Numerical Modeling of Turbulent Buoyant Wall Jets in Stationary Ambient Water

H. Kheirkhah Gildeh¹; A. Mohammadian²; I. Nistor, M.ASCE³; and H. Qiblawey⁴

Abstract: The main focus of this study is on the near-field flow and mixing characteristics of the thermal and saline wall jets. A numerical study of the buoyant wall jets discharged from submerged outfalls (e.g., from desalination plants) has been conducted. The performance of different Reynolds-averaged Navier-Stokes (RANS) turbulence models has been investigated and various k-ε, k-ω, and other turbulence models have been studied. The results of cling length, plume trajectory, temperature dilutions, and temperature and velocity profiles are compared to both available experimental and numerical data. It was found that two models perform best among the seven models chosen in this paper. According to the results from different simulations, the paper proposes corresponding relationships and comparative graphs that are helpful for a better understanding of buoyant wall jets. DOI: 10.1061/(ASCE)HY.1943-7900.0000871. © 2014 American Society of Civil Engineers.

Author keywords: Wall jets; Salinity; Effluent discharge; Turbulence models; Linear eddy viscosity models (LEVMs); Reynolds stress models (RSMs).

Introduction

Liquid wastes discharged from industrial outfalls are categorized in two major classes based on their density. One type is the effluent that has a higher density than the ambient water body. In this case the outfall jet has a tendency to sink as a negatively buoyant plume. The second type is the effluent that has a lower density than the ambient water body and which is hence defined as a buoyant jet that causes the plume to rise (Bleninger et al. 2010). This is, for instance, liquid waste discharged from multistage flash (MSF) desalination plants, that is, high-temperature water (containing salinity as well) that is generated by the cooling systems of the plant.

When a buoyant jet is discharged into a marine environment at a point remote from any boundary of the domain considered, it rises and mixes with the ambient fluid because of the turbulent flow generated by the presence of shear stresses that develop around the jet-ambient liquid interface. Nevertheless, if the effluent is discharged in touch with a solid horizontal boundary, it is subjected to the Coanda effect and clings to the floor for a while before the effect of buoyancy forces, which cause the jet to lift and rise away from this horizontal boundary (Foster and Parker 1970) as shown in Fig. 1. This is known as a turbulent wall jet, which is “a shear flow directed along a wall where, by virtue of the initially supplied momentum, at any downstream station, the streamwise velocity over some region within the flow exceeds that in the external stream” (Launder and Rodi 1981). Such a wall jet is widely used in practice. For instance, this simple configuration of effluent discharge is practical to use as a submerged pipe, which is secured close to the bottom for ease of construction. One of the examples is the disposal of marine tailing, which has been extensively discussed by Findikakis and Law (1998). This configuration may also be of interest in the transport of pollutants inside storm-water hydraulic conveyance systems in which near-bottom discharges are common practice. This topic has been extensively studied in the literature, which is outlined in the following.

Due to the extensive applications of wall jets, a large number of studies have been performed on buoyant jets. Sharp (1975) and Sharp and Vyas (1977) are some of the pioneers who considered submerged outfall of thermal effluents as a buoyant wall jet. Their study focused on the properties of a buoyant jet discharged immediately above a horizontal surface. The jet behaved as a wall jet in the initial part of its trajectory; nonetheless, after rising from the surface to which it initially clung, the jet acted as a normal free jet. Launder and Rodi (1981, 1983) reviewed the experimental literature up to about 1980. Moreover, they did various experimental investigations on the turbulent wall jet. The inlet in their experiments was plane as well as radial. They also reviewed numerical calculations of turbulent wall jets such as Boussinesq viscosity models (BVMs) and Reynolds-stress closures. Sobey et al. (1988) studied buoyant discharge in perhaps its most elementary geometry—a round buoyant jet discharging horizontally from a vertical side wall into a stationary water body on a flat bed. Eriksson et al. (1998) investigated both the mean and turbulent velocity fields of a wall jet on a smooth surface. In the region 40 < x/H < 150, their data were found to be reasonably self-similar. Law and Herlina (2002) applied novel experimental equipment, a combination of particle image velocimetry (PIV) and planar laser-induced fluorescence (PLIF) approaches in order to study a circular three-dimensional turbulent wall jet. Kwon and Seo (2005) also used a PIV system to investigate the behavior of a nonbuoyant circular water jet discharged from a contraction nozzle. From the experimental results, they argued that the cross-sectional profile of the axial velocity for a laminar flow near the nozzle did not show a
top-hat distribution, whereas the profiles with Reynolds number higher than 437 were almost top hat. They demonstrated that the Gaussian curve was properly approximated only for the turbulent jets and not for the laminar or transitional flows. Michas and Papanicolaou (2009) investigated horizontal, round turbulent buoyant jets that discharge into a homogeneous, calm ambient fluid. They studied both jetlike and plume-like properties of flow to include the full extent of applications. The mixing and geometry characteristics such as trajectories were obtained with video imaging, turbulence properties measured with fast response thermistors, and dilution factors were evaluated. They found that flow resembles a jet-like behavior in the horizontal regime, while the mean and turbulent temperature profiles become asymmetrical in the transition and the vertical regime. More recently, Rostamy et al. (2011) investigated the effect of surface roughness on the mean velocity and skin friction characteristics of a plane turbulent wall jet experimentally using laser Doppler anemometry (LDA). They analyzed the streamwise evolution of the flow on smooth and rough walls. Their study suggests that for the transitionally rough regime, roughness effects are significant but mostly confined to the inner region of the wall jet.

All of the previously mentioned studies were conducted experimentally. However, numerical simulations on buoyant wall jets are still being studied and require further investigation. Kim et al. (2002) investigated the mixing processes of a buoyant jet discharged from a submerged single port using a three-dimensional hybrid model. In the proposed hybrid model, the initial mixing was simulated by a jet integral method, and the advection-diffusion process was simulated using a particle-tracking method. Their proposed model was verified with their laboratory experiments, which were conducted for various conditions. They concluded that the simulated horizontal concentration distributions and minimum dilutions at the water surface were generally in agreement with the collected experimental data. However, trajectories simulated by the jet integral module of the hybrid model were in agreement with the measured trajectories when the velocity ratio was low. Maele and Merci (2006) applied standard and realizable $k$-$\varepsilon$ turbulence models and examined different types of buoyant plumes. They found out that the realizable $k$-$\varepsilon$ model performs better for the cases considered. Kim and Cho (2006) investigated buoyant flow of heated water discharged from surface and submerged side outfalls in shallow and deep water with a cross flow. They used FLOW-3D, which is a commercial computational fluid dynamics (CFD) package and a renormalization group (RNG) $k$-$\varepsilon$ model was applied for turbulence closure. Xiao et al. (2009) developed a fast non-Boussinesq integral model for the horizontal turbulent buoyant jets by using a CFD code named GASFLOW. Based on Sharp’s experimental data (Sharp 1975; Sharp and Vyas 1977), recently, Huai et al. (2010) investigated horizontal buoyant wall jet numerically. They only applied one turbulence model, realizable $k$-$\varepsilon$, for their study. They presented results mainly for the near field and proposed some relationships between the distance and the dilution of velocity and temperature based on the numerical results. It is now understood that the jet velocity distribution in the vertical cross section includes two different flow regions. The region between the wall and the level of the maximum velocity is defined as the boundary layer, while the region above that level is the free mixing area. The velocity scale for the streamwise ($x$-$y$) and span-wise ($x$-$z$) profiles is represented by the maximum streamwise velocity, $U_{m0}$. As shown in Fig. 2, the vertical length scale is taken as $y_m/2$, which is the point along the $y$-coordinate where the velocity has a value $U_{m0}/2$. The lateral length scale is also represented by $z_m/2$, which is defined similarly.
Existing studies on turbulent wall jets mainly focused on the flow kinematic behavior, while mixing characteristics were rarely investigated. However, understanding of how a wall jet disperses is fundamental for buoyant wall jets. Hence, it would also be of interest to evaluate pollutant discharges in the form of jets moving close to a boundary. The turbulent wall jet is also a basic flow of fundamental interest for turbulence studies because of its two-scale character. The inner layer of the wall jet is similar to that of the turbulent boundary layer, while the outer layer resembles that of a free jet. The interaction of large turbulence scales in the outer layer with smaller scales in the inner layer creates a complicated flow field and determines the development of the wall jet. In order to investigate the mixing characteristics of these jets, various turbulence models including advanced ones are applied to evaluate the accuracy of these models to predict the flow fields. A total of seven Reynolds-averaged Navier-Stokes (RANS) turbulence models [four linear eddy viscosity models (LEVMs), one nonlinear eddy viscosity model, and two Reynolds stress models (RSMs)] have been studied and then the best model is introduced based on the model verification with previous experimental and numerical data. Therefore, the main objective of the paper is to evaluate the accuracy of various turbulence models in simulating buoyant jets. For example, RSM models such as Launder-Reece-Rodriguez-Law (LRR) and Launder-Gibson [e.g., Launder et al. (1975) and Gibson and Launder (1978)] have been rarely studied before and this paper concludes that they are suitable choices for such applications. For paper brevity, the results of nonlinear k-ε turbulence model are not presented (this model performed poorly compared with other models and overpredicted most parameters studied in this paper).

Mathematical Formulation

Governing Equations

The governing equations are the well-known Navier-Stokes equations for three-dimensional, incompressible fluids as follows:

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

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

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

where \( u, v, \) and \( w \) are mean velocity components in the \( x-, y-, \) and \( z- \)directions, respectively; \( t \) is time; \( P \) is fluid pressure; \( \nu_\text{eff} = \) effective kinematic viscosity \( (\nu_\text{eff} = v_t + \nu) \); \( v_t \) is turbulent kinematic viscosity; \( g \) is the gravity acceleration; \( \rho \) is fluid density; and \( \rho_0 \) is reference fluid density.

The equations are divided by \( \rho \) and the buoyancy term is added to the momentum equation in vertical direction (\( y \)-coordinate) to account for variable density effects.

Temperature and concentration evolutions are modeled using the advection-diffusion equation as

\[
\frac{\partial C}{\partial t} + u \frac{\partial C}{\partial x} + v \frac{\partial C}{\partial y} + w \frac{\partial C}{\partial z} = D \left( \frac{\partial^2 C}{\partial x^2} + \frac{\partial^2 C}{\partial y^2} + \frac{\partial^2 C}{\partial z^2} \right)
\]

with

\[
k_\text{eff} = \frac{\nu_t}{P_r} + \frac{u}{P}
\]

where \( T \) is fluid temperature; \( k_\text{eff} \) is heat transfer coefficient; \( C \) is fluid concentration (salinity); \( D \) is diffusion coefficient; \( P \) is Prandtl number; and \( P_r \) is turbulent Prandtl number. The Prandtl number is a dimensionless number that is used often in thermal jet studies. This number is the ratio of momentum diffusivity (kinematic viscosity) to thermal diffusivity. In the present study, it was numerically found that the results are not significantly sensitive to \( P \) and \( P_r \) within the range of (0.6–1). Thus, both coefficients were set to 1.0.

Density Calculation

In the case of the buoyant wall jet released from submerged outfalls of industrial plants, the flow is characterized by several important parameters. The outfall diameter \( D \), jet initial velocity \( U_0 \), jet initial density \( \rho_0 \), jet initial temperature \( T_0 \), jet initial concentration \( C_0 \), and the densimetric Froude number \( F_d \) are the most important parameters of discharge. The densimetric Froude number is calculated as

\[
F_d = \frac{U_0}{\sqrt{g' D}}
\]

\[
g' = g \frac{\rho_a - \rho_0}{\rho_0}
\]

The ambient water is assumed to have a uniform temperature and concentration that are less than that of the jet (\( T_a < T_0 \) and \( C_a < C_0 \)), while its density \( \rho_a \) is higher than that of the jet (\( \rho_a > \rho_0 \)). In this paper, the density is calculated for both the jet and the ambient water according to the equation of state of seawater proposed by Millero and Poisson (1981).

Temperature and concentration (salinity) change the density of the discharges of many industrial plants simultaneously. Moreover, the densimetric Froude number with which many flow characteristics are correlated is a function of density. Therefore, it is density of the discharge that controls the jet characteristics. Thus, it is expected that the jets with various temperature and salinity values but the same density behave similar to each other when they are discharged to the same ambient water.
Computational Domain and Boundary Conditions

The sketch of the numerical model has been shown in Fig. 1 with its coordinate system. Only half of the wall-jet domain is considered in this study due to the symmetry of the problem. The dimensions of the computational domain are chosen based on the available experimental setups. The numerical simulations were performed in a tank with dimensions of 2-m length, 0.8-m width, and 1.2-m depth. A refined mesh is used for all simulations to better capture velocity and temperature and concentration characteristics in the near-field zone. For most of the results presented in this paper, a total of 800,000 hexahedrals (structured) cells were used and found using mesh independency analysis to be adequate in resolving the flow features.

For the inlet, the boundary conditions are \( u = U_0, v = w = 0, T = T_0, C = C_0, k = 0.06u^2, \varepsilon = 0.06u^3/D, \) and \( \omega = \varepsilon/k. \) The inlet values for \( k \) and \( \varepsilon \) are chosen based on Huai et al. (2010). Moreover, in order to make sure the inlet velocity profile is fully developed at the nozzle, a 0.5-m pipe has been modeled first. Regarding the solver at the outlet section, a zero-gradient boundary condition perpendicular to the outlet plane is defined for \( u, v, w, k, \varepsilon, \omega, T, \) and \( C. \) Moreover, for the wall boundaries the standard wall functions are used for \( k, \varepsilon, \) and \( \omega \) and the no-slip condition is considered. The wall functions used in the present research are based on the standard wall function model presented by Launder and Spalding (1972) and the rough wall model derived from it (the effect of roughness is discussed in “Application of the Model”). Finally, the symmetry boundary was modeled using zero-gradient conditions because the symmetry plane condition specifies that component of the gradient normal to the plane should be zero.

Numerical Algorithm

The governing equations are numerically solved using the finite-volume method. The solver, which is used within OpenFOAM, is the modified pisoFoam (OpenFOAM 2.1.1). This solver is a transient solver for incompressible flow. The code first predicts the velocity field by solving the momentum equations. Pressure is then found by solving Poisson’s equation in Issa’s pressure-implicit with splitting of operators (PISO) algorithm via an iterative process. Rather than solving all of the coupled equations in a coupled or iterative sequential fashion, PISO splits the operators into an implicit predictor and multiple explicit corrector steps. At each time step, velocity and temperature are predicted, and then pressure and velocity are corrected. The velocity is predicted implicitly because of the greater stability of implicit methods, which means that a set of coupled linear equations, expressed in matrix-vector form as \( Ax = b, \) are solved. More information about Issa’s PISO algorithm can be found in Ferziger and Peric (2002), Issa (1985), Issa et al. (1986), and Oliveira and Issa (2001). The temperature and concentration equations [Eqs. (5) and (6)] are then solved using the finite-volume method.

The temporal term has been discretized by first-order implicit Euler scheme. The advection and diffusion terms are discretized by the standard finite-volume method using Gaussian integration with a linear interpolation scheme for calculating values at face centers from cell centers.

For the pressure field, the preconditioned conjugate gradient (PCG) method is used to solve the linear system. The preconditioned bioconjugate gradient (PBiCG) method has been used for other fields: \( U, T, C, k, \varepsilon, \) and \( \omega. \) In order to enhance the rate of convergence for iterative solvers, the diagonal incomplete Cholesky (DIC) preconditioner is used to calculate the pressure field. This is a simplified diagonal-based preconditioner for symmetric matrices. The diagonal incomplete lower upper (DILU) preconditioner is used for the other fields: \( U, T, C, k, \varepsilon, \) and \( \omega, \) which mostly include asymmetric matrices to be solved.

For all the simulations in this study, a physical time step of \( 1 \times 10^{-3} \text{s} \) was used, which leads to a Courant-Friedrichs-Lewy (CFL) number less than 0.5 for stability considerations. Smaller time steps were also tried but the results remained the same. The convergence criteria for each time step were set such that the residuals for the velocity components and pressure are \( 1 \times 10^{-5} \) and \( 1 \times 10^{-6} \), respectively.

Results and Discussions

Five different cases have been numerically simulated. All seven turbulence models have been applied to each case and the comparative results are presented. The first three cases are pure thermal jets, but the last two cases include salinity as well. The density of the jet is a function of both temperature and salinity. However, it is numerically found that the final jet density is important when jet properties are correlated to the densimetric Froude number regardless of the contribution of temperature and/or salinity based on Eq. (8). In order to investigate this, Case 4 is set as follows: A value for salinity is added to the jet in Case 3 (\( \rho = 973.89 \text{ kg/m}^3 \)) such that the density remains the same as Case 1 (\( \rho = 978.48 \text{ kg/m}^3 \)). The characteristics of five cases are summarized in Table 1.

Cling Length and Trajectory

As the fluid leaves the inlet that is attached to the horizontal wall, water entrainment occurs from all directions to the jet except for the wall region. This causes a lower pressure on the wall than at the top of the jet. This keeps the jet on the wall up to a point where the top suction pressure decreases and the buoyancy force becomes larger than the pressure difference. Therefore, the wall buoyant jets can be divided in three regions: (1) initial jet region, (2) wall jet region, and (3) free jet region. The initial jet region is the distance from the inlet to the point where the velocity profile is almost uniform and equal to the maximum initial velocity. The wall jet region itself is divided into two regions as explained in the following. The first region is the Wall Jet Region I, which spans from the end of the initial jet region to the point where the jet centerline leaves

<table>
<thead>
<tr>
<th>Table 1. Characteristics of the Different Simulated Cases</th>
</tr>
</thead>
<tbody>
<tr>
<td>Case</td>
</tr>
<tr>
<td>1</td>
</tr>
<tr>
<td>2</td>
</tr>
<tr>
<td>3</td>
</tr>
<tr>
<td>4</td>
</tr>
<tr>
<td>5</td>
</tr>
</tbody>
</table>
the horizontal level and starts rising. Wall Jet Region II spans from the latter point to the point where the outer layer of jet leaves the floor. The free jet region starts after the wall jet region. These regions are shown in Fig. 1.

Cling length is often defined as the distance between the inlet and the position where the floor (wall) temperature has the condition of \( \frac{T - T_a}{T_0 - T_a} = 3\% \) [e.g., Huai et al. (2010)]. The numerical results obtained for the cling length are presented in Fig. 3 and are compared with experimental and other numerical results. The axes are dimensionless and \( x \)-axis represents the densimetric Froude number.

The results show good agreement with both the experimental and other researchers’ numerical data. However, results show that for higher Froude numbers, the experimental cling length values obtained by Sharp and Vyas (1977) are smaller than the numerical results published by Huai et al. (2010) and are more consistent with the numerical results obtained in the present study. Moreover, there is no big difference between Case 1 and Case 4 as was expected. The cling length for the jet with salinity is less than 5% shorter than the one with only temperature. The relationship between \( L/D \) and \( F_d \) for each turbulence model is given in Table 2. Sharp and Vyas (1977) suggested the same relationship, \( L/D = 3.2F_d \).

As shown in Table 2, all the turbulence models have a smaller value for \( L/D \) than the experimental data. The cling length value for shear stress transport (SST) \( k-\omega \) is the smallest one, while the other models are close to each other. However, the Launder-Gibson model is the closest one to the experimental result.

Predicting the trajectory of jets is one of the key objectives in jet studies. This is also important in the design procedures for disposal outfalls because it provides the distance from the nozzle to where the jet reaches to water surface. This is critical, especially in regions with shallow water depths where depth is not enough to completely dilute the effluent. The trajectory of the two cases (Cases 2 and 3) as well as the results of several other studies are shown in Fig. 4. The axes are normalized using \( L_M \), which is momentum length scale and is calculated as

\[
L_M = \frac{M_0^{3/4}}{B_0^{1/2}}
\]

where \( M_0 \) and \( B_0 \) = kinematic momentum flux and the buoyancy flux, respectively, and are defined as

\[
M_0 = U_0^2 \frac{D^2}{4}
\]

\[
B_0 = Q_0 \rho_0 g
\]

in which \( Q_0 \) is the discharge volume flux and is calculated as

\[
Q_0 = \frac{U_0^2 \pi D^2}{4}
\]

Table 2. Cling Length Relationship according to the Turbulence Model Used

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>Standard k-( \varepsilon )</th>
<th>RNG k-( \varepsilon )</th>
<th>Realizable k-( \varepsilon )</th>
<th>SST k-( \omega )</th>
<th>Launder-Gibson</th>
<th>LRR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cling length</td>
<td>( L/D = 2.68F_d )</td>
<td>( L/D = 2.76F_d )</td>
<td>( L/D = 2.65F_d )</td>
<td>( L/D = 2.50F_d )</td>
<td>( L/D = 2.79F_d )</td>
<td>( L/D = 2.70F_d )</td>
</tr>
</tbody>
</table>

Fig. 4. Centerline trajectory: Froude number approximately (a) 12, (b) 20
The trajectory results obtained using RSMs as well as realizable \( k-\varepsilon \) turbulence models are much more accurate than the other models. Except for the SST \( k-\omega \) model, the results of this study are in better agreement with experimental data than the numerical results of Huai et al. (2010). It can be concluded that isotropic character of eddy viscosity in the \( k-\varepsilon \) models makes them insensitive to the orientation of the turbulence structure and its transporting and mixing mechanisms. However, RSMs account automatically for the effects of the stress anisotropy, which are important for stress-induced secondary flows as well as changing in the streamline trajectory (Hanjalic 1994). This can result in a more accurate calculation of stresses, especially on the jet-ambient interface where the pressure difference exists and controls the jet trajectory. Because the RSMs consider different viscosity value for each direction, therefore, the jet behavior seems to be calculated more accurately in these models.

Velocity Characteristics

Streamwise Velocity Profiles

The streamwise (\( x-y \)) velocity profiles along the centerline of the buoyant wall jet were extracted from different simulations. Because each case has seven different submodels (seven turbulence models for each case), only the results for one case (Case 3) are presented in the following in most figures for brevity. The results of velocity field have been obtained along different jet sections in the \( x \)-direction (different values of \( x/D \)) at the plane of symmetry. In Fig. 5, \( U_m \) is the velocity component in the \( x \)-direction (along \( y \) at central plane), \( U_{m0} \) is the maximum of \( U_m \) values and its ordinate is \( y \). Moreover, \( y_{m/2} \) is the velocity half-height, which is the height of \( U_m = U_{m0}/2 \). On the abscissa and ordinate, \( U_{m0} \) and \( y_{m/2} \) are taken as the velocity and length scales, respectively. All streamwise velocity profiles show self-similarity and are in good agreement with experimental results by Law and Herlina (2002) as seen in Fig. 5. Other previous studies such as Rajaratnam and Pani (1974), Padmanabham and Gowda (1991), and Abrahamsson et al. (1997) are also in agreement with the results shown in Fig. 5. Verhoff’s (1963) empirical equation, which is proposed for a two-dimensional wall jet, is also in good agreement with the results obtained in the current study. The following equation was suggested by Verhoff:

\[
\frac{U_m}{U_{m0}} = 1.48 \left( \frac{y}{y_{m/2}} \right)^{1/7} \left[ 1 - \text{erf} \left( \frac{0.68 y}{y_{m/2}} \right) \right]
\]  

From the existing agreement between Verhoff’s formula and experimental and present study results, it can be concluded that there is no significant difference in velocity profiles between two-dimensional and three-dimensional wall jets at the symmetry plane (central plane).

Among different turbulence models’ results shown in Fig. 5, the results of the SST \( k-\omega \) model are not as accurate as those of the other models. On the other hand, the LRR and realizable \( k-\varepsilon \) models’ results are in better agreement with the experimental data and theory than the results of other turbulence models. As seen in Fig. 5, despite the slight scatter the profiles exhibit a good similarity in different values of \( x/D \). The deviation for larger values of \( x/D \) may be due to the buoyancy-induced distortion because these deviations mostly occur at higher elevations (i.e., \( y/y_{m/2} \) > 1) where buoyancy dominates the flow.

Spanwise Velocity Profiles

The spanwise velocity profiles were obtained at the height where the inlet centerline is located. This is the height at which the maximum values of the velocity and temperature are expected. The results for different turbulence models are shown in Fig. 6 where \( z_{m/2} \) is the velocity half-width corresponding to \( U_m = U_{m0}/2 \). The profiles further downstream become more Gaussian with the maximum value at the centerline. They are also in better agreement with experimental data compared with the profiles from areas closer to the inlet. All turbulence models show better results than the results of the numerical model of Huai et al. (2010). The difference in \( k-\varepsilon \) turbulence models is mainly due to the calculation of the eddy viscosity. In the standard \( k-\varepsilon \) model, the eddy viscosity is based on a single turbulent length scale, while in reality all scales of motion affect the turbulent diffusion. Using the renormalization group method, the RNG \( k-\varepsilon \) model is able to deal with smaller-scale effects. However, in the realizable \( k-\varepsilon \) model, considering a non-constant value for \( C_\mu \) might result in a more accurate estimation of turbulent viscosity that is different and more accurate than standard and RNG \( k-\varepsilon \) models.

The Goertler solution for the free jet adapted from Schlichting (1979) is also shown in Fig. 6 with solid line. The present study is in a better agreement with the Goertler solution than the experimental data published by Law and Herlina (2002). Moreover, the other experimental measurements obtained by Rajaratnam and Pani (1974) and Padmanabham and Gowda (1991) showed very good agreement with the analytical solution proposed by Goertler and the results of this study. Similar to previous results presented in this paper, the SST \( k-\omega \) model does not accurately predict and underestimates the velocity field. Similar to \( k-\varepsilon \) models, the \( k-\omega \) turbulence model presumes a linear relationship between the turbulent stresses and mean rate of strain. This can cause the inability to mimic the preferentially oriented and geometry-dependent effects of eddy-flattening and squeezing mechanisms due to the proximity of solid surface (Hanjalic 1999). The results of realizable \( k-\varepsilon \) and LRR turbulence models are close to each other and more accurate. These two models are slightly better than other models especially for \( z/b > 1 \). Like streamwise profiles, the spanwise profiles are independent of the Froude number.

Fig. 7 shows the \( w \)-velocity distribution for one of the cases (Case 3). The \( w \)-velocity distribution profiles also appear to be independent of Froude number. As shown in Fig. 7, the maximum value of the \( w \)-velocity component occurs near the plane of symmetry.

Villafruela et al. (2008) reported that \( w \)-velocity values increase from the plane of symmetry (\( z = 0 \)) to a maximum value at approximately \( z = 0.8 z_{m/2} \). This is similar to the experimental results published by Abrahamsson et al. (1997). The maximum value for experimental results by Law and Herlina (2002) was reported to have occurred at the half-width of the wall jet. However, as shown in Fig. 7, the results of the current study are in a better agreement with those of Abrahamsson et al. (1997) than those of Law and Herlina (2002). The present study results are similar to the experimental data for larger values of \( x/D \). Again, the LRR turbulence model provides the most accurate results for the \( w \)-velocity distribution. Based on experimental data, Law and Herlina (2002) and Launder and Rodi (1981) inferred the possibility of the formation of a secondary vortex, where the flow is directed away from the symmetry plane further away from the centerline. Because the stress anisotropy plays a dominant role in governing secondary motions like the longitudinal vortices, the RSMs (mostly LRR model here) show better agreement with experimental data than LEVMs. Moreover, the influence of viscosity and wall proximity on the turbulent motion is different by nature, and because of high anisotropy.
Fig. 5. Self-similarity of streamwise velocity profiles for different turbulence models: (a) $k$-$\varepsilon$; (b) RNG $k$-$\varepsilon$; (c) realizable $k$-$\varepsilon$; (d) SST $k-\omega$; (e) Launder-Gibson; (f) LRR
Fig. 6. Comparison of spanwise self-similarity of $U$ profile at $y = y_m$ for different turbulence models: (a) $k$-$\varepsilon$; (b) RNG $k$-$\varepsilon$; (c) realizable $k$-$\varepsilon$; (d) SST $k\omega$; (e) Launder-Gibson; (f) LRR
Fig. 7. Comparison of spanwise $w$-velocity (velocity in $z$-direction) profile at $y = y_m$: (a) $k-\varepsilon$; (b) RNG $k-\varepsilon$; (c) realizable $k-\varepsilon$; (d) SST $k\omega$; (e) Launder-Gibson; (f) LRR
of turbulence in the near-wall region, the RSMs that allow computation of each stress component have much better prospects for simulating wall proximity effects (Bradshaw 1987).

Decay of Maximum Velocity

According to Sforza and Herbst (1970), longitudinal spreading of a wall jet is divided into three regions based on the maximum velocity decay rate. These regions include the potential core region, the characteristic decay region, and the radial-type decay region. Fig. 8 focuses on the third region, which is farther away from the inlet. The velocity decay is almost linear in this region. The rate of decay along the plane of symmetry, between 20D and 50D, is shown for six turbulence models.

The results of this study are in good agreement with the experimental results, especially for realizable $k$-$\varepsilon$ and LRR turbulence models. The best curve fit for the LRR model is shown in Fig. 8, which is very close to the curve fit of experimental data and is formulated as follows:

$$U_{d0}/U_0 = 13.99 \left( \frac{D}{x} \right)^{-1.16}$$

The best-fit curves for different studies and all the different turbulence models used are presented in Table 3. As shown, the results of numerical models are in good agreement with the results of almost all experimental studies, especially with those of Davis and Winarto (1980) and of Padmanabham and Gowda (1991).

Dilution and Temperature Characteristics

Dilution Characteristics

Dilution is related to the amount of water entrainment achieved by the jet. Dilution is defined as [e.g., Abassi et al. (2010) and Huai et al. (2010)]

$$S = \frac{T_0 - T_a}{T - T_a}$$

where $T_0$ = initial jet temperature; $T_a$ = ambient water temperature; and $T$ = temperature at the computational mesh. Generally, when the jet is discharged, it behaves like a pure jet (called jetlike flow) for a while and after passing a transient condition, which is named jet-to-plume-like flow, due to buoyancy forces, it behaves like a pure plume (called plume-like flow). Because of the high momentum force, the dilution in the jetlike region is less than in the other two regions. Water entrainment reaches the jet centerline at the end of the transient region and a higher dilution rate will subsequently occur.

Fig. 9 shows the temperature dilution values along the centerline at the plane of symmetry and at the cross section $y/D = 35$. The results are in good agreement with the experimental data of Sharp (1975) for the lower Froude numbers for which the experimental data is available. At lower Froude numbers, the results of realizable $k$-$\varepsilon$ and LRR are in good agreement with the theoretical solution proposed by Sharp and Vyas (1977). However, the results of the current study underpredict the dilution rate for higher Froude numbers when compared with the results of the theoretical solution. Because experimental data are not available for higher Froude numbers, the numerical results of Huai et al. (2010) are used for comparison in that range. As shown in Fig. 9, the numerical results of Huai et al. (2010) are in good agreement with the present study for a Froude number of approximately 40. As seen from this figure, when the jet includes salinity (Case 4), the temperature dilution is slightly lower (approximately 3%). This obviously occurs due to presence of concentration, which needs to be diluted, as well as temperature.

Streamwise Temperature Profiles

Dimensionless results for temperature profiles are presented in this section. The self-similarity temperature profiles along the centerline at the symmetry plane are shown in Fig. 10. As can be seen in this figure, the temperature decays exponentially from a maximum value of $T_m/T_{m0}$ at the floor to zero at a higher height. The results are normalized in both axes. $T_m$ is the temperature

![Fig. 8. Comparison of the maximum velocity decay; $U_0$ is the velocity at the inlet.](image)

![Fig. 9. Comparison of temperature dilution at the symmetry plane for different $F_d$ numbers, $y/D = 35$; for $F_d = 42.33$, the black color denotes Case 1](image)

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>$n$</td>
<td>1.38</td>
<td>1.19</td>
<td>1.18</td>
<td>1.23</td>
<td>1.20</td>
<td>1.16</td>
<td>1.07</td>
<td>1.10</td>
<td>1.00</td>
<td>1.15</td>
<td>1.15</td>
</tr>
<tr>
<td>$U_{d0} = ax^{-n}$</td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
<td></td>
</tr>
</tbody>
</table>
Fig. 10. Comparison of self-similarity streamwise temperature profiles for different turbulence models: (a) $k$-$\varepsilon$; (b) RNG $k$-$\varepsilon$; (c) realizable $k$-$\varepsilon$; (d) SST $k$-$\omega$; (e) Launder-Gibson; (f) LRR
Fig. 11. Comparison of self-similarity spanwise temperature profile at $y = y_m$ for different turbulence models: (a) $k$-$\varepsilon$; (b) RNG $k$-$\varepsilon$; (c) realizable $k$-$\varepsilon$; (d) SST $k$-$\omega$; (e) Launder-Gibson; (f) LRR
along the y-direction at the symmetry plane, $T_{m0}$ the maximum temperature at the centerline along y, and $y_{m}/2$ the temperature half-height.

Generally, a Gaussian profile is assumed (Shao and Law 2010) for the temperature distribution along the cross section at the central plane when the jet has entered the zone of established flow (ZEF). As shown in Fig. 5, the general trend of all six turbulence models’ results follows the trend of the experimental data. Similar to the velocity field, for $1.5 \leq y/y_{m}/2 \leq 2.5$ and larger values of $x/D$, the temperature results of the present numerical study are higher than experimental results. Similar to the velocity profiles, the deviations for larger values of $x/D$ are assumed reasonable due to the buoyancy forces that become more significant compared to momentum effects when the jet advances and starts rising up from the wall surface. However, the realizable $k-\varepsilon$ and LRR models can be considered to have performed best among the selected turbulence models, while the SST $k-\omega$ performed the worst. Similar to the velocity profiles, temperature profiles appear to be independent of the Froude number.

Spanwise Temperature Profiles

Spanwise temperature profiles were extracted from the horizontal plane at the height of the inlet centerline ($y = y_{m}$). The spanwise self-similarity profiles are presented in Fig. 11, which shows that in the range of $3 \leq x/D \leq 50$ the profiles exhibit a shape similar to a top hat. Launder-Gibson and LRR models provide better results than the other turbulence models used and are in a better agreement with experimental data. This is probably due to the anisotropic nature of RSMs, as discussed previously, which makes these models more capable than others. Similar to the streamwise temperature profiles, it was found that the spanwise profiles are also independent of Froude number.

Maximum Temperature Decay

The temperature decay along the centerline for different turbulence models are compared with the experimental results by Law and Herlina (2002) in Fig. 12. Reasonable agreement is between the numerical and experimental results, except for the SST $k-\omega$ model, which overpredicts the temperature values at the centerline for the interval $10 \leq x/D \leq 30$. The best-fit curve for all turbulence models’ results as well as experimental results is presented in Table 4. The best turbulence models for this case are, again, the realizable $k-\varepsilon$ and LRR as expected based on previous comparisons.

In order to quantify the agreement between the current numerical study with the previous experimental data by Law and Herlina (2002), the $R$-squared values for all turbulence models are summarized in Table 5.

**Application of the Model**

In this section, the physical changes of the model and their effect on the jet geometry, mixing, and dispersion process are evaluated for a few parameters. Choosing these parameters was attempted to be as practical as possible. Two turbulence models have been selected to perform the new numerical experiments with: realizable $k-\varepsilon$ turbulence model from the LEVMs and LRR turbulence model from the RSMs. The following experiments have been examined in the numerical model.

**Effects of Bed Inclination**

One of the engineering interest points of view in discharging of effluent is the bed nonuniformities regarding the bathymetry of the disposal area. In fact, the seabed is not flat in the zone of near field and this may affect the buoyant jet properties. To investigate this,
Due to water entrainment from the bottom of the jet, the relationship proposed by Huai et al. (2010) is satisfied at a point closer to the nozzle than the one of the flat bottom (Table 6). Therefore, seabed inclination plays an important role in establishing the jet trajectory and consequently mixing properties of the jet.

**Effects of Bed Roughness**

Another application of the current numerical model is to simulate the bottom roughness. The majority of the previous experimental and numerical studies on the wall jet focused on the smooth wall. In this section, to investigate the effect of roughness in the model, two different rough walls were applied and then the numerical results for maximum velocity and temperature decays have been compared with those from previous sections (with smooth walls).

The effect of roughness is considered by means of two variables: the roughness height \(K_s\) and the roughness constant \(C_r\). In the solver, given the roughness parameters \(K_s\) and \(C_r\), required values are evaluated using the corresponding formulas. The modified law of the wall is then used to evaluate the shear stress at the wall and other wall functions for the mean temperature and turbulent quantities. In this study the value for \(C_r\) corresponds to uniform sand-grain roughness. However, in order to see the effect of roughness changes and comparing the results to smooth wall, two different values for \(K_s\) have been chosen (Folk 1954).

Maximum velocity and temperature decay for two turbulence models on smooth as well as rough walls is shown in Fig. 14. In general, the results of rough walls lie under the results of smooth wall jet. Therefore, it can be inferred that the decay rate (deceleration rate) of maximum jet velocity on rough walls is faster than that on smooth walls. This has been observed in the experimental studies of Dey et al. (2010) as well. It is important to discuss that, even though the velocity decay profile belongs to the cross sections \(20 < x/D < 50\), the values for smooth and rough walls are very close to each other in the range of \(x/D < 25\). The roughness effect on the temperature field seems to be less than that on the velocity field. However, similar to the velocity profiles, the effect of roughness is greater for further downstream. In fact, it is for \(x/D > 20\) that the difference of smooth and rough wall becomes meaningful.

Table 6. Cling Length Relationship for Sloped Bed

<table>
<thead>
<tr>
<th>Turbulence model</th>
<th>Realizable k-ε</th>
<th>LRR</th>
</tr>
</thead>
<tbody>
<tr>
<td>Cling length</td>
<td>(L/D = 1.79 F_d)</td>
<td>(L/D = 1.86 F_d)</td>
</tr>
</tbody>
</table>

Fig. 13. Centerline trajectory; Froude number approximately 20

Fig. 14. Comparison of the maximum velocity and temperature decay: \(K_s\) value in Rough1 is smaller than that in Rough2 (i.e., Rough1 is less rough than Rough2)
However, the difference between the two rough walls is small. Similar to the velocity decay, the temperature decays slightly faster when the roughness value is larger in both turbulence models.

**Effects of Space in the Model**

As mentioned previously, the dimension of the numerical model is chosen based on the physical flumes and half of the experiments are modeled due to symmetry of the problem. However, there may be differences in the experimental results when the space limitation is considered in the tank dimensions. For instance, the side walls’ distance from the jet centerline may affect the circulation in the tank (flume). This phenomenon also may exist in the physical experiments. Many different scenarios have been examined in the numerical experiments to see the effect of tank-size changes on the flow field as well as mixing characteristics of the plumes. In the first experiment, the effect of tank-width change has been studied by modeling the tank with closer and further side walls. As shown in Fig. 15, the original tank half-width was 0.4 m [Fig. 15(b)], and then to compare the results, the narrower half-width has been chosen as 0.2 m [Fig. 15(a)] and the wider half-width is 0.6 m [Fig. 15(c)]. As seen clearly from Fig. 15, the fluid circulation happens on the two sides of the jet centerline at the central plane, when the tank half width is 0.2 m. However, when the tank width is doubled, those circulations are vanished almost completely. Finally, for a tank width equal to the tank height, there is no circulation regarding the side walls. Although doubling the tank width does not double the numerical cost, it still increases the simulation time and one needs to consider the available computational resources and then choose proper dimensions for the numerical model.

The tank width has a slight effect on the temperature dilution as well. It is numerically found that the dilution rate in narrower tanks is lower than that in the wider ones. This may be due to decrease in tank width that does not allow the jet to be developed in width properly and also increases the surrounding forces from the ambient water on the jet edges. The result of the narrower tank is in better agreement with the Huai et al. (2010) numerical model (not shown in this paper) in which their tank width was close to this case.

A few other experiments were also conducted numerically to see the effect of side walls’ type. One of them is a case in which both side walls are assumed as a symmetry plane. The other one is a case in which the entire domain is simulated to see the probable changes when the full tank is modeled. The results from these experiments have been very close to the original cases and are not shown here.

The effect of jet evolution on the top of the tank was also investigated numerically with simulating cases with longer time as well as larger tank height (Fig. 16). As seen from Fig. 16(a), when the simulation time is increased, the top face of the jet touched the water surface and the intermediate fields are observed gradually. However, the intermediate field is beyond the scope of this study and the effect of this field on the near field downstream of the jet becomes negligible. On the other hand, when the height of the tank is increased [Fig. 16(b)] for the same simulation time as the original cases, the jet properties remain similar. This shows that when the tank height is increased more than a specific height, the difference in the properties of near field of the jet becomes negligible.

**Conclusions**

A detailed numerical study was conducted to investigate both the velocity and temperature fields of a three-dimensional thermal and saline wall jet. A finite-volume method was applied to solve the equations numerically. Seven different turbulence models were employed in order to evaluate the accuracy of RANS models when simulating the discharge of thermal or saline fluid disposals.
The numerical results were compared to previous experimental and numerical data. The results showed good agreement with the recent experimental data for velocity and temperature fields. The density of the jet is a function of both temperature and salinity. It is numerically found that the final jet density is important when jet properties are correlated to the densimetric Froude number regardless of the contribution of temperature and/or salinity based on Eq. (8). Dilution rates for temperature and maximum temperature decay were highly accurate when compared with experimental data. Jet trajectory differs from case to case and is related to discharge and ambient water characteristics. Efficient design for discharge systems may hence be possible based on the results of the jet trajectory. The numerical results of maximum velocity decay were found to be in good agreement with results of several experimental studies. Streamwise and spanwise profiles for velocity and temperature showed self-similarity after an initial distance from discharge point. The general shape of these profiles was also found to be independent of the Froude number. The streamwise temperature profiles followed a general Gaussian form in various values of $x/D$. In the case of spanwise profiles, temperature exhibited the same characteristics as the velocity. Controversies about the results of different turbulence models arise sometimes from the accuracy of the applied numerical schemes. Different schemes were tried to solve the different terms in the partial differential equations (PDEs) to get the most accurate results of the model. Model application for bed inclination resulted to a shorter clung length. Bed roughness was found to affect both velocity and temperature field. This influence was greater on the velocity decay of the wall jet. The numerical model could also capture the circulation in the tank regarding the decrease in the width. Moreover, he jet evolution on the intermediate field was numerically found to have no effect on the near-field properties of the turbulent wall jet. Among the different turbulence models, RSMs (mainly the LRR model) could successfully capture secondary flows and buoyancy-induced forces because these models account for the effects of the stress anisotropy. Finally, among the seven models examined in this study, the realizable $k$-$\varepsilon$ and LRR turbulence models were found to be the most accurate and capable of accurately modeling thermal and saline wall jets discharged into stationary ambient water.

Acknowledgments

This publication was made possible by NPRP grant #4-935-2-354 from the Qatar National Research Fund (a member of Qatar Foundation). The statements made herein are solely the responsibility of the authors. The authors are also grateful to the anonymous reviewers for their helpful comments.

Notation

The following symbols are used in this paper:

\[ L_M = \text{momentum length scale}; \]
\[ M_0 = \text{kinematic momentum flux}; \]
\[ P = \text{pressure}; \]
\[ P = \text{Prandtl number}; \]
\[ P_t = \text{turbulent Prandtl number}; \]
\[ Q_0 = \text{discharge volume flux}; \]
\[ R_0 = \text{discharge Reynolds number}; \]
\[ S = \text{salinity and dilution [Eq. (16)]}; \]
\[ T = \text{temperature on the numerical mesh}; \]
\[ T_m = \text{temperature values along a section}; \]
\[ T_0 = \text{temperature at source}; \]
\[ U_m = \text{velocity values along a section}; \]
\[ U_0 = \text{velocity at source}; \]
\[ u, v, w = \text{velocity in the } x-, y-, z-\text{directions, respectively}; \]
\[ x, y, z = \text{coordinates}; \]
\[ y_{m/2} = \text{velocity half-height}; \]
\[ y_{tm/2} = \text{temperature half-height}; \]
\[ z_{m/2} = \text{velocity half-width}; \]
\[ z_{tm/2} = \text{temperature half-width}; \]
\[ \varepsilon = \text{dissipation rate for } k \text{ in } k-\varepsilon; \]
\[ \mu = \text{dynamic viscosity}; \]
\[ \nu = \text{kinematic viscosity}; \]
\[ \rho = \text{density on the computational mesh}; \]
\[ \rho_a = \text{ambient water density}; \]
\[ \rho_0 = \text{discharge density}; \]
\[ \omega = \text{dissipation rate for } k \text{ in } k-\omega. \]

References


