Numerical Modeling of Vertical Buoyant Jets discharged from multiport diffusers

B. Ghodoosipour, A. Mohammadian & I. Nistor
Department of Civil Engineering, University of Ottawa, Canada

Abstract

Multiport diffusers are widely used for marine wastewater discharges. In order to study the behavior of such jets after the discharge point, the numerical model OpenFOAM was used and two different RANS (Reynolds-Averaged Navier-Stokes) turbulence models namely $k - \varepsilon$ and LRR models were tested to simulate jets discharged from uni-directional multiport diffusers. Results from cross-sectional variation of mean axial velocity and longitudinal variation of axial velocity were compared to existing experimental results performed with similar jet properties. In general results from both models show good agreement between numerical and experimental data when simulating cross-sectional variation of mean axial velocity which is the direction that no merging happens; however, LRR model was shown to work better in simulating the longitudinal variation of axial velocity which shows the fact that LRR is more successful in simulating jet behavior in the direction that merging happens.

Keywords: Numerical Modeling; Vertical Buoyant Jets; Multiport diffusers; Turbulence Models

1 INTRODUCTION

Outfalls and diffusers are being commonly used for discharging wastewater into water bodies. In order to reduce environmental impacts of the discharged effluent and to meet water quality standards, multiport diffusers are widely used. These diffusers consist of several ports placed close to each other injecting the wastewater as an array of turbulent buoyant jets. Multiport diffusers maximize the initial dilution of the waste by reducing pollutant concentration. Based on most regulations, high dilution should take place in a limited mixing zone. Hence the wastewater will only impact a certain small area. Various mixing processes will affect the concentration distribution of the disposed wastewater in the ambient water. Depending on the ports arrangement, several types of multiport diffusers exist. Among all, uni-directional diffuser is the most commonly used type (Lyu, 2003).

In practice, because of huge amount of wastewater being discharged into very deep oceans, it is usually impossible to design diffusers with sufficient port spacing which allows for their discharge to the water surface without interacting with the neighboring jets (Davidson and Pun, 2000). As the adjacent jets start to merge mixing process reduces considerably which decreases the efficiency of the diffuser. (Wang and Davidson, 1999). When it comes to analysis, multiple diffusers with small port spacing are usually substituted with an equivalent two-dimensional discharge known as slot jet (Jirka and Harleman, 1979). Such analysis will ignore the merging process which usually happens in some distance from the source.
However, flow characteristics in merging jets is different from that of slot jet especially in the beginning of merging process which may cause the analysis to be invalid in many cases.

CFD (Computational Fluid Dynamics) has shown to be able to model single port buoyant jets characteristics with a reliable accuracy (Kheirkhah Gildeh 2014, a,b), however, no attempt has been made for CFD modeling of multiport diffusers because of its complex geometry which requires very fine grids and as a result it is computationally expensive. Recently, having access to powerful computer resources, three-dimensional CFD simulation of multiport diffusers has started to become a feasible task.

This study aims to perform a 3-D CFD numerical modeling on uni-directional multiport diffusers using different turbulence models.

This paper is organized as follows. Section 2 explains flow properties of buoyant jets. The method used and the numerical model details are explained in Sections 3 & 4, respectively. Results from different turbulence models are presented and discussed in section 5. Some concluding remarks complete the study.

2 FLOW PROPERTIES OF BUOYANT JETS:

Local mean properties of jets change during the jet discharge. A schematic view of a vertically discharged uni-directional multiport diffuser is shown in figure 1. Flow in this case has an axisymmetric zone at the beginning of discharge. Then, jets begin to merge as they are travelling upward; this region is called transitional or merging zone. After some distance, jets totally merge and their behavior is no longer three-dimensional; this region is called two-dimensional region. Many researchers have studied jet behavior in axisymmetric and two-dimensional zones (Kotsovinos 1975). However there are few studies on the flow behavior in the merging zone (Wood et al., 1993).

Figure 1. Behavior of Buoyant Jets Discharged from Unidirectional diffuser
Time averaged velocity in the axisymmetric and also two-dimensional zones is self-similar and can be described by Gaussian distribution as:

\[ \frac{U}{U_c} = \exp\left[ -C \left( \frac{x}{z} \right)^2 \right] \]

In which \( U \) is the local axial velocity, \( U_c \) represents the local centerline velocity and \( x \) is the distance from the centerline. \( C \) is the Gaussian constant and is derived experimentally. As the jets start to merge, time averaged velocity no longer follows Gaussian distribution.

3 METHODOLOGY

This paper aims to study the accuracy of different turbulence models in predicting behavior of merging buoyant jets from multiport diffusers. For this purpose, experiments conducted by Lyu et al. (2013) were numerically modeled. These experiments were performed in a rectangular tank, 6m in length, 1.2 m in width and 0.8 m in height. Nozzles are 3 mm in diameter and 30 mm in length. The jets were vertically discharged at the depth of 700 mm. Heated water was used in these experiments to represent the wastewater effluent from the diffusers. The experimental conditions that are used both by Lyu et al. (2013) and in numerical simulation models during this study are presented in Table 1.

Table 1. Experimental Conditions used in numerical models (ambient depth, \( H = 0.7 \) m, port diameter, \( d_p = 3 \) mm, Lyu et al. (2013))

<table>
<thead>
<tr>
<th>Case</th>
<th>Port spacing ( \frac{P_s}{d_p} )</th>
<th>( \rho_a \left( \frac{kg}{m^3} \right) )</th>
<th>( \Delta \rho \left( \frac{kg}{m^3} \right) )</th>
<th>( \Delta_0 \left( \frac{m}{s^2} \right) )</th>
<th>( U_0 \left( \frac{m}{s} \right) )</th>
<th>( Fr_0 )</th>
</tr>
</thead>
<tbody>
<tr>
<td>MV10-2</td>
<td>10</td>
<td>992.6</td>
<td>6.037</td>
<td>0.059</td>
<td>0.101</td>
<td>7.59</td>
</tr>
</tbody>
</table>

Experimental conditions were repeated in the numerical simulations using the numerical model but to make the simulations feasible, port diameters were increased to 1cm and a scale ratio of 1/0.3=3.33 was used for the dimensions in a way that flume height was increased to 70 \( \times \) 3.33=233 cm, \( s/d \) and densimetric Froude number were kept constant as non-dimensional values, so port spacing was increased to 10 cm and \( U_0 \) was increased to 0.26 (to be able to keep \( Fr_0=7.59 \) by increasing the port diameter).

4 NUMERICAL MODEL

The Finite Volume Model (FVM) OpenFOAM (OPEN Field Operation And Manipulation) CFD model, with structured grid, was used to simulate the buoyant wall jet discharge. OpenFOAM is widely used for modelling and solving scientific problems described by partial differential equations. Two equations were added to the source code which will account for temperature transport and the impact of buoyancy on vertical momentum equation.
Two different Reynolds-averaged Navier-Stokes (RANS) turbulence models $k - \varepsilon$ and Launder-Reece-Rodi (LRR), that were shown to produce acceptable results in simulating single port buoyant jets by Kheirkhah Gildeh et al. (2014a,b) were used in this study.

### 4.1 Governing Equations

The model uses three-dimensional RANS equations as the governing equations for incompressible fluids, as follows:

**Continuity Equation:**

$$\frac{\partial u_x}{\partial x} + \frac{\partial u_y}{\partial y} + \frac{\partial u_z}{\partial z} = 0 \quad [2]$$

**Momentum Equations:**

$$\begin{align*}
\frac{\partial u_x}{\partial t} + u_x \frac{\partial u_x}{\partial x} + u_y \frac{\partial u_x}{\partial y} + u_z \frac{\partial u_x}{\partial z} &= \frac{\partial}{\partial x} \left[ \nu_e \frac{\partial u_x}{\partial x} \right] + \frac{\partial}{\partial y} \left[ \nu_e \frac{\partial u_x}{\partial y} \right] + \frac{\partial}{\partial z} \left[ \nu_e \frac{\partial u_x}{\partial z} \right] - \frac{1}{\rho} \frac{\partial P}{\partial x} \\
\frac{\partial u_y}{\partial t} + u_x \frac{\partial u_y}{\partial x} + u_y \frac{\partial u_y}{\partial y} + u_z \frac{\partial u_y}{\partial z} &= \frac{\partial}{\partial x} \left[ \nu_e \frac{\partial u_y}{\partial x} \right] + \frac{\partial}{\partial y} \left[ \nu_e \frac{\partial u_y}{\partial y} \right] + \frac{\partial}{\partial z} \left[ \nu_e \frac{\partial u_y}{\partial z} \right] - \frac{1}{\rho} \frac{\partial P}{\partial y} - g' \\
\frac{\partial u_z}{\partial t} + u_x \frac{\partial u_z}{\partial x} + u_y \frac{\partial u_z}{\partial y} + u_z \frac{\partial u_z}{\partial z} &= \frac{\partial}{\partial x} \left[ \nu_e \frac{\partial u_z}{\partial x} \right] + \frac{\partial}{\partial y} \left[ \nu_e \frac{\partial u_z}{\partial y} \right] + \frac{\partial}{\partial z} \left[ \nu_e \frac{\partial u_z}{\partial z} \right] - \frac{1}{\rho} \frac{\partial P}{\partial z}
\end{align*} \quad [3-4]$$

where:

$$g' = g \left( \frac{\rho - \rho_0}{\rho} \right) \quad [5]$$

$$\nu_e = \nu_t + v \quad [7]$$

with

$$\nu_t = \frac{k}{\varepsilon} \quad [6]$$

where $u_x, u_y, u_z$ are the components of the mean velocity in the $x, y, z$ directions, respectively, $P$ is the fluid pressure, $t$ is the time, $g'$ is the modified gravity acceleration, $g$ is the acceleration of gravity, $\rho$ is the fluid density, $\rho_0$ is the reference fluid density, $\nu_e$ is the effective kinematic viscosity of water, and $\nu_t$ is the turbulent kinematic viscosity. The effect of the variable density (buoyancy) in the vertical direction (y-coordinate) is considered and added in Eq. 4.

**Temperature equation:**

$$\frac{\partial T}{\partial t} + u_x \frac{\partial T}{\partial x} + u_y \frac{\partial T}{\partial y} + u_z \frac{\partial T}{\partial z} = k_e \left( \frac{\partial}{\partial x} \left[ \frac{\partial T}{\partial x} \right] + \frac{\partial}{\partial y} \left[ \frac{\partial T}{\partial y} \right] + \frac{\partial}{\partial z} \left[ \frac{\partial T}{\partial z} \right] \right) \quad [8]$$
where:

\[ k_e = \frac{V_e}{Pr_i} + \frac{V}{Pr} \]

and \( T \) is the fluid temperature, \( k_e \) is the heat transfer coefficient, \( Pr \) is the Prandtl number, and \( Pr_i \) is the turbulent Prandtl number. In this study, the \( Pr_i \) and \( Pr \) values are chosen to be 1.0 based on Kheirkhah Gildeh et al. (2014a,b).

4.2. Boundary conditions

The boundary conditions for the buoyant wall jet discharged in a rectangular tank are shown in Figure 2.

The wall jet is discharged with initial velocity \( U_0 \), temperature \( T_0 \), density \( \rho_0 \) and nozzle diameter \( D \) into a still ambient water with temperature \( T_a \) and density \( \rho_a \). The nozzle (outfall) boundary conditions are chosen based on Kheirkhah Gildeh et al. (2014a), as \( u_x = U_0 \), \( u_y = u_z = 0 \), \( T = T_0 \), \( k = 0.06u^2 \), \( \varepsilon = 0.06u^3 / D \), \( \omega = \varepsilon / k \). A fixed value boundary condition perpendicular to the outlet plane is defined for \( u_x, u_y, u_z, k, \varepsilon, \omega \) and \( T \) for the flow at the inlet and outlet boundary section and values are set to be equal to the ambient flow, i.e \( T=39^\circ C \). For the wall boundaries, boundary conditions defined as \( z = 0 \), \( u_z = 0 \), no-slip condition is considered in this study, thus \( k, \varepsilon \) and \( \mu \) are assumed to follow standard wall functions. For the atmosphere, inletOutlet boundary was assigned which forces \( U \) and \( P \) to shift between fixedValue and zeroGradient depending on direction of \( U \). Symmetry plane was used for the front and back walls which allows for limiting the simulation to one port which will be duplicated several times for the model to represent a multiport diffuser.
5 RESULTS AND DISCUSSION

5.1 Cross Sectional Variations of Local Flow Properties of Merging Buoyant Jets

Figure 3 illustrates the cross-sectional variation of the mean axial velocity, normalized by the maximum cross-sectional velocity, at different locations. According to previous investigations, normalized velocity should be self-similar and therefore should follow a Gaussian distribution curve. Such graphs for cross-sectional profiles will diverge from Gaussian distribution during merging processes which can be explained by the local flow properties starting to change during merging process.

Lyu et al. (2013) used best-fit curve in their experiments to derive the Gaussian constant (C) in Equation (1). With the jet characteristics shown in Table 1, they derived C= 77.68.

In order to numerically model the uni-directional multiport diffuser in this study both turbulence models, k-ε and LRR were applied to each case. Figure 3 shows both results for the Gaussian distributions (with C derived from experiments by Lyu et al. (2013), C=77.68) and results from numerical simulations for z/s=2.

![Figure 3. Cross sectional variation of Mean axial velocity](image)

Results show good agreement between numerical simulations and Gaussian distribution curve. The K-ε model proves to better represent plume behavior compared to LRR model as model results are in better agreement with the Gaussian distribution curve.

5.2 Longitudinal Variations of Local Flow Properties of Merging Buoyant Jets

Figure 4 shows the longitudinal variation of axial velocity along the centerline. According to the findings in many previous studies, axial velocity decreases with the decay of -1 power to the distance from the source. Such a line is shown in Figure 4, together with the results from Lyu et al. (2013) experiments and results from numerical simulations using both LRR and k-ε model.
The experimental data show a good agreement with the decay of -1 power to the distance from the source, widely verified by many researchers. Results show better agreement between the experiments and LRR model especially at the beginning of the discharge, however, k-\( \varepsilon \) model shows better agreement with the -1 power line and experimental results around \( \frac{z}{s} = 6 \) and as the jet travels upward. Figure 5 illustrates the time averaged velocity contours for both LRR and k-\( \varepsilon \) models. It can be noticed that velocity decays faster in k-\( \varepsilon \) model compared to LRR model which explains why it fits better experimental data as the jets travels further from the source. Both models are in reasonable agreement with the experimental data.
5.3. Merging zone

From graphs in Figures 3 and 4, one can conclude that merging zone differs between the two turbulence models. Jets merge faster when simulated using LRR as the data show convergence from the $-1$ power line about $\frac{z}{s} = 8$ while experimental data begin to diverge from the line at about $\frac{z}{s} = 10$. $k-\varepsilon$ model shows to be unsuccessful in predicting jet behavior in the direction that merging happens.

6 CONCLUSIONS

Results from an experimental study on behavior of buoyant jets discharged from a unidirectional (vertical) multiport diffuser were successfully reproduced numerically using two different turbulence models namely LRR and $k-\varepsilon$ model.

Both models showed to be able to represent the cross-sectional variation of the mean axial velocity since their results are close to the Gaussian curve. The LRR model resulted in better agreement between experimental and numerical results in this case.

Results for longitudinal variation of axial velocity along the centerline show that LRR model better represents behavior of the jet especially close to the source.

Merging zone in which longitudinal axial velocity variation scatters from the $-1$ power line showed to happen faster in LRR model ($\frac{z}{s} \approx 8$) compared to experimental results in which merging happens around ($\frac{z}{s} \approx 10$). The study is in progress using more accurate grid (un-structured grid). LES (Large Eddy Simulation) approach is also being tested and results will be compared with LRR and $k-\varepsilon$ models. Further results will be presented in the subsequent papers.

ACKNOWLEDGEMENTS

This publication was made possible by NPRP Grant 4-935-2-354 from the Qatar National Research Fund (a member of the Qatar Foundation). The statements made herein are solely the responsibility of the authors.

REFERENCES


