Numerical Modeling of Vertical Buoyant Jets Subjected to Lateral Confinement

Xiaohui Yan, S.M.ASCE; and Abdolmajid Mohammadian, Ph.D., P.Eng.

Abstract: The near-field flow and mixing properties of vertical buoyant jets subjected to lateral confinement are studied numerically for different cases, including different confinement indexes and jet densimetric Froude numbers. The performances of different turbulence models are investigated, such as the standard $k-\varepsilon$ turbulence model and buoyancy-modified $k-\varepsilon$ model. The modeled results are compared to previous and present experimental observations. The present paper confirms that the universally accepted model ($k-\varepsilon$ turbulence model) can be satisfactorily accurate, eliminating the need for an advanced modeling approach, as long as suitable modifications are performed. In contrast to previous studies, which used one single and constant value of $Pr$ and $Fr$ numbers, the present study links these two numbers to the $Fr$ number, which is more practical and can produce very good results. This study also makes it possible to roughly quantify the rate at which the jet concentration spread width grows and identify the location where impingement occurs, which enables engineers or researchers to perform a quick estimation of the evolution and profile of a laterally confined vertical buoyant jet. DOI: 10.1061/(ASCE)HY.1943-7900.0001307, © 2017 American Society of Civil Engineers.

Author keywords: Numerical modeling; Vertical buoyant jets; Lateral confinement; Turbulence models.

Introduction

Wastewater effluents that have a lower density than the ambient water body are often discharged into marine environments in the form of turbulent buoyant jets. Examples include municipal wastewater discharges and liquid wastes discharged from desalination plants (e.g., Gildeh et al. 2015b; Jones et al. 2007). The inadequate disposal of wastewater effluents can result in serious environmental issues in coastal areas (Zhang et al. 2016; Lattemann and Höpner 2008; Drani et al. 2011) so a clear understanding of the mixing processes of turbulent buoyant jets is necessary for sound outfall design and environmental impact assessment.

When a buoyant jet is discharged into a marine environment with a sufficient clearance from any boundary of the domain considered, the ambient fluid is entrained into the jet because of the shear stresses; this is known as a free buoyant jet. In practice, however, a submarine buoyant jet is often dynamically affected by boundaries, such as a jet from a submarine outfall pipeline that is laid in a dredged trench (Lee and Lee 1998).

As shown in Fig. 1, wastewater effluents with an initial velocity of $W_j$, a density of $\rho_j$, and a concentration of $C_j$ are discharged from a discharge port with a diameter of $D$. A vertical riser tube is used to protect the discharge port, and is of diameter of $D_r$, which typically ranges from 2 to 4 times of the diameter of the discharge port. The length of the riser tube, $H_r$, is equal to the height of the trench. In contrast to a free buoyant jet, the entrainment of ambient fluid into this type of jets can be significantly restricted due to the existence of the confinement. The confinement geometry can be represented by a parameter called the confinement index, the expression of which is

$$B = H_r/(D_r - D).$$

The confined jet has been a topic of significant research interest during the past several decades. Gosman and Simitovic (1986) performed measurements of the mixing between a primary gas flow in a rectangular-sectioned duct and a secondary flow discharged from an orifice in one wall. Considering various injection angles and rates, they measured the local concentration distributions, inlet velocity profiles, and the static pressure variations. Khoo et al. (1992) investigated the turbulence in the bulk-free region of a confined jet, which was discharged vertically upward towards the interface from a location deep below the surface. Based on the measurements, they calculated the spatially resolved integral length scales and proposed a relation for Eulerian-type length scales. Lee and Lee (1998) investigated the effects of lateral confinement on the initial dilution of vertical round buoyant jets. A wide range of jet flow characteristics (e.g., confinement geometries and jet densimetric Froude numbers) were taken into account, and the vertical variations of centerline dilution were observed using laser-induced fluorescence (LIF) techniques. Their work presents a comprehensive data set on laterally confined buoyant jets. Shinneeb et al. (2011) investigated the effects of vertical confinement on a round jet discharging horizontally from a vertical wall into a flat-bottomed, shallow layer of water. The mean velocity field and turbulence statistics were measured using the particle image velocimetry (PIV) technique, which is a quantitative visualization technique that can provide instantaneous global velocity fields. Their study demonstrated that the vertical confinement has a significant effect on the turbulence structure of shallow water jets.

Numerical simulations on confined jets have rarely been done, and require further investigation. Numerous researchers have developed and employed integral models for turbulent buoyant jets (Jones et al. 2007; Chu 1985), and Jirka (2007) developed a jet integral model, CorSurf, that includes the confinement effects of shallow water and/or lateral boundaries on the jet dynamics. The model has been validated using extensive high-resolution laboratory data and proved to be a convenient and efficient tool...
for surface discharge jet analysis. It is commonly accepted that, despite their usefulness and efficiency in describing turbulent buoyant jets, integral models do not adequately capture some observed mixing characteristics, such as dilution (Pincince and List 1973; El-Amin et al. 2010). In contrast, 3D numerical simulations based on the solutions of the Navier-Stokes equations can provide more reliable and detailed results for confined jets. El-Amin et al. (2010) analyzed a turbulent buoyant confined jet discharged into a cylindrical tank. The distributions of velocity, pressure, temperature, and turbulence were measured and modeled using the realizable $k$-$\varepsilon$ turbulence model. The modeled results matched the measurements fairly well. Gilageh et al. (2014) conducted a numerical study on the velocity and temperature fields of the thermal and saline wall jets. The cling length, plume trajectory, temperature dilutions, temperature, and velocity were simulated using different turbulence models. The simulated results showed good agreement with the recent experimental data, and two models performed best among the seven models chosen for the study.

The application of direct numerical simulation (DNS) for buoyant confined jets is not practical in the foreseeable future because of the heavy computational cost. Based on different methods of filtering, two classes of turbulence modeling can be distinguished: large eddy simulations (LES) and Reynolds-averaged Navier-Stokes (RANS) simulations (Van Mael and Merci 2006). In the LES, the large-scale motions are resolved directly, while small-scale motions are modeled. In the RANS simulations, a Reynolds-averaging procedure is used to filter out all turbulent fluctuations, and a turbulence model is employed to take into account the effect of turbulence on the mean flow field. The LES generally produces more accurate predictions but is more computationally expensive. As powerful computing resources have become more available in recent years, the LES is increasingly used. However, research on the practice of RANS simulations is still quite meaningful in the field of hydraulic engineering, especially as typically, computationally more efficient submodels can be developed. In RANS simulations, the $k$-$\varepsilon$ turbulence model is the most extensively used model due to its good balance between accuracy and efficiency. It computes the turbulent kinetic energy ($k$) and the rate of dissipation of the turbulent kinetic energy ($\varepsilon$) by solving the transport equations (Worthy et al. 2001). As turbulence buoyant jets are essentially driven by buoyancy forces, the effect of buoyancy on turbulence needs to be considered, which is commonly accomplished by adding source terms to the $k$ and $\varepsilon$ equations. In the current study, buoyancy source terms based on the generalized gradient diffusion hypothesis (GGDH) of Daly and Harlow (1970) is incorporated in the $k$-$\varepsilon$ turbulence model; this modified model is hereafter referred to as $k$-$\varepsilon$ GGDH turbulence model.

The present paper deals with the near-field flow and mixing properties of vertical buoyant jets subjected to lateral confinement using a 3D numerical model. The study simulates the experiments of Lee and Lee (1998) by utilizing the $k$-$\varepsilon$ and $k$-$\varepsilon$ GGDH models for turbulence closure, and compares the mixing characteristics between measurements and simulations. To the best of the authors’ knowledge, vertical buoyant jets subjected to lateral confinement have hitherto not been studied numerically. The rest of the current paper is organized as follows: firstly, the methodology is introduced; then the validation, results, and discussions are presented; and thirdly, the conclusions are drawn.

### Methodology

The present study consists of a series of 24 numerical simulations of various jet densimetric Froude numbers and confinement indices. Data collected from both the present and previous experiments are used to validate the numerical predictions.

### Governing Equations

The conservation equations for mass and momentum for an incompressible fluid with a Boussinesq assumption for buoyancy effects can be expressed as (Gilageh et al. 2015b):

\[
\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + w = 0
\]

\[
\frac{\partial u}{\partial t} + u \frac{\partial u}{\partial x} + v \frac{\partial u}{\partial y} + w \frac{\partial u}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial x} + \frac{\partial}{\partial x} \left[ \nu_{\text{eff}} \frac{\partial u}{\partial x} \right] + \frac{\partial}{\partial y} \left[ \nu_{\text{eff}} \frac{\partial u}{\partial y} \right] + \frac{\partial}{\partial z} \left[ \nu_{\text{eff}} \frac{\partial u}{\partial z} \right]
\]

\[
\frac{\partial v}{\partial t} + u \frac{\partial v}{\partial x} + v \frac{\partial v}{\partial y} + w \frac{\partial v}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial y} + \frac{\partial}{\partial x} \left[ \nu_{\text{eff}} \frac{\partial v}{\partial x} \right] + \frac{\partial}{\partial y} \left[ \nu_{\text{eff}} \frac{\partial v}{\partial y} \right] + \frac{\partial}{\partial z} \left[ \nu_{\text{eff}} \frac{\partial v}{\partial z} \right] - \frac{\rho_0 - \rho}{\rho}
\]

\[
\frac{\partial w}{\partial t} + u \frac{\partial w}{\partial x} + v \frac{\partial w}{\partial y} + w \frac{\partial w}{\partial z} = -\frac{1}{\rho} \frac{\partial P}{\partial z} + \frac{\partial}{\partial x} \left[ \nu_{\text{eff}} \frac{\partial w}{\partial x} \right] + \frac{\partial}{\partial y} \left[ \nu_{\text{eff}} \frac{\partial w}{\partial y} \right] + \frac{\partial}{\partial z} \left[ \nu_{\text{eff}} \frac{\partial w}{\partial z} \right]
\]

with

\[
\nu_{\text{eff}} = v_t + \nu
\]

where $u$, $v$, and $w$ = Reynolds-averaged velocities, and $x$, $y$ (vertical direction), and $z$ = Cartesian coordinate axes; $t$ = time; $\rho$ = fluid density; and $\rho_0$ = reference fluid density. Pressure is represented by $P$. $\nu_{\text{eff}}$ = effective kinematic viscosity; $v_t$ = turbulent kinematic viscosity; $\nu$ = kinematic viscosity; and $g$ = gravity acceleration.

The temperature equations are modeled using the advection-diffusion equation, as

\[
\frac{\partial T}{\partial t} + u \frac{\partial T}{\partial x} + v \frac{\partial T}{\partial y} + w \frac{\partial T}{\partial z} = k_{\text{eff}} \left( \frac{\partial^2 T}{\partial x^2} + \frac{\partial^2 T}{\partial y^2} + \frac{\partial^2 T}{\partial z^2} \right)
\]

with

\[
k_{\text{eff}} = \frac{v_t}{Pr_t} + \frac{\nu}{Pr}
\]

where $T$ = fluid temperature; $k_{\text{eff}}$ = heat transfer coefficient; $Pr_t$ = turbulent Prandtl number; and $Pr$ = Prandtl number. It is known that the conventional ranges of $Pr_t$ and $Pr$ are both 0.6–1 [Gilageh et al. 2014, 2015a, b]. Five different cases are first simulated, and the optimal $Pr$ and $Pr_t$ numbers for each case are determined. With low $F$ number values, i.e., where buoyancy force dominates over inertial force, turbulence is suppressed (Brennan 2001) and the heat transfer coefficient is reduced. Therefore, the $Pr_t$ and $Pr$ numbers are expected to be a function of the $F$ number, and an empirical
expression is derived using the least-squares regression method:

\[ P_t = P_f = (0.032F + 0.89)^{-1}. \]

**Turbulence Models**

The most extensively used model for turbulence modeling is the standard \( k-\varepsilon \) model proposed by Jones and Launder (1972). Its extensive use can be attributed to its good balance between accuracy and efficiency. It is a two-equation model, where the turbulence kinetic energy \( k \) and its dissipation rate \( \varepsilon \) are calculated from their respective transport equations. The details of the \( k-\varepsilon \) turbulence model are not described here for brevity. The \( k-\varepsilon \) GGDH turbulence model can be expressed as

\[
\frac{\partial k}{\partial t} + U_j \frac{\partial k}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\nu}{\sigma_k} + \nu \right) \frac{\partial k}{\partial x_j} \right] + G + \hat{B} - \varepsilon \quad (8)
\]

\[
\frac{\partial \varepsilon}{\partial t} + U_j \frac{\partial \varepsilon}{\partial x_j} = \frac{\partial}{\partial x_j} \left[ \left( \frac{\nu}{\sigma_\varepsilon} + \nu \right) \frac{\partial \varepsilon}{\partial x_j} \right] + c_4 \frac{\varepsilon}{k} G + c_5 (1 - c_3) \frac{\varepsilon}{k} \hat{B} - c_2 \frac{\varepsilon^2}{k} \quad (9)
\]

with

\[
\hat{B} = \frac{3}{2} \frac{\nu_i}{\sigma_{ij} \rho} \left( \frac{\partial U_i}{\partial x_j} \right) \frac{\partial P}{\partial x_j} + \rho \dot{g} \left( \frac{\partial U_i}{\partial x_j} \right) \quad (10)
\]

\[
G = \nu_i \left( \frac{\partial U_i}{\partial x_j} + \frac{\partial U_j}{\partial x_i} \right) \frac{\partial U_i}{\partial x_j} \quad (11)
\]

\[
\nu_i = c_1 \frac{k^2}{\varepsilon} \quad (12)
\]

where \( U_i \) = instantaneous velocity component in the direction \( x_i \); \( \hat{B} \) = production of turbulence due to the buoyancy effect; and \( u/u_i \) = velocity fluctuation. \( G \) is the production of turbulence due to shear. The model constants \( c_{ij}, c_{1ij}, c_{2}, \sigma_k, \sigma_\varepsilon \) are given the standard values of 0.09, 1.44, 1.92, 1, and 1.3. It is numerically found that there are no significant differences in the results when \( c_{3} \) and \( c_{4} \) vary between 0.6 and 1, so these coefficients are set to 0.8 and 1.0, respectively.

The \( k-\varepsilon \) GGDH turbulence model is a buoyancy-corrected \( k-\varepsilon \) model. The standard \( k-\varepsilon \) turbulence model was originally developed for constant density flows, but turbulent buoyant jets are essentially driven by buoyancy forces due to the density variations. Thus, the \( k-\varepsilon \) model has to be modified to account for the effect of buoyancy on the turbulence. This is commonly accomplished by adding source terms to the \( k \) and \( \varepsilon \) equations. The source term that denotes the production of turbulence due to buoyancy is usually modeled by the simple gradient diffusion hypothesis (SGDH). However, the approach based on the SGDH has a drawback of underestimating the magnitude of the buoyancy production term. To overcome the drawback of the SGDH, the general gradient diffusion hypothesis (GGDH) has proved to be a more accurate approximation (Van Maele and Merci 2006; Worthy et al. 2001; Kumar and Dewan 2013). The GGDH approach incorporates buoyancy source terms for both the \( k \) equation (Van Maele and Merci 2006; Daly and Harlow 1970) and the \( \varepsilon \) equation (Van Maele and Merci 2006; Rodi 1993) in the \( k-\varepsilon \) turbulence model. A major difference between the standard \( k-\varepsilon \) and \( k-\varepsilon \) GGDH turbulence models is that the \( k-\varepsilon \) GGDH model enhances the turbulence level in unstably stratified flow regions and reduces it in stably stratified flow regions.

**Density Calculation**

In the case of the vertical buoyant jet subjected to lateral confinement, the densimetric Froude number \( F \) is one of the most important parameters in characterizing the flow. The \( F \) number is the ratio of inertia to buoyancy force, which can be expressed as

\[
F = \frac{W_j}{\sqrt{g'D}} \quad (13)
\]

with

\[
g' = g \left( \frac{\rho_a - \rho_j}{\rho_a} \right) \quad (14)
\]

where \( W_j \) = initial velocity; \( D \) = diameter of the discharge port; \( g \) = gravitational acceleration; \( \rho_a \) = ambient density; and \( \rho_j \) = jet’s initial density.

The density difference, which is incorporated in the densimetric Froude number, produces the buoyancy. As density is a function of temperature and salinity, there are generally two ways to generate a positively buoyant jet flow: the first is to use higher-salinity ambient water and lower-salinity jet water; the second method is to use lower-temperature ambient water and higher-temperature jet water. In the cases considered in the paper, the discharges are heated and ambient conditions remain roughly unchanged. In other words, heat is used as the source of buoyancy. Therefore, the density is calculated based on the equation of state of seawater of Millero and Poisson (1981) with salinity = 0.

**Model Setup**

The equations are numerically solved using the finite volume method, and the simulations are performed with the open-source CFD code OpenFOAM. The implementation is performed with a modified transient solver for incompressible fluid developed for OpenFOAM. This solver is based on the pisoFoam solver within OpenFOAM and is extended to solve the temperature transport equation and to include the Boussinesq term. Additionally, the solver calculates the density as a function of temperature. As can be observed from the governing equations, the velocity cannot be calculated until the pressure is solved, and the pressure cannot be computed until the velocity is known. The solver starts by estimating the pressure, and then uses this estimated pressure to calculate an intermediate velocity field and the mass fluxes at the cells faces. With this information, it then solves the pressure equation. The fluxes are then corrected to satisfy continuity, velocities are corrected on the basis of the new pressure field, and the transport equation is solved using the finite volume method. The \( k-\varepsilon \) GGDH turbulence model is written based on the standard \( k-\varepsilon \) turbulence model within OpenFOAM. The modified turbulence model reads the density and pressure fields from the modified solver, and then solves the buoyancy-corrected \( k-\varepsilon \) equations.

The schematic diagram of the model is shown in Fig. 1. The numerical simulations are performed in a computational domain corresponding to a \( 1 \times 1 \times 0.45 \) m deep water tank. The other dimensions (e.g., port diameter, riser height, and riser diameter) are determined according to the available experimental setups, and the parameters for each simulated cases are summarized in Table 1.

A structured mesh with confinement is used for discretizing the computational domain, as shown in Fig. 2. There is no universally accepted rule for the selection of grid resolution. For small-scale simulations, which focus on the details (e.g., instability features), very fine meshes should be used. For large-scale simulations, used when the small-scale features are not of primary interest, coarse meshes are usually used (Stoll and Porté-Argel 2006). In the present
paper, the uncertainty in simulations caused by the grid resolution is kept to be less than 2%. For example, if the difference in predicted results when using two grids exceeded 2%, a finer mesh would be used; if the reduction of grid size did not have significant effect on the predicted results, the grid resolution would be regarded as satisfactory and no further refinement would be needed. The average number of grid cells is about 1.79 million, which is much smaller than the number for comparable large eddy simulations (Zhang et al. 2016). Additionally, all the cases presented in this paper have been tested using the laminar and large eddy simulations with the present grid sizes. It was observed that the results of large eddy simulations are very close to those of the laminar simulations. Consequently, this demonstrated that the large eddy simulation requires much higher grid resolution than the RANS simulation to produce satisfactory results, which is one of the primary reasons that the LES is more computationally expensive, and that the practice of RANS simulations is still quite meaningful in the field of hydraulic engineering.

The boundary condition at the top surface is set to inletOutlet (a type of boundary condition in the OpenFOAM), which is normally the same as the zero-gradient open boundary condition but switches to fixed-value boundary condition if there is any backward flow. A major advantage of this type of boundary condition is that it does not require modelers to determine whether the boundary condition is of Neumann or Dirichlet type. Wall boundary conditions are used for the velocity and pressure at the other five outer boundaries and the riser walls, but a fixed-value boundary condition is used for the temperature to correctly represent the physical experiments (the ambient temperature was remained roughly unchanged). The boundary condition at the nozzle surface is set a velocity inlet, where $v = W_j$; $u = w = 0$; $T = T_j$; and $\rho = \rho_j$. The temporal term, divergence term, and Laplacian term are discretized by the Euler scheme, Gauss upwind scheme, and Gauss linear scheme, respectively. The preconditioned conjugate gradient (PCG) method and diagonal incomplete Cholesky (DIC) preconditioner are used for the pressure field with a tolerance of $10^{-7}$, while the preconditioned bi conjugate gradient (PBiCG) method and diagonal incomplete LU (DILU) preconditioner are used for the other fields with a tolerance of $10^{-5}$. The default time step interval is set to 0.01, and the time step is set to be adjustable in order to ensure the numerical stability. Lower convergence criteria values and smaller default time step intervals were attempted, but the results remained roughly the same as the original ones. It is numerically found that the flow is almost steady after 20 s for all the simulated cases, so the simulations are run up to 30 s.

### Validation

Although 3D numerical modeling of jets is quite promising in terms of balancing cost and accuracy, it does not yet rise to the level of a

### Table 1. Parameters of the Simulated Cases

<table>
<thead>
<tr>
<th>Cases</th>
<th>$D$ (mm)</th>
<th>$D_r$ (mm)</th>
<th>$H_r$ (mm)</th>
<th>$T_j$ (°C)</th>
<th>$T_a$ (°C)</th>
<th>$\Delta \rho$ (kg/m$^3$)</th>
<th>$W_j$ (cm/s)</th>
<th>$B$</th>
<th>$F$</th>
</tr>
</thead>
<tbody>
<tr>
<td>C1</td>
<td>8</td>
<td>36</td>
<td>120</td>
<td>56.7</td>
<td>21.4</td>
<td>13.1</td>
<td>10.6</td>
<td>4.3</td>
<td>3.3</td>
</tr>
<tr>
<td>C2</td>
<td>8</td>
<td>36</td>
<td>120</td>
<td>58.7</td>
<td>21.4</td>
<td>14.1</td>
<td>13.3</td>
<td>4.3</td>
<td>4.0</td>
</tr>
<tr>
<td>C3</td>
<td>8</td>
<td>36</td>
<td>120</td>
<td>55.9</td>
<td>21.4</td>
<td>12.7</td>
<td>15.9</td>
<td>4.3</td>
<td>5.0</td>
</tr>
<tr>
<td>C4</td>
<td>8</td>
<td>36</td>
<td>120</td>
<td>57.6</td>
<td>21.3</td>
<td>13.5</td>
<td>21.6</td>
<td>4.3</td>
<td>6.5</td>
</tr>
<tr>
<td>C5</td>
<td>8</td>
<td>36</td>
<td>120</td>
<td>45.4</td>
<td>21.1</td>
<td>7.9</td>
<td>21.4</td>
<td>4.3</td>
<td>8.6</td>
</tr>
<tr>
<td>C6</td>
<td>8</td>
<td>36</td>
<td>120</td>
<td>47.0</td>
<td>21.2</td>
<td>8.6</td>
<td>31.8</td>
<td>4.3</td>
<td>12.3</td>
</tr>
<tr>
<td>C7</td>
<td>10</td>
<td>64</td>
<td>150</td>
<td>51.4</td>
<td>17.8</td>
<td>11.3</td>
<td>10.2</td>
<td>2.8</td>
<td>3.1</td>
</tr>
<tr>
<td>C8</td>
<td>10</td>
<td>64</td>
<td>150</td>
<td>39.5</td>
<td>17.3</td>
<td>6.3</td>
<td>31.2</td>
<td>2.8</td>
<td>12.6</td>
</tr>
<tr>
<td>C9</td>
<td>10</td>
<td>64</td>
<td>200</td>
<td>55.6</td>
<td>19.9</td>
<td>12.8</td>
<td>13.6</td>
<td>3.7</td>
<td>3.9</td>
</tr>
<tr>
<td>C10</td>
<td>10</td>
<td>64</td>
<td>200</td>
<td>42.3</td>
<td>19.6</td>
<td>6.0</td>
<td>25.8</td>
<td>3.7</td>
<td>9.9</td>
</tr>
<tr>
<td>C11</td>
<td>10</td>
<td>36</td>
<td>200</td>
<td>58.7</td>
<td>19.7</td>
<td>14.4</td>
<td>19.0</td>
<td>7.7</td>
<td>5.1</td>
</tr>
<tr>
<td>C12</td>
<td>10</td>
<td>36</td>
<td>200</td>
<td>59.7</td>
<td>19.8</td>
<td>14.9</td>
<td>22.6</td>
<td>7.7</td>
<td>5.9</td>
</tr>
<tr>
<td>C13</td>
<td>10</td>
<td>36</td>
<td>150</td>
<td>46.1</td>
<td>16.9</td>
<td>9.1</td>
<td>10.2</td>
<td>5.8</td>
<td>3.4</td>
</tr>
<tr>
<td>C14</td>
<td>10</td>
<td>36</td>
<td>150</td>
<td>46.9</td>
<td>16.8</td>
<td>9.4</td>
<td>12.0</td>
<td>5.8</td>
<td>3.9</td>
</tr>
<tr>
<td>C15</td>
<td>10</td>
<td>36</td>
<td>150</td>
<td>55.6</td>
<td>16.7</td>
<td>13.4</td>
<td>19.0</td>
<td>5.8</td>
<td>5.2</td>
</tr>
<tr>
<td>C16</td>
<td>10</td>
<td>36</td>
<td>150</td>
<td>55.9</td>
<td>16.6</td>
<td>13.6</td>
<td>24.5</td>
<td>5.8</td>
<td>6.7</td>
</tr>
<tr>
<td>C17</td>
<td>10</td>
<td>36</td>
<td>150</td>
<td>38.1</td>
<td>16.3</td>
<td>6.0</td>
<td>20.4</td>
<td>5.8</td>
<td>8.4</td>
</tr>
<tr>
<td>C18</td>
<td>10</td>
<td>36</td>
<td>150</td>
<td>39.0</td>
<td>16.9</td>
<td>6.2</td>
<td>31.2</td>
<td>5.8</td>
<td>12.7</td>
</tr>
<tr>
<td>C19</td>
<td>8</td>
<td>21</td>
<td>120</td>
<td>55.5</td>
<td>21.2</td>
<td>12.5</td>
<td>10.6</td>
<td>9.2</td>
<td>3.4</td>
</tr>
<tr>
<td>C20</td>
<td>8</td>
<td>21</td>
<td>120</td>
<td>57.1</td>
<td>21.5</td>
<td>13.3</td>
<td>13.3</td>
<td>9.2</td>
<td>4.1</td>
</tr>
<tr>
<td>C21</td>
<td>8</td>
<td>21</td>
<td>120</td>
<td>58.2</td>
<td>21.5</td>
<td>13.8</td>
<td>15.9</td>
<td>9.2</td>
<td>4.8</td>
</tr>
<tr>
<td>C22</td>
<td>8</td>
<td>21</td>
<td>120</td>
<td>57.4</td>
<td>21.8</td>
<td>13.3</td>
<td>21.3</td>
<td>9.2</td>
<td>6.6</td>
</tr>
<tr>
<td>C23</td>
<td>8</td>
<td>21</td>
<td>120</td>
<td>44.6</td>
<td>21.1</td>
<td>7.6</td>
<td>21.4</td>
<td>9.2</td>
<td>8.8</td>
</tr>
<tr>
<td>C24</td>
<td>8</td>
<td>21</td>
<td>120</td>
<td>45.7</td>
<td>21.2</td>
<td>8.0</td>
<td>31.8</td>
<td>9.2</td>
<td>12.7</td>
</tr>
</tbody>
</table>

Note: $B =$ confinement index; $D =$ jet diameter; $D_r =$ riser diameter; $F =$ densimetric froude number; $H_r =$ riser height; $T_a =$ ambient temperature; $T_j =$ jet temperature; $W_j =$ jet velocity; $\Delta \rho =$ source density deficit.
deterministic tool because many of the factors involved—such as input data, model structures, and parameters—involves unacceptable levels of uncertainty. Therefore, it is necessary to deal with the uncertainties with care, and verify the outputs before accepting the simulated results. The simulations discussed in this paper follow a clearly defined path, which contains four steps toward producing solutions with a high degree of confidence. Step 1 determines the grid resolution to minimize uncertainties due to grid cell sizes; Step 2 qualitatively compares the numerical outputs to the present experimental observations to assure that the model structures are physically correct; Step 3 calibrates a key but uncertain model parameter based on available data in the literature; and Step 4 quantitatively verifies the simulated results to give a sense of confidence that the model can properly represent what would occur in reality.

**Grid Resolution**

To eliminate uncertainties due to grid cell sizes, a grid independency study with a tolerance of 2% is performed. The grid resolution of each simulation in the present study is determined in the following manner. First, a preliminary simulation with relatively coarse grid resolution was established, and then simulations with finer resolution were conducted (until the model achieves the 2% confidence criteria). Taking Case 1 as an example, eight different meshes were tested, as shown in Fig. 3 and Table 2. The employed procedure can be summarized as follows. A preliminary simulation was performed with Mesh 1, which contains about $0.256 \times 106$ grid cells. Another simulation was performed with Mesh 2. The mesh contains about $0.402 \times 106$ grid cells and its level of grid resolution is 1.4 (that is, the number of grid cells in each block is multiplied by a factor of 1.4 compared to that of Mesh 1). The two simulations were compared. As can be observed in Fig. 3, the difference between the Mesh 1 and Mesh 2 simulations is significant; thus, Mesh 1 is not considered acceptable. A new simulation was conducted using Mesh 3, which has a level of grid resolution of 1.5. As one can see in Fig. 3, the outputs of the Mesh 2 simulation do not deviate far from those of the Mesh 3 simulation, so a quantitative analysis is necessary. The quantitative comparison comprises two steps: first, the results are interpolated to make the $x$, $y$, and $z$ values consistent; second, the deviation between the two simulations is calculated. As summarized in Table 2, the deviation is
−2.013%, which exceeds the 2% criteria. The results from Mesh 3 deviates from those of Mesh 4 within only 2%, but the error regarding interpolation for Mesh 4 exceeds 2%. Thus, the results were not considered reliable; neither Mesh 5 nor Mesh 6 were adopted due to the same concern (interpolation error). Finally, Mesh 7 and Mesh 8 met the 2% standard, and the results were deemed reliable. To be conservative, the results from Mesh 8 were employed.

**Qualitative Verification**

To ensure that the model structure and mathematical basis are physically correct, qualitative verifications are employed before detailed and extensive quantitative analyses are conducted. Qualitative verification entails two steps: first, the model must reproduce the correct mixing pattern; second, the model should correctly estimate the influence of $F$ and $B$ on the mixing pattern. The mixing pattern of laterally confined buoyant jets can be observed in Fig. 4, which shows a normalized concentration $C/C_j$ contour plot at the central plane, obtained from the present experimental measurements. The setup of these experiments is similar to that of Lee and Lee (1998), except that the output of the jets is recorded using the imaging module of the particle image velocimetry (PIV) system developed at the University of Ottawa and then processed using a laser-induced fluorescence (LIF) code. However, as experimental investigations are not the focus of this paper, these preliminary experiments were conducted only for basic qualitative analyses, so the details are not described here. It can be observed from Fig. 4 that the dilution of the effluent is relatively weak inside the riser, as the initial momentum of the jet is significantly large and the entrainment of ambient water into the jet is restricted by the riser. The mixing pattern of the mixed effluent outside the riser is like that produced by a free buoyant jet, where shear layers are formed and the ambient water is freely entrained into the effluent. The same observations can be obtained from the simulated results. Two general conclusions regarding the mixing pattern of lateral confined buoyant jets can be obtained from these experimental observations. First, the higher the $F$ number, the lower the dilution will be because inertial force dominates over buoyancy force at higher $F$ numbers. Second, the larger the confinement index $B$, the lower

**Table 2. Meshes and Parameters**

<table>
<thead>
<tr>
<th>Mesh</th>
<th>Number of cells (million)</th>
<th>Level of grid resolution</th>
<th>Mean value of $S$</th>
<th>Mean value of $S$ (interpolated)</th>
<th>Error of interpolation (%)</th>
<th>Deviation from higher resolution (%)</th>
</tr>
</thead>
<tbody>
<tr>
<td>Mesh 1</td>
<td>0.259</td>
<td>1</td>
<td>—</td>
<td>—</td>
<td>—</td>
<td>—</td>
</tr>
<tr>
<td>Mesh 2</td>
<td>0.699</td>
<td>1.4</td>
<td>0.145</td>
<td>0.143</td>
<td>−1.294</td>
<td>−2.013</td>
</tr>
<tr>
<td>Mesh 3</td>
<td>0.885</td>
<td>1.5</td>
<td>0.141</td>
<td>0.139</td>
<td>−1.982</td>
<td>−0.310</td>
</tr>
<tr>
<td>Mesh 4</td>
<td>1.038</td>
<td>1.6</td>
<td>0.137</td>
<td>0.144</td>
<td>4.436</td>
<td>−0.222</td>
</tr>
<tr>
<td>Mesh 5</td>
<td>1.237</td>
<td>1.7</td>
<td>0.142</td>
<td>0.141</td>
<td>−0.400</td>
<td>−0.198</td>
</tr>
<tr>
<td>Mesh 6</td>
<td>1.473</td>
<td>1.8</td>
<td>0.137</td>
<td>0.142</td>
<td>3.356</td>
<td>−1.895</td>
</tr>
<tr>
<td>Mesh 7</td>
<td>1.773</td>
<td>1.9</td>
<td>0.141</td>
<td>0.139</td>
<td>−1.455</td>
<td>−0.207</td>
</tr>
<tr>
<td>Mesh 8</td>
<td>2.014</td>
<td>2</td>
<td>0.136</td>
<td>0.135</td>
<td>−0.858</td>
<td>—</td>
</tr>
</tbody>
</table>

© ASCE 04017016-6 J. Hydraul. Eng.
Fig. 5. Comparison of dilution at various cross sections for different Fr numbers: (a) Fr = 4.3, k = ε; (b) Fr = 4.3, k = ε GGDH; (c) Fr = 5.8, k = ε; (d) Fr = 5.8, k = ε GGDH; (e) Fr = 9.2, k = ε; (f) Fr = 9.2, k = ε GGDH

the dilution will be because the restriction of ambient intrusion into the riser is stronger when B is larger. Theoretically, the jet is a buoyant jet when B = 0, and the riser becomes a larger nozzle as B approaches infinity.

The influences of F and B on the mixing pattern can also be observed from the modeled results from the numerical simulations. Fig. 5 shows the temperature dilution plots at various cross sections for different F numbers, where Tj is jet initial temperature, Tu is ambient temperature, and T_max is maximum temperature at a particular cross-section (Z/D). It can be observed that the dilution becomes lower when the Fr number increases. As Fig. 6 shows, the dilution becomes lower when B is larger.

Calibration
A major weakness of 3D numerical modeling of jets is that the results depend highly on a wide variety of uncertain variables, including the initial values of k and ε, the kinematic viscosity ν, the internal empirical coefficients of the model, and the P_r and Pr numbers. This weakness is a main reason why 3D numerical modeling of jets is considered less accurate than either field measurement or physical modeling. However, numerical modeling can incorporate these two techniques into the procedures by adjusting certain parameters to reproduce the measured results. In the present study, the initial values of k and ε are calculated from the empirical equations $k = 0.06u^2$ and $ε = 0.06u^3/D$ (Gildeh et al. 2014, 2015b, a; Huai et al. 2010); the kinematic viscosity ν is calculated using the initial Reynolds number, jet velocity, and jet diameter. The default values are used for the internal empirical coefficients of the model. There is no commonly used approach to determine the P_r and Pr numbers, so they have been selected to be the key uncertain parameters to calibrate.

The expected result of calibration can be either a single value within a reasonable range or an empirical expression. The former is much more straightforward, but the accuracy is relatively low because it ignores the impact of flow and fluid properties on thermal
Table 3. Uncertain Parameters and their Values in the Present Study for Each Case

<table>
<thead>
<tr>
<th>Case</th>
<th>Initial $k$ ($\times 10^{-3}$)</th>
<th>Initial $\varepsilon$ ($\times 10^{-2}$)</th>
<th>$\nu$ ($\times 10^{-2}$)</th>
<th>$P_{tr}$</th>
<th>$Pr$</th>
</tr>
</thead>
<tbody>
<tr>
<td>C1</td>
<td>0.64</td>
<td>0.82</td>
<td>5.03</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>C2</td>
<td>1.06</td>
<td>1.76</td>
<td>4.91</td>
<td>1.02</td>
<td>1.02</td>
</tr>
<tr>
<td>C3</td>
<td>1.52</td>
<td>3.01</td>
<td>5.09</td>
<td>1.05</td>
<td>1.05</td>
</tr>
<tr>
<td>C4</td>
<td>2.80</td>
<td>7.56</td>
<td>4.98</td>
<td>1.10</td>
<td>1.10</td>
</tr>
<tr>
<td>C5</td>
<td>2.75</td>
<td>7.35</td>
<td>5.97</td>
<td>1.17</td>
<td>1.17</td>
</tr>
<tr>
<td>C6</td>
<td>6.07</td>
<td>24.12</td>
<td>5.82</td>
<td>1.28</td>
<td>1.28</td>
</tr>
<tr>
<td>C7</td>
<td>0.62</td>
<td>0.64</td>
<td>5.45</td>
<td>0.99</td>
<td>0.99</td>
</tr>
<tr>
<td>C8</td>
<td>5.84</td>
<td>18.22</td>
<td>6.63</td>
<td>1.29</td>
<td>1.29</td>
</tr>
<tr>
<td>C9</td>
<td>1.11</td>
<td>1.51</td>
<td>5.11</td>
<td>1.01</td>
<td>1.01</td>
</tr>
<tr>
<td>C10</td>
<td>3.99</td>
<td>10.30</td>
<td>6.31</td>
<td>1.21</td>
<td>1.21</td>
</tr>
<tr>
<td>C11</td>
<td>2.17</td>
<td>4.12</td>
<td>4.90</td>
<td>1.05</td>
<td>1.05</td>
</tr>
<tr>
<td>C12</td>
<td>3.06</td>
<td>6.93</td>
<td>4.81</td>
<td>1.08</td>
<td>1.08</td>
</tr>
<tr>
<td>C13</td>
<td>0.62</td>
<td>0.64</td>
<td>5.92</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>C14</td>
<td>0.86</td>
<td>1.04</td>
<td>5.85</td>
<td>1.01</td>
<td>1.01</td>
</tr>
<tr>
<td>C15</td>
<td>2.17</td>
<td>4.12</td>
<td>5.11</td>
<td>1.06</td>
<td>1.06</td>
</tr>
<tr>
<td>C16</td>
<td>3.60</td>
<td>8.82</td>
<td>5.10</td>
<td>1.10</td>
<td>1.10</td>
</tr>
<tr>
<td>C17</td>
<td>2.50</td>
<td>5.09</td>
<td>6.82</td>
<td>1.16</td>
<td>1.16</td>
</tr>
<tr>
<td>C18</td>
<td>5.84</td>
<td>18.22</td>
<td>6.69</td>
<td>1.30</td>
<td>1.30</td>
</tr>
<tr>
<td>C19</td>
<td>0.67</td>
<td>0.89</td>
<td>5.12</td>
<td>1.00</td>
<td>1.00</td>
</tr>
<tr>
<td>C20</td>
<td>1.06</td>
<td>1.76</td>
<td>5.02</td>
<td>1.02</td>
<td>1.02</td>
</tr>
<tr>
<td>C21</td>
<td>1.52</td>
<td>3.01</td>
<td>4.92</td>
<td>1.04</td>
<td>1.04</td>
</tr>
<tr>
<td>C22</td>
<td>2.72</td>
<td>7.25</td>
<td>4.99</td>
<td>1.10</td>
<td>1.10</td>
</tr>
<tr>
<td>C23</td>
<td>2.75</td>
<td>7.35</td>
<td>6.05</td>
<td>1.17</td>
<td>1.17</td>
</tr>
<tr>
<td>C24</td>
<td>6.07</td>
<td>24.12</td>
<td>5.95</td>
<td>1.30</td>
<td>1.30</td>
</tr>
</tbody>
</table>

Quantitative Verification

The last step of validation is to quantitatively compare the simulated results against available experimental data to demonstrate the models can accurately represent what would occur in reality. Lee and Lee (1998) reported concentrations at various locations along the central line; the baseline results are obtained from the plots presented in their findings. The following paragraphs will discuss the comparisons among the simulation results obtained in this study, the measured results, and the results estimated in the empirical equations.

In the experiment, the concentration was measured using the LIF system. The spatial variation between the measured and simulated concentrations and a comparison of the two for different cases are presented in Figs. 7 and 8, which show the variation of centerline concentrations according to location. The results are obtained from the measurements, the simulations, and the empirical estimations. The jets with the lowest $F$ number had a concentration excess value $S$ of about 0.28 at the riser exit, while the jets with the highest $F$ number had a concentration excess value $S$ of about 0.48 at the riser exit. This implies that the dilution is less when $F$ is the larger. As expected, the concentrations at locations farther from the riser exit are lower than those nearer, suggesting that the jets dilute continuously along their trajectory. The results obtained by the standard $k-\varepsilon$ and $k-\varepsilon$ GGDH models showed a very good match with the experimental data, but the empirical model deviated farther from the data.

To obtain a quantitative measure of the difference between data and prediction, scatter plots of measured versus simulated centerline concentration for different cases (Figs. 9 and 10) were used. The results were grouped according to the proximity of the measurement points. The root-mean-square deviation (RMSD) values are also shown in the figures. The $k-\varepsilon$ GGDH model consistently shows lower RMSD values than the $k-\varepsilon$ model. The results show that both the $k-\varepsilon$ GGDH and $k-\varepsilon$ modeling approaches provide a good match with the measurements, as well as an excellent match between the measured and simulated concentrations. A close examination of the plots shows that of the two models, the $k-\varepsilon$ GGDH model produces better results.

Results

Cross-Sectional Concentration Profile

The mixing properties of a confined buoyant jet can be clearly observed using concentration profiles at various cross sections along the trajectory. Fig. 11 shows the time-averaged image of the concentration field and representative cross sections of a confined jet ($B = 4.3$, $F = 3.3$) based on the $k-\varepsilon$ GGDH model as an illustration. The $x$-coordinate denotes the radial distance from the centerline, and $y$-coordinate represents the values of $Z/D$, where $Z$ is the vertical distance from the nozzle exit and $D$ is the jet diameter. The range of $x$-coordinates is $-25$ to $25$ cm, outside of which the change of flow characteristics is almost negligible. The $y$-coordinates range from $0$ to $45$ cm, where $y = 0$ cm denotes the bottom of the study domain and $y = 45$ cm is the initial water level.

The cross-sections begin at $Z/D = 20$, the interval between the sections is $\Delta(Z/D) = 4$, and the cross-sections terminate at $Z/D = 40$. It is observed that the jet is typically jet-to-plume-like in this range, where the transverse mixing processes are influenced by both momentum and buoyancy forces. This region can be roughly regarded as a transient region, and the flow pattern can be divided by it into jet-like, jet-to-plume-like, and plume-like regions. These three regions were also named the initial stage, transient stage, and developed stage in previous studies regarding waste-water discharges. The jet below this region is significantly dominated by momentum forces, so it generally behaves like a pure jet, where the dilution rate is very low. The jet above this region is mainly influenced by buoyancy forces, so it behaves like a pure plume; water entrainment reaches the jet centerline and the dilution rate becomes much higher. A similar phenomenon was observed in many previous studies and can be attributed to the variations in momentum and buoyancy forces along the jet trajectory. The study of transverse mixing processes in the transient region is very challenging due to the complexity of relations between momentum and buoyancy forces, and this is of great research interest.

Fig. 12 presents transverse distributions of time-mean tracer concentrations at the representative cross-sections in the central plane for different cases. In order to obtain self-similarity profiles, nondimensionalized concentrations $C/Cm$ at various locations are...
extracted and plotted against dimensionless normal coordinates $r/b_{gc}$, where $C$ is the time-mean concentration at a point, $C_m$ is the maximum concentration in a representative cross section, $r$ is the radial distance of a point from the trajectory (negative represents left half), and $b_{gc}$ is the concentration $1/e$ width, defined at the location where the concentration drops to $1/e$ of the corresponding maximum value. This figure can be used to evaluate the evolution of the mixing patterns. For all cases, the locus of

Fig. 7. Concentration profile of confined buoyant jet with $B = 4.3$, $S$ versus $Z/D$: (a) $F = 3.3$; (b) $F = 4.0$; (c) $F = 5.0$; (d) $F = 6.5$; (e) $F = 8.6$; (f) $F = 12.3$
Fig. 8. Concentration profile of confined buoyant jet, $S$ versus $Z/D$: (a) $B = 2.8$, $F = 3.1$; (b) $B = 2.8$, $F = 12.6$; (c) $B = 3.7$, $F = 3.9$; (d) $B = 3.7$, $F = 9.9$; (e) $B = 7.7$, $F = 5.1$; (f) $B = 7.7$, $F = 5.9$
concentration maximums overlaps with the jet centerline, so $C_m$ is exactly the concentration at a point where the $x$-coordinate is 0, and $r$ is the absolute of $x$-coordinate of a point. As can be observed, the value of normalized concentration approaches zero when the value of nondimensionalized coordinate is $C_6$, so $bgc$ can also be regarded as the half concentration width.

It can be observed in Fig. 12 that the concentrations in different representative cross sections collapse nicely on a typical Gaussian profile, which can be formulated as follows:

$$C = C_m e^{-(r/b_{gc})^2}$$  (15)

The cross-sectional profiles for other cases also show such self-similarity and Gaussian distribution, which suggests that self-similarity is independent of the cross sections, Froude number, and confinement index. However, a close examination of the plots shows that the agreement between modeled and Gaussian results is variable.

The cross-sectional concentration profiles at the level close to the riser ($Z/D = 20$) deviate farther from the Gaussian profile than the others, which should be due to the flow instability caused by the confinement. In the medium region (e.g., $Z/D = 32$), the concentration maximums overlaps with the jet centerline.
concentration profiles agree with the Gaussian profile very well, which is consistent with the flow pattern of free buoyant jets (where no confinement is imposed). The profile departure increases as the jet approaches the water surface (e.g., $Z/D = 40$), but the intensification in near-surface region is not as obvious as that in the near-riser region. Thus, it can be concluded that the cross-sectional concentration distribution of a confined jet can be influenced by boundaries, but the impact of water surface is relatively weaker than that of confinement.

Cases C1 and C6 have the same confinement index of 4.3, but the densimetric Froude number of the latter is almost 3.7 times that of the former. The cross-sectional concentration profiles of these two cases are compared in Fig. 13(a). Contrary to the previous analysis regarding the impact of densimetric Froude number, the difference between these two profiles can be clearly observed, implying that the impact of confinement index on the cross-sectional concentration distribution of confined jets is significant. The results are more scattered for C24 than for C18, suggesting that the profile departure increases with the confinement index.

Cases C18 and C24 have the same densimetric Froude number of 12.7, but the confinement index of the latter is about 1.6 times that of the former. The cross-sectional concentration profiles of these two cases are compared in Fig. 13(b). Contrary to the previous analysis regarding the impact of densimetric Froude number, the near-riser cross-section ($Z/D = 16$) has the largest deviation, demonstrating that cross-sectional concentration distributions in the near-riser regions are more sensitive to the confinement index.

In order to quantify the agreement between the present simulated results with the Gaussian distributions, the root-mean-square deviation (RMSD) values of cross-sectional concentration distributions for different cases are summarized in Table 4. The smallest average RMSD value occurs at the cross section $Z/D = 32$, which is 0.023, so it is confirmed that the profile agreement increases with the distance from the boundaries within the range of the investigation in the current study. The mean RMSD value at the

Fig. 12. Normalized concentration profiles at various cross-sections for different cases: (a) C1 ($B = 4.3$, $F = 3.3$); (b) C6 ($B = 4.3$, $F = 12.3$); (c) C18 ($B = 5.8$, $F = 12.7$); (d) C24 ($B = 9.2$, $F = 12.7$)
cross section $Z/D = 40$ is about 41% of that at the near-riser level, implying that the influence of water-air interface is smaller than that of riser walls. The densimetric Froude number difference between C6 and C1 is as large as 9 but their RMSD values are very close. On the contrary, the confinement index increases by 3.4 from C18 to C24 but the increase of RMSD is as large as 56.7%. Therefore, the cross-sectional concentration distribution of confined jets is more influenced by the confinement index than the densimetric Froude number. The same conclusions can also be drawn from the other cases, which are not presented here for brevity’s sake.

Throughout the entire trajectory, the time-mean tracer concentration field is absolutely axis-symmetric with respect to the centerline. Therefore, in order to speed up numerical simulations of the present problem, one can just consider a quarter of the study domain in modeling processes.

### Table 4. RMSD Values of Cross-Sectional Concentration Distributions for Different Cases

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>C1</td>
<td>0.035</td>
<td>0.023</td>
<td>0.035</td>
<td>0.024</td>
<td>0.024</td>
<td>0.03</td>
<td>0.029</td>
</tr>
<tr>
<td>C6</td>
<td>0.045</td>
<td>0.034</td>
<td>0.029</td>
<td>0.017</td>
<td>0.019</td>
<td>0.019</td>
<td>0.027</td>
</tr>
<tr>
<td>C18</td>
<td>0.058</td>
<td>0.03</td>
<td>0.025</td>
<td>0.021</td>
<td>0.023</td>
<td>0.024</td>
<td>0.03</td>
</tr>
<tr>
<td>C24</td>
<td>0.093</td>
<td>0.04</td>
<td>0.056</td>
<td>0.03</td>
<td>0.032</td>
<td>0.031</td>
<td>0.047</td>
</tr>
<tr>
<td>Average</td>
<td>0.058</td>
<td>0.032</td>
<td>0.036</td>
<td>0.023</td>
<td>0.024</td>
<td>0.026</td>
<td>0.033</td>
</tr>
</tbody>
</table>

**Fig. 13.** Comparison of normalized concentration profiles for different cases: (a) C1 and C6; (b) C18 and C24

**Fig. 14.** Variation of jet concentration spread width along the trajectory

### Concentration Spread

The study of jet concentration spread widths can be used to investigate the properties of jet concentration dispersion and growth. Jet concentration spread widths can commonly be characterized either by the concentration 5% width or the concentration half-width. The former is defined at the location where the concentration drops to 5% of the corresponding maximum value, while the latter is based on the distance from the trajectory to the $1/e$ value of the trajectory maximum.

In the present study, the latter method is adopted as it is more frequently used. The growth rates of confined buoyant jets are quantified based on the concentration distribution map determined in the numerical experiments. The variation of the jet concentration spread width at various cross-sections along the centerline is examined in Fig. 14. In the figure, the dimensionless concentration spread widths ($b_{bc}/F/B/D$) of the $k$-$\varepsilon$ GGDH predictions are plotted against vertical locations normalized by respective densimetric Froude numbers, confinement indexes, and jet diameters ($Z/F/B/D$).

As seen from the figure, the spread widths increase almost linearly with distance, resembling that of a free buoyant jet. However, the rate at which the jet concentration spread width grows is roughly equal to $b_{bc}/s = 0.0938$, which is relatively smaller than that of a free buoyant jet (about 0.12). This is in line with the fact that the jet concentration spread rate decreased due to the existence of confinement.

In order to obtain a quantitative measure of the difference between different cases, the jet concentration spread widths (ratio of case-average width to overall-average width) are calculated and plotted in Fig. 15. It can be clearly observed that Cases C7 and C8 have a higher jet concentration spread rate compared to the others. This is due to the fact that the confinement has little influence on the buoyant jets when $B = 2.8$. In addition, the values of Cases C1–C6 are higher than...
average but lower than those of Cases C7 and C8, which is consistent with the conclusion drawn from the experimental observations; when the confinement index is moderate, e.g., \( B = 4.3 \), the buoyant jet is said to be moderately confined. The values of the Cases C19–C24 are lower than average, which is also in accordance with the observations in the experiments: when \( B \) is large, the jet concentration spread rate is lower than others, and the buoyant jet can be regarded as highly confined.

As mentioned in the previous section, the flow can be divided into jet-like, jet-to-plume-like, and plume-like regions, but it is observed that the jet concentration spread rate remains almost unchanged throughout the entire trajectory for each case, demonstrating that the spreading rate does not obviously vary between stages.

**Velocity Characteristics**

The velocity distributions can be obtained based on the \( k-\varepsilon \) GGDH model results. The normalized streamwise velocity \( U_y/U_0 \) at each cross-section along the centerline is extracted and plotted against the vertical location of the corresponding cross section \( Z \) in Fig. 16.

From the figure, it can be observed that the jet could be divided into different zones based on the characteristics of velocity magnitude decay. In Zone 1, the streamwise velocity magnitudes increase along the centerline, which is attributed to the positive buoyancy and process of flow establishment. In Zone 2, the streamwise velocity magnitudes decrease significantly because of the rapid dilution in this region and the shear stresses induced by the riser wall. At the end of Zone 2, impingement occurs, where the jet hits the riser wall, and the flow is deflected towards the centerline. Therefore, the streamwise velocity magnitudes increase to a small degree due to the decrease of the jet cross-section area in Zone 3. Note that Zone 3 does not exist for Cases C7 and C8, the confinement index of which are both 2.8. In Zone 4, the velocities decay gradually and continuously, caused by momentum transfer and jet dilution.

As mentioned above, there is no Zone 3 in C7 and C8, where the confinement has little effect on the jet. This implies that Zone 3 is a unique property of confined jets and thus can be used as an indicator of the effect of lateral confinement on buoyant jets. However, there is no generally accepted method of estimating the location of the impingement points. Therefore, correlation analyses are done using the simulated data to find out the main factor influencing the location of impingement points. The results show that the location of impingement points is mainly determined by the diameter of the riser \( D_r \). Thus, the vertical distance is normalized by the diameter of the riser and the results are plotted again in Fig. 17.

As can be observed from the figure, the location where impingement occurs is about \( Z/D_r = 3 \) for all cases. This observation provides a much simpler way to characterize the impact of lateral confinement on buoyant jets. For example, for Cases C7 and C8, \( H_r < 3D_r \), so no impingement occurs within the riser, implying the buoyant jet is weakly confined; for Cases C1–C6, \( H_r \approx 3D_r \), so the jet width just touches the riser top, suggesting that the buoyant jet is moderately confined; for Cases C19 to C24, \( H_r > 3D_r \), so the jet fills up the riser completely, meaning that the buoyant jet is highly confined.

**Discussion**

**Free and Confined Buoyant Jets**

To improve the understanding of laterally confined vertical buoyant jets, the differences between buoyant jets subjected to lateral confinement and nonlateral confinement (i.e., free buoyant jets) must...
be examined. The differences can be quantified by comparing confined buoyant jets and free buoyant jets with the same Fr number. The modeled results of Cases C7 and C8 in the $k$-$\varepsilon$ GGDH model are used for comparison because they both have a $B$ number of 2.8 (the smallest among all the cases). The dilutions of free buoyant jets are estimated using the approach proposed by Chen and Rodi (1980). The concentration profiles of free and confined buoyant jets are presented in Fig. 18. Note that the concentration is an increasing function of both $B$ and $F$, which is consistent with the conclusion drawn from the comparisons of various confined buoyant jets. Therefore, one may conclude that a free buoyant jet is a special case of confined buoyant jet, the $B$ number of which is very small. This conclusion is of major importance in experimental design and practical engineering practice because it provides a new venue for determining whether a jet will be influenced by boundaries.

**Various Turbulence Models**

Numerical modeling of vertical buoyant jets subjected to lateral confinement can be performed using a variety of models, ranging from simple to complex; it is quite interesting to test the performance of various models, such as $k$-$\omega$, Launder-Reece-Rodi (LRR), and Launder-Gibson. The authors and collaborators have extensively compared various models in different applications. For example, Gildeh et al. (2014) compared seven Reynolds-averaged Navier-Stokes turbulence models in modeling turbulent buoyant wall jets in stationary ambient water; by comparing the modelled results of clinging length, plume trajectory, temperature dilutions, and temperature and velocity profiles to available data, they concluded that the realizable $k$-$\varepsilon$ and LRR turbulence models are the most accurate among all the tested models. The present research aims to improve the practice of numerical modeling in another way: namely, by modifying a widely used model and calibrating a key but uncertain parameter. If this universally accepted model suffices for the work at hand, there will be no need to use other more complex models. This approach has two advantages: first, the results are more accurate than more complex models because it does not require the assumption that the $P_r$ and $Pr$ numbers are constants; second, it is more appropriate for numerical experiments because it is more computationally efficient.

**Uncertainties and Calibration**

The successful application of computational fluid dynamics (CFD) modeling in hydro-environmental research (e.g., mixing characteristics of buoyant jets) is still hindered by the uncertainties inherent to CFD models. Therefore, modelers need an in-depth understanding of the uncertainties to reproduce the processes accurately. There are several sources of uncertainty in a CFD model, including empirical constants, assumptions, and numerical approximations. To make numerical simulations simple and fast, it is useful to employ default settings, following general rules and standard constants in a model. However, there are no commonly accepted rules for determining the $P_r$ and $Pr$ in the present models.

The present study focuses on the uncertainties relating to the $P_r$ and $Pr$ numbers. The model was calibrated against the concentration measurements in laboratory experiments. It was decided to calibrate against concentrations because the accurate prediction of these values is very important for modeling the mixing processes of confined buoyant jets. First, optimal $P_r$ and $Pr$ numbers were found for Cases C1, C7, C11, C13, and C19. Second, correlation and regression analysis techniques were used to derive an empirical equation. Lastly, the empirical equation was validated by the rest of the cases. It was found that the calibrated values of the $P_r$ and $Pr$ numbers obtained by the standard $k$-$\varepsilon$ model deviate further from the values estimated by the empirical equation than those obtained by the $k$-$\varepsilon$ GGDH model. Therefore, it can be concluded that the $k$-$\varepsilon$ GGDH model is more capable of predicting the dilution of confined jets than the standard $k$-$\varepsilon$ model.

**Outcome**

Because uncertainties can be related to modeling approaches, a more advanced model is usually regarded as more useful in terms of reducing uncertainties. However, there are two problems that impose substantial barriers to the usage of such models in common engineering problems: the computational cost is typically too high, and the requirement for high-resolution input data is too strict. One major outcome of this paper is to demonstrate the capacity of a 3D numerical model to provide an accurate forecast of the mixing and dilution performance of laterally confined vertical buoyant jets. Another contribution of this study is the proposed formula for determining the $P_r$ and $Pr$ numbers. In contrast to previous studies, which used a single and constant value of $P_r$ and $Pr$ numbers, the present study links these two numbers to the $F$ number, which is more physically reasonable and can produce very good results. This study also makes it possible to roughly quantify the rate at which the jet concentration spread width grows and identify the location where impingement occurs.

**Summary and Conclusion**

A study of laterally confined buoyant jets was conducted. The modelled results from different turbulence models are compared with the present and previous experimental results, and the comparisons demonstrate that the buoyancy corrected $k$-$\varepsilon$ model performs fairly well in the mixing characteristics of a laterally confined buoyant jet. This outcome is of major importance in pollutant dispersion forecasting and environmental assessment problems because it confirms that the universally accepted model ($k$-$\varepsilon$) can be satisfactorily accurate, eliminating the need for an advanced modeling approach, as long as suitable modification and calibration are performed. The cross-sectional concentration profiles, jet concentration spread, and velocity characteristics are also investigated based on the $k$-$\varepsilon$ GGDH model predictions. The results clearly show the impacts of confinement and the densimetric Froude number on the mixing pattern of laterally confined buoyant jets. The important conclusions of the present study are listed below:
The turbulent Prandtl number $Pr_t$ and Prandtl number $Pr$ are expected to be a function of the densimetric Froude number $Fr: Pr_t = Pr = (0.032F + 0.89)^{-1}$. This assumption is more practical and can produce very good results. This outcome also provides a possible method for improving the determinacy of numerical modeling of buoyant jets, which can save modellers much time for calibration.

The cross-sectional concentration distribution of a confined jet can be influenced by boundaries, but the impact of water surface is relatively weaker than that of confinement.

The rate at which the jet concentration spread width grows is uncertain factors on modeled results and proper determination of search objectives mainly involve the quantification of impacts of uncertain factors on modeled results and proper determination of these factors for various cases.

Acknowledgments

The research was supported by NSERC (Natural Sciences and Engineering Research Council of Canada) and QNRF (Qatar National Research Fund).

References


