11528

EFFECTS OF A VENTILATION DUCT ON THE PERFORMANCE OF A FUME CUPBOARD

P. M. J. Trevelyan, L. Elliott and D. B. Ingham

Department of Applied Mathematics
The University of Leeds, Leeds LS2 9JT

ABSTRACT

When a fume cupboard is placed in a room with a ventilation duct, the air movement inside and around the fume cupboard is fully three-dimensional turbulent flow. However, in order to understand the fluid flow away from the fume cupboard a much simpler model can be used. This leads to a steady 2D model, with the computational domain including only the sash of the fume cupboard, the room and the entrance into the ventilation duct. In this paper we have used both the k-ε turbulence model and the wall function technique to calculate the steady 2D turbulent fluid flow. In addition, a mathematical technique has been employed to map the simpler model onto the upper half of the complex plane, so that the complex potential can be found using a source and a sink on the real axis to represent the ventilation inlet and the exhaust outlet of the room, respectively.

The objective of this paper is to reduce the cpu time by restricting the computational domain to a region which only includes the fume cupboard and a small region outside. We establish from a modified version of the potential flow a solution which we can use near the fume cupboard as a specified velocity boundary condition whilst solving the turbulent fluid flow model within the fume cupboard.

KEYWORDS

CFD, fume cupboard, modelling, air flow pattern.

INTRODUCTION

In a laboratory which contains a fume cupboard, the 1988 Control of Substances Hazardous to Health Regulation must be satisfied, which means that the understanding of the air flow, and hence the convection of the contaminant, in the vicinity of the working aperture is crucial, see Khezzar and Whitelaw (1994), Hu et al. (1997) and Durst et al. (1997). To improve the performance of fume cupboards in laboratories, ventilation ducts are usually introduced. Although in practice laboratories may also contain several other sources of air supply, where the associated problems have fluid flows which are time dependent, three-dimensional and turbulent, on solving such complicated problems one frequently finds that they fail to reveal the main characteristics of the fluid flow. Hence, in this paper we have considered the much simplier situation of steady 2D turbulent fluid flow.

Due to the safety laws it is important to find the fluid flow field near the fume cupboard, and hence a detailed numerical study of the fluid flow in the fume cupboard is necessary. However, since the entrance boundary condition of the fume cupboard is unspecified it is necessary to include the rest of the room, namely, the ventilation duct, the fume cupboard and the entire room, in the computational domain, so that their surfaces define the boundary.

Due to the structure of the turbulent fluid flow, a fine grid in the fume cupboard is necessary, therefore the cpu time is large. Hence the objective of this paper is to reduce this cpu time by restricting the computational domain. If we employ a simplified model throughout the room, the ventilation duct and the fume cupboard, to give a solution to the fluid flow, we could use this solution near the fume cupboard as a specified velocity boundary condition. Then the fluid flow within the fume cupboard, using the k-ε turbulent model and the wall function technique within the commercial software package FLUENT can be implemented.

PHYSICAL REGION

The dimensions of the room have been chosen as typical values for a room, namely the height and length are $3m$ and $4m$, respectively. Further, a typical fume cupboard of height and width $1m$ and $1m$, respectively, has been set $1m$ above the floor and an exhaust duct of height and width $4m$ and $0.25m$, respectively, is placed at the back of the fume cupboard. The ventilation duct of height and width $3m$ and $0.25m$, respectively, is located in the ceiling of the room and the distance from the wall containing the fume cupboard to the ventilation duct opening is $1m$, see figure 1.

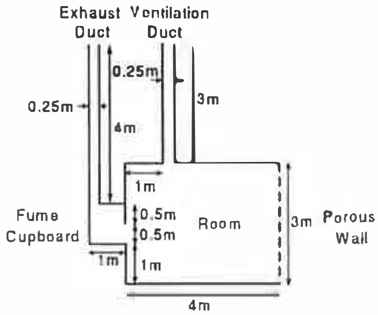


Figure 1: Physical dimensions of the room, fume cupboard and ventilation duct.

Fletcher and Johnson (1992a,b) assumed the wall opposite the fume cupboard to be porous so that when they used an electric fan to extract the air from inside the fume cupboard into the exhaust duct the fluid entering the room through this wall could be treated as having a uniform speed.

In this paper the emphasis has been placed on determining the fluid flow **outside** the fume cupboard. Hence the handle of the fume cupboard has been ignored and the design of the sash (fume cupboard opening) kept as simple as possible. It is assumed that the average velocity across the sash, which varies from 0.3 to 1.0 ms^{-1}, is known. However, since in this range of fluid velocities the numerical predictions of the turbulent flow are qualitatively the same, only results when the typical value of the average sash velocity is 0.5 ms^{-1} are presented. The average fluid velocity into the ventilation duct is in the range 0 to 0.5 ms^{-1}, with the corresponding average fluid velocity through the porous wall in the range 0.1 to 0.04 ms^{-1}, so as to ensure that the fluid velocity through the sash is maintained at 0.5 ms^{-1}.

TURBULENT MODEL

We have used the k-ε turbulence model and the wall function technique within the commercial software package FLUENT to calculate the 2D turbulent fluid flow. The system of equations solved is based on the time-averaged Navier-Stokes equations, which are closed by the standard version of the k-ε turbulence model, and in the usual notation are expressed in the following form:

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\rho u_i \frac{\partial u_j}{\partial x_i} = -\frac{\partial P}{\partial x_j} + \frac{\partial}{\partial x_i}\left[(\mu_l + \mu_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] \tag{2}$$

$$\rho u_i \frac{\partial k}{\partial x_i} = \frac{\partial}{\partial x_i}\left[(\mu_l + \frac{\mu_t}{\sigma_k})\frac{\partial k}{\partial x_i}\right] + G_k - \rho\varepsilon \tag{3}$$

$$G_k = \mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right)\frac{\partial u_j}{\partial x_i} \quad (4)$$

$$\rho u_i \frac{\partial \varepsilon}{\partial x_i} = \frac{\partial}{\partial x_i}\left[(\mu_l + \frac{\mu_t}{\sigma_\varepsilon})\frac{\partial \varepsilon}{\partial x_i}\right]+$$
$$c_1 \frac{\varepsilon}{k}G_k - c_2 \rho \frac{\varepsilon^2}{k} \quad (5)$$

$$\rho \overline{u_i' u_j'} = \frac{2}{3}\rho k \delta_{ij} - \mu_t\left(\frac{\partial u_i}{\partial x_j}+\frac{\partial u_j}{\partial x_i}\right) \quad (6)$$

$$\mu_t = \rho c_\mu \frac{k^2}{\varepsilon} \quad ; \quad P = p + \frac{2}{3}\rho k \quad (7)$$

$$c_1 = 1.44, \quad c_2 = 1.92, \quad c_\mu = 0.09,$$
$$\sigma_k = 1.0, \quad \sigma_\varepsilon = 1.3 \quad (8)$$

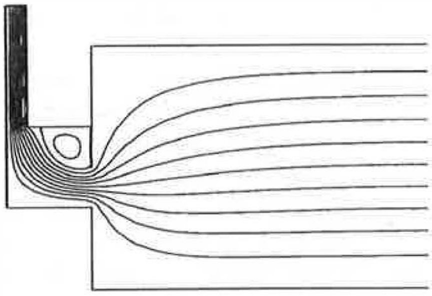
The boundary conditions applied at the surface of the walls make use of the wall function approach developed by Launder and Spalding (1974). In order to test the performance of the fume cupboard, when it is influenced by the ventilation duct the flow field must be found.

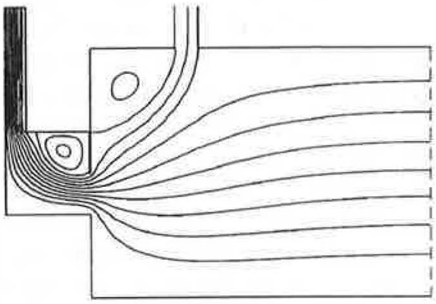
When applying the **fully developed** velocity profiles at the porous wall of a room of length $4m$, and comparing the fluid flow with that from a room of length $5m$, we found that at $x=3m$ the fluid velocities were almost identical, which suggests that the influence of the fume cupboard and the ventilation duct are restricted to the region $0<x<3m$. However, from applying the uniform velocity profiles at the porous wall, i.e. at $x=4m$ and $x=5m$, along the same line $x=3m$ the velocity profiles are quite different, which, from the information for fully developed profiles, suggests that these profiles are still being influenced by the floor and the ceiling, i.e. the fluid flow is still developing. The velocity profiles at $x=0.5m$ have much more consistent nature, regardless of the type of upstream boundary condition and the position where it was being applied. This suggests that at $x=0.5m$ the velocity profile is dominated by the fume cupboard and the ventilation duct. Thus, provided the flux of fluid entering through the ventilation duct and through the porous wall remain fixed, the upstream effects in this region remain small.

When comparing the experimental results with the numerical predictions, then the uniform flow boundary condition is the most appropriate since this is what is used in wind tunnels. However, in any practical room situation containing a fume cupboard then the assumptions of both the porous wall and the uniform flow are inaccurate. However, in order to focus on the region near to the fume cupboard and to remove any possible fluctuations in the flow caused by variations in the length of the room, uniform flow through the porous wall has been replaced by that resulting from the fully developed turbulent flow. Although this assumption is still inaccurate, it enables all results found from a room of length greater than about $3m$ to be appropriate for other rooms where the length exceeds $3m$, and so simplifies the problem being solved. The fluid flow calculations have also been performed in the absence of any ventilation duct.

By considering a long open channel of height $3m$, the fully developed turbulent fluid velocity profile for various strengths of flux into the ventilation duct were obtained. Then the 'region of influence' of the fume cupboard and the ventilation duct on the flow upwind of the inlet was determined by comparing the u component of the fluid velocity in two long rooms of different lengths arbitrarily chosen as $10m$ and $20m$, but with the same fully developed boundary condition at the porous wall. The region of influence of the fume cupboard and the ventilation duct was found to be less than about $3m$, which agrees with the previous results.

Now that the region of influence has been determined, the fully developed velocity profile, as found from the flow in an open channel, can be imposed on the porous wall, with the wall $4m$ from the fume cupboard so that the porous wall is not contained within the region of influence.


(a)


(b)

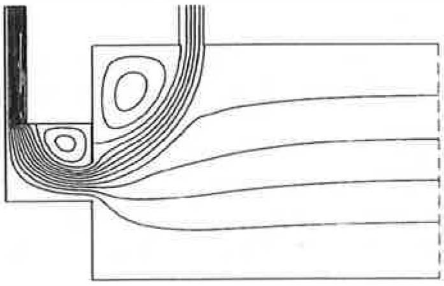

(c)

Figure 2: Turbulent fluid flow streamlines in the physical region, see figure 1, with the flux of the fluid through exhaust duct being $0.25\,m^2s^{-1}$ and the flux of the fluid through the ventilation duct being (a) $0\,m^2s^{-1}$, (b) $0.05\,m^2s^{-1}$, and (c) $0.125\,m^2s^{-1}$. The values of the stream function along each streamline are from -0.3 to -0.025 in steps of 0.025.

The fluid flow for the whole room was then calculated. To illustrate the fluid flow we present in figure 2 the streamlines for various strengths of the flux of fluid into the ventilation duct. We observe that the larger the face velocity at the ventilation duct, the larger is the recirculating flow between the fume cupboard and the ventilation duct.

POTENTIAL MODEL

As the Reynolds number is very large, we initially assume that the flow may be approximated by neglecting the viscosity and considering the flow to be potential. In addition, if we are to maintain a realistic geometrical model, the physical region in which a solution is to be found will be simplified to that shown in figure 3, which includes just the sash, the ventilation duct and the porous wall at the opposite end of the room to the fume cupboard.

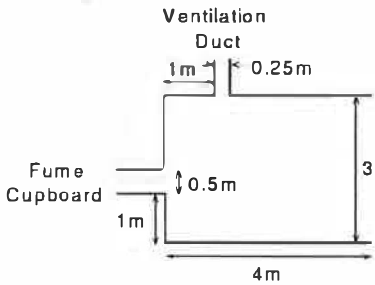


Figure 3: Simplified geometrical model.

It is assumed that the porous wall and the ventilation duct can be treated as uniform inlets and the sash can be treated as a uniform fluid flow outlet, provided that the flux of fluid entering from the ventilation duct doesn't exceed that being removed at the exhaust duct.

This solution domain has been mapped onto the upper half of the complex plane by applying the Schwartz-Christoffel transformation, with a mathematical technique being performed to determine the unknown parameters of the transformation from the dimensions of the simplified geometrical model. The fluid flow is finally found using complex potential theory. In

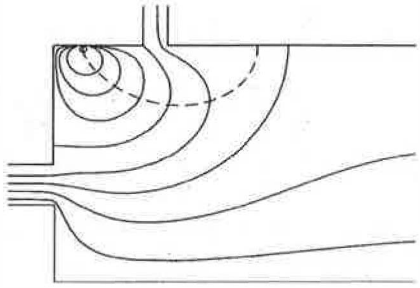


Figure 5: Potential fluid flow streamlines in the geometrical model, see figure 3, after introducing a vortex at the point (0.3325, 2.9790) with a strength of $54.4915\,m^2 s^{-1}$, with the sash of strength $0.25\,m^2 s^{-1}$ and the ventilation duct of strength $0.125\,m^2 s^{-1}$. The dotted line represents the location of the vortex of a predetermined strength such that a streamline exists with the same end points as the dividing streamline in figure 2(c). The values of the stream function along each streamline are -5, -1, -0.55, -0.35, -0.25, and then the remainder are from -0.175 to -0.025 in steps of 0.05.

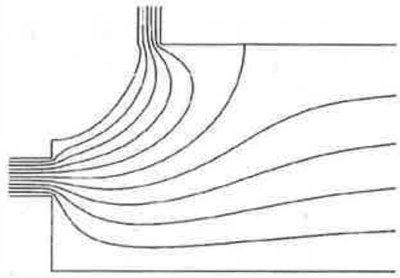
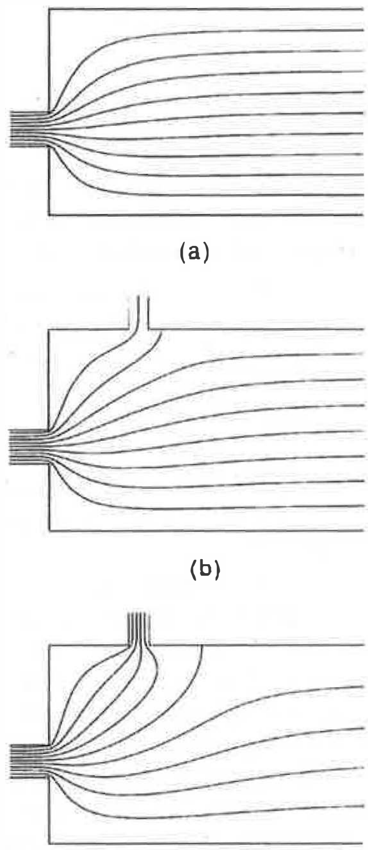


Figure 6: Potential fluid flow streamlines in the geometrical model, see figure 3, using the mapped ellipse method, with the sash of strength $0.25\,m^2 s^{-1}$ and the ventilation duct of strength $0.125\,m^2 s^{-1}$. The values of the stream function along each streamline are from -0.225 to -0.025 in steps of 0.025.

Another approach is to modify the boundary of the room by replacing the straight walls by a curved wall which approximates the edge of the recirculation zone. This can be achieved by mapping the region shown in figure 6 to the region above an ellipse in the upper half of the complex plane, while still having the height of the ellipse left free to enable the ellipse to approximate the curvature of the dividing streamline. Following a further mapping to an upper half plane, we can now find the complex potential in this new plane, and hence determine the streamlines and fluid velocity. We illustrate the fluid flow with the streamlines in figure 6.

This model requires more information from the turbulent fluid flow predictions than does the previous modification, and so, as one would expect, this model gives the best approximation to the turbulent fluid flow. However, the idea of studying potential flow was to be able to determine the fluid flow field without having to use the results from the turbulent fluid flow predictions, and so these modifications have digressed slightly from this aim.

We now return to the objective of this paper, which is to reduce the cpu time by restricting the size of the computational domain, we have now used this modified version of the potential flow, with a curved boundary approximating the edge of the recirculation zone, to give a solution to the fluid flow. We can now compare the horizontal and vertical components of the fluid velocity given by the turbulent model with those found using the second modified version of the potential flow. In order to achieve reasonable agreement between these models it was found that if a specified fluid velocity boundary condition was to be imposed then it should be more than $0.05m$ away from the fume cupboard, since the potential model has an infinite velocity at the corners of the opening to the fume cupboard. However, as the aim of this paper was to reduce the computational domain, the boundary condition should not be imposed at more than $0.5m$ from the fume cupboard. To illustrate this, we show a selection of the u component of the velocities for both the turbulent model and this modified version of the potential flow, for a ventilation flux of figure 4 we illustrate the potential flow for two different inlet strengths, i.e. the flux of fluid into the ventilation duct.



(a)

(b)

(c)

Figure 4: Potential fluid flow streamlines in the geometrical model, see figure 3, with the sash of strength $0.25\,m^2s^{-1}$ and the ventilation duct of strength (a) $0\,m^2s^{-1}$, (b) $0.05\,m^2s^{-1}$, and (c) $0.125\,m^2s^{-1}$. The values of the stream function along each streamline are from -0.225 to -0.025 in steps of 0.025.

The turbulent fluid flow and the potential flow can be seen to agree fairly well when there is no flux of fluid through the ventilation duct in figures 2(a) and 4(a). However, as the flux the fluid through the ventilation duct increases, the two fluid flows become less comparable, as highlighted in figures 2(b) and 2(c) and 4(b) and 4(c).

From the streamline patterns obtained for the turbulent fluid flow, see figure 2(b) and 2(c), we observe that there exists a large recirculating fluid flow in the upper left corner of the room, i.e. between the fume cupboard and the ventilation duct, which we will refer to as the recirculation zone.

However, when observing the streamline patterns for the potential fluid flow in figure 4(b) and 4(c), we observe that there is no such recirculation region. This, of course, is as we would expect from physical considerations, but it gives us some indication as to why the potential fluid flow fails to be comparable to the turbulent fluid flow. Before any detailed comparisons can be made between the turbulent and the potential fluid flows we need to modify the potential fluid flow so as to be able to incorporate the recirculation zone.

We considered various types of modification to the potential model, with the aim that the revised fluid flows will give more realistic approximations to the turbulent fluid flow situations over a large range of ventilation positions and strengths. The types of modifications included attempting to incorporate the recirculation zone by artificially placing a vortex inside the region where the recirculation zone is located, such that the vortex ensures that the dividing streamline approximates the edge of the recirculation zone. We found a set of points for the location of the vortex which, with the required strength, enabled there to exist a streamline with the same end points as the dividing streamline in figure 2(c), and is represented in figure 5 by the dotted line. We note that the dividing streamline models the edge of the recirculation zone most effectively when the vortex is closest to the ceiling between the ventilation duct and the wall containing the fume cupboard. The potential streamlines of the fluid flow, for the most effective location of the vortex, is shown in figure 5.

$0.125\, m^2 s^{-1}$ with a fume cupboard flux of $0.25\, m^2 s^{-1}$, at various distances from the fume cupboard, see figure 7.

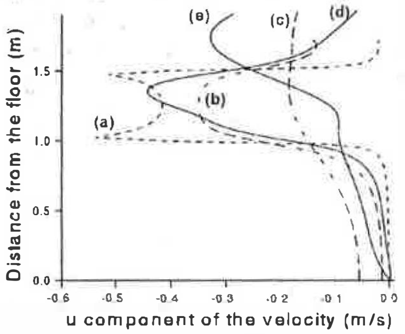


Figure 7: The u component of the fluid velocities for both the turbulent model and the second modified version of the potential flow at various distances from the fume cupboard, with the flux of the fluid through the exhaust duct being $0.25\, m^2 s^{-1}$ and the flux of the fluid through the ventilation duct being $0.125\, m^2 s^{-1}$. The potential u velocity component is shown at distances from the fume cupboard of (a) 1.5*cm*, (b) 11*cm*, and (c) 52*cm*. The turbulent u velocity component is shown at distances from the fume cupboard of (d) 11*cm*, and (e) 52*cm*.

It should be noted that a boundary condition applied along a vertical line throughout the room, may not be the best way of reducing the computational grid. Work is still ongoing in finding a means to apply this solution as a specified fluid velocity boundary condition near the fume cupboard while solving the fluid flow within the fume cupboard using the k-ε turbulent model and the wall function technique within the commercial software package FLUENT.

CONCLUSIONS

This paper has considered ways of modifying the potential fluid flow in order to acheive a fluid flow which is more comparable with the turbulent fluid flow in the region outside the fume cupboard. However, the aim of providing the most effective specified fluid velocity boundary condition in order to reduce the computational domain for the turbulent model is still ongoing.

ACKNOWLEDGEMENTS

The author would like to acknowledge the financial support of the Engineering and Physical Sciences Research Council.

REFERENCES

Durst F., Denev J.A. and Mohr B. (1997), Room ventilation and its influence on the performance of fume cupboards: A parametric numerical study. *Ind. Eng. Chem. Res.*, **36**, 458-466.

Hu P., Ingham D.B. and Wen X. (1997), Three-dimensional turbulent flow inside a fume cupboard and a room, *IChemE97*, **1**, 369-372.

Khezzar, L. and Whitelaw, J.H., (1994), Numerical simulation of fume cupboard performance, Proc. Instn. Mech. Engngs., Part C: J. of Mech. Engng. Science, **208**, 155-166.

Fletcher B. and Johnson A.E., (1992a), Containment testing of fume cupboards-I methods, *Ann. Occup. Hyg.*, **36**, 239-252.

Fletcher B. and Johnson A.E., (1992b), Containment testing of fume cupboards-II methods, *Ann. Occup. Hyg.*, **36**, 395-405.

Launder, B.E. and Spalding, D.B., (1974), The numerical computation of turbulent flows, Computer Methods in Applied Mechanics and Engineering, **3**, 269-289.