Numerical Investigation of Turbulent Diffusion in Push–Pull and Exhaust Fume Cupboards

MING-JYH CHERN* and WEI-YING CHENG

Department of Mechanical Engineering, National Taiwan University of Science and Technology, 43 Sec. 4 Keelung Road, Taipei 10607, Taiwan

Received 5 August 2006; in final form 27 March 2007; published online 11 July 2007

The aim of this study is to investigate airflow motions and associated pollutant distributions in fume hoods. Currently, most exhaust fume hoods are designed to use an airflow induced by a fan at the top to remove pollutants. Ambient fluids are drawn, flowing toward the opening and subsequently turning to the outlet at the roof. Pollutants are supposedly captured by the airflow and brought out from the cupboard. The present numerical study based on the finite-volume method and the standard $k$–$\varepsilon$ turbulence model simulates flow patterns and pollutant distributions in an exhaust fume hood with and without a manikin present. Subsequently, a push–pull air curtain technique is applied to a fume cupboard. To investigate the capturing performance of a push–pull fume cupboard, numerical approaches are used to simulate flow and concentration variations. Numerical results reveal that four characteristic flow modes exist for a variety of speed ratios of push–pull flows and openings. A concave curtain mode which has a fast pull flow and a weak push flow is suggested for the operation of a push–pull fume cupboard. According to ANSI-ASHRAE Standard 110-1995, the local concentration at the specified point is $<0.1$ parts per million (p.p.m.). Meanwhile, we also examine concentration variations at 12 selected points in front of the sash, and all where the concentration is $<0.1$ p.p.m. A manikin is put in front of the sash to observe its effect. As a result, the flow and the concentration contours in a push–pull fume cupboard are not affected by a manikin. In terms of those predicted results, it turns out that a push–pull fume cupboard successfully captures pollutants and prevents an operator from breathing pollutants.

Keywords: air curtain; fume hood; local ventilation; push–pull; turbulent diffusion

INTRODUCTION

A fume hood is a common device used in a laboratory. The main purpose of a fume hood is to enclose toxic containments or vapors and then to remove such material to the outdoor environment using an airflow. As a result, one can safely perform experiments which may generate toxic products in a fume hood. A conventional way to induce an airflow in order to remove toxic containments is to install a fan on the roof. A design example for an exhaust fume hood can be found in ACGIH (2004). This way to induce an airflow using a top fan has been employed for many years. However, whether an exhaust fume hood is able to evacuate all pollutants to the environment is still an important issue in occupational safety. Essentially, a fume hood should be able to enclose pollutants in a fixed space and then remove all enclosed pollutants from the working environment using the minimum suction speed and prevent an operator from breathing toxic pollutants.

Several researchers have performed experiments on exhaust fume hoods to investigate the capability to capture and extract toxic pollutants, e.g. Fletcher and Johnson (1992a,b). Ivary et al. (1989) utilized SF$_6$ to trace leakage of pollutants in a fume hood. Durst and Pereira (1991) investigated the variation of concentration of SF$_6$ in an exhaust fume cupboard from a start to an operation state using experimental and numerical approaches. Ekberg and Melin (1991) used N$_2$O as a tracer and studied the effect of the sash motion on the variation of velocity and leakage of gas in front of an exhaust fume cupboard. Saunders et al. (1994) measured the wind speed in the vicinity of the sash and its handle of an exhaust fume hood. Ozdemir et al. (1993) investigated variations of
velocity and concentration inside an exhaust fume hood and in the vicinity of its sash. Tseng et al. (2006) employed a flow-visualization technique to illustrate flow patterns inside an exhaust fume hood and also measured the concentration profile of SF₆ inside an exhaust fume cupboard.

Due to fume cupboard experimental limitations with regard to measurements of instant distribution of concentration, numerical simulations are being introduced to study flows in an exhaust fume cupboard. For example, Durst and Pereira (1991) and Ekberg and Melin (1991) utilized numerical approaches to explore flows, in an exhaust fume cupboard. Hu et al. (1993a,b) simulated two-dimensional (2-D) and 3-D turbulent fluid flows in an exhaust fume cupboard and studied the variation of concentration fields. Subsequently, Hu et al. (1996) studied effects of various locations of the top fan and the handle and the effects of obstacles in front of the sash on flow patterns using a 2-D numerical model. To understand the effects of baffles and a louvered bypass inside an exhaust fume hood on flow patterns and contaminants, Hu et al. (1998) employed a 2-D model and found that the vortex behind the sash becomes smaller than in the case without baffles inside an exhaust fume hood. Kirkpatrick and Reither (1998) considered the effect of a manikin and various openings of a fume hood on the capturing capability for pollutants. He found that leakage situations were very serious in a fully open fume hood. Graham et al. (2000) investigated several factors including the vortex behind a sash, the doorsill, baffles and a manikin using a numerical model. Lan and Viswanathan (2001) established 2-D and 3-D finite-volume models to explore flow and concentration fields affected by a manikin in an exhaust cupboard. They clearly demonstrated flow fields and concentration profiles in various vertical planes and found the main vortex behind a sash.

In order to improve the flow structure and enhance the performance of a fume cupboard, a push–pull air curtain can be applied to a fume cupboard. Push– pull air curtain technology has been widely applied to the capture of chemical pollutants evaporating from large open-surface tanks. The application of a push–pull air curtain to an open tank dates back to the 1950s (see, e.g. Malin, 1945; Battista, 1947; Ege and Silverman, 1950; Hama, 1957). The cited studies suggested the required minimum push and pull flow speeds and nozzle sizes. Subsequently, the National Institute for Occupational Safety and Health in United States carried out a series of studies of push–pull systems during the 1980s. Huebener and Hughes (1985) suggested the required minimum flow speed for both inlet and outlet sides for different sizes of tank based on laboratory conditions. A later study by Klein (1986) confirmed that the results of Huebener and Hughes were valid, reliable and applicable to industrial processes. Since the 1990s, Computational Fluid Dynamics has been used to predict the flow field in push–pull geometries. For example, Robinson and Ingham (1995a,b) simulated push–pull systems using numerical approaches and discovered that the optimal push and pull flow rates depend on the push nozzle height and the tank length. Robinson and Ingham also found that the most important parameter in determining the inlet flow conditions is the initial momentum of the push flow. Rota et al. (2001) recommended new design guidelines for pollutant tanks in an external crosswind. They investigated the influence of tank width, initial jet momentum and crosswind velocity. Detailed design guidelines and rules for cupboard designers have been written up in ACGIH (2004). This manual presents methods for the design and testing of industrial exhaust ventilation systems. Huang et al. (2005) employed a smoke flow visualization technique and Laser Doppler Velocimetry to demonstrate a variety of flow patterns in a push–pull ventilation system. Chern and Ma (2007) established an in-house numerical model based on the finite-volume method and a turbulence model to simulate flow fields in a push–pull ventilation system. They proposed suggestions regarding the operation of a push–pull system for a larger surface tank. The push–pull air curtain technique was further applied to a fume cupboard by Huang et al. (2007). A push flow and a pull flow come from the bottom of a sash and a slit behind a doorsill, respectively. The roof is open to the environment in a push–pull fume cupboard. Huang et al.
(2007) conducted flow-visualization experiments to observe flow fields inside a push–pull fume cupboard. The authors were inspired by Huang et al. (2007) and encouraged by them to perform numerical simulations for the assessment of a push–pull fume cupboard.

From the available literature regarding the performance of an exhaust fume cupboard, it seems that details of the various flow structures inside a fume cupboard and related concentration distributions are not very clear. The aim of this study is to use a 3-D numerical model to visualize the flow patterns and concentration fields inside and outside a fume cupboard. Turbulence diffusion in the airflow inside a fume hood is considered. Furthermore, a push–pull air curtain is applied to a fume hood. Figures 1 and 2 present the schematics of an exhaust fume hood and a push–pull fume hood at the central vertical plane. An SF₆ ejector is installed according to ANSI/ASHRAE Standard 110 (ASHRAE, 1995). In addition, a manikin, as shown in Fig. 3, is utilized to explore its effects on the airflow. ANSI/ASHRAE Standard 110 (ASHRAE, 1995) and a stricter standard are employed to judge whether an exhaust fume cupboard or a push–pull fume hood is able to achieve the goal of removing all pollutants to the outdoor environment.

**MATHEMATICAL FORMULAS AND NUMERICAL MODEL**

To study flows in a fume hood numerically, governing equations for fluid motions have to be established first. An incompressible fluid is considered in this study. All fluids simulated in the numerical model should obey the laws of conservation of mass and momentum. Under normal circumstances, the airflow inside a fume cupboard should be turbulent, so Reynolds
decomposition (see Versteeg and Malalasekera, 1995) is used to obtain a Reynolds averaged equation for the mass conservation, which is denoted as

\[
\frac{\partial U_j}{\partial x_j} = 0, \quad j = 1, 2, 3
\]

where \( U_j \) is a mean Cartesian velocity component \((\text{m s}^{-1})\). Furthermore, the Unsteady Reynolds Averaged Navier–Stokes (URANS) equations are presented as

\[
\rho \left[ \frac{\partial U_i}{\partial t} + \frac{\partial (U_i U_j)}{\partial x_j} \right] = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left( \mu \frac{\partial U_i}{\partial x_j} - \rho \overline{u_i u_j} \right), \quad i, j = 1, 2, 3, \quad \text{(2)}
\]

where \( P \) is the mean pressure \((\text{Pa})\), \( \rho \) is the density of the fluid \((\text{kg m}^{-3})\), \( t \) is time \((\text{s})\) and \( \mu \) is the dynamic viscosity of the fluid \((\text{kg m}^{-1}\text{s}^{-1})\). It is found that new variables, \(-\overline{u_i u_j}\), called the Reynolds stresses appear in equation (2) \( (\alpha' \text{ fluctuation of Cartesian velocity component, m s}^{-1}) \). We do not have sufficient equations to solve these unknowns. To close this problem, Boussinesq’s approximation is adopted whereby

\[
-\overline{u_i u_j} = \nu_t \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij}, \quad \text{(3)}
\]

where \( \nu_t \) and \( k \) are the turbulent kinematic viscosity \((\text{m}^2 \text{s}^{-1})\) and the turbulence kinetic energy \((\text{m}^2 \text{s}^{-2})\), respectively, and \( \delta_{ij} \) is the Kronecker delta. Moreover, the \( k-\varepsilon \) two-equation model proposed by Jones and Launder (1972) is used to determine the kinematic eddy viscosity \( \nu_t \) from

\[
\nu_t = C_{\mu} \frac{k^2}{\varepsilon}, \quad \text{(4)}
\]

where \( C_{\mu} \) is an empirical coefficient. The turbulence dissipation rate \( \varepsilon \) \((\text{m}^2 \text{s}^{-3})\) is denoted as

\[
\varepsilon = \nu_t \frac{\partial u_i u_j}{\partial x_j \partial x_i}, \quad \text{(5)}
\]
The \( k \) and \( \varepsilon \) equations can be written as:
\[
\begin{align*}
\frac{\partial k}{\partial t} + \frac{\partial k U_j}{\partial x_j} &= \frac{\partial}{\partial x_j} \left[ \left( v + \frac{v_i}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] \\
&+ \left[ \frac{v_i}{\sigma_k} \left( \frac{\partial U_j}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \right] \frac{\partial U_i}{\partial x_j} - \varepsilon, \quad (6)
\end{align*}
\]
and
\[
\begin{align*}
\frac{\partial \varepsilon}{\partial t} + \frac{\partial \varepsilon U_j}{\partial x_j} &= \frac{\partial}{\partial x_j} \left[ \left( v + \frac{v_i}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] \\
&+ C_{\varepsilon 1} \frac{\varepsilon}{k} \left[ \frac{v_i}{\sigma_\varepsilon} \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) - \frac{2}{3} k \delta_{ij} \right] \frac{\partial U_i}{\partial x_j} - C_{\varepsilon 2} \frac{\varepsilon^2}{k}, \quad (7)
\end{align*}
\]
where \( \sigma_k, C_{\varepsilon 1}, \sigma_\varepsilon \) and \( C_{\varepsilon 2} \) are empirical coefficients.
The following empirical values for those coefficients are used in the turbulence model: \( C_\mu = 0.09, \sigma_k = 1.44, \sigma_\varepsilon = 1.92, C_{\varepsilon 1} = 1.0 \) and \( C_{\varepsilon 2} = 1.3 \).

The turbulent mass-transfer equation based on Reynolds’ decomposition approach is used to explore concentration variations and is denoted as
\[
\begin{align*}
\frac{\partial C}{\partial t} + \frac{\partial U_j C}{\partial x_j} &= \frac{\partial}{\partial x_j} \left( D_m \frac{\partial C}{\partial x_j} - u_j \varepsilon \right), \quad j = 1 \sim 3, \quad (8)
\end{align*}
\]
where \( C \) and \( \varepsilon \) are the temporal mean and fluctuating value of concentration, respectively. The concentration fluctuation term \(-u_j \varepsilon\) is proportional to the concentration gradient, i.e.
\[
\frac{\partial C}{\partial x_j} \equiv D_t \frac{\partial C}{\partial x_j}, \quad (9)
\]
where \( D_t \) is the turbulent diffusion coefficient (m\(^2\) s\(^{-1}\)).

Since turbulent flows are complex and highly changeable, regarding \( D_t \) as a constant is usually unreliable for most applications. Therefore, we adopt the turbulent Schmidt number \((Sc)\) defined as:
\[
Sc = \frac{v_t}{D_t} = \frac{\text{turbulent momentum diffusion rate}}{\text{turbulent mass diffusion rate}}, \quad (10)
\]
where \( v_t \) is the turbulent kinetic energy.

The turbulent Schmidt number relates to the ratio of the viscous diffusion to mass diffusion and can be assumed as a constant. Depending on the situation...
under consideration, this value will vary from 0.8 to 1.2 (see Durst and Pereira, 1991; Çengel, 1998). We use $Sc = 1$ for the following studies.

The continuity equation, URANS equations and the turbulent mass-transfer equation are solved numerically for the present study. Air is the working fluid, with density and dynamic viscosity as $1.1842 \text{ kg m}^{-3}$ and $1.855 \times 10^{-5} \text{ kg m}^{-1} \text{s}^{-1}$ at $25^\circ C$, respectively. The physical domain is $3.73 \times 3.89 \times 3.16 \text{ m}^3$. Dimensions of exhaust and push–pull fume cupboards can be found in Figs 1 and 2. A rectangular Cartesian coordinate system is used for the solution and its origin is located at the center of the doorsill. Computational meshes consisting of 541 632 and 545 228 cells for the exhaust and push–pull fume cupboards are used. Figures 4 and 5 show the 2-D central planes of the meshes for the exhaust and push–pull fume cupboards, respectively. Cells are clustered densely inside the cupboard. Constant pressure 1 atm is imposed at the far field boundaries. A fixed upward speed of $0.05 \text{ m s}^{-1}$ is imposed at the outlet of the exhaust fume cupboard. Various speeds of push and pull flows are used for the boundary conditions of a push–pull fume cupboard. The push flow condition is imposed at the exit of the channel instead of the slot behind the doorsill.

The wall functions are used to determine boundary conditions at solid walls. Variations in velocity and concentration are predominantly normal to solid walls. The tangential velocity profile in terms of the normal distance $y$ from the wall is

$$u^+ = \frac{u}{(\tau_w/\rho)^{1/2}} = \begin{cases} y^+, & y^+ \leq y_m^+ \\ \frac{1}{\kappa} \ln(Ey^+), & y^+ > y_m^+ \end{cases}$$

(11)

where $u$ is the tangential fluid velocity, $\tau_w$ is the wall shear stress, $y^+$ is $\rho C_{m1/4} k^{1/2} y / \mu$ and $\kappa$ and $E$ are 0.4187 and 9.793. $y_m^+$ satisfies the following equation

$$y_m^+ - \frac{1}{\kappa} \ln(Ey_m^+) = 0.$$  

(12)

and the value of $y_m^+$ is 11.63 in the study. Boundary conditions for $k$ and $\varepsilon$ are set according to Versteeg and Malalasekera (1995). The fluxes of $k$ and $C$ at solid walls are set to zero. The boundary value of $\varepsilon$ at a solid wall is determined by

$$\varepsilon^+ = \left( \frac{\varepsilon y}{k^{3/2}} \right)^{3/4} = \frac{C_{\mu}^{3/4}}{\kappa}.$$  

(13)

According to Huang et al. (2007), a turbulent intensity 10% is used when $k$ is determined. In

![Fig. 6. Comparison of velocity profiles of numerical and experimental results. The sash is fully open. Root mean square error $\phi$ (a) 8.5%, (b) 17.6%, (c) 26.9% and (d) 32.6%.

[Diagram showing velocity profiles]
addition, SF6 is used as a tracer gas in the numerical model. Its density and molecular diffusivity in air are 6.04 kg m$^{-3}$ and 3 $\times$ 10$^{-5}$ m$^2$ s$^{-1}$. The gravitational effect is not considered in this study. The boundary condition of concentration at the exit of the ejector is 100%. A quiescent flow field is used as the initial condition in the model. Simulations do not terminate until steady-state solutions are reached. For each steady solution, the ratio of the maximum temporal variation of velocity to its magnitude is $<10^{-4}$.

The software STAR-CD based on the finite-volume method is employed to solve equations (1), (2), (6), (7), and (8). The second-order Crank–Nicolson scheme and the quadratic upstream interpolation of convective kinematics scheme are used for the temporal and advective terms, respectively. The pressure implicit with splitting operators scheme is utilized for the pressure–velocity iteration. The maximum mass residual from all computational cells must be $<10^{-5}$.

RESULTS AND DISCUSSION

Validation of model

The verification and validation of the established models are performed according to Babuska and Oden (2004). Two various meshes consisting of $\approx$540 000 and 590 000 cells for the exhaust fume hood model are used to assess the sensitivity of numerical solutions to the number of cells. The last one has more cells inside the fume hood. Predicted velocity profiles at $y/H = 0.125$, 0.375, 0.625 and 0.875 in front of the sash are compared and shown in Figs 6 and 7 for 100 and 75% open cases. As a result, those predicted velocity profiles by those meshes are very close. Hence, the grid independence exists in those solutions. The mesh consisting of $\approx$540 000 cells is enough to obtain a grid-independent solution.

Several parameters, including the volumetric flow rate induced by the top fan and the sash opening, govern the operation of an exhaust fume hood. In the present study, the volumetric flow rate at the top of an exhaust fume hood is fixed at 0.5 m s$^{-3}$ in order to compare experimental results provided by L. C. Tseng and Professor R. F. Huang of the authors’ department in 2006. The averaged upward speed of the cross-section of the duct is 12.5 m s$^{-1}$. Four sash openings, 100, 75, 50 and 25% are considered to explore the effect of the opening on flow structures inside a fume hood.

The established numerical model is validated using experimental results. Figure 8 reveals the
Fig. 8. Comparison of numerical streamline plot and experimental visualized results of flow fields of the central vertical plane with various openings. A manikin stands in front of the exhaust fume cupboard.

Fig. 9. Streamline plots of various vertical planes of a fully open push–pull fume cupboard in a concave mode. $V_b = 3 \text{ m s}^{-1}$ and $V_s = 14 \text{ m s}^{-1}$.
Fig. 10. Streamline plots of various horizontal planes of a fully open push–pull fume cupboard in a straight mode. 
\( V_b = 5 \text{ m s}^{-1} \) and \( V_s = 12 \text{ m s}^{-1} \).

Fig. 11. Streamline plots of various horizontal planes of a fully open push–pull fume cupboard in a under-suction mode. 
\( V_b = 2 \text{ m s}^{-1} \) and \( V_s = 4 \text{ m s}^{-1} \).

Fig. 12. Streamline plots of various horizontal planes of a fully open push–pull fume cupboard in a under-suction mode. 
\( V_b = 6 \text{ m s}^{-1} \) and \( V_s = 2 \text{ m s}^{-1} \).
streamline patterns of the central vertical plane and flow-visualized pictures, respectively. Two vortices are established: one behind the sash and the other behind the doorsill. This is observed both in the numerical simulations and experimental measurements. Qualitatively, the numerical results agree with the flow-visualization patterns. Hu et al. (1996) also explained those two vortices in their numerical results of the airflow inside an exhaust fume cupboard. Furthermore, Figs 6 and 7 show comparisons between the numerical and experimental profiles of the horizontal velocity component in the $x$-direction. For the case with a fully open sash, the numerical results are generally lower than experimental results. The root mean square error $\phi$ between numerical and experimental results ranges from 8.5 to 32.6%. For the case with a 75% open sash, the numerical and experimental results are close. The root mean square error $\phi$ between numerical and experimental results ranges from 8.1 to 17.7%. In terms of those comparisons, it appears that the numerical model is able to predict flow fields inside a fume hood, but it should be noted that the difference between the numerical solutions and experimental data exists.

Flow modes and concentration distribution of push–pull fume cupboard

A variety of parameters including push and pull flow speeds denoted as $V_b$ and $V_s$ and sash opening are considered in this study to investigate the performance of a push–pull fume cupboard. Essentially, all flow variations can be categorized into four modes, i.e. concave curtain, straight curtain, under-suction and over-blow modes. Figure 9 presents vertical streamline plots of a concave curtain example in a fully open fume cupboard. The pull flow is much faster than the push flow ($V_b = 3 \text{ m s}^{-1}$ and $V_s = 14 \text{ m s}^{-1}$). The resultant air curtain starts straight at the bottom of the sash and becomes concave in the vicinity of the pull side, i.e. the front bottom of the cupboard. Figure 9a–d shows the streamline plots of various vertical planes from the central section to the sidewall successively. Only a small vortex denoted as $E_a$ occurs at the rear bottom of the cabinet and consists of clean air inhaled from the environment. The concave curtain mode is recommended for operating a push–pull fume cupboard.

When the speed ratio of push to pull flows becomes large, the flow in the fume cupboard becomes...
a straight curtain mode as shown in Fig. 10a–d ($V_b = 5 \text{ m s}^{-1}$ and $V_s = 12 \text{ m s}^{-1}$). It is found that a vortex denoted as $E_{bR}$ occurs above the doorsill and in front of the air curtain. Due to the air curtain, the vortex $E_{bR}$ consists of clean ambient fluid. This is the only difference compared with a concave curtain mode. Figure 10a–d also reveals that all inhaled air from the top can be pulled to the bottom along smooth streamlines. As a result, pollutants coming from the ejector are captured by the flow and pulled into the slot behind the doorsill as shown in Fig. 10a. However, a straight curtain mode is not appropriate for the operation of a push–pull fume cupboard due to the manikin effect discussed later.

Furthermore, an under-suction example appears as shown in Fig. 11a–d as the speed ratio increases ($V_b = 2 \text{ m s}^{-1}$ and $V_s = 4 \text{ m s}^{-1}$). The air curtain still exists and becomes concave near the bottom of the cupboard. Nonetheless, a primary vortex denoted as $E_d$ is found inside the fume cupboard. Consequently, inhaled fluids from the top cannot be transported to the pull slit and brought out. In other words, no mechanism can remove pollutants from the fume cupboard in a under-suction mode. Therefore, it is recommended that a fume cupboard not be operated in an under-suction mode.

If the push flow is faster than the pull flow, then an over-blow mode will be observed. Figure 12a–d shows the flow patterns of an over-blow example ($V_b = 6 \text{ m s}^{-1}$ and $V_s = 2 \text{ m s}^{-1}$). Not only does a huge primary vortex occur inside the fume cupboard but also a vortex denoted as $E_b$ also develops in front of the air curtain. Due to the primary vortex $E_b$, pollutants cannot be brought out from the fume cupboard. In fact, pollutants may be transported to the environment by $E_d$ from the top of the fume cupboard. This mode is certainly not recommended for operating a push–pull fume cupboard.

To observe whether pollutants can be brought to the pull slit, streamlines starting from the exit of the ejector are drawn to visualize possible pollutant paths. Figure 13a–d presents 3-D streamlines in various modes. Obviously, streamlines are pulled to the slit at the bottom soon after exit from the ejector in the concave curtain and straight curtain modes. Nonetheless, streamlines in a under-suction or over-blow mode are strongly affected by vortices inside the fume cupboard in Fig. 13c,d. Pollutants coming from the ejector remain inside the fume cupboard.

### Characteristic figures of push–pull fume cupboards

Flow structures inside a push–pull fume cupboard are affected by the speed ratio of push–pull flows and the opening. Various ratios and openings are considered in this study in order to observe their effects. Figure 14 indicates the characteristics of flow
patterns obtained at various speed ratios in a fully open fume cupboard. Numerically predicted results are compared with the experimental results of Huang et al. (2007). The agreement between numerical and experimental results is acceptable as can be seen in Fig. 14. An operator may refer to those figures when they use a push–pull fume cupboard. Characteristic figures for other openings from 25 to 75% are available upon request. As mentioned in the previous section, one should adjust the speed ratio of push–pull flows to a concave curtain mode according to various openings.

Comparisons between exhaust and push–pull fume cupboards

The manikin effect on the capturing capability of an exhaust fume cupboard is extremely negative. This is because a manikin plays the role of an obstacle in a uniform fluid flow. As a result, a pair of vortices occurs between the manikin and the fume cupboard and pollutants accumulate in front of the manikin. To investigate the manikin effect on a push–pull fume cupboard, a manikin is located 7.5 cm in front of the sash according to ANSI/ASHRAE Standard 110-1995. As under-suction and over-blow modes are not appropriate for operation, only a concave curtain mode is examined and compared with an exhaust fume cupboard including the manikin effect.

The push–pull fume cupboard is designed to overcome the disadvantages of an exhaust fume cupboard. We now examine whether this aim has been achieved. Figure 15a–h shows vertical streamline plots for fully open exhaust and push–pull fume cupboards. These cupboards are in operation at the same volumetric flow rate of suction: 0.5 m³ s⁻¹. An upward flow appears in front of the manikin in an exhaust fume cupboard as shown in Fig. 15a,b, so an operator may breathe pollutants transported by the upward flow. Figure 15e–h shows that there is no such upward flow in a push–pull fume cupboard. The flow in front of the manikin is smoothly pulled into the slit at the bottom. Moreover, Fig. 16a–h reveals the vertical concentration contours of those fume cupboards. The local concentration values in front of the manikin in an exhaust fume cupboard are extremely high as shown in Fig. 16a–d. In contrast, the local concentration values in front of a manikin in a push–pull fume cupboard are <1 parts per billion (p.p.b.).

Since pollutants may also leak out of the cupboard from gateposts and the doorsill, the ANSI-ASHRAE
Standard 110-1995 does not define the required minimum concentration values at the rest areas on the front face of the sash. In order to investigate variations of concentration at the rest areas, 12 points on the vertical face of the sash are examined. Figure 17 gives a schematic of the locations of those 12 points on the sash face. Figure 18 shows an example in a concave curtain mode. Hollow and solid symbols refer to the local concentration values in a push–pull fume cupboard and an exhaust fume cupboard, respectively. Consequently, all local concentration values in a push–pull fume cupboard are $<0.1$ parts per million (p.p.m.). The values at P2, P3, P6 and P7 are even $<1$ p.p.b., so no hollow symbols appear. (a) $H/H_{max}=100\%$, (b) $H/H_{max}=75\%$, (c) $H/H_{max}=50\%$, (d) $H/H_{max}=25\%$.

**Fig. 16.** Comparisons of vertical concentration contours between a conventional (a)–(d) and a push–pull fume cupboard (e)–(h) at the same flow rate $0.5 \text{ m}^3/\text{s}$. The push and pull speeds are 2 and 12 m/s. This speed ratio is in a concave curtain mode.

**Fig. 17.** Schematic of 12 sampling points of concentration in front of the fume cupboard.
in Fig. 18. In addition, an exhaust fume cupboard obviously does not satisfy the requirement of 0.1 p.p.m. except for P6 and P7. In general, an exhaust fume cupboard does not perform better than the corresponding push–pull fume cupboard in terms of the local concentration values at those selected points. All those results show that a push–pull air curtain fulfills its purpose successfully.

CONCLUSIONS

The push–pull air curtain is a new form of fume cupboard. Numerical approaches based on the finite-volume method are established and used to simulate flows inside exhaust and idealized push–pull fume cupboards. Four characteristic flow modes (i.e. concave curtain, straight curtain, under-suction and over-blow) are considered, for a variety of speed ratios of push to pull flows and openings. The concave curtain mode is recommended to the practical operation of a push–pull fume cupboard in terms of its flow pattern and concentration distribution. The presence of a manikin located in front of the sash is also considered. The numerical predictions indicate that the flow pattern is not affected by the standing manikin in a concave curtain mode and neither is the concentration distribution. Comparisons between an exhaust and push–pull fume cupboard are also undertaken. The disadvantages of an exhaust fume cupboard with or without a manikin are not found in a push–pull fume cupboard. Moreover, the push–pull fume cupboard not only satisfies the ANSI/ASHRAE Standard 110-1995 but also is found to have concentration values <0.1 p.p.m. at 12 selected points. This means that a push–pull fume cupboard is able to prevent pollutants from escaping from the cupboard and thus protect an operator in front of the sash.

FUNDING

National Science Council, Taiwan (NSC 94-2212-E-011-038); computing resources from National Center for High Performance Computing, Taiwan; computing center of National Taiwan University of Science and Technology.

Acknowledgements—Generous English editing given by Professor A. G. L. Borthwick in the Department of Engineering Science of University of Oxford is highly appreciated.

REFERENCES


