Proceedings of the 39th IAHR World Congress
19-24 June 2022
Granada, Spain
DOI number

Large Eddy Simulation of Multiple Inclined Brine Discharges

Seyed Ahmad Reza Saeidi Hosseini(1), Mostafa Taherian(1) and Abdolmajid Mohammadian(1)

(1) Department of Civil Engineering, University of Ottawa, Ottawa, Canada
ssaei041@uottawa.ca (S.S.); mtahe039@uottawa.ca (M.T.); majid.mohammadian@uottawa.ca (A.M.)

Abstract

Brine effluents are commonly discharged into water bodies through diffusers and may cause almost permanent adverse effects on marine habitats. Since the extension of such negative effects closely depends on the features of the outfall diffusers, it is essential to predict the mixing behavior of the jets for different types of diffusers. Although single-port discharges are theoretically superior in terms of mixing efficiency, currently, multiport diffusers are frequently applied due to the limitations in practical projects. The mixing behavior of multiport discharges compared to single-port ones is much more complicated because of the complex mechanisms resulting from jet interaction and the Coanda effect. Most of the numerical studies on discharges have focused on single-port discharges, and less attention has been given to the multiple jets. Also, the limited available numerical studies on the multiport diffusers have applied the Reynolds-averaged Navier-Stokes (RANS) turbulence models. In this study, the dilution and geometrical characteristics of the multiple 60° inclined dense jets were numerically investigated in detail using the large eddy simulation (LES) approach with the Smagorinsky subgrid-scale (SGS) model. The LES predictions for the impact point location, impact point dilution, and terminal rise height were compared to the existing measurements in the literature. The results showed that the LES outcomes were in good agreement with the experimental data, and the LES approach was found to be a reliable tool for the investigation of multiport diffusers. The merging process of discharges was also studied based on the presented model.

Keywords: Large eddy simulations; Negatively buoyant jets; Multiport diffusers; Turbulence modeling; Mixing characteristics

1. INTRODUCTION

Improper release of dense wastewaters into water bodies causes adverse environmental effects. Dense effluents are commonly discharged into water bodies through outfall diffusers as highly turbulent jets. Outfall discharges are mainly employed to provide rapid mixing, thereby protecting marine habitat. The mixing process of the discharges is significantly affected by the features of the diffusers (Yan and Mohammadian, 2019; Lai et al., 2015). Therefore, the study of the mixing process of the discharges for different types of diffusers can be of interest, as it can provide useful insights for outfall design purposes.

Wastewaters may be discharged into the seas and oceans as single-port or multi-port diffusers considering different factors in practical projects such as effluent flow rate, budget limit, and environmental regulatory demands. Although a free mix of single jets with the surrounding water causes them to be superior in terms of the mixing behavior, high wastewater flow rates may require multiple-port diffusers (Lyu et al., 2013; Wang and Davidson, 2003), as applied in Perth, Boston, Sydney, and Melbourne. As multiple jets are discharged, jet interaction and the Coanda effect lead to an intricate mechanism, which significantly affects the geometrical and mixing characteristics of the jets. Therefore, the study of the mixing process of multiple jets may be more complicated and differs from the single ones, highlighting the importance of a separate study on multiport diffusers. Unlike single dense discharges, the mixing behavior of multiple dense discharges has attracted less attention in prior research.

Most of the previous studies on multiple jets have been conducted experimentally (Xu et al., 2019; Shrivastava et al., 2019; Abessi et al., 2017; Abessi and Roberts, 2014; Marti et al., 2011; Adams, 1972), which have significantly shed light on the mixing process of multiple jets. However, experimental measurements are expensive and time-consuming options for the study of multiple jets. Integral entrainment models have always been an alternative method for the study of the mixing characteristics of jets. Many commercial tools, e.g. UM3 (Frick, 2004), CoreJet (Jirka, 2004), and JetLag (Lee and Chu, 2003; Lee et al., 1990), for the assessment of jet mixing process have been designed based on these models. The integral models are formed based on simplified mass and momentum equations (Taherian and Mohammadian, 2021), and they cannot resolve the Coanda effect and re-entrainment (Taherian and Mohammadian, 2021; Baum and Gibbes, 2020). The incapability of capturing the Coanda effect may be an important deficiency of these models when they are employed for the investigation of multiple jets since jet merging combined with the Coanda effect diminish terminal rise height and dilution of discharges (Roberts, 2015; Abessi and Roberts, 2014). In previous studies, major discrepancies for the estimation of dense jets' terminal rise height and dilution were found, where CoreJet, JetLag, and UM3 tools were employed for the simulations (Palomar et al., 2012).

Computational fluid dynamics (CFD) modeling considers less simplifying assumptions compared to the entrainment integral models (Baum and Gibbes, 2020), and its application in the prediction of the jet mixing process has increased in the last two decades (Taherian et al., 2022). However, CFD modeling has been limitedly employed for the prediction of multiport diffusers. Its application for multiple jets may be limited to some studies conducted in recent years, which have mainly applied the Reynolds-averaged Navier-Stokes (RANS) turbulence models. One study investigated the mixing behavior of multiple buoyant jets using two different models including an eddy viscosity model (EVM) and a CORMIX model and showed that the EVM was superior in terms of capturing the changes of plumes' depth along the trajectory (Tang et al., 2008). A recent study (Baum and Gibbes, 2020) applied k-ω shear stress transport (SST) model to investigate the dilution properties of multiple inclined dense jets in cross-flow, and validated the performance of the model against field measurements. Another study (Yan and Mohammadian, 2019) evaluated the mixing characteristics of multiple inclined dense jets, employing the re-normalization group (RNG) and standard k-ε turbulence models. It was shown that the predictions of terminal rise height and impact point dilution were in good agreement with the experimental measurements obtained by Abessi and Roberts (2014). The RNG k-ε model could provide more accurate results, with less than 15% error, compared to the standard k-ε model, over 18% error. Moreover, multiple vertical jets were numerically modeled using different RANS models in another study (Yan et al., 2020), and it was concluded that the RNG k-ε model outperformed the standard k-ω, k-ω (SST), and standard k-ε turbulence models.

According to the presented review and the authors' knowledge, multiple dense discharges have not been yet simulated employing the large eddy simulation (LES) approach. The reason may be the high computational cost and convolution when it comes to the simulation of the complex geometry of multiple jets. However, as the large coherent eddies are directly resolved in the LES approach, it is expected that this model provides more accurate predictions, compared to the RANS models. It can also provide more detailed information on the flow field and eddy structures. Therefore, the present study employs the Smagorisnky LES model for the prediction of the mixing behavior of multiple inclined dense jets and evaluates its prediction capabilities by comparing the results against the available experimental measurements. The validated model is further utilized for the study of the jet merging process.

2. METHODOLOGY

2.1 Multiple Jets Analysis

Figure 1 illustrates the schematic plan and side views of multiple inclined dense discharges. The initial vertical momentum flux of the jets moves them upward to a maximum height, which is called the terminal rise height, y_t. Along the upward movement of jets, the entrainment and negative buoyancy continuously diminish the initial momentum, and there is almost no trace of the initial vertical momentum anymore at y_t. Beyond this point, the jets move downward and finally impact the bottom boundary at the impact point location, x_i. The dilution at this point is referred to as the impact point dilution, S_i. These parameters are the most important ones while investigating the outfall systems as they are essential for design purposes. The values of these factors are numerically calculated using the LES approach and compared to the experiments on the 60° multiple inclined dense jets conducted by Abessi et al. (2014) to evaluate the performance of the LES model.

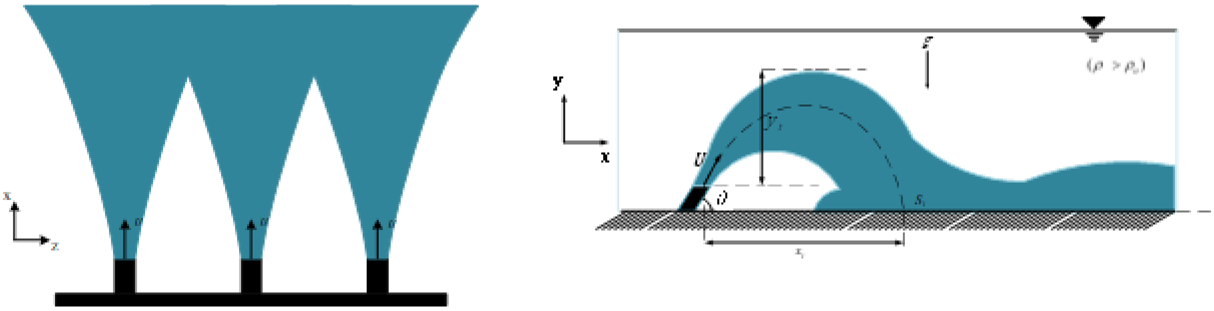


Figure 1. Schematic plan and side views of multiple dense jets

2.2 Governing Equations

In the LES approach, large and small eddies are distinguished using a spatial filter; the unsteady nature of large eddies is directly computed, while the small eddies are modeled. The filtered Navier-Stokes equations including continuity and momentum equations can be presented for an incompressible fluid as Eqs. [1] and [2], respectively.

$$\frac{\partial(\bar{u}_i)}{\partial x_i} = 0 \qquad [1]$$

$$\frac{\partial(\bar{u}_i)}{\partial t} + \frac{\partial\left(\bar{u}_i\bar{u}_j\right)}{\partial x_j} = -\frac{1}{\rho}\frac{\partial\bar{p}}{\partial x_i} + \nu\frac{\partial^2\bar{u}_i}{\partial x_j\partial x_j} - \frac{\partial\tau_{ij}}{\partial x_j} \qquad [2]$$

where $\bar{u}_i$, t, ν, and $\bar{p}$ are the filtered velocity field, time, kinematic viscosity, and filtered pressure field, respectively. Also, τ_{ij} indicates the subgrid-scale (SGS) stresses which can be defined as follows:

$$\tau_{ij} = \overline{u_i u_j} - \bar{u}_i\bar{u}_j \qquad [3]$$

In the present study, the Smagorinsky SGS model was adopted. The Smagorinsky model estimates the SGS stress tensor for an incompressible flow as below:

$$\tau_{ij}^{SGS} = -2\mu_t\tilde{S}_{ij} \qquad [4]$$

where $\tilde{S}_{ij}$ is the rate of strain tensor for the resolved scale. The unknown term of SGS eddy viscosity, μ_t, can be determined using the following equations:

$$\mu_t = \rho(C_s\Delta)^2\left|\tilde{S}\right| \qquad [5]$$

$$\left|\tilde{S}\right| = \sqrt{2\tilde{S}_{ij}\tilde{S}_{ij}} \qquad [6]$$

where C_s is the Smagorinsky constant and Δ is the LES filter size.

2.3 Computational Setup

As the experiments were conducted based on the different numbers of ports, the computational domain was considered to include three ports. These ports could be the representatives of all possible jets on a diffuser since the side planes of the domain were considered as symmetry planes, which could mirror the flow field of the domain. Boundary conditions play a significant role in the stability and accuracy of fluid numerical calculations. The simulation domain was structured using seven planes including the inlet, back wall, outlet, atmosphere, lower wall, and two side planes (see Figure 2). The boundary conditions at the back and lower walls were set to slip; the atmosphere and outlet planes to zero gradients; and side planes to the symmetry plane. Also, a fixed value of velocity was assigned to the inlets. The structured mesh, which was used for the discretization of the continuous domain, is shown in Figure 2. The cell sizes were set to be much smaller at the locations close to the nozzles compared to other areas so that they could capture the velocity and pressure fields better. The detailed simulation parameters and domain characteristics are presented in Table 1.

Table 1. Simulation parameters

NOZZLE DIAMETER (CM)	EFFLUENT DENSITY (KG/M^3)	AMBIENT DENSITY (KG/M^3)	JET VELOCITY (M/S)	F_D	DISCHARGE ANGLE (°)	PORT SPACING (CM)	DOMAIN DIMENSIONS (CM)	NUMBER OF CELLS (MILLION)
0.193	1013.8	999.8	0.85	52	60	5.7	40x40x17.6	1.35

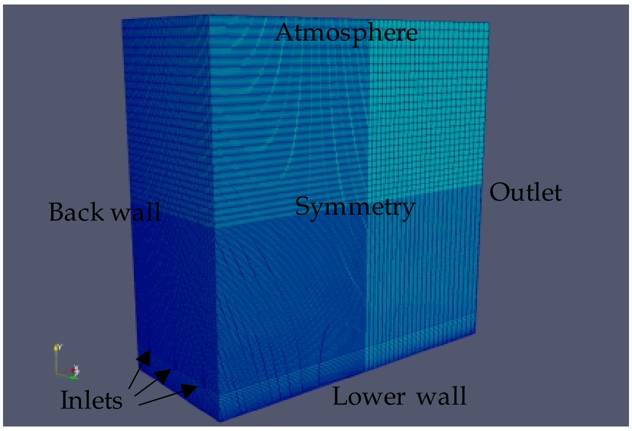


Figure 2. Computational domain, mesh setup, and boundary conditions

The finite volume method (FVM) was adopted for solving the presented equations, and an open-source code OpenFOAM was utilized to perform the simulations. The twoLiquidMixingFoam solver, which is a transient solver for multiphase fluids, in OpenFOAM software was used to solve the Navier-Stokes equations. This solver has been adopted for the simulation of various flow fields in previous studies (Yan et al., 2020; Jiang et al., 2019; Yan and Mohammadian, 2019; Li et al., 2016; Zhang et al., 2016; H Lai et al., 2015). The Euler scheme was utilized to discretize the temporal term; Gauss linear scheme was adopted for the discretization of the gradient and Laplacian terms; Gauss upwind, Gauss Vanleer, and Gauss linear schemes were applied to discretize the divergence terms. The time step was considered to be adjustable during the simulation based on the courant number, and the maximum courant number was set to 0.5 to provide small time steps. The outcomes were extracted based on time-averaging over the range of 70s to 130s since the results were found to be stable after 50s and the flow was fully developed. The simulation was performed in the advanced research computing (ARC) center of Compute Canada, and 16 processors were employed for parallel simulations.

3. RESULTS AND DISCUSSION

The Smagorisky LES predictions were compared to the experimental measurements conducted by Abessi and Roberts (2014). Table 2 demonstrates the model predictions and measurements for the impact point location (x_i), impact point dilution (s_i), and terminal rise height (y_t). Considering the errors, the LES predictions were in good agreement with the experimental data. The error for the geometrical characteristics of multiple dense jets was less than 10%, and it was less than 6% for the impact point dilution.

Table 2. Comparison of experimental and numerical results

	EXPERIMENAL RESULTS	NUMERICAL PREDICTIONS	ERROR(%)
IMPACT POINT LOCATION (CM)	17.36	17.15	1.2
IMPACT POINT DILUTION	26	24.45	5.97
TERMINAL RISE HEIGHT (CM)	16.1	17.7	9.94

Concentration contours at the side view of the middle jet are shown in Figure 3, where C0 denotes the initial concentration, C signifies the local concentration of the discharges, and C/C0 is the normalized concentration. In the ascending phase, the vertical component of the initial momentum resulted in the upward movement of the jets. As the discharges were dense jets, the negative buoyancy continuously hindered this momentum. At the end of this phase, there was no trace of it, and the jets started to turn downward. In the descending phase, the negative buoyancy was dominant, and this force moved the jets downward until they impinged to the bottom boundary. The contours show that the concentration continuously decreased along the trajectory, and the lower half of the jets expanded more widely compared to the upper half.

As discussed in Abessi and Roberts (2014), the multiple jets compared to the single ones have a reduced terminal rise height and impact point location, which have roots in the jet merging and Coanda effects. Figure 3 visualizes the inward bending of multiple jets, subsequently resulting in a reduced impact point location.

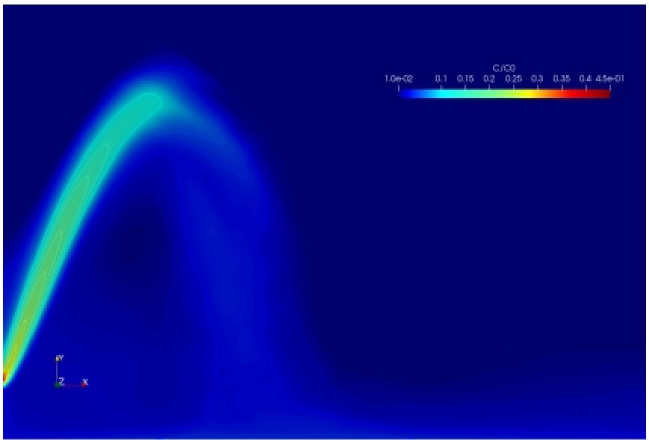
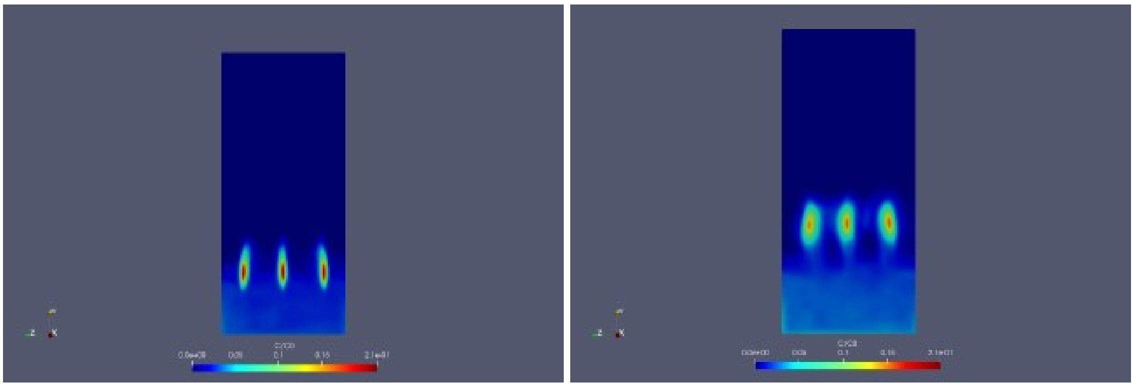
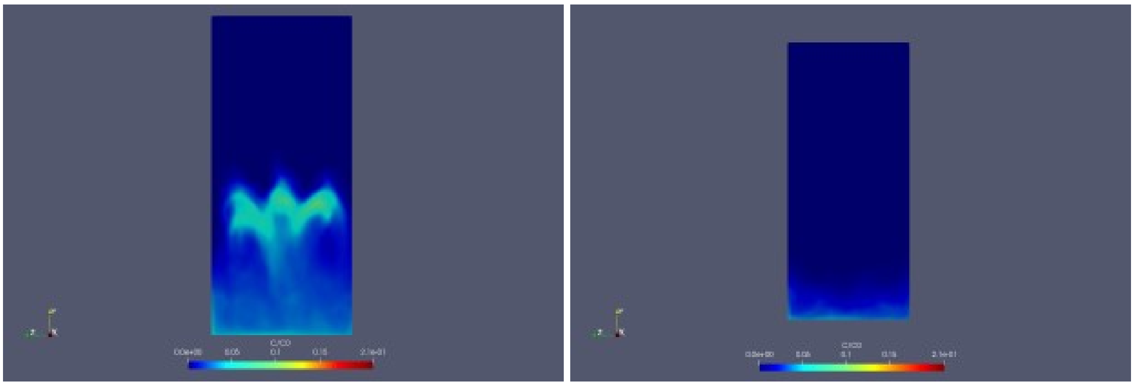


Figure 3. Time-averaged concentration contours at the central plane

Figure 4 demonstrates the concentration fields at different cross-sections, covering the locations close to the ports and downstream areas. These cross-sections could provide an insight into the jet merging process. At the locations close to the nozzles, e.g. X/DF=0.23, the discharges were not expanded enough to merge, and they were almost like separate jets. The concentration at the center of the jets was very high. In farther areas from the ports, e.g. X/DF=0.5, shear stresses on the edges of the jets caused the entrainment of the ambient water. Subsequently, the concentration decreased and the jets expanded. However, at this distance, the jets barely interacted with each other since they were not still diffused enough to merge. Farther away, e.g. X/DF=0.95, the jets were much more diffused and jet merging occurred. The concentrations at the jets' centers and the space between the jets were almost identical, and there was little freshwater between the jets. The jet merging leads to the formation of a virtually impenetrable wall that hinders the entrainment of surrounding water to inner surfaces and intensifies the Coanda effect, resulting in sharp inward bending of the jets (Abessi and Roberts, 2014). After the impact point, e.g. X/DF=1.92, the separate jets were indistinguishable.



(X/DF=0.23) (X/DF=0.5)



(X/DF=0.95) (X/DF=1.92)

Figure 4. Concentration fields at various non-dimensional distances from the nozzles

4. CONCLUSIONS

The main objective of the current study was to evaluate the prediction capabilities of the LES approach for multiple dense jets. The multiple 60° inclined dense jets were numerically simulated employing LES with the Smagorisky SGS model, for the first time. The LES predictions for impact point location, impact point dilution, and terminal rise height were compared against the existing experimental data. The model was further employed to investigate the jet merging process, which is of significant importance in the study of multiport diffusers. The results showed that the LES model could predict geometrical and dilution properties of the multiple dense jets with errors less than 10% and 6%, respectively. Therefore, it can be concluded that the LES approach can be a reliable tool for the prediction of the mixing behavior of multiport diffusers. The merging process was discussed through the visualization of different cross-sections.

5. ACKNOWLEDGMENT

The authors would like to thank the advanced research computing (ARC) center of Compute Canada (a not-for-profit-organization) for providing the computational resources used in this study. The research of A.M. was supported by Natural Sciences and Engineering Research Council of Canada (NSERC).

6. REFERENCES

Abessi, O., Roberts, P.J.W. (2017). Multiport diffusers for dense discharge in flowing ambient water. *Journal of Hydraulic Engineering*, American Society of Civil Engineers (ASCE), 143(6), 04017003.

Abessi, O., and Roberts, P.J.W. (2014). Multiport Diffusers for Dense Discharges. *Journal of Hydraulic Engineering*, American Society of Civil Engineers (ASCE), 140(8), 04014032.

Adams, E.E. (1972). Submerged multiport diffusers in shallow water with current (Doctoral dissertation, Massachusetts Institute of Technology).

Baum, M.J., and Gibbes, B. (2020). Field-Scale Numerical Modeling of a Dense Multiport Diffuser Outfall in Crossflow. *Journal of Hydraulic Engineering*, 146(1), 05019006.

Frick, W.E. (2004). Visual Plumes mixing zone modeling software. *Environmental Modelling & Software*, 19(7-8), 645-654.

Huai, W.X., Li, Z.W., Qian, Z.D., Zeng, Y.H., Han, J., & Peng, W.Q. (2010). Numerical simulation of horizontal buoyant wall jet. *Journal of Hydrodynamics*, *22*(1), 58-65.

Jiang, M., Law, A.W.K., and Lai, A.C.H. (2019). Turbulence characteristics of 45° inclined dense jets. *Environmental Fluid Mechanics*, Springer Netherlands, 19(1), 27–54.

Jirka, G.H. (2004). Integral Model for Turbulent Buoyant Jets in Unbounded Stratified Flows. Part I: Single Round Jet. *Environmental Fluid Mechanics*, 4(1), 1–56.

Joseph W Lee, B.H., Member, A., and Cheung, V. (1990). Generalized Lagrangian Model for Buoyant Jets in Current. *Journal of Environmental Engineering*, American Society of Civil Engineers (ASCE), 116(6), 1085–1106.

Lai, A.C., Zhao, B., Law, A.W.K., & Adams, E.E. (2015). A numerical and analytical study of the effect of aspect ratio on the behavior of a round thermal. *Environmental Fluid Mechanics*, *15*(1), 85-108.

Lee, J.H.W., Chu, V., & Chu, V.H. (2003). *Turbulent jets and plumes: a Lagrangian approach* (Vol. 1). Springer Science & Business Media.

Li, Y., Geng, X., Wang, H., Zhuang, X., and Ouyang, J. (2016). Simulating the frontal instability of lock-exchange density currents with dissipative particle dynamics. *Modern Physics Letters B*, 30(17), 1–9.

Lyu, S., Seo, I.W., & Kim, Y.D. (2013). Experimental investigation on behavior of multiple vertical buoyant jets discharged into a stagnant ambient. *KSCE Journal of Civil Engineering*, *17*(7), 1820-1829.

Marti, C.L., Antenucci, J.P., David Luketina, Okely, P., Imberger, J., and Asce, M. (2011). Near-Field Dilution Characteristics of a Negatively Buoyant Hypersaline Jet Generated by a Desalination Plant. *Journal of Hydraulic Engineering*, American Society of Civil Engineers (ASCE), 137(1), 57–65.

Palomar, P., Lara, J., and Losada, I. (2012). Near field brine discharge modeling part 2: Validation of commercial tools. *Desalination*, 290, 28–42.

Roberts, P.J.W. (2015). Near field flow dynamics of concentrate discharges and diffuser design. *Environmental Science and Engineering* (Subseries: Environmental Science), Kluwer Academic Publishers, 369–396.

Shrivastava, I., & Adams, E.E. (2019). Mixing of tee diffusers in shallow water with crossflow: A new look. *Journal of Hydraulic Engineering*, *145*(4), 04019006.

Taherian, M., and Mohammadian, A. (2021). Buoyant Jets in Cross-Flows: Review, Developments, and Applications. *Journal of Marine Science and Engineering*, Multidisciplinary Digital Publishing Institute, 9(1), 61.

Taherian, M., Saeidi Hosseini, S.A.R., and Mohammadian, A. (2022). Overview of outfall discharge modeling with a focus on turbulence modeling approaches. Advances in Fluid Mechanics: Modeling and Simulation, edited by Zeidan, D., Zhang, L., da Silva, E.G., and Merker., J. Singapore: Springer Nature, Accepted.

Tang, H.S., Paik, J., Sotiropoulos, F., and Khangaonkar, T. (2008). Three-Dimensional Numerical Modeling of Initial Mixing of Thermal Discharges at Real-Life Configurations. *Journal of Hydraulic Engineering*, 134(9), 1210–1224.

Wang, H.J., and Davidson, M.J. (2003). Jet Interaction in a Still Ambient Fluid. *Journal of Hydraulic Engineering*, American Society of Civil Engineers (ASCE), 129(5), 349–357.

Xu, Z., Otoo, E., Chen, Y., & Ding, H. (2019). 2D PIV Measurement of Twin Buoyant Jets in Wavy Cross-Flow Environment. *Water*, 11(2), 399.

Yan, X., Ghodoosipour, B., and Mohammadian, A. (2020). Three-Dimensional Numerical Study of Multiple Vertical Buoyant Jets in Stationary Ambient Water. *Journal of Hydraulic Engineering*, 146(7), 04020049.

Yan, X., and Mohammadian, A. (2019). Numerical modeling of multiple inclined dense jets discharged from moderately spaced ports. *Water* (Switzerland), 11(10), 14–16.

Zhang, S., Jiang, B., Law, A.W.K., and Zhao, B. (2016). Large eddy simulations of 45° inclined dense jets. *Environmental Fluid Mechanics*, Springer Netherlands, 16(1), 101–121.