LBNL-55561

Investigation of Room Ventilation for Improved Operation of a Downdraft Table

B. Jayaraman[1], A. Kristoffersen[1,2],
E. Finlayson[1], A. Gadgil[1]

[1]Indoor Environment Department
Lawrence Berkeley National Laboratory
Berkeley, CA 94720, USA.

[2]Norwegian Building Research Institute
Forskningsvn.3b, 0314 OSLO, NORWAY.

May 2004

This work was supported by Lawrence Livermore National Laboratory and performed for the U.S. Department of Energy under Contract No. DE-AC03-76SF00098; the Norwegian Research Council through the Strategic Institute Program, Environmentally Favourable Energy Use in Buildings, at the Norwegian Building Research Institute, Project No. 133692/420; and U.S.- Norway Fulbright Foundation for Educational Exchange.

Investigation of Room Ventilation for Improved Operation of a Downdraft Table

B. Jayaraman[1], A. Kristoffersen[1,2], E. Finlayson[1], A. Gadgil[1]

[1]Indoor Environment Department, Lawrence Berkeley National Laboratory, Berkeley, CA 94720, USA.
Email: EUFinlayson@lbl.gov http://eetd.lbl.gov/IED/APT/APT.html
[2]Norwegian Building Research Institute, Forskningsvn.3b, 0314 OSLO, NORWAY.

Summary: *We report a computational fluid dynamics (CFD) study on containment of airborne hazardous materials in a ventilated room containing a downdraft table. Specifically, we investigate the containment of hazardous airborne material obtainable under a range of ventilation configurations. The desirable ventilation configuration should ensure excellent containment of the hazardous material released from the workspace above the downdraft table. However, increased airflow raises operation costs, so the airflow should be as low as feasible without compromising containment. The airflow is modeled using Reynolds Averaged Navier Stokes equations with a high Reynolds number k-epsilon turbulence model. CFD predictions are examined for several ventilation configurations. Based on this study, we find that substantial improvements in containment are possible concurrent with a significant reduction in airflow, compared to the existing design of ventilation configuration.*

Keywords: *CFD modeling, downdraft table, ventilation, contamination control.*
Category: *Industrial and agricultural ventilation.*

Introduction

Downdraft tables are used to handle hazardous materials that can become airborne. The ventilation configuration in the room containing the downdraft table affects the downdraft table performance, however we could find no guidelines in the literature for room ventilation design to ensure good performance of a downdraft table. Furthermore, there are few studies regarding downdraft tables in a ventilated room. Although both numerical and experimental work has been published investigating the performance of fume hoods (e.g. ref [1]), fume hoods are unsuitable where manipulating operations are required. The present study investigates possible modifications to the ventilation system of a downdraft facility using computational fluid dynamics (CFD). Previous work on this facility can be found in [2].

The facility, shown schematically in Figure. 1, consists of two rooms connected by a doorway. In the first room, the change room, the worker puts on protective clothing. This room provides an entrance to the second room, the downdraft room (2.3m x 2.0m x 2.5m high), which contains the downdraft table located against the wall opposite the door. This wall has in it a pass-through window directly above the downdraft table. After the contaminated packages are passed into the room through this window, the window is closed. As the facility is currently operated, air enters the room through a vertical slot in the door behind the worker, and from a rectangular inlet in the ceiling above the table. All airflow exits through the downdraft table. The rooms, constructed in the 1960s, currently have a ventilation configuration that supplies a total of 1700 l/s (3600 CFM) air to the room. Of this 1230 l/s (2600 CFM) are supplied through an opening in the door connecting the two rooms and 470 l/s (1000 CFM) from the ceiling towards the table. The vertical slot in the door has dimension 0.81 m x 0.46 m (32" x 18"), with its lower edge located at 10 cm (4 inches) from the floor level. This opening in the door is referred to as the door slot inlet.

Any assessment of improving containment and reducing airflow must be undertaken with simulations, because the facility remains contaminated from previous use and inaccessible for detailed experiments.

Method

The change room is excluded from the CFD computational domain. When the door is closed the details of the airflow within the change room are assumed to have no effect on the airflow in the downdraft room. Air flows through the door slot inlet into the downdraft room, normal to the doorway, and is treated as a boundary condition. The airflow from the inlet in the ceiling is assumed to be straight down, and is also a boundary condition.

Contamination is investigated for a given velocity field by first checking the predicted flow paths of massless particles, and then, in more detail, checking the predicted concentration of tracer gas. The massless particles and the tracer gas are both released from the package and the two additional locations where "worst-case" containment is expected: from the rim of the downdraft table and the perimeter of the (closed) window in the wall behind the downdraft table. See figure 2. Tracks of airborne massless particles give a good indication of whether contamination will be contained with respect to the mean flow. This is a reasonable minimum criterion to assure containment. Particle tracks can only show the effects of the mean velocity. The additional mixing in the room from turbulent fluctuations can cause reduced containment and increased contamination. To evaluate the additional diffusion caused by turbulence, we simulate a release of a neutrally-buoyant tracer gas (as a passive scalar) at the rim of the downdraft tabletop and the perimeter of the pass through window. Tracer gas has very high diffusivity, much larger than that of particulate contaminants, and represents a maximum criterion for containment.

We considered several modifications to the existing room operation and geometry, by changing the area and airflow from the inlets.

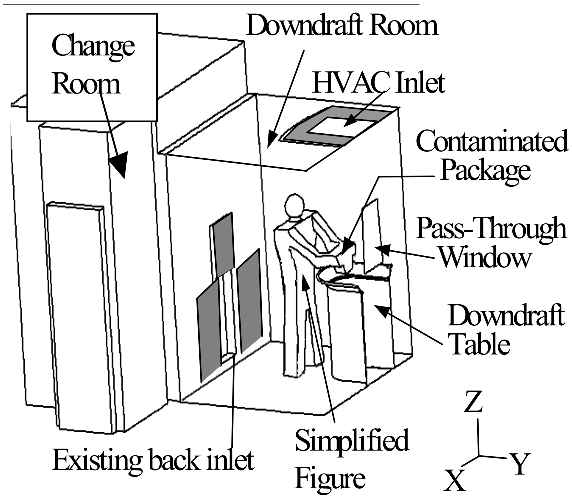

Fig. 1. Facility geometry including the change room, to the left, and the downdraft room to the right (2.3m x 2.0m x 2.5m). The geometry includes the downdraft table, simplified worker figure in a protective suit holding the contaminated package, door inlets, pass-through window, and an inlet in the ceiling.

A simplified model of a worker is included to simulate the effect of flow blockage by the worker. The worker is assumed to be in a protective suit and thermal plume of the worker is neglected (owing to larger than an order of magnitude difference between the plume velocity in still air, and the downdraft room airspeeds). The worker is holding an object representing a contaminated package. The package is held above the downdraft table surface and away from the edge in order to represent standard working conditions.

The geometry of the room was obtained from plans produced using Pro-E software [3]. These files were converted into a hexahedral finite volume mesh for the computations.

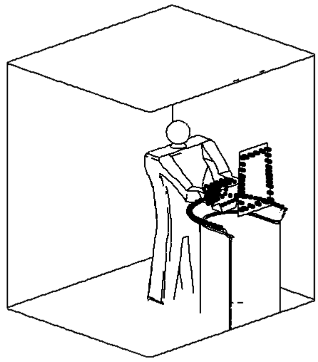

Fig. 2: Locations of release of massless particles and tracer gas used to determine containment are shown as black spots.

The room containing the downdraft table is modeled using the Reynolds Averaged Navier Stokes equations (RANS) with a high Reynolds number k-epsilon turbulence model. The flow is considered incompressible and isothermal with constant air properties. A finite volume formulation of the standard Navier Stokes equations with the above turbulence model is solved using the commercial software Star-CD [4].

Equations for the airflow are solved to obtain a steady state velocity field in the downdraft room using the SIMPLE method [5]. The spatial differencing employs a second order scheme based on a Godunov method modified for incompressible flow (MARS) [6]. The advantage of this scheme is that it suppresses numerical diffusion without causing instability. The convergence criterion used for the velocity field is that the normalized sum of the residuals is less than 1.0e-4 for each of the variables.

A Large Eddy Simulation (LES) would have increased the computational and human effort for analysis by a factor of more than 10, well beyond the resources available for this work. On the other hand, our research [7] shows that a RANS model with a second order differencing scheme that suppresses numerical diffusion can provide acceptable (i.e., within a factor of two compared to experimental measurements) detailed predictions for pollutant dispersion. Justification for our current approach rests on the successful CFD comparison to experiment reported in [7].

After solving the airflow in the room, the solution of the tracer gas transport is solved using the MARS scheme[6]. The convergence criterion used for this calculation is that the normalized sum of the residuals is less than 5.0e-5. Examination of the simulated tracer gas concentration throughout the downdraft room gives an indication of how contamination could be spread due to turbulent diffusion.

Figures 3 and 4 show the section planes that will be used to display the results.

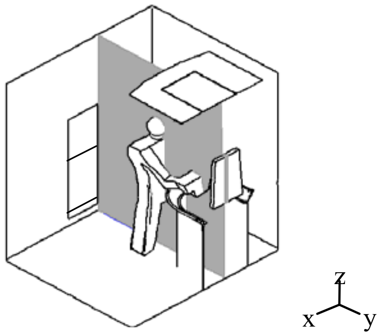

Figure 3: The Y-Z section used for displaying results. The plane passes through the middle of the figure showing the asymmetry of the downdraft table within the room. Center plane of downdraft table is 33"(0.84m) from the left wall and 56"(1.42m) from the right wall. This section illustrates the containment in front of the worker, and around the package.

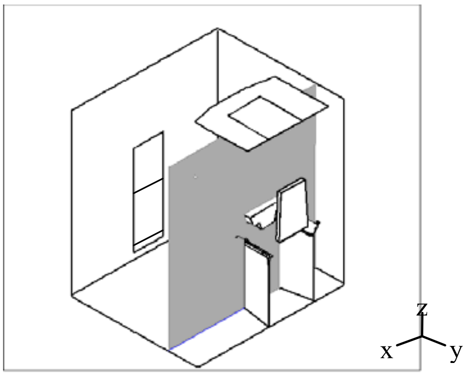

Figure 4. The X-Z section used for displaying results. The plane cuts through the middle of the contaminated package. This view shows how the contamination is spread into the room on either side of the table.

We now introduce a measure of the contamination spill, S, in the room. The spill is calculated throughout the computational domain excluding the cells in the downdraft table, and the cells where the tracer gas is released. For a release of 1kg/s tracer into the domain the spill is defined as the integral of the tracer-concentration-weighted cell volumes divided by the same volume according to

$$S = \frac{[\int MdV]}{[\int dV]} \cdot 10^4 .$$

A fully mixed room will give $S=10^4$ for a fresh air supply of 1 kg/s. If the contaminants are fully contained into the downdraft table S=0, and we have no spill. The spill value is multiplied by 10^4 to investigate more clearly what happens when the spill is close to zero. The spill factor gives the average concentration of the tracer in the room if tracer is injected at a rate of 1 kg/s.

We use about 670,000 nodes in the simulation. Good numerical predictions require a grid sufficiently refined that the CFD solution does not depend on the grid used. The grid was successively refined at locations believed to affect the airflow calculation until less than 10% changes were recorded in the overall mean velocities, max, min velocities and pressure.

Computational results for several ventilation configurations are presented below.

Discussion of existing configuration

As a first step we explored the containment capability of the existing configuration. Figure 5 shows the particles fully contained using the current configuration with airflow of 470 l/s (1000 CFM) downward and 1230 l/s (2600 CFM) through the door.

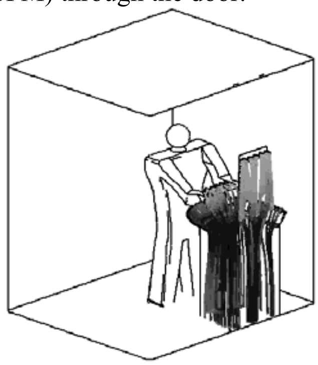

Figure 5. Particle tracks showing that all particles released are contained within the downdraft table for flow supplied through the back inlet at 1230 l/s (2600 CFM) and the ceiling at 470 l/s (1000 CFM).

Figure 6 shows the tracer concentration contours for the existing configuration. The concentration contours are presented in the log scale. In the y-z view, the tracer leaves the downdraft table rim, touches the worker's chest and spreads into the room. In the x-z view, the tracer is spread on both sides of the table. The airflow pattern is shown in figure 7. A recirculating airflow pattern can be seen on the left hand side of the table in the x-z view. The contamination spill for the existing configuration was found to be S=4.22. If tracer of 1 kg/s were injected into the room, the average concentration of the tracer in the room would be 0.422 grams of tracer per kg air. This can be unacceptably high for hazardous materials.

Alternate configurations

We explored several modifications to the existing room operation and geometry. The description of geometry changes is shown in Table 1.

Inlet type	**Description**	**Area** m^2 (in^2)
Ceiling inlet	Existing	0.32 (500)
	Large	1.07 (1659)
	Perforated	4.42 (6843)
Door inlet	Existing	0.37 (576)
	Large	0.91 (1405)

Table 1. Definitions of inlet geometries

The existing ceiling inlet is shown in Figure 1 as the nonshaded inlet in the ceiling. The perforated ceiling inlet assumes that the entire ceiling is a perforated plate, and acts as a supply inlet. The perforated ceiling inlet is modeled with a perforated full-floor exhaust. The large ceiling inlet includes the shaded area in the ceiling in figure 1. The existing door inlet is shown as the nonshaded inlet behind the worker. The large door inlet includes the shaded area on the left and right side of the existing inlet. If one draws a vertical plane similar to the one identified on figure 3, the left edge of the large door inlet behind the worker will be coplanar with the left edge of the large ceiling inlet. The width of the existing door inlet is 46 cm (18 inches), and the large door inlet is 73 cm (28.5 inch) wide.

Table 2 lists the different geometrical configurations simulated. Configuration A is the existing configuration with air supply through the door and through the ceiling inlet. Configuration B has the large ceiling inlet and existing door inlet. Configuration C has large ceiling and large door inlets. Configuration D has the whole ceiling as a perforated inlet and the whole floor as a perforated exhaust. This configuration has no door inlet. Part of the flow exits through the perforated floor, and the rest exits through the downdraft table.

Config-urations	Ceiling inlet	Door inlet	Floor outlet
A	Existing	Existing	None
B	Large	Existing	None
C	Large	Large	None
D	Perforated	None	Perforated

Table 2: Summary of different geometry configurations investigated.

Previous work on this facility [2] shows that decreasing the air flow with the same geometry as the existing set up, will reduce the contamination of the room. Figure 8 shows the contamination when airflow is reduced for configuration A. Ceiling inlet flow is reduced from 470 l/s (1000 CFM) to 380 l/s (800 CFM), and the door flow is reduced from 1230 l/s (2600 CFM) to 755 l/s (1600 CFM). By reducing the flow in the room the effecective turbulent mixing is reduced.

We examined various flow rates through different inlets for each of the above configurations. 10 cases for configuration A, 19 for configuration B, 12 for configuration C and 3 for configuration D. We report findings in the next section.

Results and discussions

Simulations with air supply only from the large ceiling inlet were reported in [2]. The results presented show that the massless particles released from the rim were not contained by the table thus failing our minimum criterion for containment. Flow from the ceiling inlet bypasses the rim of the downdraft table circulating through the room before entering the downdraft table. This flow pattern is a potential source of contaminant spread in the room. We have observed that the flow from the door slot rises in the space between the worker and the downdraft table. Bypass of the rim by the flow is minimized when the strength of the flow from the door slot and the ceiling inlet is balanced so that the vertical component of the mean velocity is zero at the level of the downdraft table rim. The upper edge of the door flow must be at the level of the downdraft table to force air towards the table at the level of the table rim. If additional airflow is added horizontally at the level of the downdraft table improved containment is observed.

Configuration B is simulated with a large ceiling inlet, and the existing door slot. The best solution with this configuration improved containment of the tracer compared to configuration A. The big ceiling inlet decreased the extent of the recirculating airflow on the left side of the downdraft table.

The next configuration, C, has a large ceiling inlet and a large door inlet. The wider door provided a better containment of the tracer than configuration B by suppressing the recirculating flow in the room. Figure 9 shows tracer gas concentrations contours at airflow of 470 l/s [1000 CFM] from the ceiling and 470 l/s [1000 CFM] from the door slot .

Configuration D with perforated floor and perforated ceiling did not improve the containment compared to configuration C. In this configuration we defined boundary conditions such that 30 % of the airflow from the perforated ceiling exits through the perforated floor, and the remaining 70 % exits through the downdraft table. Some of the airflow from the ceiling behind the worker exits through the downdraft table rather than the floor, creating an upward flow in the space between the worker and the table. The rising flow meets the downward airflow from the ceiling causing a recirculating loop in front of the worker where tracer accumulates, degrading the containment.

The best run of each configuration is listed in Table 3.

Config-uration	Ceiling inlet l/s (CFM)	Door inlet, l/s (CFM)	Spill measure
Existing	470 (1000)	760 (1600)	4.22
A	380 (800)	760 (1600)	3.20
B	470 (1000)	380 (800)	2.36
C	470 (1000)	470 (1000)	1.95
D	800 (1700)	None	2.40

Table 3: Spill measure for the existing configuration and the best cases with the lowest spill measure for each of the configurations.

Conclusion

Our CFD simulations suggest that substantial (54%) reduction in spill and concurrent significant (23%) reduction in airflow (and therefore operating costs) can be achieved in the downdraft facility under investigation by modifying the geometry and the ventilation configuration.

Acknowledgement

This work was supported by: Lawrence Livermore National Laboratory and performed for the U.S. Department of Energy under Contract No. DE-AC03-76SF00098; the Norwegian Research Council through the Strategic Institute Program "Environmentally Favourable Energy Use in Buildings" at the Norwegian Building Research Institute, Project No. 133692/420; and U.S.-Norway Fulbright Foundation for Educational Exchange.

References

[1] N. S. Lan and S. Viswanathan, Numerical simulation of airflow around a variable volume/constant face velocity fume cupboard. *Amer. Industrial Hygiene Assoc.* 62(3) (2001) pp. 303-31.

[2] E. Finlayson, B. Jayaraman, A. Kristoffersen and A. Gadgil, CFD analysis of LLNL Downdraft Table. *LBNL Report 53883.* (2003).

[3] PTC, 140 Kendrick Street, Needham 02494 MA, USA (www.ptc.com).

[4] Star-CD Version 3.15 Methodology, *Adapco*, 2001.

[5] S. V. Patankar, Numerical heat Transfer and Fluid Flow, *Hemisphere, Washington D.C.,* 1980.

[6] P.N. Asproulis, High Resolution Numerical Predictions of Hypersonic Flows on Unstructured Meshes, *PhD. Dissertation, Imperial College, Dept of Aeronautics,* London, England (1994).

[7] E. U. Finlayson, A. J. Gadgil, T. L. Thatcher and R. G. Sextro, Pollutant Dispersion in A Large Indoor Space: Computational Fluid Dynamics (CFD) Predictions and Comparison with A Scale Model Experiment for Isothermal Flow, *Lawrence Berkeley National Laboratory Report LBNL-50105,* April 2003. (Accepted for publication in Indoor Air

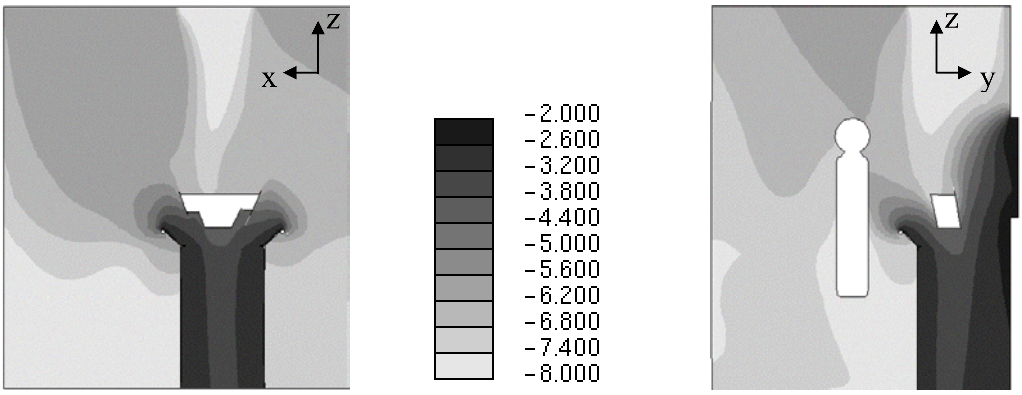

Figure 6: Tracer gas concentration (log of concentration) of existing geometry with currently operated airflow rates: 470 l/s (1000 CFM) from the ceiling inlet and 1230 l/s (2600 CFM) from the door slot, when releasing 1.98e-3 kg/s of tracer into the room.

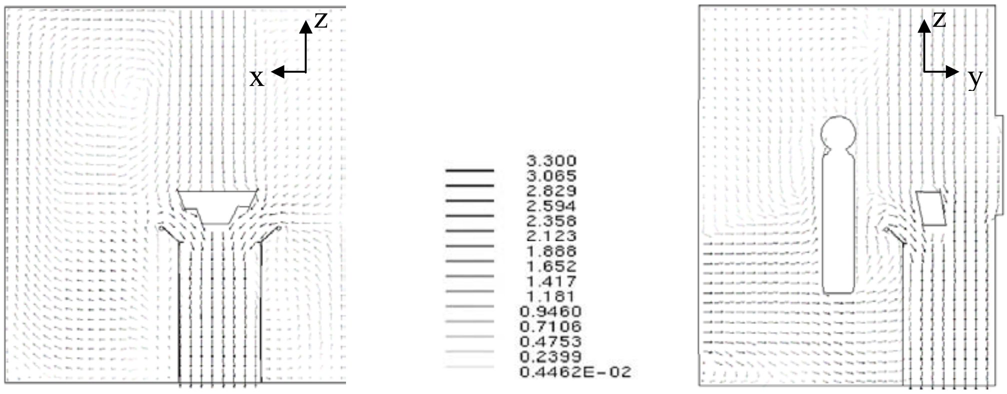

Figure 7: Velocity of existing geometry with currently operated airflow rates: 470 l/s (1000 CFM) from the ceiling inlet and 1230 l/s (2600 CFM) from the door slot.

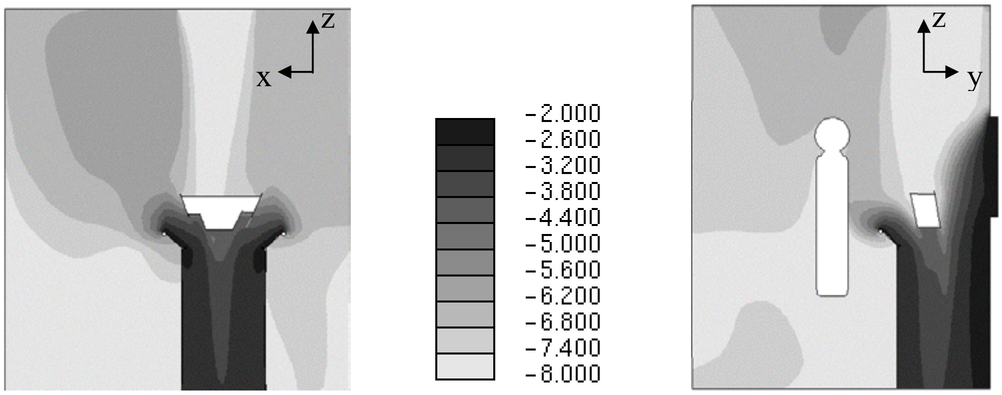

Figure 8. Tracer gas concentration of existing geometry, configuration A, with reduced airflow: 380 l/s (800 CFM) from the ceiling inlet, and 755 l/s (1600 CFM) from the door slot, when releasing 1.98e-3 kg/s of tracer into the room.

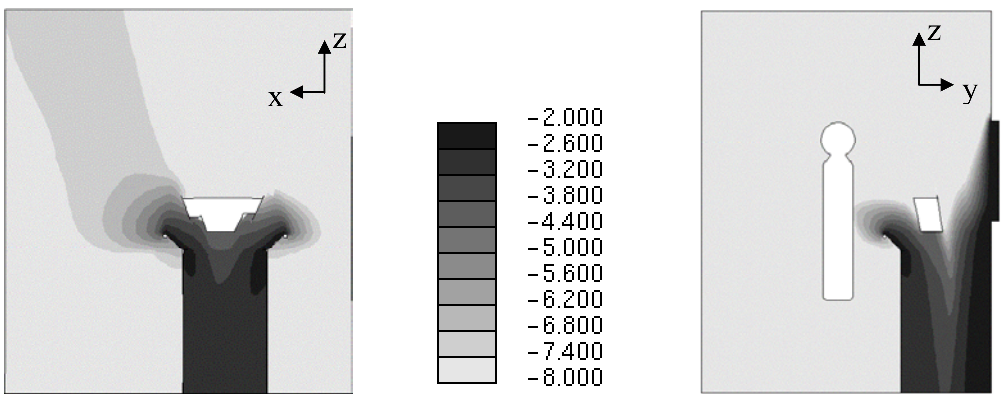

Figure 9. Tracer gas concentration of configuration C geometry: 470 l/s (1000 CFM) from the ceiling inlet, and 470 l/s (1000 CFM) from the door slot, when releasing 1.98e-3 kg/s of tracer into the room.