The Effect of Turbulent Structures on Hood Design—A Review of CFD and Flow Visualization Studies

J.O. Varley, P.E.
Member ASHRAE

B. Ghorashi, Ph.D.

This paper reviews separate flow visualization studies and numerical techniques as related to the fluid behavior of industrial ventilation systems and hood design. The results of these separate studies are brought together in order to shed some light on the fluid behavior in these systems. It is observed that empirically-derived velocity contours continue to be the standard for the design of ventilation systems. However, the practice of sizing exhaust hoods based on velocity contours neglects the existence of events that would inevitably influence the exhaust hood performance. For example, the turbulent structures created by the presence of crossdrafts, room air turbulence, and flow separation around objects within the vicinity of the hood are all but ignored when one solely relies on the velocity-contour technique. This review study presents methods that can be utilized to account for and evaluate the influences of these events.

It is concluded that CFD is the most valuable tool for modeling the flow for ventilation systems. In addition, a paradigm is identified and suggested for future research in this area.

INTRODUCTION

Ventilation is an important tool for eliminating airborne contaminants from the industrial workplace. The induced airflow created by a ventilation system provides a mechanism for contaminant transport, thereby eliminating the threat of human exposure to hazardous contaminants. A typical ventilation system consists of a hood operating under a negative pressure, ductwork, and a fan. The hood provides the means of local contaminant capture. Fan-generated airflow provides the transport means for the contaminant as it moves through the ductwork. It then is discharged directly to the atmosphere, or is filtered prior to discharge.

The ventilation design problem begins with a contaminant source that generates a solid, liquid, and/or vapor, in any combination. After it has been dispersed and suspended in air, the contaminant has the potential to be inhaled and subsequently ingested by humans located within the affected zone. Concentrations deemed hazardous to occupants require the consideration of process changes and/or additional ventilation.

Early hood design relied on tables of recommended airflows for various hood configurations. These tables were compiled through years of experience, but, nevertheless, they were only a qualitative tool. DallaValle (1932), reported the results of an empirical study showing that openings under negative pressure could be characterized by constant velocity contours radiating outward from the hood opening. These contours depicted velocity decaying radially outward from the hood opening. This information provided ventilation system designers with a quantitative technique to size systems. Today, ventilation system designers continue to rely on velocity contours. The design methodology is as follows:

John O. Varley was a graduate student in the Department of Chemical Engineering, Cleveland State University, OH before becoming a project engineer with the Brewer-Garrett Company, Middleburg Heights, OH. **Bahman Ghorashi** is a professor of chemical engineering in the Department of Chemical Engineering, Cleveland State University.

- A consensus of opinion is reached on an acceptable hood capture velocity at the contaminant source.
- An acceptable hood shape is chosen. (For example, in laboratory facilities, a standard laboratory fume hood is typically chosen.)
- Velocity contours provided in the literature are then used to size system airflows and, thus, system components.

Unfortunately, the above methodology ignores the effects of turbulent structures on contaminant capture and transport. These structures are present in virtually every turbulent flow encountered in ventilation design. Specifically, these effects can be summarized as follows:

- Turbulent structures created in the wake of objects (or people) near the hood
- Turbulent structures caused by hood geometry
- Impinging airflows on the velocity fields generated by the hood
- Free-stream turbulent structures

Each of these fluid flow phenomena has been investigated without application to ventilation. The purpose of this paper is to shed some light on the fluid behavior in these systems. In addition, it identifies a paradigm for future research in this area.

DESIGN VARIABLES

With respect to contaminant capture at a local hood, three design variables must be considered: the contaminant source, hood geometry, and the surrounding environment.

The contaminant source dictates the nature and dispersion qualities of the contaminant. The source provides the initial momentum and concentration of the contaminant. The momentum is the product of the contaminant mass and release velocity from the source. The mechanism of vapor release can be quantified using vapor phase equilibria thermodynamics. On the other hand, the release rate of liquids (mists) or solids (particulate) can be decided by evaluating the physics of the process. Momentum and concentration are transferred to the surroundings as the contaminant moves away from its source. The dispersive driving forces are molecular diffusion, convective diffusion, and turbulent diffusion. The release rate and dispersion rate are interrelated in that high contaminant release rates tend to be associated with jets that entrain surrounding air and subsequently distort air patterns around the dispersed contaminant. These rates greatly influence a hood's ability to "capture" the dispersed contaminant. The term "capture velocity" refers to the required level of air velocity generated by a hood to capture contaminant and transport it along streamlines created by the hood. Capture velocity depends on distance from the hood as derived from the aforementioned studies by DallaValle (1932, 1952). This relationship was mathematically derived for a variety of hood configurations. At distances away from hood openings capture velocities are affected by the turbulence effects listed in the introduction and expanded below.

Hood geometry is critical in determining the amount of airflow required for a specific application. The previously-mentioned empirical work showed that the volume of airflow required to generate the necessary capture velocity at a specified distance from the hood varies greatly with hood geometry. For example, a flanged slotted hood requires significantly less airflow than an unflanged hood of the same dimensions to generate an equivalent capture velocity. Potential flow theory and computational fluid dynamics have been used recently to evaluate these effects, as discussed below.

The environment that surrounds a local exhaust hood contributes to turbulent structures that impact the velocity profile of the hood and the diffusion rate of the contaminant. This contribution is made by drafts created by the air supplied to the area by mechanical or natural means, by air flow separation generated by air flowing around objects located near the hood, or by turbulence in the free-stream of air as it approaches the hood. The impact of turbulence on air flow and contaminant transport is addressed in this paper.

Roach (1981) considered the effects of turbulence on diffusion and formulated a simple model to evaluate its influence. This model mathematically combined molecular and turbulent diffusion to form effective diffusivity. He then developed simple equations that modeled contaminant concentration under various flow conditions. Esmen et al. (1991) modeled turbulent diffusion by treating it as a first order mixing process. They assumed that small pockets of air could be considered as impenetrable volumes containing and not containing the contaminant. Although these studies have considered turbulent diffusion to be the prime component of the total diffusion rate of the contaminant, recent research by Kontomaris and Hanratty (1993) shows that high molecular diffusion rates can reduce the effects of turbulent diffusion. This result was not considered in the literature reviewed for this paper, and poses a significant obstacle to modeling contaminant transport in turbulent flows where high rates of molecular diffusion are present. Nevertheless, the above-referenced work suggests that the design of the surrounding environment has a profound impact on reducing the effects of turbulent structures.

IDEALIZED FLOW

As implied in the preceding section, the contaminant source can be easily characterized using chemical equilibrium thermodynamics and transport theory. In addition, the hood geometry can be established by employing proper ergonomic considerations and common sense. The primary obstacle to good hood design is characterizing the influence of external challenge, such as competing airflows and turbulent structures, to the velocity fields created by the hood.

To develop a mathematical understanding of air motion in ventilation systems first requires a simplified approach to the fluid mechanical system. In recent years, studies have focused on applying potential flow theory to the empirical information gathered by previous research. These efforts have provided a rigorous mathematical model for designers to quantify the containment characteristics of hoods before fabrication. Potential flow theory assumes that the fluid is inviscid, incompressible, and irrotational (Gerhardt and Gross 1985, Kreyszig 1993). These assumptions result in the following mathematical expression:

$$\nabla \times u = 0 \tag{1}$$

where u is a velocity vector.

The assumption of irrotational flow leads to the following:

$$u = \nabla\Phi \tag{2}$$

where Φ is the velocity potential.

Using Cartesian coordinates,

$$u_x = \frac{\partial\Phi}{\partial x} \qquad u_y = \frac{\partial\Phi}{\partial y} \qquad u_z = \frac{\partial\Phi}{\partial z} \tag{3}$$

Assuming incompressible flow, substitution into the continuity equation leads to the Laplace equation:

$$\frac{\partial^2\Phi}{\partial x^2} + \frac{\partial^2\Phi}{\partial y^2} + \frac{\partial^2\Phi}{\partial z^2} = \Delta^2\Phi = 0 \tag{4}$$

The gradient of the velocity potential can be equated to velocity of a particle of fluid at a particular point in space. Lines of equipotential are perpendicular to the stream curves, Ψ (see Figure 1). The specific mathematical form of the potential function depends upon boundary conditions.

Potential flow theory has received significant attention by many authors who have applied it to the modeling of air flow in front of hoods for a variety of configurations. Rationale for using this approach is that the elliptical nature of the velocity contours depicted in Figure 1 are coincident with equipotential surfaces. Flynn and Ellenbecker (1985), evaluated the air flow into a flanged circular hood using potential flow theory. Using the cylindrical coordinate system and a previously derived relation for potential flow for a circular aperture they generated streamlines that were perpendicular to the equipotential lines. The resultant equations are:

$$V_{R1} = -\frac{Q}{\pi}\left[\frac{(a+r)\lambda_2 + (r-a)\lambda_1}{(\lambda_1+\lambda_2)\lambda_1\lambda_2\sqrt{(\lambda_1+\lambda_2)^2 - 4a^2}}\right] \tag{5}$$

$$V_{z1} = \frac{Qz}{\pi\lambda_1\lambda_2\sqrt{(\lambda_1+\lambda_2)^2 - 4a^2}} \tag{6}$$

where

$$\lambda_1 = \sqrt{z^2 + (a+r)^2}, \quad \lambda_2 = \sqrt{z^2 + (a-r)^2} \tag{7}$$

and where Q is the hood air flow, a is the hood radius, z is the axis corresponding to the hood centerline perpendicular to the hood face, and r is the radial distance perpendicular to the z axis in any direction.

Because the resultant equation for velocity provided erroneous results at the hood face and hood edge, the equation was modified by differentiating the potential with respect to the eccentricity of the ellipse. The second model assumes that the total velocity at any point must equal Q divided by the surface area at the equal velocity contour on which the point lies

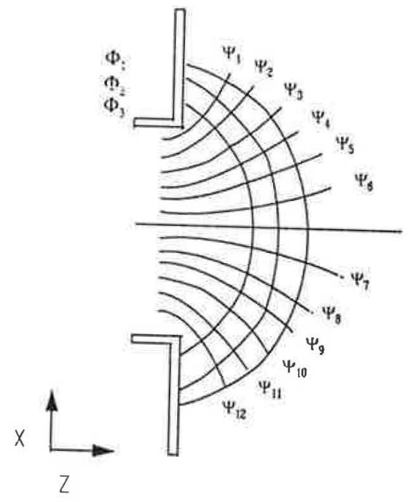

Figure 1. Equipotential lines and streamlines

$$V_T = \frac{\sqrt{3}Q\varepsilon^2}{2\pi a^2\sqrt{3 - 2\varepsilon^2}} \tag{8}$$

where V_T is the total theoretical flow at a given point and ε is the eccentricity defined as the ratio of the hood diameter to the sum of the distances from the corners of the hood to the point being evaluated.

In addition, Flynn and Ellenbecker (1987) applied a uniform crossdraft perpendicular to the hood centerline by simple vector addition. These results compared favorably to DallaValle's empirical data. Next, Flynn and Ellenbecker validated their models. Using measurements taken with a two-wire anemometer to allow for airflow measurement in two directions, they found good correlation with all three models, and particularly good correlation with the model depicting crossdraft. Besides these analytically approaches to using potential flow theory, Flynn and Miller (1989) employed a numerical technique called the "Boundary Integral Equation Method" to solve Laplace's equation in three dimensions.

Conroy et al. (1989) evaluated the capture efficiency of flanged slot hoods using potential theory. They employed a potential model developed for an elliptical opening to represent a rectangular opening of a slot. In addition, they demonstrated that crossdrafts may be applied by using vector addition of the crossdraft potential with the hood potential for this hood configuration. Streamlines were then generated by assuming that they are perpendicular to the potential lines. To model contaminant transport in the presence of a crossdraft they assumed that contaminant was released with no velocity into a crossdraft. Streamlines entering the hood achieved 100% capture efficiency while streamlines not entering received zero efficiency.

The Flynn and Ellenbecker model was validated experimentally in two parts. First, a hot film two-wire anemometer was used to validate calculated airflow contours. The next phase of the study validated the calculated efficiencies by performing measurements to decide the rate at which the contaminant was spread after dispersion, and then measuring the concentration of the contaminant at various points within influence of the hood. The experimental data suggested that an inscribed ellipse potential flow model provided good correlation, and that contaminant transport can also use potential flow theory.

Although research in the area of potential theory has assumed ideal conditions, it provides an important link between the empirical work of DallaValle and classic fluid theory. Further, the application of a crossdraft to the model provides an idealized consideration to the problem of impinging airflows created by the environment. This approach offers a powerful tool to designers who need to model hoods placed in an environment subject to challenge at the face. The primary flaw in using this approach to design local exhaust hoods is the fact that it ignores turbulence or viscous effects. In addition, most of the methods that use it assume that dispersive forces are insignificant. In real world applications these conditions are often present and cannot be ignored. As a result, ventilation system design based on this tool should use safety factors to account for these non-ideal conditions.

TURBULENT FLOW

When a hood is introduced to real world conditions, competing airflows will distort the generated velocity contours. These conditions include:

- Flow separation around a person or object in front of the hood
- Crossdrafts created by motion in the space where the hood is located (i.e. people movement, door movement)
- Drafts created by space air diffusion
- Drafts created by equipment loads

Drafts created by thermal gradients within the space are also possible and have been studied (Nielson et al. 1978, 1979). However, these effects can be neglected on the assumption that room air is well mixed, and that motion generated by thermal buoyancy is negligible when compared with fluid movement generated by pressure gradients.

Separation of airflow streamlines occurs as the air approaches an object. As air moves across an object, a boundary layer appears on the upstream side of the object to the middle of the object. At this point, air decelerates as pressure increases. This tendency causes flow streamlines to be separated from the boundary of the object. Flow streamlines generated by a local exhaust hood will be affected by this phenomenon. A hood is typically positioned in front of a process and a worker assigned to the process. This configuration generates streamline separation around the worker as well as other objects in the air stream and subsequent turbulence at the hood face. As a result, the velocity contours depicted in Figure 1 will be distorted. In addition, the hood geometry creates boundary layer effects. For example, air flow streamlines tend to constrict at the hood entry forming a vena contracta. Further, enclosing hoods, such as those used in laboratories, contain recirculation zones within the hood. The complexity involved in mathematically describing these phenomena limits the ability of even the most sophisticated system designer to accurately model a local exhaust hood placed in real world conditions. However, recent research has begun to apply numerical approaches to this problem.

The nature of airflow around a human form has been studied extensively (Kim and Flynn 1991a,b; Chang 1993, 1994). Kim and Flynn (1991a) demonstrated that airflow around a human form can be broken down into three zones:

- A downwash effect dominates airflow above the chest
- A combination of downwash and vortex shedding characterizes airflow from the chest to the elbows.
- Vortex shedding governs the airflow from the waist to the hip.

The authors caution that motion of the human body, surface irregularities such as clothing, and free-stream disturbances will distort their findings. However, at this time, the Kim and Flynn model is the best published approach to modeling airflow around a human form.

Contaminant removal in the area above the chest can be characterized by using turbulent diffusion theory. As a result, the equations related to the phenomenon of turbulent diffusion govern contaminant removal from this region of the body. To date, no research effort has applied this theory to airflow around a human figure. However, the flow patterns described by Flynn and Kim (1991a) appear to resemble flow patterns observed in work related to flow around bluff bodies (Humphries and Vincent 1976a,b,c, Carmody 1964, Vincent 1978, MacLennan and Vincent 1982, Pal 1985, Neish and Smith 1992, and Zhou and Antonia 1993). The study of turbulent diffusion theory relative to separation around a bluff body when it is immersed in a free stream may provide insight to the problem of modeling a human subject located in a free stream.

The work of Humphries and Vincent (1976a,b) focused on flow separation around an axisymmetric disk (see Figure 2). In this figure turbulent diffusion in terms of the kinetic energy and mixing length of the turbulent motion is described.

Using dimensional analysis, transport in the wake region can be defined by the Equation (9):

$$\Phi\left(\frac{Ut_d}{D}, \frac{DU}{\nu}, \frac{l_f}{D}, \frac{K_f}{U^2}, \frac{D_b}{\nu}, \frac{Df}{U}\right) \tag{9}$$

where

U = mainstream air velocity
t_d = detention time

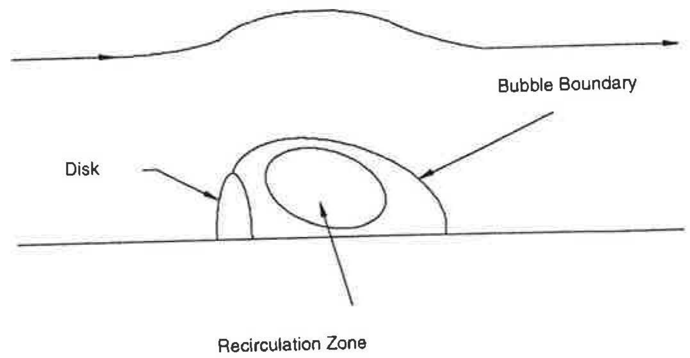

Figure 2. Axisymmetric flow around a disk

D = disk diameter
k_f = characteristic energy of free-stream turbulence
l_f = characteristic mixing length of free-stream turbulence
ν = molecular kinematic viscosity of air
D_B = molecular diffusion coefficient for scalar
f = frequency of vortex shedding
$Ut_d/D = H$, which is the dimensionless number describing the detention time of a scalar trapped in a bubble
DU/ν = disk Reynolds number (Re)
t_f/D = dimensionless mixing length
K_f/u^2 = dimensionless turbulent kinetic energy
D_b/ν = molecular diffusion
Df/U = Strouhal number (s)

Next, molecular diffusion and vortex shedding are assumed to be negligible.

Rearranging yields:

$$\underline{H} = \Phi_1\left(\text{Re}, \frac{l_f}{D}, \frac{k_f}{U^2}\right) \tag{10}$$

$$\Phi\left(\underline{H}, \text{Re}, \frac{l_f}{D}, \frac{k_f}{U^2}\right) = 0 \tag{11}$$

The continuity equation governs transport of the contaminant in the wake:

$$\frac{\partial \bar{c}}{\partial t} + \nabla \cdot (\overline{uc}) = \nabla \cdot (\overline{\eta_B \nabla_c}) \cdot \nabla \cdot (\overline{u'c'}) \tag{12}$$

where

c = concentration of contaminant in bubble boundary
t = time
u = fluid velocity in the streamwise direction
u' = fluid velocity fluctuation
c' fluctuation of contaminant concentration

Humphries and Vincent (1976a,b), by arguing that turbulent motion in the shear layers is in approximate equilibrium with the mean flow along the streamlines, considered the convective and diffusional terms in Equation (12) to be negligible. Thus, the actual flux by turbulent diffusion is:

$$j = -\overline{u'c'} \tag{13}$$

Using a two-flux model developed by Launder and Spalding (1972), Humphries and Vincent (1976a,b) then defined the local net flux across the bubble boundary as:

$$j_y = -l_f \frac{\partial}{\partial y}[(\overline{u_y^2})^{1/2}\bar{c}] \tag{14}$$

which leads to:

$$j_y = -(2b)^{1/2} l_f \left(k_f \frac{\partial \bar{c}}{\partial y} + \bar{c}\frac{\partial k_f^{1/2}}{\partial y}\right) \tag{15}$$

where b is the fraction of the total kinetic energy of the turbulence.

At steady state, if the source is removed at some time, $t = 0$, the bubble will empty at a rate:

$$\frac{dN}{dt} = \int j_y dA \tag{16}$$

where N is the number of moles of contaminant.

Inserting Equation (15) into Equation (16) creates:

$$\frac{dN}{dt} = -\int_A \left(2bk_f^{1/2} l_f \frac{\partial \bar{c}}{\partial y}\right) dA \tag{17}$$

where

$$\bar{c}\frac{\partial k_f^{1/2}}{\partial y} \ll k_f^{1/2}\frac{\partial \bar{c}}{\partial y} \tag{18}$$

Equation (18) is based on Carmody's (1964) experimental data, which suggest that the turbulent kinetic energy is a maximum over most of the bubble surface.

Equation (17) is simplified by assuming the functional form for the concentration gradient:

$$\frac{dN}{dt} = -l_f(2bk_f)^{1/2}\frac{\bar{C}_t}{\delta}A \tag{19}$$

where δ is a length scale indicating the mean distance over the turbulent mixing layer across which the concentration gradient drops to zero and C_t is the mean spatial concentration inside the bubble at time t.

Using the above analysis, Humphries and Vincent (1976b) defined the base pressure coefficient C_{pb} as:

$$C_{pb} \equiv \frac{p_b - p_s}{\frac{1}{2}\rho U^2} \tag{20}$$

where p_b is the base pressure on the rear face of the plate and p_s is the free-stream static pressure.

Next, they quantified the detention time H as:

$$H = \frac{f_1 f_2}{\left[\frac{l_c k_c^{1/2}}{DU}\right]} \tag{21}$$

where k_c and l_c are characteristic wake values for turbulent energy and length scale respectively. The functional relationships f_1 and f_2 were determined experimentally. The detention time depends on the free-stream turbulence parameter Λ (where $\Lambda = l_f k_f^{1/2}/DU$) and provides a means of describing and scaling transport properties of a fluid in the near wake regions of bluff bodies.

Humphries and Vincent (1976a,b) used flow visualization techniques to learn the functional dependence of H. Their experiments determined the following:

- The fraction of turbulent energy carried across the bubble boundary is approximately 0.5
- The ratio of the characteristic mixing length of the fluid to the diameter of the disk is constant within the range $0.01 \le l_c/D \le 0.10$
- The length scale over which the concentration of particles drops to zero across the bubble boundary has as its lower limit the length scale of the turbulent motion of the particles and as its upper limit the width of the mixing layer
- The free stream turbulence parameter L controls H, the base pressure coefficient, bubble length parameter, and bubble shape parameter.

This work was extended to account for free-stream turbulence effects (Humphries and Vincent 1976c). They concluded that as the level of free-stream turbulence increases, H decreases while the base pressure coefficient becomes more negative.

Detention time offers insight into transport within the wake and, thus, contaminant concentration in this region. Sadly, the author found no work that extended Humphries and Vincent's efforts to other shapes or systems. This promising research may provide the basis for future work, developing simple mathematical models that quantify contaminant transport around obstacles.

Vincent and MacLennan (1982) evaluated near wake properties of rectangular flat plates and discovered that these shapes are dominated by the periodic shedding of vortices. Turbulent diffusion is only a minor secondary influence and free-stream turbulence does not significantly influence fluid flow. These results suggest that the airflow patterns from the waist to hip around a person, as depicted in the Kim and Flynn model (1991a), are similar to those around rectangular flat plates.

Work in the aeronautical field (Pal 1985) has produced studies on wake properties of flat plates. Although this work was performed at Reynolds numbers exceeding those typically found in ventilation systems, it may shed insight into the behavior of wakes and their impact on ventilation system effectiveness.

Neish and Smith (1992) considered the effects of free-stream turbulence on flow separation around a circular cylinder and discovered that, as free-stream turbulence increases, the scale of separation is reduced until it becomes nearly insignificant and is confined to the trailing edge.

Zhou and Antonia (1993) studied turbulent vortices in the near wake of a cylinder immersed in a low turbulent free stream. They concluded that the vorticity and circumferential velocity could be modeled using the Oseen vortex. The circumferential velocity v_θ can be assumed to be:

$$v_\theta = \frac{\Gamma_0}{2\pi r}\left[1 - \exp\left(-1.26\frac{r^2}{r_1^2}\right)\right] \tag{22}$$

where

r = vortex radius
r_1 = 0.52 D_c
G_0 = 1.5UD_c by experiment
D_c = cylinder diameter

Although this work did not discuss mass transport phenomena, it offers insight into near wake behavior that can be linked to future work.

Some recent work has begun to apply the above analysis to airflow separation around the human form in ventilation systems. Kim and Flynn (1991b) simplified the mathematics of airflow around a human standing in front of a local exhaust hood by applying a simple model of vortex shedding around an elliptical cylinder to model the periodic nature of vortex shedding from the waist to hip of a human form. Their work assumed the following:

- The human subject can be represented by a long circular cylinder
- The whole body is subject to the vortex shedding process
- Vortex shedding is the principle contaminant removal process
- The area of a vortex covers one-half the formation region

The first two assumptions are based on their previous flow visualization work. The third assumption was tested using flow visualization techniques. Kim and Flynn (1991b) determined that vortex frequencies are consistent at the levels of the waist and hip. As a result, they concluded that transport in this region is dominated by vortex shedding. In turn, they devised a mathematical model to predict breathing zone concentration as a function of vortex shedding. Their approach was as follows:

- The spatially and temporally averaged concentration in the zone was:

$$C_{st} = Q_s/Q_v \tag{23}$$

where C_{st} is the steady-state contaminant concentration, Q_s is the flow of contaminant into the zone, and Q_v is the air-contaminant mixture flow out of the zone.
- Assuming the vortex area covers one-half the formation region the vortex volume V is:

$$V = (\pi/8)D^2H \tag{24}$$

- By the previous assumptions, the frequency with which a vortex is shed from one side of the human subject is:

$$f = SU/D_c \tag{25}$$

where S is the Strouhal number, U is the free-stream velocity, and D_c is the cylinder diameter.

By hot-film anemometry and flow visualization techniques using a mannequin, they determined that the Strouhal number between the chest and hip (at approximately 0.3H, where H is the height of the mannequin) is 0.19. In addition, the area of each vortex is 0.7 times the area of a circle with the same diameter of the mannequin at the chest. Therefore,

$$V = 0.7(\pi D_c^2/4) \tag{26}$$

and

$$C_{st} = 50Q_s/(\pi UHD_c) \tag{27}$$

where V is the revised vortex volume.

The assumption is that vorticity shedding dominates the transport process and simplifies the mathematical approach to the problem of contaminant transport in the wake area. In addition, based on the previous work of others, Flynn and Miller (1991) assumed that the coherent vortex structures created in the wake region are similar for turbulent and laminar flow, thereby, eliminating turbulence effects. As a result, the Navier-Stokes Equation can be simplified to the vorticity transport equation by making the additional assumption of incompressible flow (Appendix A).

Conditions at the boundary of an object immersed in the free stream are:

$$u \cdot \bar{n} = 0 \tag{28}$$

$$u \cdot \bar{\tau} = 0 \tag{29}$$

where $\bar{n}$ is the outward normal and $\bar{\tau}$ is the counterclockwise unit tangential.

In addition, conditions at infinity are:

$$u(\infty, y, t) = u_0 \tag{30}$$

$$u(x, \infty, t) = u_0 \tag{31}$$

Using a Lagrangian numerical technique known as the Discrete Vortex Method (DVM), Flynn and Miller modeled flow around an elliptical cylinder (which represented a human form) by applying Equations (28) through (31) and were able to determine the frequency of vortex shedding. Equations (23) through (27) were then used to determine the concentration in the breathing zone. The DVM method (Chorin 1973, 1978, Cheer 1989) evaluates separated flows using time increments as opposed to grids such as the numerical methods based on k-ε.

Dunnett (1994) used a CFD approach based on the finite element method to model vortex shedding around an elliptical cylinder placed in a uniform free-stream in front of an exhaust opening. Using a k-ε model in two dimensions to provide closure to the Reynolds-averaged turbulent Navier-Stokes equation, Dunnet evaluated several configurations using the following boundary conditions:

- Flow enters the upstream boundary in the negative y direction with a constant velocity u_0
- On the walls of the room and on the surface of the elliptical cylinder, fluid velocity is zero
- Flow is drawn through the opening at a velocity u_1

The significant result of Dunnett's work was the conclusion that when a human subject is placed near an exhaust opening the flow field is steady and a small zone of recirculation exists in the human subject's wake. On the other hand, as the human subject is moved away from the opening, the flow becomes unsteady and vortex shedding predominates. This was an advance to previous work, and demonstrated that turbulence models can be used to model local exhaust hood flows. However, due to the limitations of computer memory, the computational requirements associated with this Eulerian approach are still restrictive. The Lagrangian approach offered by DVM is still the more viable option. Future technological advances may change this.

The assumption that vortex shedding is the dominant transport mechanism is based on the suppositions that the contaminant source will be located within the zone of vortex formation and that it will be situated between the waist and hips. This assumption is easily violated be real-world conditions. For example, the formation zone may not extend to the source due to free-stream conditions or, perhaps, the source is placed outside the waist and hip region because the hood design assumed ergonomics based on unrealistic human subject height. In either event, the turbulent effects located in the downwash regions described by Kim and Flynn (1991a) must be considered.

Boundary layer separation at and within a hood has not been studied as extensively as boundary layer separation around obstacles external to the hood. Ozdemir et al. (1993), however, presented on excellent review on flow structures created inside and enclosing hood, such as a laboratory fume hood. Using flow visualization and measurements they mapped the flow patterns and scalar transport with the hood structure. They observed that the flow field generated by the hood created a span-wise vortex at the upper front of the hood. They noted that, in the absence of external challenge to the hood, leakage from the hood occurred at periodic bursts at the edge of the vortex. Cross drafts resulted in random bursts and asymmetrical leakage from the hood.

Recent effort by Kulmala (1994, 1995) has applied computational fluid mechanics to the characterization of airflow into local exhaust hoods. In both studies, Kulmala employed a k-ε turbulence model. In the first study, airflow patterns were simulated for an enclosing hood with a dedicated air supply. Calculations were performed using the SIMPLE algorithm. A non-uniform grid was used that was finer at the exhaust inlet and supply air opening. Kulmala assumed constant velocity values at the air inlet, the exhaust opening and the top horizontal plane. The k-ε model assumes isotropic turbulence. Kulmala used a 10% turbulence intensity for this model. The model was validated using a tracer gas method. Kulmala found good agreement between the measured and calculated values. This study provided the following findings:

- A filter at the supply air opening reduces entrainment of room air with hood supply air
- Low turbulence intensity in the supply air reduces the required air velocity to provide clean air

In Kulmala's recently published work, the FLUENT computer code was used to apply the k-ε turbulence model to unflanged rectangular openings. The objective of this work was to obtain velocity profiles similar to those obtained by DallaValle for hoods with four different aspect ratios (1:1, 4:3, 2:1, 3:1). Results of this effort suggest that a grid-independent solution is achievable. However, the numerical predictions underestimated the centerline velocities close to the exhaust opening and overestimated the velocities at greater distances. The maximum difference occurred at the same dimensionless point for all four hoods. The difference increased with respect to increases in aspect ratio. Kulmala could not isolate the cause of these deviations.

As Kulmala discusses, the major weakness of the k-ε turbulence model is that it assumes isotropic turbulence. This assumption is easily defeated by crossdrafts and the effects of molecular diffusion. In addition, the release rate of the contaminant will also obviate this assumption. Although computer hardware limits the use of other simulation programs, research in other scientific areas suggests that more accurate modeling will be available in the future. For example, a paper by Murakami et al. (1996) compared the use of large eddy simulation (LES), the differential stress model (DSM), the algebraic stress model (ASM), and the k-ε model, concerning modeling airflow around buildings. This study demonstrated that the LES model produced the most reliable results in terms of velocity distributions, mean surface pressure, and turbulence energy k. The k-ε turbulence model was highly unreliable. The study authors attribute this result to the assumption of isotropic turbulence. This work may shed light on where future work in the area of modeling local exhaust hoods should be focused.

CONCLUDING REMARKS

Empirically derived velocity contours continue to be the standard approach to the design of ventilation systems. In fact, recent research (Bellia et al. 1996) has generated new equations for velocity contours at a hood using linear regression methods. Despite the increasing use of numerical techniques the authors justified their approach by claiming that these methods are not easy to use. Nevertheless, the current practice of sizing exhaust hoods based on velocity contours ignores important influences that affect exhaust hood performance. These influences include the presence of crossdrafts, room air turbulence, and flow separation around objects within the area of a hood's influence. The turbulent structures created by these phenomena must be considered by a ventilation system designer before the specification of a design.

As seen above, many methods can be used to evaluate these influences. Appendix B summarizes these techniques and their deficiencies. Interestingly, the matrix suggests that CFD, although limited, in some cases, by simplifying assumptions, is the most diverse tool available to the engineer to accurately model the phenomena associated with ventilation systems. Although this technique has witnessed increasing acceptance in the research world, it has not received even modest notice among the ventilation design world. Nevertheless, the march of computer technology suggests that within a few years the most rigorous CFD methods will be available to the ventilation system designer for validation of design before construction.

In conclusion, it is suggested that, although research in the area of ventilation design tools has progressed without a clear focus on their final application, the ventilation design engineer is at the cusp of a new paradigm. Within a decade designers should be able to validate the transport aspects of their designs prior to construction, without the need for prototypes. If that is the case, the designer would no longer be asked to achieve a specified capture velocity. Instead, a level of contaminant concentration in the room or at the breathing zone of a human subject could be specified, and the designer would be able to use the computational tools available to numerically confirm the correctness of the design.

This potential suggests that the professional bodies in charge of ventilation system design should push research toward the simplification of CFD tools. To begin working towards this, continuing education programs could be established to prepare experienced designers for the transition from a manual design approach to an automated one. Universities could also include appropriate training for future engineers in this regard.

APPENDIX A.

Mathematical Derivation of Vorticity Transport Equation

The governing equations for vorticity transport are as follows:

Continuity:

$$\nabla \cdot u = 0 \tag{32}$$

Vorticity Transport Equation:

$$\frac{\partial \omega}{\partial t} + (\boldsymbol{u} \cdot \nabla)\omega = \nu \nabla^2 \omega \tag{33}$$

where $\boldsymbol{u}$ = velocity vector in x-direction, t = time, and ω = vorticity vector, and where

$$\omega = \frac{\partial v}{\partial x} - \frac{\partial u}{\partial y} \tag{34}$$

Equation (33) is derived by taking the curl of the momentum equation (Chung 1988, Brodkey 1968):

$$\nabla \times \left(\frac{\partial \boldsymbol{u}}{\partial t} + (\boldsymbol{u} \cdot \nabla)\boldsymbol{u} - \boldsymbol{F} + \nabla p - \nu\left[\nabla^2 \boldsymbol{u} + \frac{1}{3}\nabla(\nabla \cdot \boldsymbol{u})\right]\right) = 0 \tag{35}$$

where $\boldsymbol{F}$ is the body force and p the pressure deviation from hydrostatic conditions.

The following vector identities allow Equation (35) to be reduced. First,

$$(\boldsymbol{u} \cdot \nabla)\boldsymbol{u} = \nabla\left[\left(\frac{1}{2}\right)\boldsymbol{u} \cdot \boldsymbol{u}\right] - \boldsymbol{u} \times \omega \tag{36}$$

In addition,

$$\nabla \times (\boldsymbol{u} \times \omega) = \omega \cdot (\nabla \boldsymbol{u}) - \boldsymbol{u} \cdot (\nabla \cdot \omega) + \boldsymbol{u}(\nabla \cdot \omega) - \omega(\nabla \cdot \boldsymbol{u}) \tag{37}$$

where, for incompressible flow

$$\omega \cdot (\nabla \boldsymbol{u}) = 0 \tag{38}$$

For two dimensional flow,

$$\boldsymbol{u}(\nabla \cdot \omega) = 0 \tag{39}$$

Therefore,

$$\nabla \times (\boldsymbol{u} \times \omega) = \boldsymbol{u}(\nabla \cdot \omega) = (\boldsymbol{u} \cdot \nabla)\omega \tag{40}$$

Combining Equations (34) through (40) reduces the momentum equation to the vorticity transport equation (33).

APPENDIX B

Summary of Modeling Techniques

Modeling Tool	Advantages	Deficiencies
	Velocity Contours Created by a Hood	
Empirically derived models	• Simple to use • Accurate for ideal environments	• Ignores turbulence effects • Ignores molecular and turbulent diffusion • Assumes inviscid, incompressible, and irrotational flow
Potential theory	• Allows creation of 3-D computer generated models for various hood configurations • Can model contaminant transport using simplifying assumptions • Vector addition also allows consideration of effects of crossdraft	• Ignores molecular and turbulent diffusion
CFD	• Can incorporate effects of turbulent structures on hood-generated airflows	• Some techniques cannot model nonisotropic turbulence and those that can model such systems have too large a computational requirement for most personal computers • Most designers do not have expertise to use CFD packages
	Flow Separation Around Objects	
Detention time *H* [as defined by Humphries and Vincent (1976)]	• Provides simple mathematical expression to determine contaminant transport in wake of object • Considers turbulence effects • Can be applied to the modeling of ventilation systems • Provides an understanding of contaminant transport by a particular turbulent structure	• Ignores molecular diffusion
Vortex shedding model (Use of Discrete Vortex Method to numerically predict vortex shedding from cylindrical object.)	• Lagrangian approach of model (as opposed to Eulerian approach such as a CFD package) can be handled by personal computer	• Assumes that contaminant is released in very narrow area of human body • Cannot consider turbulent, molecular, or convective diffusion
CFD models	• See above	• See above
	Flow Separation Created by Hood Geometry	
Flow visualization studies (Have provided some rules-of- thumb for designers of hoods.)	• Designers now have simple rules to generate hood designs	• These rules do not provide a means of modeling systems prior to construction

Summary of Modeling Techniques

Modeling Tool	Advantages	Deficiencies
CFD models (Have been applied to enclosing hoods, such as work by Kulmala.)	• See above	• See above
	Cross Drafts	
Potential flow theory	• Provides ideal picture of effects of crossdraft on flow field generated by a hood	• Cannot model effects of turbulent structures on flow field
CFD models	• See above	• See above
	Free-Stream Turbulence	
Detention time H (Can be used with Humphries and Vincent's analysis to determine effects of free-stream effects on wake effects.)	• See above	• See above
CFD models use free-stream turbulence as boundary condition	• See above	• See above

NOMENCLATURE

a = hood radius
A = surface area of bubble
b = fraction of total kinetic energy of turbulence
c' = concentration fluctuation
C_{st} = steady-state contaminant concentration
C_t = mean spatial concentration inside bubble at time t
D = diameter
D_B = molecular diffusion coefficient for scalar
D_c = cylinder diameter, diameter of mannequin's chest.
f = frequency of vortex shedding
F = body force
H = worker/cylinder height
$\underline{H}$ = dimensionless detention time
j = flux
k_c = characteristic wake value for turbulent energy
k_f = characteristic energy of free-stream turbulence
l_c = characteristic wake value for length scale
l_f = characteristic mixing length of free-stream turbulence
n = outward normal
N = number of moles of contaminant
p = pressure deviation from hydrostatic conditions
p_b = base pressure on rear face of plate
p_s = free-stream static pressure
Q = hood airflow
Q_s = flow of contaminant into zone
Q_v = air contaminant mixture flow out of zone
r = radial distance perpendicular to z axis in any direction
S = Strouhal number
t = counterclockwise unit tangential
t_d = detention time
u' = velocity fluctuation
u = velocity vector
U = free stream velocity
u = velocity scalar
V = revised vortex volume
V_T = total theoretical flow at given point
v_θ = circumferential velocity
x,y,z = coordinates
z = axis corresponding to hood centerline perpendicular to hood face
Γ_0 = vortex strength
Λ = free-stream turbulence
δ = length scale indicating mean distance over turbulent mixing layer across which concentration gradient drops to zero
Ψ = stream function
Φ = velocity potential.
ε = eccentricity, ratio of hood diameter to sum of distances from corners of hood to point being evaluated

ν = kinematic viscosity
ω = vorticity vector
overbar over symbol indicates time-mean value of variable
tilde over symbol indicates vector quantity

REFERENCES

Alden, J.L., and J. M. Kane. 1982. *Design of Industrial Ventilation Systems*. New York: Industrial Press.

American Conference of Governmental Industrial Hygienists (ACGIH). 1995. *Industrial Ventilation: A Manual of Recommended Practice*, 22nd Edition. Cincinnati, OH: ACGIH

Bellia, L., V. Betta, and F. Cascetta. 1996. New Empirical Equations for the Calculation of Centerline Air Velocity in Front of Exhaust Openings. *ASHRAE Transactions* 102(1): 230-238.

Brodkey, R.S. 1968. *The Phenomena of Fluid Motions*. Reading, MA: Addison-Wesley.

Carmody, T. 1964. Establishment of the Wake behind a Disc. *Trans. of ASME Paper* 64-FE-3.

Chang, S. KW. 1994. Air Velocity Profiles around a Person Standing in front of Exhaust Hoods. *ASHRAE Transactions* 100(2): 439-447.

Chang, S. KW., and R.R. Gonzalez. 1993. Air Velocity Profiles around the Human Body. *ASHRAE Transactions* 99 (1): 450-458.

Cheer, A.Y. 1989. Unsteady Separated Wake Behind an Impulsively Started Cylinder in Slightly Viscous Fluid. *J. Fluid Mechanics* 201: 485-505.

Chorin, A.J. 1978. Vortex Sheet Approximation of Boundary Layers. *J. of Comp.Physics* 27: 428-442.

Chorin, A.J. 1973. Numerical Study of Slightly Viscous Flow. *J. Fluid Mechanics* 57: 785-796.

Chung, T.J. 1988. *Continuum Mechanics*. Englewood Cliffs, NJ: Prentice Hall.

Conroy, L.M., M.J. Ellenbecker, and M.R. Flynn. 1988. Prediction and Measurement of Velocity into Flanged Slot Hoods. *Am. Ind. Hyg. Assoc. J.* 49: 226-234.

Conroy, L.M., and M.J. Ellenbecker. 1989. Capture Efficiency of Flanged Slot Hoods Under the Influence of a Uniform Cross Draft: Model Development and Validation. *Appl. Ind. Hyg.* 4: 135-142.

DallaValle, J.M. 1952. *Exhaust Hoods*. New York: Industrial Press.

DallaValle, J.M., and T. Hatch. 1932. Studies in the Design of Local Exhaust Hoods. *Transactions of ASME* 54: 31-37.

Dunnett, S.J. 1994. A Numerical Investigation into the Flow Field around a Worker Positioned by an Exhaust Opening. *Ann. Occup. Hyg.* 38: 663-686.

Esmen, N.A., T.A. Grauel, D.A. Weyel, and A.G. Ilori. 1991. The Effect of Turbulence on Exhaust Hood Capture Efficiency, In *Proc. of Ventilation '91*. American Conference of Governmental Industrial Hygienists, Cincinnati, OH, pp. 321-327.

Esmen, N.A, D.A. Weyel, and F.P. McGuigan. 1986. Aerodynamic Properties of Exhaust Hoods. *Am. Ind. Hyg. Assoc. J.* 47: 448-454.

Flynn, M.R. and M.J. Ellenbecker. 1985. The Potential Flow Solution for Air Flow into a Flanged Circular Hood. *Am. Ind. Hyg. Assoc. J.* 46: 318-322.

Flynn, M.R. and M.J. Ellenbecker. 1987. Empirical Validation of Theoretical Velocity Fields into Flanged Circular Hoods. *Am. Ind. Hyg. Assoc. J.* 48: 380-389.

Flynn, M.R., and C.T. Miller. 1989. The Boundary Integral Equation Method (BIEM) for Modeling Local Exhaust Hood Flows. *Am. Ind. Hyg. Assoc. J.* 50: 281-288.

Flynn, M.R., and C.T. Miller. 1991. Discrete Vortex Methods for the Simulation of Boundary Layer Separation Effects on Worker Exposure. *Ann. Occup. Hyg.* 35: 35-50.

George, D.K., M.R. Flynn, and R. Goodman. August 1990. The Impact of Boundary Layer Separation on Local Exhaust Design and Worker Exposure. *Appl. Occup. Environ. Hyg.* 5: 501-509.

Gerhardt, P.M., and R.J. Gross. 1985. *Fundamentals of Fluid Mechanics*. Reading, MA: Addison-Wesley.

Hemeon, W.C.L. 1963. *Plant and Process Ventilation*. New York: Industrial Press.

Humphries, W., and J.H. Vincent. 1976a. Experiments to Investigate Transport Processes in the Near Wakes of Disks in Turbulent Air Flow. *J. Fluid Mechanics* 75: 737-749.

Humphries, W., and J.H. Vincent. 1976b. An Experimental Investigation of the Detention of Airborne Smoke in the Wake Bubble Behind a Disk. *J. Fluid Mechanics* 73: 453-464.

Humphries, W., and J.H. Vincent. 1976c. Near Wake Properties of Axisymmetric Bluff Body Flows. *Appl. Sci. Res.* 32: 649-669.
Kim, T., and M.R. Flynn. 1991a. Airflow Pattern Around a Worker in a Uniform Freestream. *Am. Ind. Hyg. Assoc. J.* 52: 287-296.
Kim, T., and M.R. Flynn. 1991b. Modeling a Worker's Exposure From a Hand-Held Source in a Uniform Freestream. *Am. Ind. Hyg. Assoc. J.* 52: 458-463.
Kontomaris, K., and T.J Hanratty. 1993. Effect of Molecular Diffusivity on Turbulent Diffusion in Isotropic Turbulence. *Int. J. Heat Mass Trans.* 36: 1403-1412.
Kreyszig, E. 1993. *Advanced Engineering Mathematics*, 7th Ed. New York: Wiley.
Kulmala, I. 1994. Numerical Simulation of a Local Ventilation Unit. *Ann. Occup. Hyg.* 38: 337-349.
Kulmala, I. 1995. Numerical Simulation of Unflanged Rectangular Exhaust Openings. *Am. Ind. Hyg. Assoc. J.* 56: 1099-1106.
Launder, B.E., and D.B. Spalding. 1972. *Lectures in Mathematical Models of Turbulence*. New York: Academic Press.
MacLennan, A.S.M., and J.H. Vincent. 1982. Transport in the Near Aerodynamic Wakes of Flat Plates. *J. Flu id Mechanics* 120: 185-197.
Murakami, S., A. Mochida, R. Ooka, S. Kato, and S. Lizuka. 1996. Numerical Prediction of Flow Around a Building with Various Turbulence Models. *ASHRAE Transactions* 102(1).
Neish, A., and F.T. Smith. 1992. On Turbulent Separation in the Flow Past a Bluff Body. *J. Fluid Mechanics* 241: 443-467.
Nielsen, P.V., A. Restivo, and J.H. Whitelaw. 1978. The Velocity Characteristics of Ventilated Rooms. *J. of Fluids Engineering: Transactions of ASME* 100: 291-298.
Nielsen, P.V., A. Restivo, and J.H. Whitelaw. 1979. Buoyancy-Affected Flows in Ventilated Rooms. *Numerical Heat Transfer* 2: 115-127.
Ozdemir, I.B., J.H. Whitelaw, and A.F. Bicen. 1993. Flow Structures and their Relevance to Passive Scalar Transport in Fume Cupboards. In *Proc. Instn. Mech. Engnrs* 207: 103-115.
Pal, S. 1985. Freestream Turbulence Effects on Wake Properties of a Flat Plate at an Incidence. *AIAA Journal* 23: 1868-1871.
Roach, S.A. 1981. On the Role of Turbulent Diffusion in Ventilation. *Annals Occup. Hyg.* 24: 105-132.
Vincent, J.H. 1978. Scalar Transport in the Near Aerodynamic Wakes of Surface-Mounted Cubes. *Atmospheric Environment* 12: 1319-1322.
Zhou, Y., and R.A. Antonia. 1993. A Study of Turbulent Vortices in the Near Wake of a Cylinder. *J. Fluid Mechanics* 253: 643-661.

10963