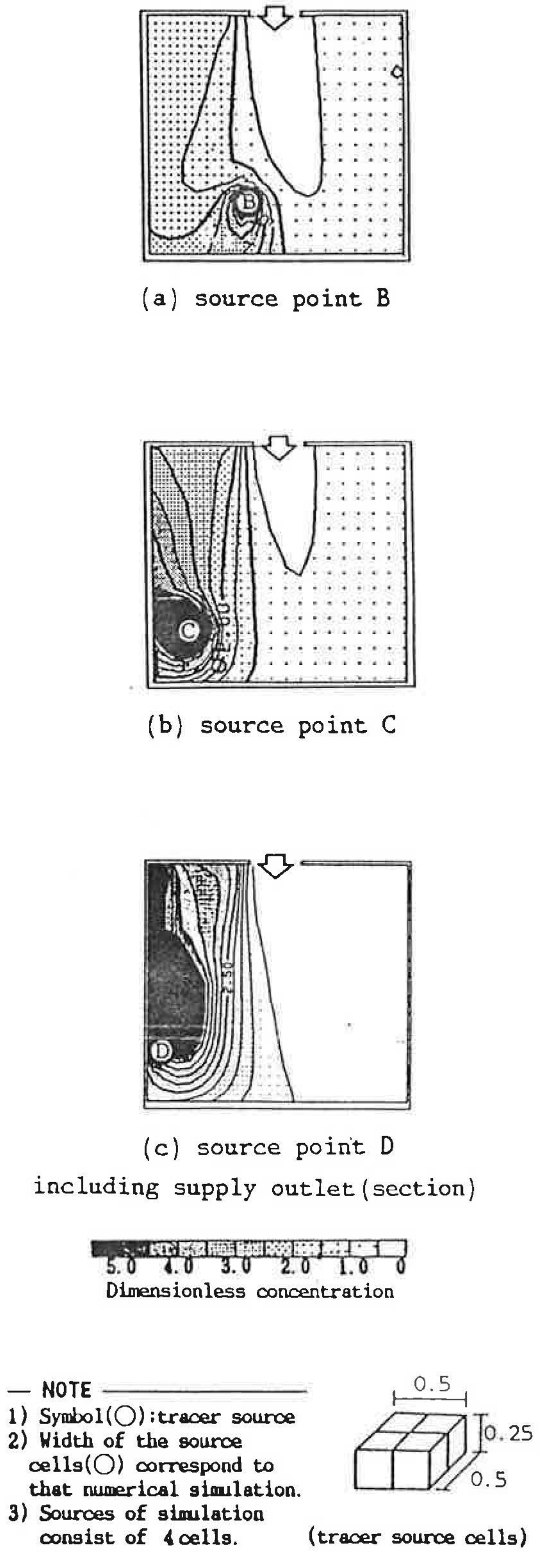


Figure 4 Comparison of contaminant distribution for various source points in TYPE 1 (simulation, source : point B~D)

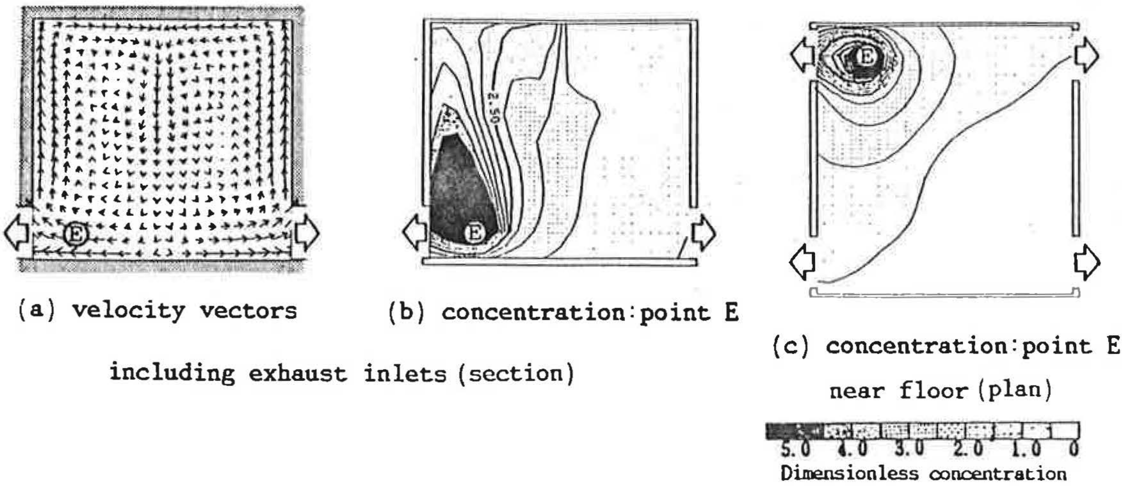


Figure 5 Velocity vectors and contaminant distribution in TYPE 1 (simulation, source : point E)

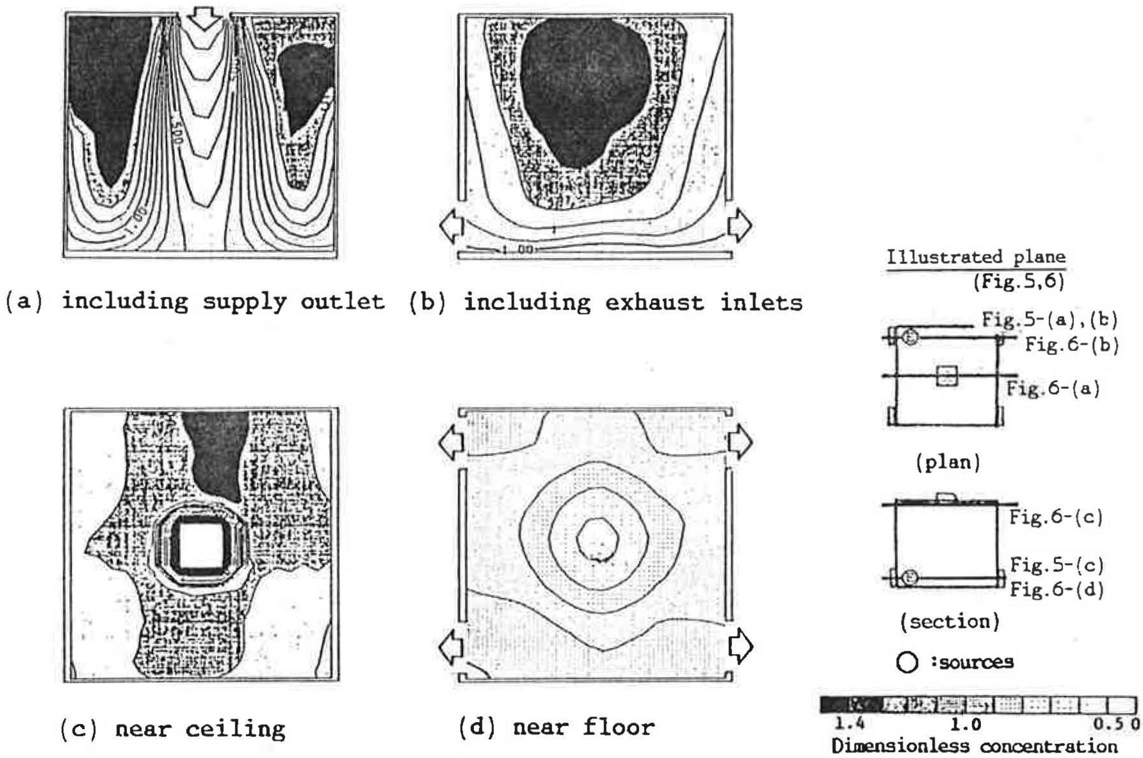


Figure 6 Contaminant distribution in TYPE 1 (simulation, source : uniform generation in whole room)

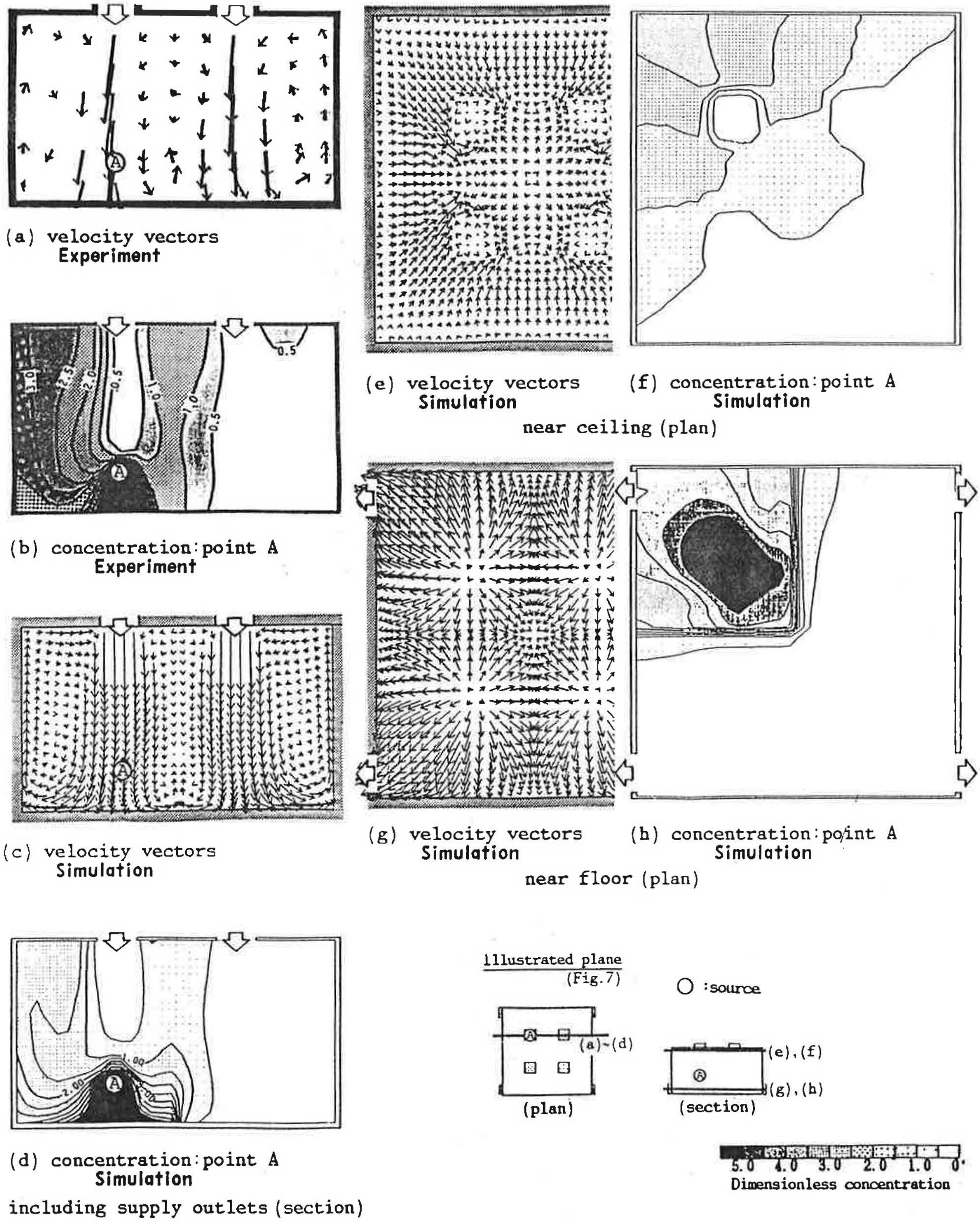
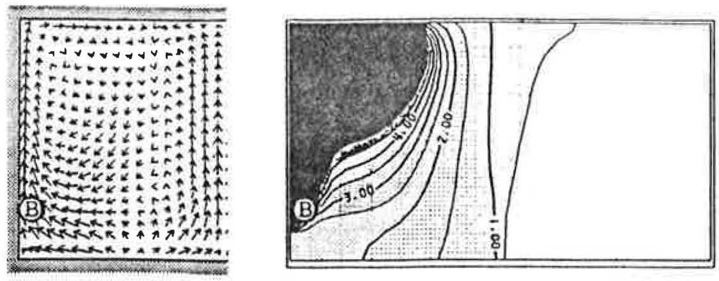
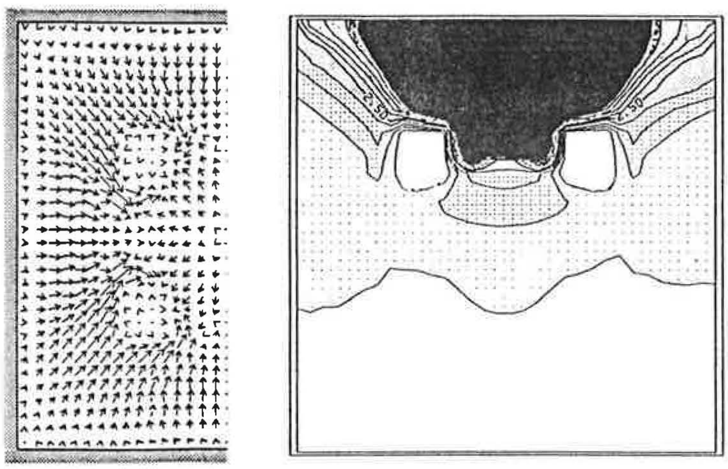


Figure 7 Velocity vectors and contaminant distribution in TYPE 2 clean room model (4 outlets & 4 inlets, source : point A)

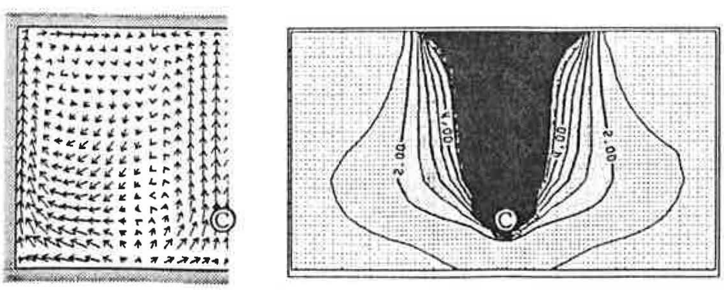


(a) velocity vectors (b) concentration:point B
center of room (section)



(c) velocity vectors (d) concentration:point B
near ceiling (plan)

Figure 8 Velocity vectors and contaminant distribution in TYPE 2 (simulation, source : point B)



(a) velocity vectors (b) concentration:point C
center of room (section)

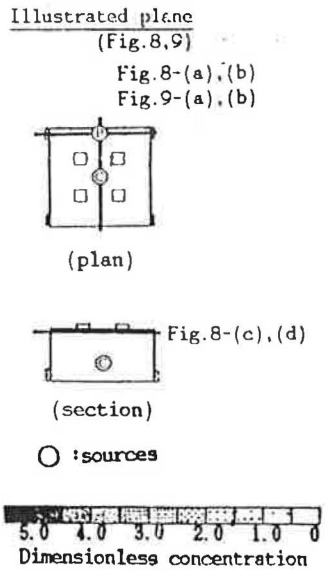


Figure 9 Velocity vectors and contaminant distribution in TYPE 2 (simulation, source : point C)

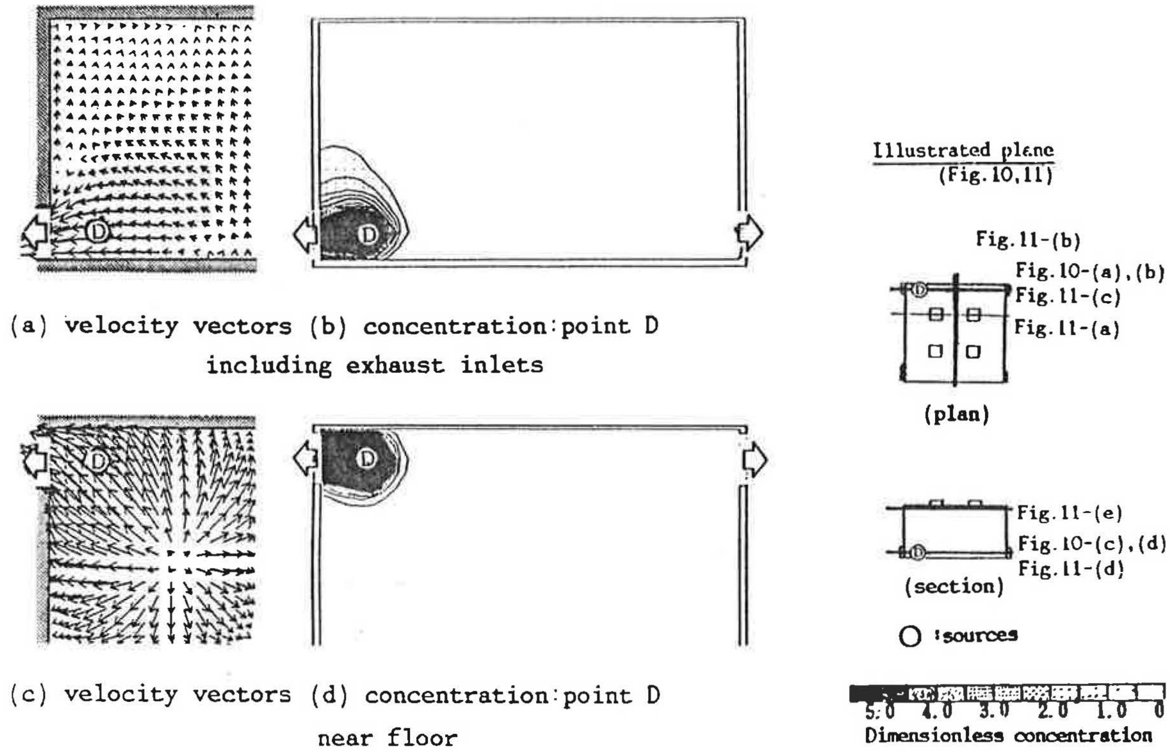


Figure 10 Velocity vectors and contaminant distribution in TYPE 2 (simulation, source : point D)

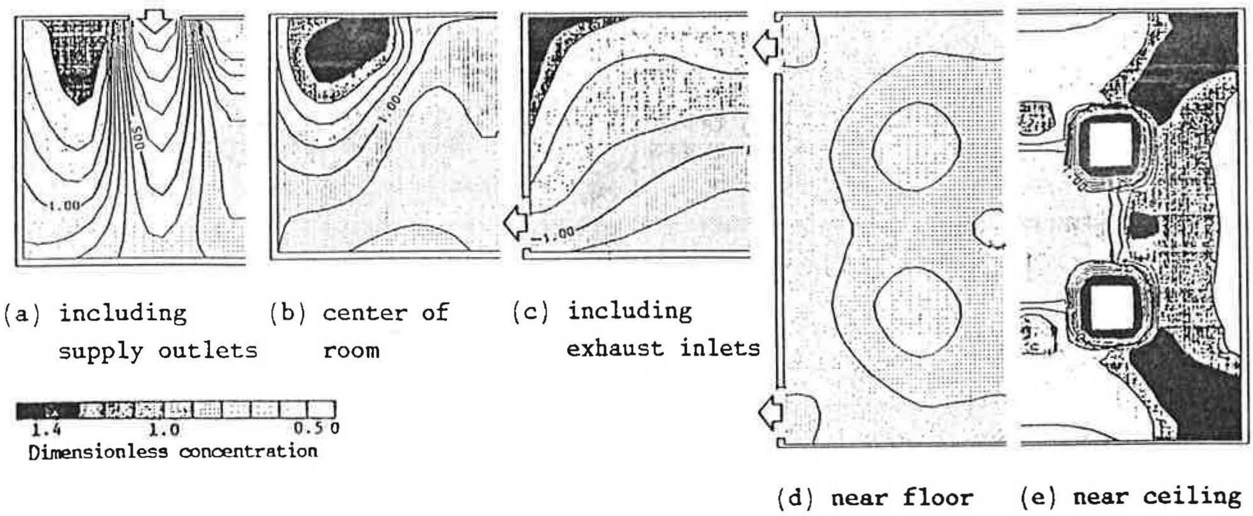


Figure 11 Contaminant distribution in TYPE 2 (simulation, source : uniform generation in whole room)

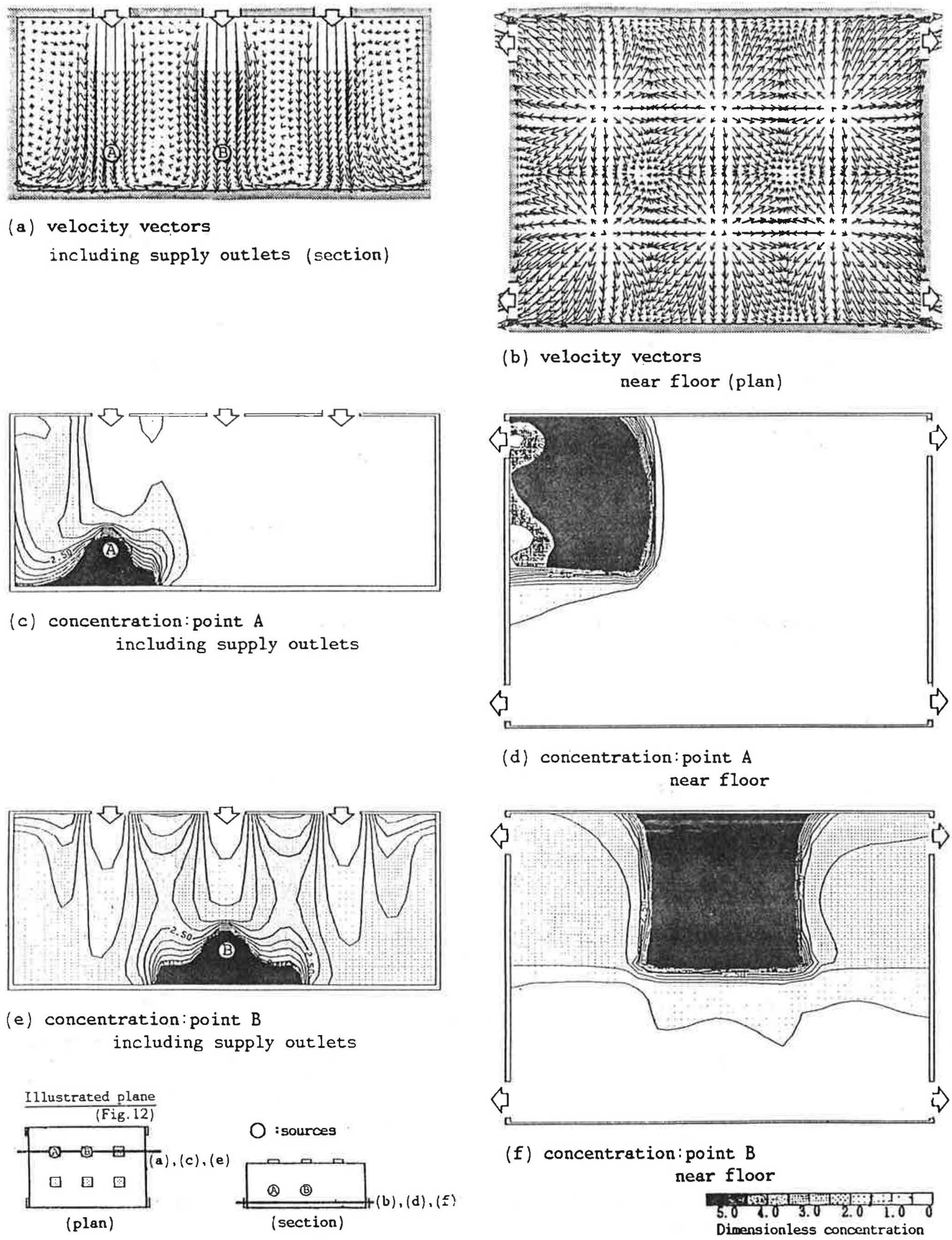


Figure 12 Velocity vectors and contaminant distribution in TYPE 3 clean room model (simulation, 6 outlets & 4 inlets, source : point A, B)

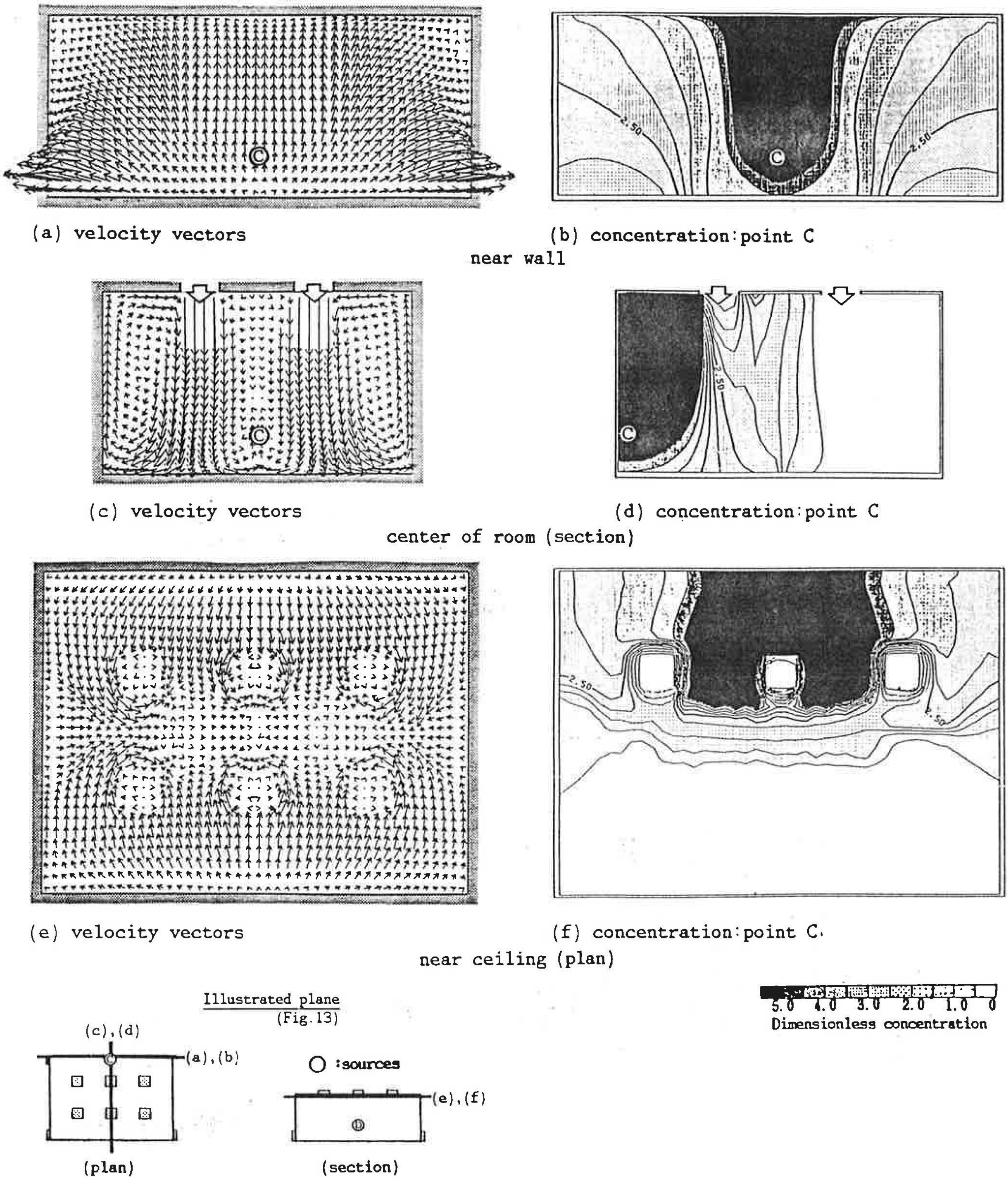
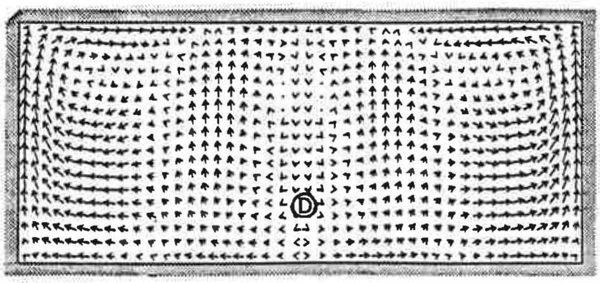
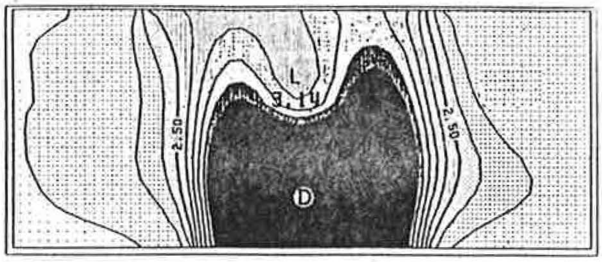


Figure 13 Velocity vectors and contaminant distribution in TYPE 3 (simulation, source : point C)

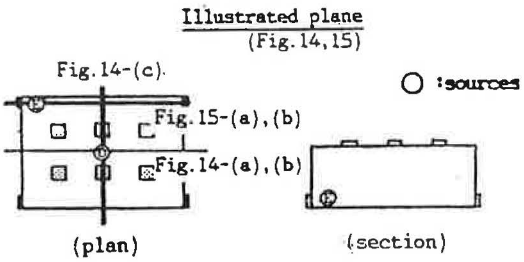


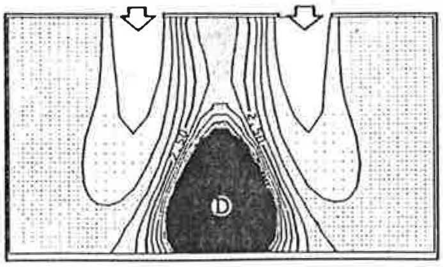
(a) velocity vectors



(b) concentration:point D

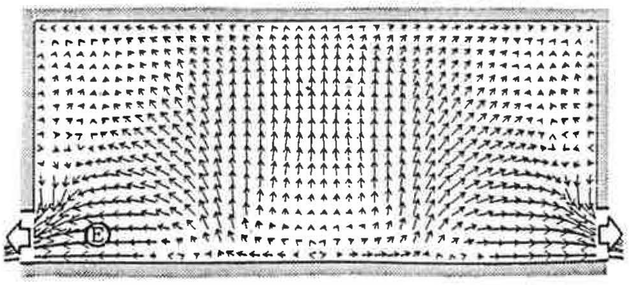
center of room (section)



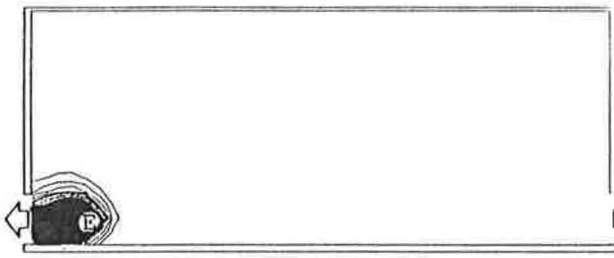


(c) concentration:point D
center of room

Figure 14 Velocity vectors and contaminant distribution in TYPE 3 (simulation, source : point D)



(a) velocity vectors



(b) concentration:point E

including exhaust inlets (section)

Figure 15 Velocity vectors and contaminant distribution in TYPE 3 (simulation, source : point E)

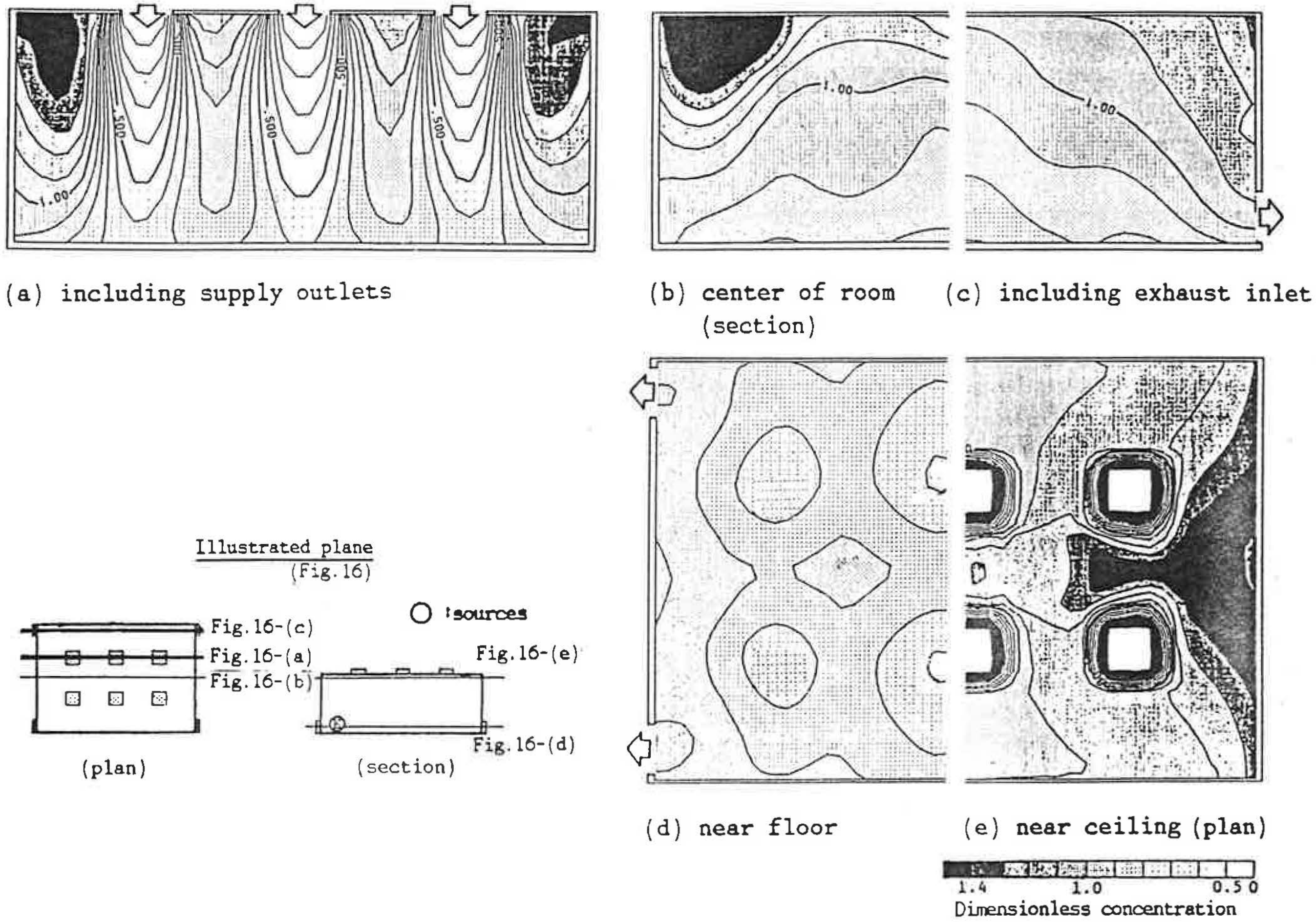


Figure 16 Contaminant distribution in TYPE 3 (simulation, source : uniform generation in the whole room)

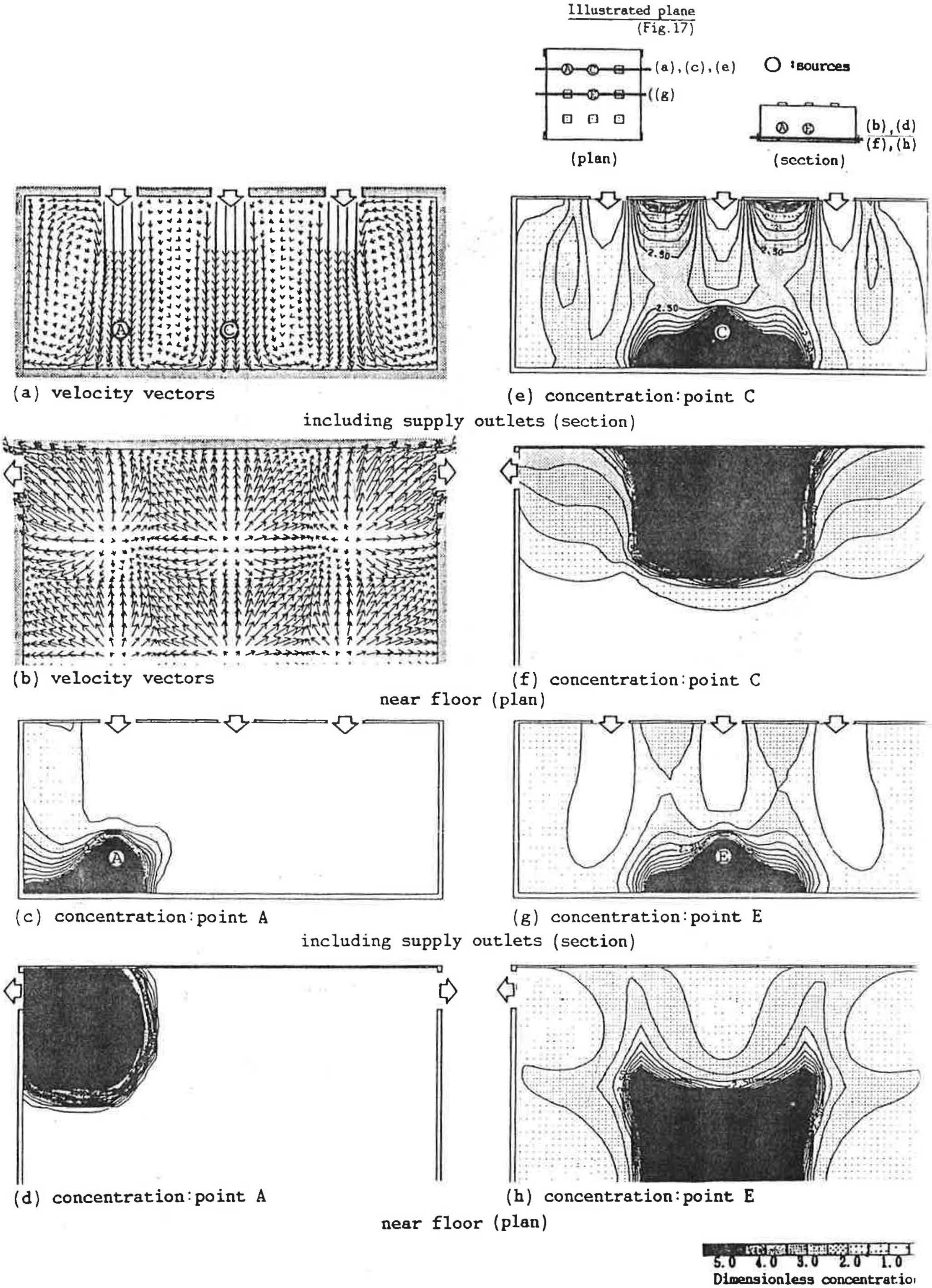


Figure 17 Velocity vectors and contaminant distribution in TYPE 4 clean room model (simulation, 9 outlets & 4 inlets, source : point A, C, E)

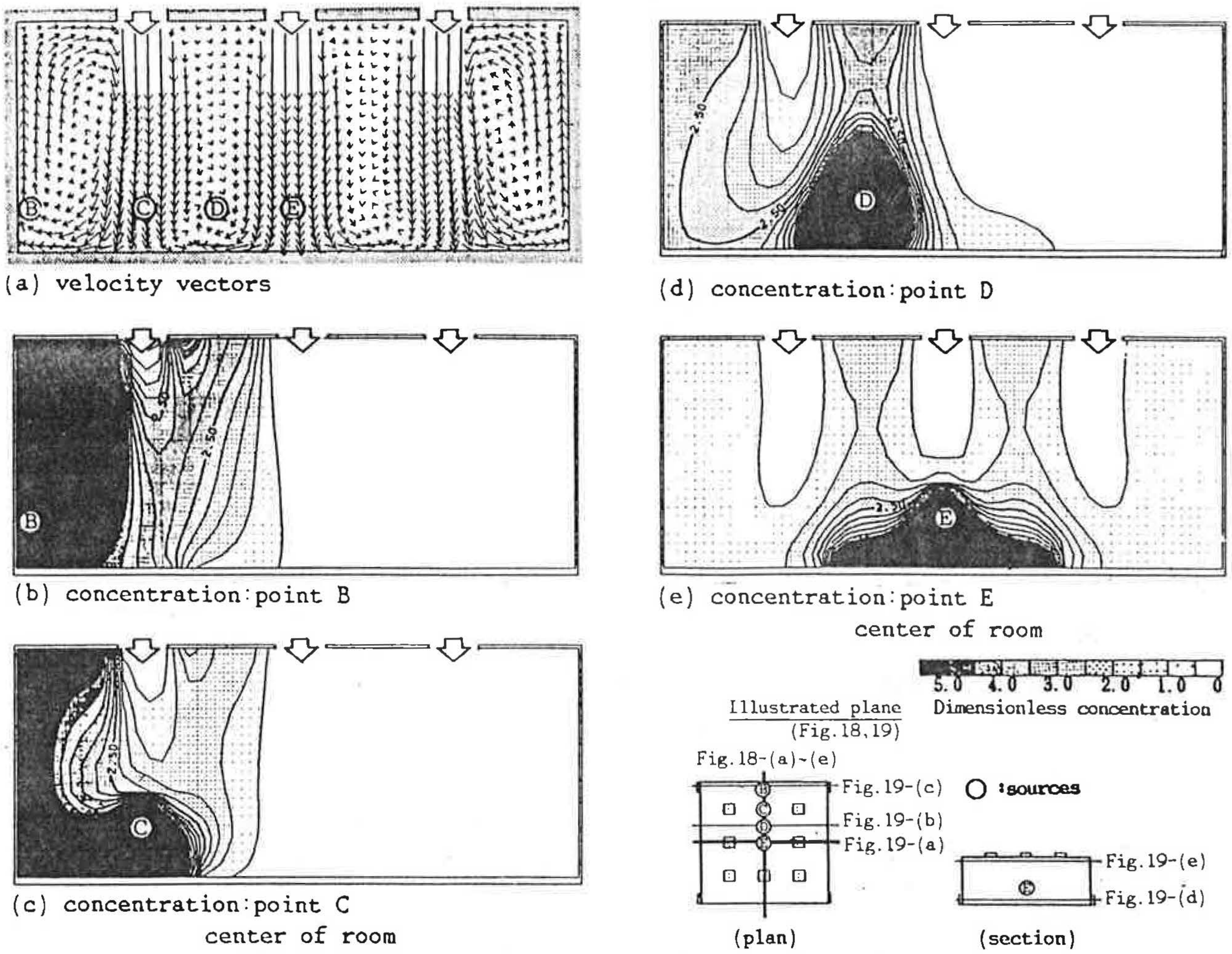


Figure 18 Comparison of contaminant distribution for various source points in TYPE 4 (simulation, source : point B~E)

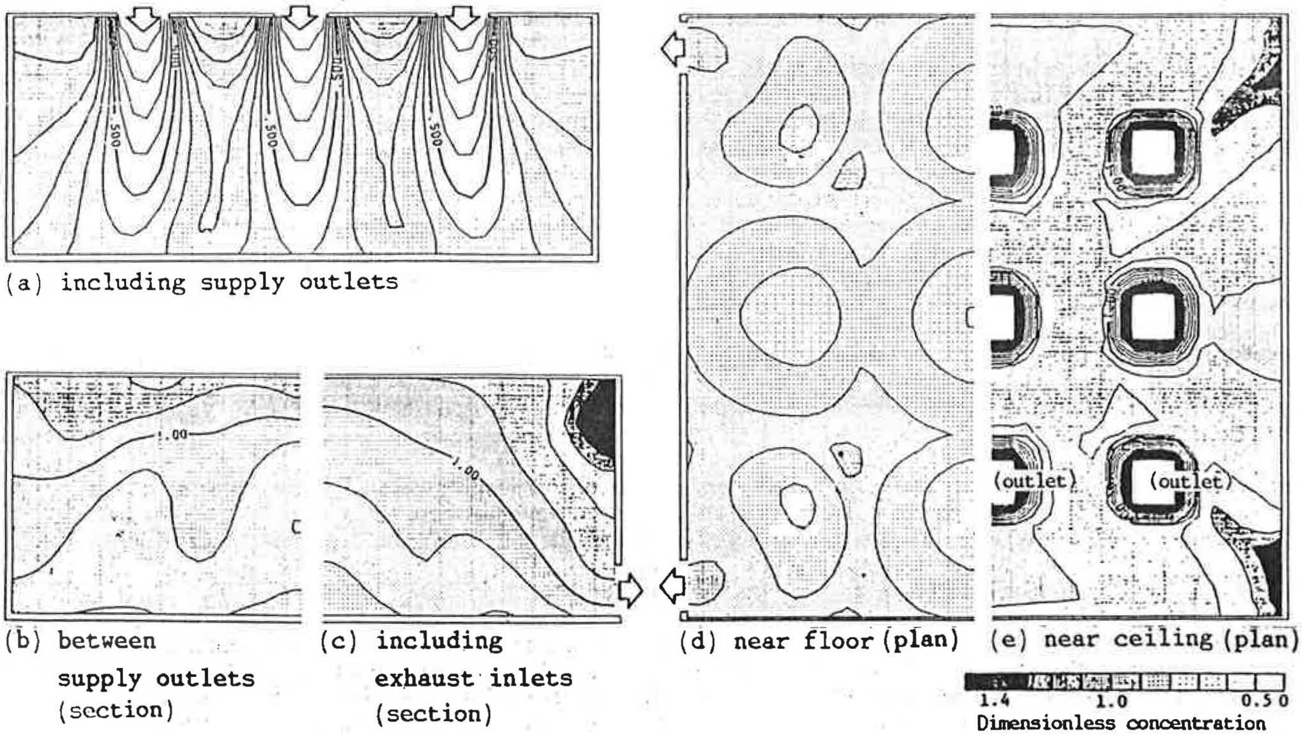


Figure 19 Contaminant distribution in TYPE 4 (simulation, source : uniform generation in whole room)

DISCUSSION

A. Fobelets, Research Assistant, John B. Pierce Foundation, New Haven, CT: In the discussion you implied that the turbulent Reynolds number and Peclet number are identical; this is not true in all situations. What assumptions have you made in this respect?

S. Murakami: We assumed here that the turbulent Schmidt number equals one (see Table 2). If we deal with a problem such as heat transport, we should surely suppose that the turbulent Prandtl number is less than one. However, as we treat here the diffusion of a passive scalar contaminant, such as fine airborne particles or a gaseous substance, it is reasonable to suppose that the turbulent Schmidt number is almost one. Therefore, the turbulent Reynolds number and the Peclet number are identical.

Fobelets: The same statement seems to imply that turbulent diffusion is negligible when compared to convection. I believe this fact to be true, at least away (more than 5 to 15 mm) from the walls. Why then do you retain the diffusion terms in your transport equations?

Murakami: In the convection-dominated flow field treated here, turbulent diffusion plays a much smaller role than mean-flow convection in transporting the contaminant. Generally, turbulent diffusion is negligible when the turbulent Peclet number is sufficiently large. However, in a room, there are regions where the mean air velocity is low and the turbulent Peclet number is small. We cannot neglect the turbulent diffusion in such regions.

Fobelets: I wonder if the Reynolds number defined in the duct flow (in the inlet) has any meaning within the room. Reynolds numbers can be derived from characteristic lengths and velocities pertaining to the room flow.

Murakami: I think you will agree with me on the fact that the supply air jet has a crucial influence on the behavior of room air flow. From this standpoint, we used the Reynolds number defined at the supply opening. Generally speaking, we can choose any characteristic lengths and velocities concerning room air flow to define the Reynolds number. It is easy to estimate another Reynolds number that is defined by a different representative scale. But the one that is defined at the supply outlet seems to be as valid as any other. I would like to ask you in return which is the most suitable length scale in the room. We cannot always deal with the same room shape. The room configuration might change from case to case. Even if you choose the ceiling height, you could encounter a case where the ceiling height varies in the room. I think the length scale of the supply opening is the most suitable one. We do not deal with a simple configuration, such as pipe or channel flow. Because we deal with the complicated shape of flow, the simplest definition of Reynolds number is appropriate.

Fobelets: The model for experimental testing is based on Reynolds scaling, but the Reynolds number is computed in the inlet duct, not in the room. I suspect this is the reason there is good agreement between simulation and experiment. You have velocities of 6 m/s in a room that is about 0.5 m in size [see work by P.V. Nielson (1973-1981) also published in ASHRAE].

Murakami: I cannot understand your argument. As mentioned in the previous answer, any Reynolds number is easily translated into another Reynolds number if the definition of Reynolds number is clear. We did conduct the model experiment on the basis of the same definition of the Reynolds number in the experiment and in the actual geometry. If the Reynolds number defined in terms of the supply opening is identical for the full-scale and the model, the Reynolds number defined by another scale is also identical.

The most important fact is that the flow in the room must be in a condition of asymptotic similarity with respect to Reynolds number; that is, the air flow must be fully turbulent, and we can neglect the molecular viscosity effect. The modeling used in this study assumes that the flow is fully turbulent. From this viewpoint we tested the Reynolds number effect in the model experiment by changing the supply air velocity, and we could not find any significant changes in the flow field in the room.

C. Qingyan, Ph.D. Student, Delft University, Holland: You used the power law velocity profile; why don't you use the log-law form? Normally, the generation of k in the boundary is in balance with the dissipation of k. Could you explain why you use (insert formula) but use sources for ? How did you get those boundary equations?

Murakami: We agree with you that the log-law velocity profile is more consistent with the boundary condition for dissipation . There is no special reason for using the power-law rather than the log-law.

We would like to say, however, that the boundary condition used here is only one of a number of possible models for the boundary condition. We have obtained rather successful results in the past with the current practice. Also, we have confirmed that the result from the power-law boundary condition corresponds well to that from the log-law.

N. Nelson, Project Manager, CHZM Hill, Rexon, VA: What was the flow rate through the room? Did you test for different flow rates? What type of outlets were in the room and what was the outlet velocity?

Murakami: The air-exchange rate of the model clean room is in the range of 53 to 90 per hour. In the model experiment, we made a preliminary test with one-sixth of the real air-exchange rate for each model and we confirmed that, in this range of air exchange, no change in the flow field can be observed. So it may be assumed that the air flow is fully turbulent if the air-exchange rate exceeds 10 per hour in this situation. Outlets are single square openings. The outlet jet velocity in the model is 6 m/s. In the full-scale room, this value corresponds to 1 m/s.