CONTAMINANT DISPERSION IN PERSONAL DISPLACEMENT VENTILATION

Caiqing Yang[1], Xudong Yang[1], Yue Xu[2], and Jelena Srebric[3]

[1]Department of Building Science, Tsinghua University, Beijing 100084, China
[2]Department of Civil, Architectural, Environmental Engineering, University of Miami, Coral Gables, USA
[3]Department of Architectural Engineering, The Pennsylvania State University, University Park, USA

ABSTRACT

Personal displacement ventilation (PDV) is a new ventilation concept that intends to combine the positive features of displacement ventilation with those of task conditioning or personalized ventilation. PDV is expected to create a micro-environment around the occupant to control the environment individually. In this study, a PDV with a contaminant source at different locations was modeled for contaminant dispersion in a full scale chamber. Computational fluid dynamics (CFD) was used to simulate the indoor airflow and pollutant transport, and the simulation results were validated against the experimental data. The contaminant concentration field for three different contaminant source locations was analyzed. It seems that this kind of PDV system cannot create the expected "micro-environment" to avoid the disturbance of the outside airflow. Further studies are needed to examine the conditions where PDV could perform better.

KEYWORDS

Personal displacement ventilation, CFD, contaminant dispersion, micro-environment

INTRODUCTION

Ventilation plays a fundamental role in maintaining good indoor air quality and thermal comfort in buildings. Mixing and displacement ventilation are widely used nowadays, however, neither of them can provide the required specifications for individual controls in a small volume around the occupant, which is called micro-environment. To solve the problem, task ventilation system, which can provide occupants with improved thermal comfort, air quality and individual control of micro-environment, was proposed (Kaczmarczyk et al. 1999). Neverthless, this improved ventilation method could create thermal discomfort becasue the supply of fresh air to breathing zone can disturb the buoyancy-driven natural convection plume around the human body (Gao 2004). Combining the positive features of displacement ventilation and those of task conditioning, personal displacement ventilation (PDV) is proposed. The major objective of PDV is to create a healthy and comfortable micro-environment within a macro-environment. Because the air is supplied directly to the occupied zone, the air quality near the occupant could be improved. In general, PDV applies the rules set by the DV principle, such as introduction of sub-cooled air, with a supply temperature only slightly lower than room temperature, over a relatively large area at low velocity (e.g., 0.2 m/s), and, therfore, the specific comfort requirement of the occupant may be individually controlled and satisfied. Loomans (1998) investigated a PDV system (desk displacement ventilation) with regard to micro/macroclimate and thermal comfort. The study concluded that comfort conditions can be achieved with desk displacement ventilation. However, this study did not include air contaminant and is thus limited to thermal comfort considerations. Therefore, our study experimentally and numericaly examines the contaminant dispersion in PDV system with different contaminant source locations.

EXPERIMENTAL MEASUREMENTS

A set of full-scale chamber experiments were conducted to study the airflow, temperature and pollutant transport with the PDV system. The facility, located at the Pennsylvania State University, Department of Architectural Engineering, is composed of an indoor environmental chamber and an outdoor climate chamber. The chambers' walls are constructed with R-30 insulation and coated with galvanized steel on both sides to isolate the experimental environment from the surrounding. There is a partition wall with a sliding window which divides the climate and environmental chambers. Each of the chambers has a separate Air Handling Unit (AHU) capable of simulating various environmental conditions. Both AHUs have a pre-filter and a High Efficiency Purification of Air (HEPA) filter which together provide 95% cleaning of outdoor air.

A model office room was built within the environmental chamber to mimic a PDV system as shown in Figure 1. The size of the chamber is 6m × 3.9m × 2.35m. One occupant and one computer were placed in front of the inlet as heat sources. The three

PDV cases studied by experiments had the same setup of human simulator, computer, and table, but different location of contaminant source (Figure 2). In Case 1, the contaminant source was located 0.15m in front of the human simulator's feet, and 0.1m above the floor. In case 2, the contaminant source was located at the same location as in Case 1, but 1.1m above the floor, which is almost at the same height as the human simulator's mouth. In Case 3, the contaminant source was located 0.15m from the back of human simulator, and 0.1m above the floor.

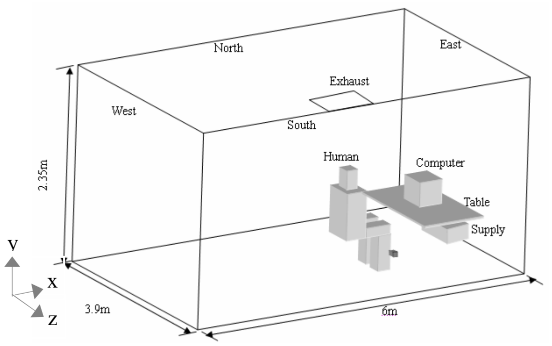


Figure 1 Schematic of the PDV test cases

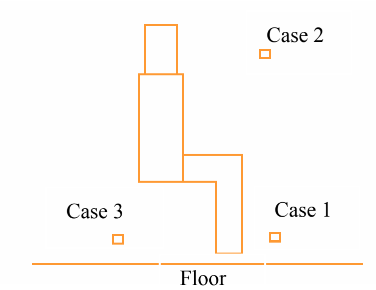


Figure 2 The contaminant source locations for the three PDV cases

The shape of human simulator has been simplified to a group of cubes, which reperesented head, chest and legs. The height of human simulator is 1.6m, and the surface area is 1.68m^2. Heating panels were p inside of human simulator to generate heat of 76W. The computer is simulated by a 0.46m (length) × 0.32m (width) × 0.37m (height) chipboard box. The box was sealed on all surfaces and placed on the table in front of human simulator during the experiments. A 40W heat source (lamp) was placed in the middle of the box.

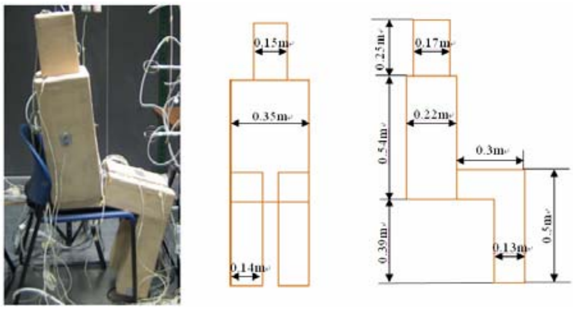


Figure 3 The human simulator in experiment

The dimensions of the supply and exhaust diffusers are 0.4 m (length) × 0.15 m (width) and 0.34 m × 0.14m, respectively. The air supply rate was 43m^3/h (0.79 ACH) with fluctuation of ±9m^3/h. The average supply air temperature was 19°C with fluctuation of ±0.5°C

To detect the air temperature and velocity distribution in micro-environment around human simulator and other part of environment chamber, 5 vertical poles with 24 probes were placed in the chamber to measure the air velocity and temperature. For each experiment, these 24 probes sampled data every 30 seconds in 30 minutes. To detect the contaminant concentration, 25 sample collection pipes were placed on the 5 poles, and 2 sample collection pipes were placed at the air supply inlet and exhaust diffusers. A sample from each pipe was collected and analyzed once every 90 minutes. It took around 24 hours to establish a stable temperature and contaminant concentration field. Detailed distribution of the measurement poles and probes for PDV is given in Figure 4.

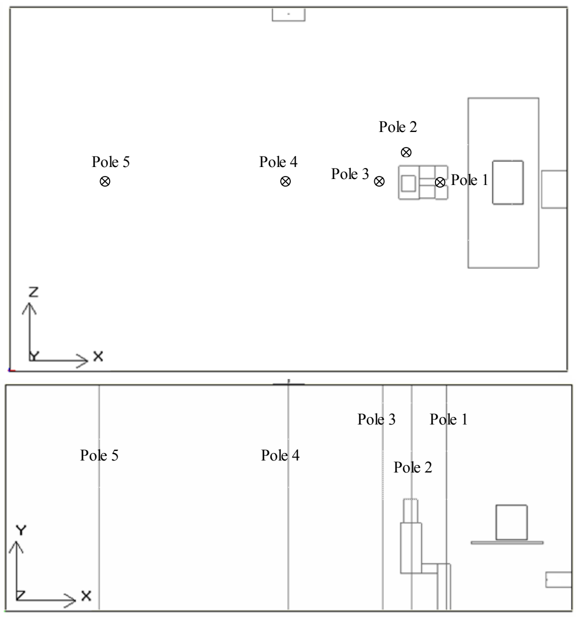


Figure 4 The arrangement of the measurement poles

SF_6 (0.1% SF_6 and 99.9% N_2) was used as the tracer gas, which was released at a certain location through injection pipe as a point contaminant source. The mass flow rate of SF_6 injection was 8.03ml/min during the experiments. The transport of the contaminant is investigated using a localized tracer injection and sampling for the presence of the tracer at various locations. SF_6 was used as the tracer gas since it is non-reactive, non-toxic, odorless, colorless, and it is detectable in small concentrations by a recognized measurement technique. The tracer gas system consists of following components: instantaneous tracer gas injection system, continuous tracer gas injection system, and sample collection and analysis system. During the experiments, SF_6 was released continuously into a certain location from a plastic pipe. Air samples were collected through pipes placed along the poles which also carried velocity and tempertaure probes. Samples were also collected at air supply and exhaust diffusers.

CFD MODELING

Selection of turbulence model

To simulate indoor airflow and contaminant dispersion, a proper turbulent model needs to be selected among many available models. Chen (1995) compared eight modified k-ε models and concluded that the Renormalisation Group (RNG) k-ε model (Yokhot et al. 1992) performs best among all the eddy-viscosity models tested for mixed convection flow. Yuan et al. (1999) used this model to successfully predict indoor contaminant distribution in a displacement ventilated room. In this study, RNG k-ε model was employed to simulate the PDV cases and a commercial CFD software (Fluent 2005) was used to solve the basic conservation equations.

Boundary conditions

The air supply diffuser was defined as velocity inlet boundary. The velocity input value was obtained from volume flow rate and the area of the supply diffuser. No recirculation air was used, so the air supplied into the room can be treated as non-contaminated. The outflow boundary condition was imposed at the exhaust diffuser.

No penetration and non-slip conditions were imposed at all solid wall boundaries. Walls, human simulator, and computer were the main heat sources. In the simulation, the measured wall temperatures have been used as the thermal boundary conditions for the walls. Furthrmore, for the human simulator and the computer, the constant heat flux thermal boundary condition was imposed at the surface. The heat flux input values were based on the experimental data, and the area of the heat source surface.

Table 1 Measured wall surface temperatures (°C)

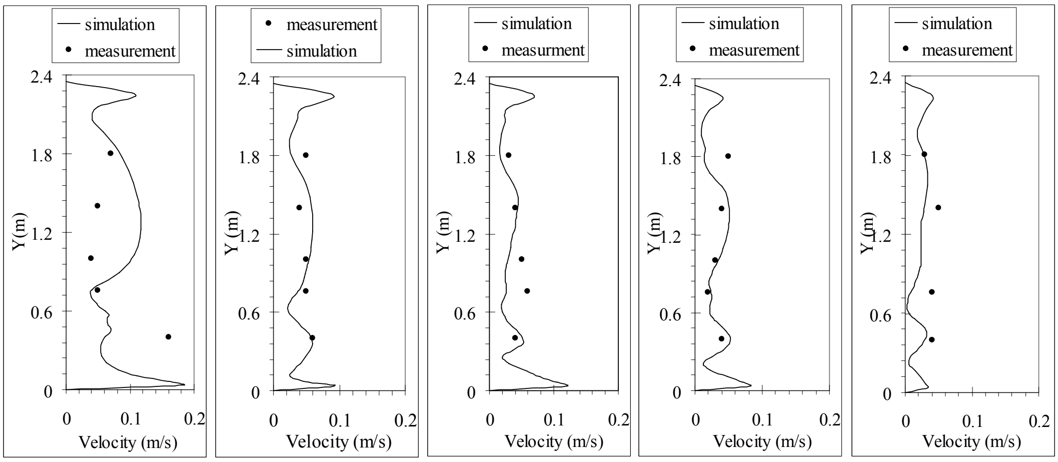
	WEST	EAST	NORTH	SOUTH	CEILING	FLOOR
T(°C)	24.5	25	24.8	24.8	24.9	23.9

RESULTS AND DISCUSSION

The source rate of the contaminant (SF_6) was 1.818×10^{-4} m^3/h, and compared to the air supply rate of $43m^3/h$, it is negligible. Therefore, the emission of the contaminant did not affecte the airflow field. Since the three PDV cases had the same setup except contaminant source location, they have the same airflow field.

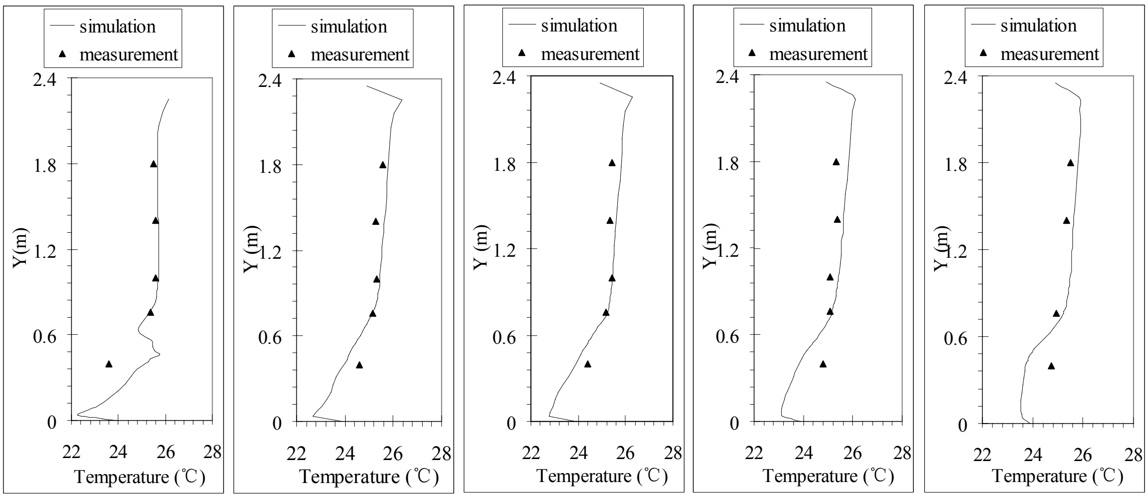
To validate the CFD model, the simulated results were compared with experimental data. Figures 5 and 6 present the velocity and temperature validation. Figures 7, 8, and 9 show the comparison of the measured and simulated contaminant concentrations for Case1, Case2 and Case 3, respectively.

As shown in Figure 5 and Figure 6, the simulated velocity and temperatue profiles match the measurements. The velocity in most of the space is lower than 0.05m/s, and the velocity measurement instrument may fail to give accurate results in such a low velocity range. For the temperature profiles, there are small temperature difference at the places below 0.6m. In this region, the model slightly over-predicts the temperature for Pole 1 and under-predicts the temperature for Poles 2 to 5. In the simulation, a constant flux was given to the surface of the human simulator, while in real experiment, the lamp to generate the heat was placed in the body of the human simulator and the heat flux of the leg surface may be lower than for the other places. Pole 1 was located between the two legs of the simulator. That may explain why the simulated tempertaure values are higher at Pole 1.



(a) Pole 1 (b) Pole 2 (c) Pole 3 (d) Pole 4 (e) Pole 5

Figure 5 Comparison of measured and simulated velocities at five vertical poles



(a) Pole 1 (b) Pole 2 (c) Pole 3 (d) Pole 4 (e) Pole 5

Figure 6 Comparison of measured and simulated temperatures at five vertical poles

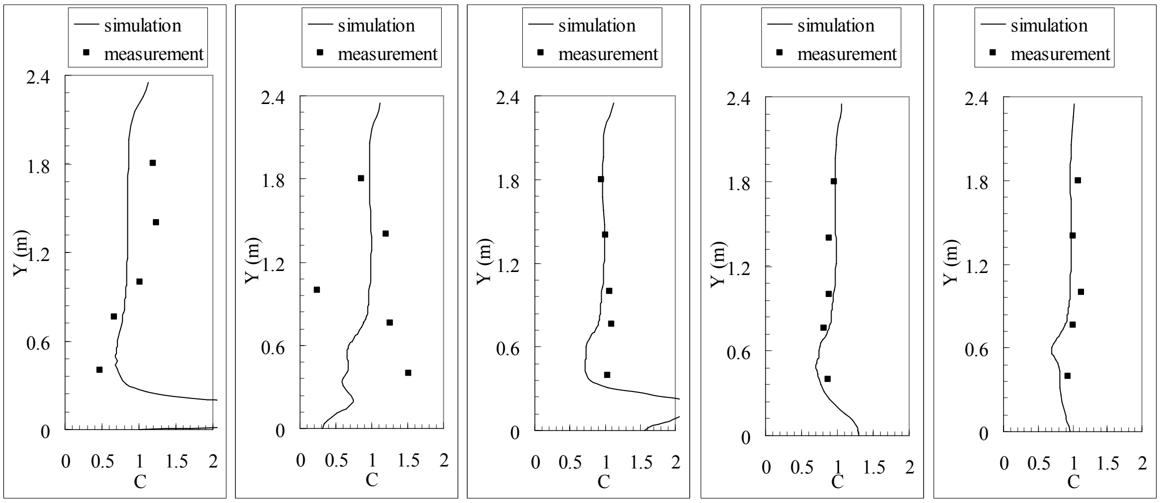
The contaminant concentration is normalized by the steady-state contaminant concentration at the exhaust:

$$C_N = \frac{C}{C_O}$$

Where C_o is the contaminant concentration at the exhaust (outlet), C_N is dimensionless concentration.

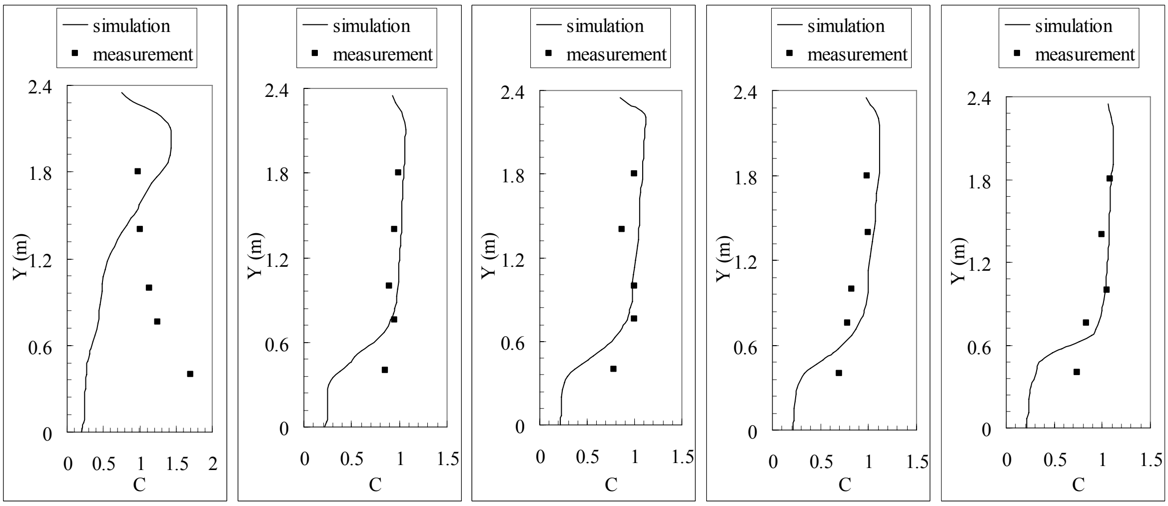
Figure 7 shows that the simulated contaminant concentrations agree very well with the measured data for Poles 3 to 5, while deviations are found for Poles 1 and 2. Pole 1 is located between the two legs of the human simulator, and the airflow pattern is very complicated. Pole 2 is near the contaminant source where both the measurement errors and simulation errors could be relatively large.

Figure 8 shows that the concentration profile and trends are well predicted for Poles 2 to 5, but not for Pole 1 due to similar reasons as in the previous case. Similar results also apply to Case 3 as shown in Figure 9.



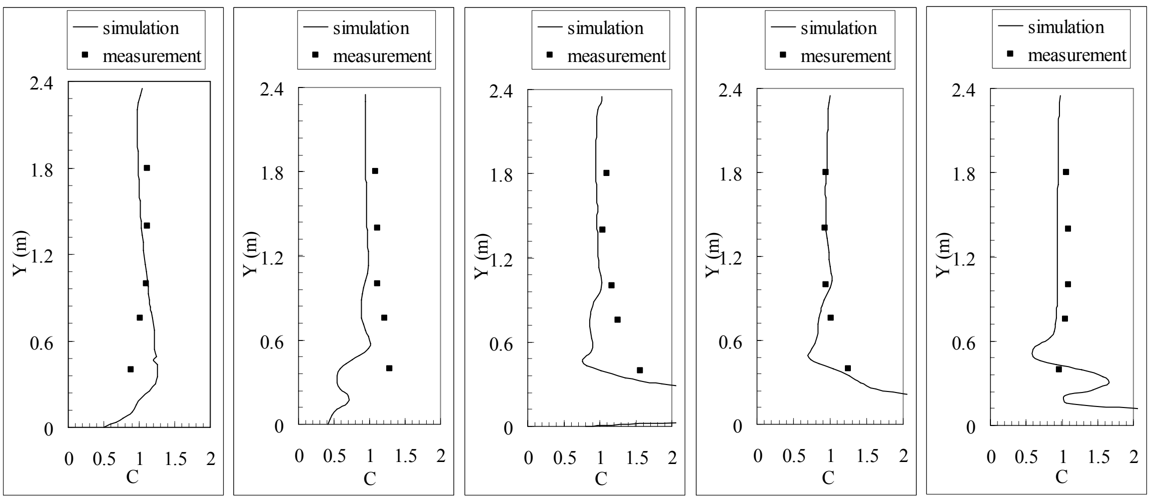
(a) Pole 1 (b) Pole 2 (c) Pole 3 (d) Pole 4 (e) Pole 5

Figure 7 Comparison of measured and simulated contaminant concentrations for Case 1 (source at front, low)



(a) Pole 1 (b) Pole 2 (c) Pole 3 (d) Pole 4 (e) Pole 5

Figure 8 Comparison of measured and simulated contaminant concentrations for Case 2 (source at front, high)



(a) Pole 1 (b) Pole 2 (c) Pole 3 (d) Pole 4 (e) Pole 5

Figure 9 Comparison of measured and simulated contaminant concentrations for Case 3 (source at back, low)

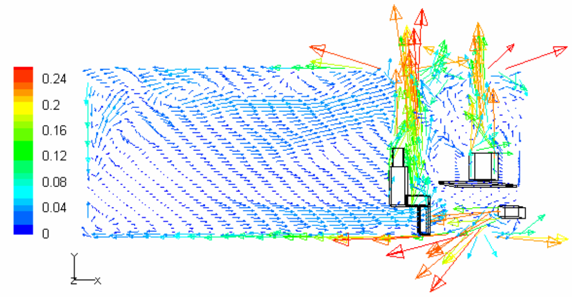
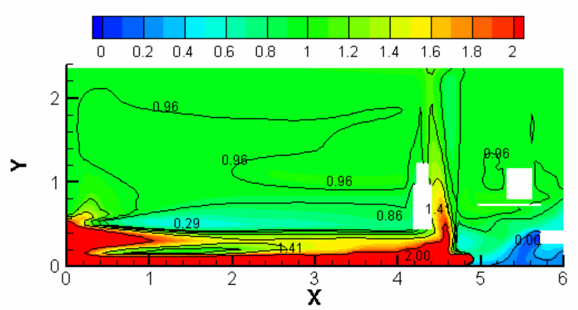


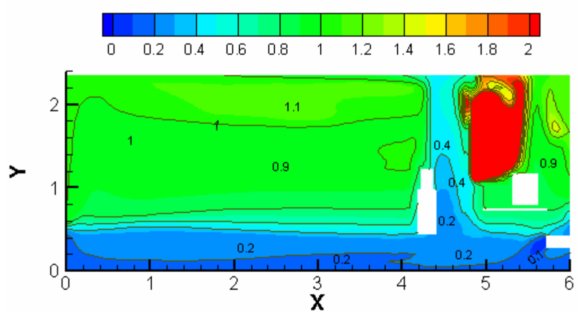
Figure 10 The velocity vector of the central plane

Figure 10 shows the simulated velocity vectors of the central plane, running through the middle of the occupant. The air from PDV diffuser first dropped to the floor due to lower supply temperature. When the air ran accross the human body (legs), part of the air moves upwards due to the buoyancy effect. However, the majority of airflow passed the occupant's legs to the back of the room. This may imply that the expected micro-environment around the human breathing zone may not be well established.

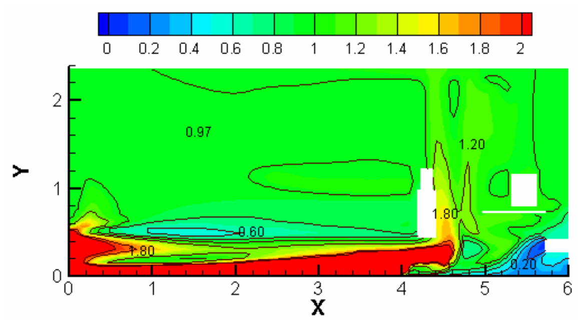
Figure 11 presents the contaminant distribution at the central plane for different contaminant source locations (Cases 1-3). In Figure 11 (a), the air from the supply diffuser carries the contaminant to the other places of the room and also the occupant's breathing zone. When contaminant source is located in front of the human simulator's mouth (Figure 11 b), the air near the floor and around the human simulator is clean because the clean air from the supply diffuser did not sweep through the pollutant source at the lower level. The airflow moves from the lower to the higher room zone and does not come back to the lower zone. Therfore, if the contaminant source is located in the higher zone, its effect on the cleanness of the space is confined to the upper zone. Figure 11 (c) shows the contaminant distribution when contaminant source is located at the back of the human simulator's feet. The polluted air moves back after hitting the wall and returns to the human simulator's zone. As a result, the air around the human simulator is also polluted even when the contamiant source is behind the occpant.



(a) Case 1 (source at front, low)



(b) Case 2 (source at front, high)



(c) Case 3 (source at back, low)

Figure 11 The concentration distribution in central plane for different contaminat source locations

From the analysis above, when the contaminant source is located at the upper room zone, the lower room zone will be clean and the contaminant will disperse only in the upper zone. When the contaminant source is located at the lower zone, the contaminant will disperse all over the room regardless of the contaminant source location. Hence, it is not clear whether this PDV system creates a micro-environment. Contrary to the original expectation, this basic PDV system does not have strong enough buoyancy to draw most of the supply air up to the breathing zone. Instead, most of the clean air from PDV diffuser just "slips through" the person's legs and mixes with the air in other part of the room.

CONCLUSION

The contaminant dispersion in a PDV system with a different contaminant source location was studied using both experiments and CFD simulations. The simulated results of the temperature, velocity and contaminant concentration were compared to the experimental data.

It seems that this kind of PDV system cannot create a clean "micro-environment" as expected. Further studies are needed to examine the conditions where PDV could perform better.

ACKNOWLEDGMENT

This project was financially supported by the Specialized Research Fund for the Doctoral Program of Higher Education of China, Approval Number 20060003065.

REFERENCES

Chen Q. 1995. “Comparison of different k-ε models for indoor air flow computations,” Numerical Heat Transfer. 28 (B): 353-369.

FLUENT Inc: FLUENT Version 6.2, 2005, Fluent Inc., Lebanon, New Hampshire, USA.

Gao NP and Niu JL. 2004. “CFD study on micro-environment around human body and personalized ventilation,” Build and Environment. 39: 795-805.

Kaczmarczyk J, Melikov AK and Fanger PO. 1999. “Human response to personalized ventilation and mixing ventilation,” Indoor Air. 14: 17-29.

Loomans M. 1998. “The Measurement and Simulation of Indoor Air Flow,” Ph.D. Thesis, University of Eindhoven (The Netherlands).

Yokhot V, Orzag SA, Thangam S, Gatski TB and Speziale CG. 1992. “Development of turbulence models for shear flows by a double expansion technique,” Physics Fluids A. 4: 1510-1520.

Yuan X, Chen Q, Glicksman LR, Hu Y and Yang X. 1999. “Measurements and computations of room airflow with displacement ventilation,” ASHRAE Transactions. 105: 340-352.