An Analytical, Numerical, and Experimental Comparison of the Fluid Velocity in the Vicinity of an Open Tank with One and Two Lateral Exhaust Slot Hoods and a Uniform Crossdraft

L. M. CONROY†*, P. M. J. TREVELYAN‡ and D. B. INGHAM‡

†Environmental and Occupational Health Sciences, University of Illinois at Chicago, School of Public Health (MC 922), 2121 W. Taylor, Chicago, IL 60612, USA; ‡Department of Applied Mathematics, University of Leeds, Leeds LS2 9JT, UK

The objective of this research was to compare mathematical models of the fluid velocity in the vicinity of an open tank with lateral slot exhaust. Two approaches were explored: a numerical solution assuming turbulent flow and an analytical solution assuming potential flow. A numerical simulation of the flow field in and around an open surface tank was performed using the commercial software FLUENT®. An analytical solution was obtained using two-dimensional potential fluid flow determined using the Schwarz-Christoffel transformation and complex potential theory. The numerical and analytical solutions were compared with numerical solutions and experimental measurements published by others. The numerical solution using FLUENT and the two numerical solutions published by others appear to reflect experimental conditions with equal accuracy. In some regions, the FLUENT solution appears better while in other regions the other two solutions appear better. Differences in geometry and boundary conditions could explain these differences. Greater differences were observed between the FLUENT and CFX-F3D® solutions than between the EOL-2D® and CFX-F3D solutions. This was unexpected since the geometry, boundary conditions, and turbulence model were more similar in the former case than in the latter. The potential flow solution, while simpler and less computationally intensive than the numerical solutions, resulted in estimates of experimental velocity that were equally as good as those of the numerical solutions. The simplicity and conservative estimates of this model make it useful for estimating exhaust hood flow fields. © 2000 British Occupational Hygiene Society. Published by Elsevier Science Ltd. All rights reserved.

Keywords: local exhaust ventilation; computational fluid dynamics; open surface tanks

INTRODUCTION

Open surface tanks are widely used in industrial settings with typical uses including electroplating and vapour degreasing. Lateral exhaust, consisting of one or two slot hoods along the length of the tank, is often used to control emissions of hazardous materials from these tanks. The advantages of this type of exhaust are that the hood is located between the contaminant source and the worker’s breathing zone and the hood usually does not interfere with access to the tank. One disadvantage of this type of hood is that it is susceptible to low efficiency in the presence of crossdrafts and worker activity (Iyiegbuniwe, 1997).

This research examines the velocity field near an open surface tank with one and two lateral exhaust slot hoods and a uniform crossdraft. Two approaches were explored: a numerical solution assuming turbulent flow and an analytical solution...
assuming potential flow. The results of this research are compared with experimental data and a numerical solution published by Braconnier et al. (1993) and a numerical solution published by Saunders and Fletcher (1997).

METHODS

Turbulent flow

A numerical simulation of the flow field in and around an open surface tank was performed using the commercial software FLUENT® (Fluent, Inc., Lebanon, NH, USA). Figures 1 and 2 show the modelled domains for the tank with one exhaust slot and two exhaust slots, respectively. The domain was selected to allow comparison with results of other investigators (Braconnier et al., 1993; Saunders and Fletcher, 1997). A two-dimensional problem was defined and the $k$–$\varepsilon$ turbulence model with standard wall functions was used. The flow field was determined for a slot hood on the upstream side of the tank and with slot hoods on

**Fluent**

(a)

\[ \begin{align*}
H_1 &= 1.98833 \text{ m} \\
L &= 0.497 \text{ m} \\
H_2 &= 2.035 \text{ m} \\
d &= 0.297 \text{ m}
\end{align*} \]

slot opening = 0.035 m  
wall thickness = 0.01167 m

**Potential Flow**

(b)

\[ \begin{align*}
H_1 &= 1.98833 \text{ m} \\
L &= 0.497 \text{ m} \\
H_2 &= 2.035 \text{ m} \\
d &= \infty
\end{align*} \]

distance to sink = 1/2 slot opening + wall thickness = 0.02917 m

Fig. 1. Modelled domains for the case with one exhaust hood: (a) geometry used in FLUENT solution; (b) geometry used in potential flow solution.
both sides of the tank. The slot hoods were modelled as velocity inlets with a velocity magnitude of 4.658 m s\(^{-1}\) at each slot. The crossflow was equal to 0.48 m s\(^{-1}\) and the crossflow inlet velocity was determined by combining the mass flux for the crossflow and the slot hood(s). In the case with just the upstream slot hood, the inlet velocity was equal to 0.573 m s\(^{-1}\) and when two slot hoods were operating the inlet velocity at the upstream boundary was 0.644 m s\(^{-1}\). The outlet was defined in such a way as to assure mass balance with the inlets and zero normal gradient conditions. The flow was assumed to be isothermal.

Initially, the geometry was modelled exactly as in the comparison paper by Saunders and Fletcher (1997). A non-uniform grid was used with the slot section divided into three grid cells in the vertical direction. The rest of the grid was established such that the aspect ratios of the cells in front of the hood(s) were equal to 1.0 and the cell size increased toward the edges of the domain in such a way that subsequent cell

**Fluent**

(a)  
\[ H_1 = 1.98833 \text{ m} \]
\[ H_2 = 1.98833 \text{ m} \]
\[ L = 0.497 \text{ m} \]
\[ d = 0.297 \text{ m} \]
\[ 0.5 \text{ m} \quad 0.28 \text{ m} \quad 1.8256 \text{ m} \]

Slot opening = 0.035 m  
Wall thickness = 0.01167 m

**Potential Flow**

(b)  
\[ H_1 = 1.98833 \text{ m} \]
\[ H_2 = 1.98833 \text{ m} \]
\[ L = 0.497 \text{ m} \]
\[ d = \infty \]

Distance to sink = 1/2 slot opening + wall thickness = 0.02917 m

Fig. 2. Modelled domains for the case with two exhaust hoods: (a) geometry used in FLUENT solution; (b) geometry used in potential flow solution.
size ratios in both directions were between 0.94 and 1.1. In order to ensure that the boundary locations do not effect the flow field, the upstream and downstream boundaries were moved away from the slots in incremental steps. The final modelled domains are shown in Fig. 1 and 2.

Once the solution was independent of the location of the upstream and downstream boundaries, the grid density was adjusted to establish grid independence. The final model used a 395 x 146 node grid for the one-sided slot hood geometry and a 264 x 98 node grid for the two-sided slot hood geometry. Air density was assumed equal to 1.2025 kg m\(^{-3}\) and viscosity equal to 1.815 \times 10^{-5} \text{ kg m}^{-1} \text{s}^{-1}. Convergence criteria were set such that the normalized residuals for each parameter were less than 5 \times 10^{-6}. The program was run in segments of 1000–5000 iterations until subsequent segments produced streamline plots that were graphically indistinguishable and residual plots that were flat, or nearly flat, for all parameters. It was found that 26 000 and 25 000 iterations were necessary for the one- and two-sided slot models, respectively.

**Potential flow**

The open surface tank was modelled as shown in Fig. 1–3. One way to approximate this flow is to use two-dimensional potential fluid flow which is determined using the Schwarz–Christoffel transformation (Milne-Thomson, 1968) and complex potential theory (Kellogg, 1953). The region is simplified in such a way that all of the parameters associated with the Schwarz–Christoffel transformation are determined analytically. This is achieved

![Diagram](https://academic.oup.com/annweh/article-abstract/44/6/407/181845/1)
by considering the region to be an infinite channel with a semi-infinite sub-channel attached normally to the floor and with a height discontinuity in the infinite channel for the case with one exhaust slot hood. Flow through the inlet is formed by introducing a source at infinity on the left-hand side of the channel. For the one-sided exhaust case, the suction slot is modelled by a sink placed on the upstream wall of the tank located at the centre of the physical slot exhaust in Fig. 1 (0.02917 m below the floor). The two-sided exhaust case was modelled by placing a sink on both walls of the tank.

To calculate the potential flow inside the region denoted by $G_z$ (Fig. 3) we have chosen to map $G_z$ using the Schwarz–Christoffel transformation onto the region $G_\zeta$ which corresponds to the upper half of the complex $\zeta$ plane. The fluid flow in the region $G_z$ is then established and from this the stream function and fluid velocities in $G_z$ can readily be determined. The $x$, $y$, $\xi$, and $\eta$ axes are taken as the horizontal and vertical directions in the $z$ and $\zeta$ planes, respectively.

The Schwarz–Christoffel theorem states that if $\zeta_1$, $\zeta_2$, $\ldots$, $\zeta_n$ are $n$ points on the real axis in the $\zeta$ plane such that $\zeta_1 < \zeta_2 < \ldots < \zeta_n$ and $\theta_1$, $\theta_2$, $\ldots$, $\theta_n$ are the interior angles of a simple closed polygon of $n$ vertices in the $z$ plane, then the transformation from the $\zeta$ plane to the $z$ plane is defined by

$$\frac{dz}{d\zeta} = K(\zeta - \zeta_1)^{\theta_1/s-1}(\zeta - \zeta_2)^{\theta_2/s-1} \cdots (\zeta - \zeta_n)^{\theta_n/s-1}. $$

(1)

It should be noted that the constant $K$ may be complex, but throughout this paper it is taken to be real since no rotations of the planes are required. Each quantity $\zeta_i$ is real and is mapped onto the vertex in $G_\zeta$, where the interior angle is $\theta_i$. Without any loss of generality we can choose the values of two of these $\zeta_i$. 

In performing the mapping we create a region, $G_\zeta$, with all the correct angles at the corners but with different dimensions from those shown in Fig. 3, which has lengths $H_1$, $H_2$, and $L$. The points $A$ and $F$, at infinity in the $z$ plane, are mapped to $-\infty$ and $\infty$ in the $\zeta$ plane, respectively. The four corners in $G_z$, $B$, $C$, $D$, and $E$, are mapped to the points $-\zeta_B$, $\zeta_C$, $0$, and $1$ in $G_\zeta$, respectively. The interior angles at $C$ and $E$ are both $\pi/2$ and at $B$ and $D$ are both zero. Hence, we have

$$\frac{dz}{d\zeta} = f(\zeta) = \frac{K(\zeta + \zeta_C)^{1/2}(\zeta - 1)^{1/2}}{\zeta(\zeta + \zeta_B)} $$

(2)

which maps $G_z$ to $G_\zeta$, where $f$ is introduced to simplify the notation.

**Analytical parameters.** This problem has been modelled as an infinite channel with a semi-infinite channel attached normally to the floor. This is equivalent to three semi-infinite channels. For each semi-infinite channel an analytical global relationship can be derived which relates the width of the channel to the parameters. These are derived by integrating around semi-circles and letting the radius tend to zero or infinity in the $\zeta$ plane. We appeal to Cauchy’s theorem (Priestley, 1993) to deform the semi-circle to the arc in the $\zeta$ plane which has been mapped from a horizontal or vertical line in the $z$ plane. Since we have three semi-infinite channels, three analytical global relationships can be derived to solve for the three parameters $\zeta_B$, $\zeta_C$, and $K$.

A vertical line which is far to the right on the right-hand sub-channel in $G_\zeta$ is mapped to a large arc, $\Gamma_0$, in $G_\zeta$. Since singular points only occur along the real axis of the $\zeta$ plane, and given that the large arc avoids them, Cauchy’s theorem means that the integral around $\Gamma_0$ is equal to the integral around the semi-circular arc, $\Gamma_1$, which has the same end points.

The boundary of $\Gamma_1$ with radius $r_1$, is given by $\zeta = r_1 e^{i\theta} + \zeta_0$, where $\theta$ is contained in the interval $[0, \pi]$ and $\zeta_0$ is the value of $\zeta$ at the centre of the semi-circle. Hence the integral around the arc $\Gamma_0$ is given by

$$\int_{\Gamma_0} f(\zeta) d\zeta = \int_{\Gamma_1} f(\zeta) d\zeta = \int_{0}^{\infty} f(r_1 e^{i\theta} + \zeta_0) r_1 e^{i\theta} d\theta.$$

(3)

Letting $r_1$ tend to infinity reduces the integral around the arc to

$$\int_{0}^{\pi} iK d\theta = iK\pi.$$

(4)

However, letting $r_1$ tend to infinity is equivalent to moving the vertical line an infinite distance to the right. Clearly the height of the vertical line remains constant since the floor and ceiling are parallel. Thus we have

$$\int_{\Gamma_1} f(\zeta) d\zeta = \int_{0}^{\pi} dz = iH_2$$

(5)

and

$$H_2 = K\pi.$$  

(6)

Similarly, we can consider semi-circles around $\zeta = 0$ and $\zeta = -\zeta_B$ and on letting their radii tend to zero results in

$$\frac{K\pi}{L} = \frac{\zeta_B}{\zeta_C^{1/3}}$$

(7)

and

$$H_1 = \frac{H_2(\zeta_B - \zeta_C)^{1/2}(\zeta_B + 1)^{1/2}}{\zeta_B}.$$  

(8)
Equations (6)–(8) are solved to give
\[ K = \frac{H_2}{\pi}, \quad \xi_B = \left( \frac{H_2}{L} \right) \left( (s^2 + 1)^{1/2} - s \right), \]
\[ \xi_C = \left( (s^2 + 1)^{1/2} - s \right)^2, \]
where
\[ s = \frac{L^2 + H_1^2 - H_2^2}{2LH_2}. \] (10)

Integration techniques. Equation (2) is a first-order complex ordinary differential equation which has to be solved in order to map \( G_2 \) to \( G_\xi \). This equation cannot be integrated analytically and therefore a numerical approach is required. This equation is of the form
\[ \frac{d\xi}{d\zeta} = f(\zeta) \] (11)
which if solved numerically maps \( G_\xi \) to \( G_{2\xi} \); however, we require \( G_\xi \) to be mapped onto \( G_\xi \). Simply by inverting Eq. (11) ensures that, when solving numerically with \( \zeta \) chosen, the value of \( \zeta \) is found. Applying the Cauchy–Riemann equations we separate the first-order complex differential equation into one real and one imaginary first-order ordinary differential equation. This only allows integration to take place in the horizontal and vertical directions and so if integration along lines at an arbitrary angle is required then a rotation of the plane is used. In order to obtain accurate solutions of the governing equation, the numerical integration of this pair of first-order real differential equations was performed with the aid of the NAG Fortran library.

If a numerical parameter determination process is needed then the initial value NAG routine D02BGF can be employed which integrates a system of first-order differential equations, subject to suitable initial conditions, over an interval using a Runge–Kutta–Merson method until a specified variable attains a given value. However, to map the regions the routine D02CJF was preferred since this routine integrates a system of first-order differential equations, with suitable initial conditions, over a given range using a variable-order, variable-step, Adams method until a user-specified function of the solution is zero. It then returns the solution at the points in the range specified by the user.

Integral boundary condition. When the boundary conditions are given at singular points, namely \( C \) and \( E \), the NAG routines require values near the singular points which are obtained using expansions for \( \zeta \) in terms of \( z \). We defined the \( z \) plane’s origin at \( C \), however, this is appropriately redefined later for comparison with the numerical and experimental data. For example, the inversion of \( f \) is not defined at \( \zeta = 1 \), but the point \( E \), namely \( z = L + i(H_1 - H_2) \) has to be mapped onto the point \( \zeta = 1 \). An expansion of \( \zeta \) near \( \zeta = 1 \) in terms of \( z \) was considered in the following form:
\[ \zeta \simeq 1 + qZ^p \] (12)
where \( p \) and \( q \) are constants to be determined and \( Z = z - L - i(H_1 - H_2) \). Since \( \xi_B, \xi_C, K, p, \) and \( q \) are all independent of \( Z \), substituting Eq. (12) into Eq. (2), rearranging, and neglecting all higher-order terms in \( Z \) we obtain
\[ \zeta \simeq 1 + \left( \frac{9Z^2(1 + \xi_B)^2}{4K^2(1 + \xi_C)} \right)^{1/3}. \] (13)

Again, since \( G_\xi \) is in the upper half of the \( \zeta \) plane, when \( Z \) is of the form \( R e^{\theta} \) we take \( \theta \) in the interval \([0, 3\pi/2] \). Similarly for corner \( C \) we have
\[ \zeta \simeq -\xi_C + \left( \frac{9Z^2(\xi_B - \xi_C)^2}{4K^2(1 + \xi_C)} \right)^{1/3} e^{\pi/3}. \] (14)

Using D02CJF we integrate over a mesh in the \( z \) plane to map the region \( G_{2\xi} \) into the \( \zeta \) plane. In the region \( G_{2\xi} \) a sink of strength \( \mu \) is placed at \( \zeta = -\xi \), such that in \( G_\xi \) it is along \( CD \) located at the centre of the physical slot entrance, together with a source of strength \( \mu \) placed at \( \zeta = \xi \), and therefore the complex potential \( W_1 \), for the one-sided slot, is given by
\[ W_1(\zeta) = \frac{\mu}{\pi} \ln(\zeta + \xi_B) - \frac{\mu}{\pi} \ln(\zeta + \xi_C), \] (15)
and the complex potential \( W_2 \), for the two-sided slot, is given by
\[ W_2(\zeta) = \frac{\mu}{\pi} \ln(\zeta + \xi_B) - \frac{\mu}{\pi} \ln(\zeta + \xi_{1s}) - \frac{\mu}{\pi} \ln(\zeta + \xi_{2s}), \] (16)
where \( \xi_{1s} \) and \( -\xi_{2s} \) are the values of \( \zeta \) along the sides of the tank located at the centres of the physical upstream and downstream slot exhausts, respectively. Although we refer to the potential flow solution as an analytical solution, it should be noted that the parameters \( \xi_B, \xi_C, \) and \( \xi_{1s} \) are found numerically and the mapping of \( G_\xi \) to \( G_{2\xi} \) is performed numerically. The values of all the parameters are given in Table 1.

RESULTS

Figures 4 and 5 show streamlines resulting from the FLUENT output and the potential flow solution for the one- and two-sided slot hoods, respectively. Figures 4(a) and 5(a) show the entire
DISCUSSION

In the problem presented here, FLUENT was used to simulate the flow field for a number of different geometries. It was found that the distance to the outlet had the largest effect on the solution. Increasing the distance from the initial condition of 0.3976 to 0.7952 m and again to 1.8256 m resulted in different solutions but the differences became smaller with each progressive increase in the length. Slight differences were seen between placing the outlet at \( x = 0.7952 \) and 1.8256 m and these differences were only observed near the outlet \( (x = 0.7952 \) m). No differences were observed in the region around the hood or the tank. Extending the length to 4.7230 m also resulted in very small differences at the outlet \( (x = 1.8256 \) m), however, this geometry was very computer intensive. Extending the distance from the slot to the upstream boundary made no difference in the solution.

Figures 4 and 5 show a comparison of the FLUENT and potential flow solutions. Figures 4(a) and 5(a) show the entire region for the case with one exhaust slot and two exhaust slots, respectively. The streamlines are closest near the upper wall of the region and larger differences are seen near the tank and exhaust hood. Figure 4 shows one streamline which just enters the hood and this is referred to as the ‘dividing streamline’. It is the streamline with a stream function equal to 0 m² s⁻¹. The distance to the dividing streamline gives some indication of hood capture efficiency. In the absence of contaminant dispersion, contaminant released inside or outside the dividing streamline would escape capture. The FLUENT solution is predicting a greater distance to the dividing streamline than the potential flow solution and this means that the potential flow solution underestimates hood capture efficiency relative to the FLUENT solution. The distance from the centre of the hood opening to the potential flow predicted dividing streamline is 0.162 m or 33% of the total tank surface distance and the FLUENT solution predicts a distance of 0.208 m or 42% of the tank surface distance.

The potential flow solution is an analytical solution of a simplified flow field and assumes inviscid, irrotational, and incompressible flow. The geometry selected in this problem allowed for a simplified region and an analytical solution. If we had not simplified the region then the Newton-Raphson method (Perry and Chilton, 1973) with the Jacobian matrix approximated by a finite-difference Quotient matrix would have been required to iterate upon some initial guesses of the values of the parameters. This method was used to determine the flow field for a similar geometry, but with a bottom added to the tank, i.e. the tank was no longer a semi-infinite channel. The resulting flow field was virtually identical in the region of the exhaust hoods and therefore only the analytical solution is presented in this paper.

For the case when there are two exhaust hoods, both the potential flow solution and the FLUENT solution predict complete capture. That is, all the streamlines that penetrate into the tank, enter one of the two hoods (Fig. 5).

As seen in Fig. 4 and 5, the potential flow streamlines are always below those predicted by FLUENT. If the upstream boundary were at the

Table 1. Parameter values for potential flow solution

<table>
<thead>
<tr>
<th>Position parameter</th>
<th>Value for one exhaust slot</th>
<th>Value for two exhaust slots</th>
</tr>
</thead>
<tbody>
<tr>
<td>( A )</td>
<td>(-\infty)</td>
<td>(-\infty)</td>
</tr>
<tr>
<td>( B )</td>
<td>(-\xi_B = -3.97641)</td>
<td>(-\xi_B = -3.53179)</td>
</tr>
<tr>
<td>( C )</td>
<td>(\xi_C = -0.94312)</td>
<td>(\xi_C = -0.77934)</td>
</tr>
<tr>
<td>Sink near ( C )</td>
<td>(-0.69086)</td>
<td>(-0.56345)</td>
</tr>
<tr>
<td>( D )</td>
<td>0</td>
<td>0</td>
</tr>
<tr>
<td>Sink near ( E )</td>
<td>(-0.67041)</td>
<td>(-0.63291)</td>
</tr>
<tr>
<td>( L )</td>
<td>1</td>
<td>1.0</td>
</tr>
<tr>
<td>( F )</td>
<td>(-\infty)</td>
<td>(-\infty)</td>
</tr>
<tr>
<td>( K )</td>
<td>0.64776</td>
<td>0.63291</td>
</tr>
<tr>
<td>( \lambda )</td>
<td>(-0.16303)</td>
<td>(-0.16303)</td>
</tr>
<tr>
<td>( \mu )</td>
<td>1.13983</td>
<td>1.28046</td>
</tr>
<tr>
<td>( H_1 )</td>
<td>1.98833</td>
<td>1.98833</td>
</tr>
<tr>
<td>( H_2 )</td>
<td>2.035</td>
<td>1.98833</td>
</tr>
<tr>
<td>( L )</td>
<td>0.497</td>
<td>0.497</td>
</tr>
</tbody>
</table>
same location, we would expect to see the potential flow streamlines to be below the FLUENT streamlines in the bottom part of the flow field and above the FLUENT streamlines near the top. That is, we would expect the potential flow streamlines to be closer to the walls near the walls. This is not the case, because in the potential flow problem, the upstream inlet is at infinity, whereas the FLUENT inlet is as shown in Fig. 4 and 5. By definition, the FLUENT problem has a uniform inlet velocity at $x = 0 \text{ m}$ in Fig. 4 and 5 and the velocity profile at $x = 0 \text{ m}$ for the potential flow problem is not uniform, as it is already being influenced by the exhaust hood(s).

Many of the commercially available CFD software packages allow the user to solve complex fluid dynamics problems with limited experience in mathematics. However, the solutions are only as good as the problem definition. A critical part of the problem definition is establishing a grid for numerical solution. The solution should be grid independent, that is, the solution should not change with increasing grid density. The process of establishing grid independence is very time consuming. For the problem with one slot hood, a coarse grid was established and the problem was run to convergence. A medium grid was then established and the problem was rerun to convergence. The solutions were not

Fig. 4. Streamlines for the case with one exhaust hood: (a) entire region; (b) region of tank and exhaust hood. FLUENT streamlines are shown as solid lines; potential flow streamlines are shown as dashed lines.
the same, so a fine grid was established and rerun to convergence. Changes in problem geometry required that the process be repeated. The final solution for the case with one slot hood required a grid with $395 \times 146$ nodes and required approximately 90 h of computation on a Silicon Graphics workstation. The final solution for the case with two slot hoods required a grid with $264 \times 98$ nodes and required approximately 50 h of computation on the same computer. The potential flow solution required approximately the same amount of time to set up the problem, that is defining the geometry and boundary conditions in the model, however, the computation time for the two cases presented here was several minutes for each case. Therefore, the potential flow solution is much less computationally intensive. Changes to geometry or boundary conditions can be evaluated much more quickly. The simplicity and conservative estimates make this a useful model for estimating exhaust hood flow fields.

In the problem presented by Braconnier et al. (1993), the standard $k-e$ model was replaced with the 'low Reynolds number' turbulence model described by Jones and Launder (1973). The authors stated that this allowed them to extend the model to areas where viscous effects were larger and did not require the use of boundary-layer laws at nodes near the walls. However, there was a need for an increase in

![Streamlines for the case with two exhaust hoods: (a) entire region; (b) region of tank and exhaust hood. FLUENT streamlines are shown as solid lines; potential flow streamlines are shown as dashed lines.](https://academic.oup.com/annweh/article-abstract/44/6/407/181845)
the number of nodes near the walls. The flow was assumed to be two-dimensional, incompressible, and isothermal and the finite-difference method was used. A rectangular grid with non-uniform spacing and $99 \times 77$ nodes was used and the solution converged in approximately 1000 iterations with residual deviations less than $5 \times 10^{-6}$. The geometry of the model is shown in Fig. 8. The fluid velocity components on the upstream and upper boundary were calculated by adding the velocity vectors of a horizontal uniform crossflow and velocity vectors resulting from a line sink at the position of the exhaust slot. Velocity values were set at the upstream

![Graph](https://academic.oup.com/annweh/article-abstract/44/6/407/181845)

**Fig. 6.** Comparison of numerical solutions, potential flow solution, and experimental data for cross-section at $y = 0.297$ m and $x = 0.78$–1.277 m: (a) $u$-component of the fluid velocity; (b) $v$-component of the fluid velocity.
and upper boundaries and at the exhaust slot. The program calculated the velocity profile at the downstream boundary.

In the problem presented by Saunders and Fletcher (1997), the commercial software CFX-F3D was used. The geometry was modified from the problem posed by Braconnier et al. (1993), to allow for simplified boundary conditions (Fig. 8) and the upper boundary was assumed to be a plane of symmetry. A uniform horizontal velocity was assigned at the upstream boundary such that the mass flow at the upstream boundary was equal to the sum of

![Graph](https://example.com/graph.png)

Fig. 7. Comparison of numerical solutions, potential flow solution, and experimental data for cross-section at $x = 0.85$ m and $y = 0-0.83$ m: (a) $u$-component of the fluid velocity; (b) $v$-component of the fluid velocity.
the mass flows of the crossflow and the suction flow. The fluid velocity was set at the upstream boundary and exhaust slots. Approximately 12,000 grid cells were used and convergence achieved in about 1000 iterations.

In this paper, the geometry was modelled with a wall at the upper boundary (Fig. 1). A uniform horizontal velocity was assigned at the upstream boundary, such that the mass flow was equal to the sum of the crossflow and the suction flow. A uniform horizontal velocity was also assigned to the suction flow.

These differences in geometry and boundary conditions could explain the differences seen in Fig. 6 and 7. Figure 6(a) shows the \( u \)-component of fluid velocity versus distance, in the \( x \)-direction, across the surface of the tank \( (y = 0.297 \text{ m}) \). There is good agreement among the three numerical solutions and the potential flow solution. All the techniques appear to be equally good at predicting experimental results. The FLUENT solution seems best very close to the hood face \( (x = 0.81 \text{ m}) \), while the transition to low velocities \( (x = 0.81-0.97 \text{ m}) \) is better predicted by the other numerical solutions and the potential flow solution. The three numerical solutions are very close to each other at large distances from the hood face \( (x > 1 \text{ m}) \). Examining the \( v \)-component of fluid velocity in the same cross-section [Fig. 6(b)] shows all predictions to be about the same with respect to the experimental data. All the \( v \)-components of fluid velocity in this cross-section are very near zero and are, therefore, difficult to experimentally measure accurately. There is excellent agreement among the potential flow, EOL-2D, and CFX-F3D solutions. The FLUENT solution agrees less well especially near the hood face.

Figure 7(a) shows the \( u \)-component of fluid velocity versus distance in the \( y \)-direction at \( x = 0.85 \text{ m} \) \( (7 \text{ cm from the hood face}) \). In this case, velocities in the tank and near the hood face are predicted equally well by the three numerical solutions and the potential flow solution. At \( y = 0.3 \text{ m} \) (approximately equal to the tank surface), the FLUENT solution overestimates the experimental fluid velocity. There is excellent agreement among the other two numerical solutions and the potential flow solution in this area, although all of these solutions underestimate the experimental fluid velocity. Above the tank surface \( (y > 0.34 \text{ m}) \), the FLUENT and potential flow solutions overestimate the experimental fluid velocity. This may be due to the assumed wall thicknesses in these two models. No information was provided regarding the thickness of the interior wall, above the slot opening, for either the experiments or the EOL-2D or CFX-F3D models. In the FLUENT simulation, this thickness was 1.167 cm. If the wall thickness was smaller in the experiments and other simulations, the cross-sectional area above the tank would be greater, resulting in lower fluid velocities. Reducing the thickness in the FLUENT simulation resulted in slightly improved agreement between the FLUENT solution and the other two solutions. Replacing the upper wall boundary with a symmetry boundary made little difference in the solution, particularly near the exhaust openings. Figure 7(b) shows the \( v \)-component of the fluid velocity for the same cross-section. There is little difference among the solutions, except near the hood face \( (y = 0.3-0.45 \text{ m}) \), where the FLUENT simulation appears to be the best at predicting the experimental \( v \)-component of fluid velocity. The low fluid velocities near the tank bottom \( (y = 0.15-0.28 \text{ m}) \) are not predicted well by any of the solutions and this is probably due to measurement inaccuracies at low velocities.

In this analysis, the tank and slot hoods were modelled in two dimensions. Inherent in this modelling approach is the assumption that the tank and hoods are infinite in length. In actual practice, tanks and hoods would have a finite length. The ex-

Fig. 8. Modelled domains for the comparison papers: (a) EOL-2D (Braconnier et al., 1993); (b) CFX-F3D (Saunders and Fletcher, 1997).
perimental data used for comparison were collected along the centreline of a tank and hood with length equal to 1 m. This is similar to tanks and hoods that would typically be found in industrial settings. The two-dimensional assumption appears adequate for tanks of this size. One limitation of this model, however, is the crossdraft direction. In order to maintain a two-dimensional system, the crossdraft direction is limited to perpendicular to the hood face. Modelling other crossdraft directions would require use of a three-dimensional model.

CONCLUSIONS

Three different numerical approaches have been employed to simulate the velocity field around an open surface tank with rim exhaust. The three approaches appear to reflect experimental conditions with equal accuracy. In some regions, the FLUENT solution appears better while in other regions the other two solutions appear better. Differences in geometry and boundary conditions could explain these differences.

Greater differences were observed between the FLUENT and CFX-F3D solutions than between the EOL-2D and CFX-F3D solutions. This was unexpected since the geometry, boundary conditions, and turbulence model were more similar in the former case than in the latter.

The potential flow solution, while simpler and less computationally intensive than the numerical solutions, resulted in estimates of experimental velocity that were equally as good as those of the numerical solutions. The simplicity and conservative estimates of this model make it extremely useful for estimating exhaust hood flow fields.

Acknowledgements—Lorraine Conroy was supported by sabbatical leave from the University of Illinois at Chicago and by Senior International Fellowship number 1F06 TW02314-01 from the Fogarty International Center of the U.S. National Institutes of Health. Philip Trevelyan was supported by the Engineering and Physical Sciences Research Council number GR/K80792.

REFERENCES


APPENDIX

Symbols

\( z \) plane of symmetry in two-dimensional model \((x, y, z)\) coordinates

\( \zeta \) Schwarz–Christoffel transformed plane in two-dimensional model \((\xi, \eta, \zeta)\) coordinates

\( G_z \) modelled region (Fig. 3)

\( G_\zeta \) Schwarz–Christoffel transformation of \( G_z \) which corresponds to the upper half of the complex \( \zeta \) plane

\( x, y, \zeta, \eta, \theta_1, \theta_2, \ldots, \theta_n \) horizontal and vertical axes in the \( z \) and \( \zeta \) planes, respectively

\( H_1, H_2, L \) interior angles of a simple closed polygon of \( n \) vertices in the \( z \) plane

\( A, F \) widths of the three semi-infinite channels (Fig. 1–3)

\( B, C, D, E \) points at infinity in the \( z \) plane (Fig. 3)

\( -\xi_B, \xi_C, 0, 1 \) four corners in \( G_z \) (Fig. 3)

\( \Gamma_0 \) points mapped in \( G_z \), corresponding to \( B, C, D, \) and \( E \), respectively

\( \Gamma_1 \) mapped large arc in \( G_\zeta \) corresponding to a vertical line which is far to the right in the right-hand sub-channel in \( G_z \)

\( \lambda \) semi-circular arc which has the same end points as \( \Gamma_0 \) and radius \( r_1 \)

\( \mu \) sink strength in the region \( G_z \) placed at \( \zeta = -\xi_a \) such that in \( G_z \) it is along \( CD \) located at the centre of the physical slot entrance

\( \psi \) source strength

\( W_1, W_2 \) complex potential for the one-sided slot and two-sided slot, respectively

\( \zeta_{10}, \zeta_{2a} \) values of \( \zeta \) along the sides of the tank located at the centres of the physical upstream and downstream slot exhausts, respectively.