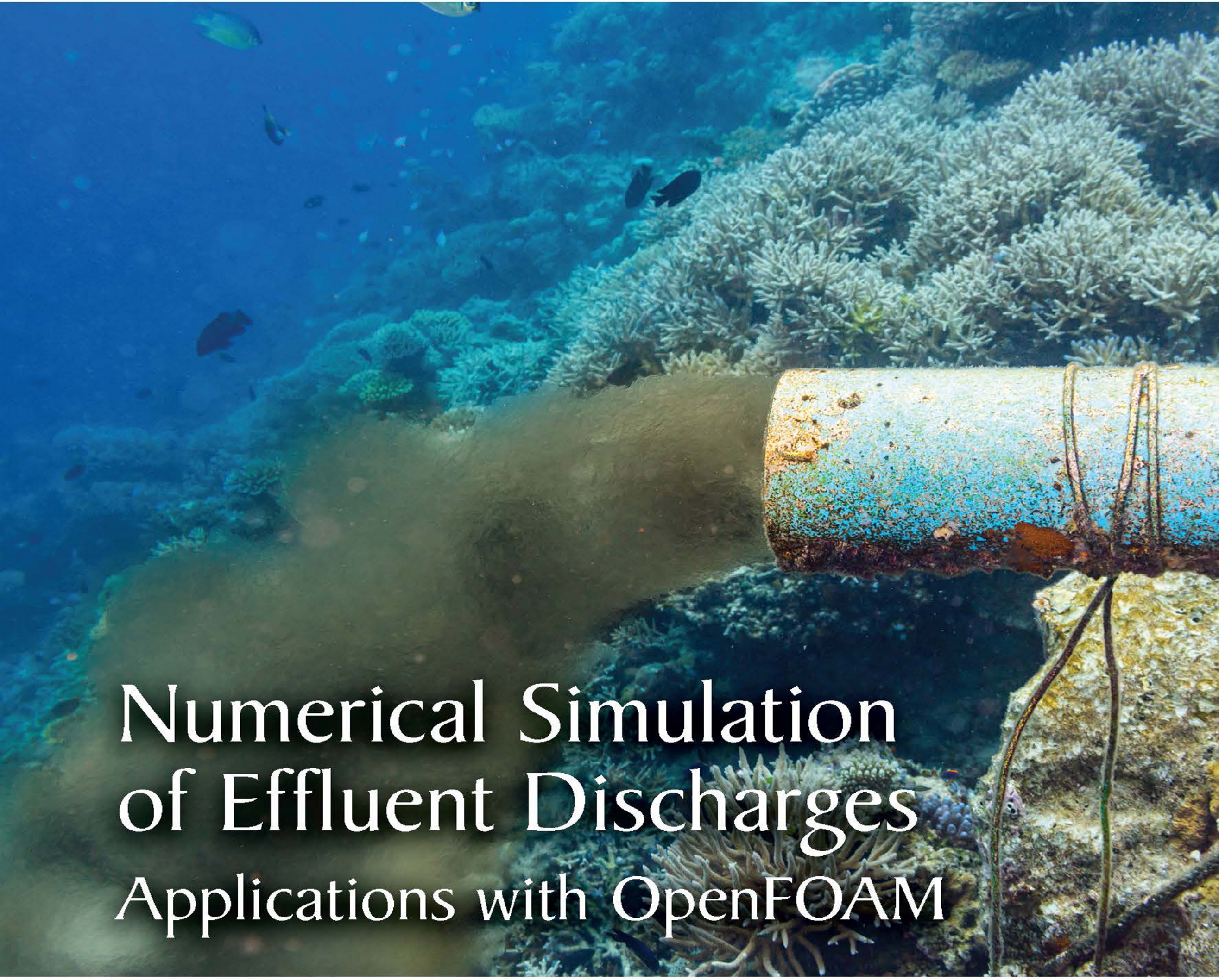
IAHR AIRH
International Association
for Hydro-Environment
Engineering and Research

Hosted by
Spain Water and IWHR, China

#iahrBooks
IAHR Books



Numerical Simulation of Effluent Discharges

Applications with OpenFOAM

**Abdolmajid Mohammadian,
Hossein Kheirkhah Gildeh,
and Xiaohui Yan**

CRC Press
Taylor & Francis Group

IAHR.org

Numerical Simulation of Effluent Discharges

Numerical Simulation of Effluent Discharges: Applications with OpenFOAM provides a resource for understanding the effluent discharge mechanisms and the approaches for modeling them. It bridges the gap between academia and industry with a focused approach in CFD modeling and providing practical examples and applications. With a detailed discussion on performing numerical modeling of effluent discharges in various ambient waters and with different discharge configurations, the book covers the application of OpenFOAM in effluent discharge modeling.

Features:

- Discusses effluent discharges into various ambient waters with different discharge configurations.
- Focuses on numerical modeling of effluent discharges.
- Covers the fundamentals in predicting the mixing characteristics of effluents resulting from desalination plants.
- Reviews the past CFD studies on the effluent discharge modeling thoroughly.
- Provides guidance to researchers and engineers on the future steps in modeling of effluent discharges.
- Includes an introduction to OpenFOAM and its application in effluent discharge modeling.

The book will benefit both academics and professional engineers practicing in the area of environmental fluid mechanics and working on the effluent discharge modeling.

IAHR Book

Series editor: Robert Ettema
Department of Civil and Environmental Engineering, Colorado State University,
Fort Collins, USA

The International Association for Hydro-Environment Engineering and Research (IAHR), founded in 1935, is a worldwide, independent organisation of engineers and water specialists working in fields related to hydraulics and its practical application. Activities range from river and maritime hydraulics to water resources development and eco-hydraulics, to ice engineering, hydroinformatics and continuing education and training. IAHR stimulates and promotes both research and its application, and, by doing so, strives to contribute to sustainable development, the optimisation of world water resources management and industrial flow processes. IAHR accomplishes its goals by a wide variety of member activities including the establishment of technical committees, working groups, congresses, specialty conferences, workshops and short courses; the commissioning and publication of journals, monographs and edited conference proceedings; involvement in international programmes, such as the UNESCO, WMO, IDNDR, GWP, ICSU and The World Water Forum; and by co-operation with other water-related (inter)national organisations.

www.iahr.org

IAHR AIRH

Supported by
Spain Water
and IWHR, China

Energy Dissipation in Hydraulic Structures
Edited By Hubert Chanson

Hydraulics of Levee Overtopping
Lin Li, Farshad Amini, Yi Pan, Saiyu Yuan, and Bora Cetin

Climate Change-Sensitive Water Resources Management
Edited by Ramesh S.V. Teegavarapu, Elpida Kolokytha, and Carlos de Oliveira Galvão

Water Projects and Technologies in Asia
Historical Perspectives
Edited by Hyoseop Woo, Hitoshi Tanaka, Gregory De Costa, and Juan Lu

For more information about this series, please visit: https://www.routledge.com/IAHR-Book/book-series/IAHRMON

Numerical Simulation of Effluent Discharges

Applications with OpenFOAM

Abdolmajid Mohammadian,
Hossein Kheirkhah Gildeh, and Xiaohui Yan

CRC Press
Taylor & Francis Group
Boca Raton London New York

CRC Press is an imprint of the
Taylor & Francis Group, an **informa** business

Designed cover image: Shutterstock | grafner.

First edition published 2023
by CRC Press
6000 Broken Sound Parkway NW, Suite 300, Boca Raton, FL 33487-2742

and by CRC Press
4 Park Square, Milton Park, Abingdon, Oxon, OX14 4RN

CRC Press is an imprint of Taylor & Francis Group, LLC

© 2023 Abdolmajid Mohammadian, Hossein Kheirkhah Gildeh, and Xiaohui Yan

Reasonable efforts have been made to publish reliable data and information, but the author and publisher cannot assume responsibility for the validity of all materials or the consequences of their use. The authors and publishers have attempted to trace the copyright holders of all material reproduced in this publication and apologize to copyright holders if permission to publish in this form has not been obtained. If any copyright material has not been acknowledged please write and let us know so we may rectify in any future reprint.

With the exception of Chapter 3, no part of this book may be reprinted or reproduced or utilised in any form or by any electronic, mechanical, or other means, now known or hereafter invented, including photocopying and recording, or in any information storage or retrieval system, without permission in writing from the publishers.

Chapter 3 of this book is available for free in PDF format as Open Access from the individual product page at www.crcpress.com. It has been made available under a Creative Commons Attribution-Non Commercial-No Derivatives 4.0 license.

Trademark notice: Product or corporate names may be trademarks or registered trademarks, and are used only for identification and explanation without intent to infringe.

British Library Cataloguing-in-Publication Data
A catalogue record for this book is available from the British Library

ISBN: 978-1-032-02048-8 (hbk)
ISBN: 978-1-032-02094-5 (pbk)
ISBN: 978-1-003-18181-1 (ebk)

DOI: 10.1201/9781003181811

Typeset in Times New Roman MT Std
by KnowledgeWorks Global Ltd.

Contents

Preface

This book is an introduction to the numerical modeling of effluent discharges using OpenFOAM. It introduces the relevant background knowledge and modeling techniques of effluents in detail.

With the increase in population, the development of regional economy and the acceleration of industrialization, wastewater effluents are increasing, which puts forward higher requirements for effluent treatment and disposal capacity. The establishment of a water effluent model can play an important role in improving effluent treatment technologies and dealing with environmental pollution. Therefore, effluent modeling is very important. This book introduces computational fluid dynamics (CFD) models of effluent discharges on the basis of understanding the research progress of jets and plumes, analyzes the advantages and disadvantages of different modeling approaches, and puts forward a series of suggestions for future research work.

Although a great deal of research has been done on the mixing properties of wastewater discharges over the past few decades, the simulation of wastewater discharges using modern mathematical and computational techniques is still in its infancy. The basic data problem and the uncertainty of model parameters in the process of model application pose challenges to the reliability of the model. The availability of open-source CFD tools has opened the door to more realistic CFD modeling of effluent discharges. Although the numerical simulation technology has been significantly developed, the turbulence modeling problem of jet or plume has not been completely studied, and further research is needed. This book discusses these gaps in the literatures.

This book is mainly for the undergraduate and graduate students in hydraulics and hydrology, as well as for practitioners. The book begins with an introduction to outfall systems (Chapter 1) and introduces the reader to the application and different configurations of outfalls, various types of effluents, and mixing zones. Chapter 2 introduces the basic principles of numerical modeling. First, it introduces the governing equations, computational domain, boundary conditions, and initial conditions. Then, computational meshing is introduced, including mesh generation and determination of mesh resolution based on mesh sensitivity analysis. Then various methods in turbulence modeling (RANS, LES, DES, DNS) are briefly discussed. The basic concepts are discussed, and the formulations of the selected methods are provided. Finally, the modification of turbulence terms for buoyant discharges is introduced. Chapter 3 is an introduction to OpenFOAM. OpenFOAM is a popular CFD tool for effluent discharge modeling. It mainly introduces the OpenFOAM solvers and mesh generation and post-processing capabilities for effluent discharge modeling. Chapter 4 reviews past numerical studies in this field, points out future research directions, and puts forward suggestions for further improving effluent discharge modeling.

This book builds on past effluent discharge researches with further discussion on CFD modeling approaches and techniques, and it is hoped that these introductions and recommendations will be a useful reference for undergraduate and graduate students in hydraulics and hydrology, as well as the practitioners.

List of symbols

C	concentration of tracers or contaminants
D	diffusivity coefficient
D	diameter of the discharge port
F_r	the densimetric Froude number
g	gravitational acceleration
g_x, g_y, g_z	gravitational acceleration components in the x, y, and z directions
h_0	water depth (m)
k_{eff}	heat transfer coefficient
p	pressure (N/m^2)
pr_t	turbulent Prandtl number
pr	Prandtl number
S	salinity (psu)
t	time (s)
t_0	initial time (s)
T	temperature
T	fluid temperature (°C)
U_j	initial velocity (m/s)
u	instantaneous velocity component (m/s)
u, v, w	velocity components (m/s)
u_x, u_y, u_z	velocity components (m/s)
v	kinematic viscosity
v_t	turbulent viscosity
x, y, z	coordinates
ω	specific dissipation rate
ρ	density
ρ_α	ambient density (g/cm^3)
ρ_j	jet's initial density
τ_{ij}	stress in the *j*-direction exerted on a plane perpendicular to the *i*-axis (N/m^2)

List of abbreviations

CFD	computational fluid dynamics
CORMIX	Cornell mixing zone expert system
DES	detached eddy simulation
DIC	diagonal incomplete Cholesky
DILU	diagonal incomplete lower upper
DNS	direct numerical simulation
EPA	environment Protection Agency
EU	European Union
FDIC	faster diagonal incomplete-Cholesky
FV	finite volume
FVC	finite volume calculus
FVM	finite volume method
FDM	finite difference method
GAMG	geometric-algebraic multi-grid
GGDH	general gradient diffusion hypothesis
LES	large eddy simulation
MSF	multistage flash
NS	Navier-Stokes equations
OSS	open source software
ODE	ordinary differential equations
PBiCG	preconditioned bioconjugate gradient
PCG	preconditioned conjugate gradient
PDE	partial differential equation
PDR	partial differential equation
PML	Prandtl mixing length
RANS	Reynolds-averaged Navier-Stokes
RNG	renormalization group
RO	reverse osmosis
RSM	Reynolds Stress Model
RWPT	random walk particle tracking
SGDH	simple gradient diffusion hypothesis
VTK	visualization toolkit
WSL	windows subsystem for Linux

Chapter 1

Introduction

1.1 Identifying the problem

Discharges of industrial effluents into coastal and estuarine waters and the emissions of incinerated urban waste into the atmosphere provide two examples of environmental flows in which water and air quality, respectively, are determined by the behavior and structure of the particle-laden, turbulent, dense/buoyant jets, or plumes generated by discharges. Industrial power plants discharge residual byproducts into water bodies (Lattemann and Höpner, 2008), mostly as submerged jets due to their higher effectiveness.

Moreover, rising populations, shortages of clean and potable water, and advancements in desalination plant technology have increased rapidly in recent decades. In arid and semi-arid countries, desalination plants are actively considered as the best alternative to respond to the high demand for drinkable water. Desalination plants remove the dissolved minerals from coastal water bodies and produce effluents with a high-salt concentration, called brines, that may also have an elevated temperature, especially for multistage flash (MSF) desalination plants. Disposal of these brines, which have higher density than the receiving water, causes many environmental impacts, especially in the near field of outfall systems, which is the natural habitat of marine species and fish cultures (Hashim and Hajjaj, 2005; Lattemann and Höpner, 2008). Some areas such as the Red Sea, Persian Gulf, and generally low energy areas with shallow waters are very sensitive to effluent discharges.

The effluent discharge systems of industrial power plants have to be designed properly in order to minimize environmental impacts and financial costs. They also must satisfy the environmental criteria and standards (e.g., US-EPA and EU regulations). Nevertheless, ocean outfall systems are mostly not optimized, either regarding environmental impacts or practical needs. In some cases, regulations also lack clear guidelines for ambient water or effluent standards (Jirka, 2004).

The density differences between the effluent and ambient water, represented by the buoyancy flux, result in various flow and mixing characteristics of the discharge. In the case of dense jets, especially brine from reverse osmosis (RO) desalination plants, the flow has the tendency to fall as negatively buoyant plumes. On the other hand, buoyant jets (e.g., effluents from MSF desalination plants) have lower density than ambient water which causes the plumes to rise.

Besides being designed to minimize environmental impacts and financial costs, discharge outfalls must be in compliance with regulatory criteria. The first step before working on the discharge outfall design is to decrease the concentrations of the waste

DOI: 10.1201/9781003181811-1

source within the industrial plant (e.g., decreasing the additive usage, enhancing plant efficiency, pretreatment technologies, etc.). The second step is the application of improved mixing technologies, like submerged diffusers, and to discharge in less sensitive regions (offshore, deep waters).

Although experimental studies on scaled physical models have been primarily used to study the mixing problems in jets and plumes, this book only focuses on the numerical aspects of such problems. Therefore, the problem at hand is clear: how numerical modeling can be of help to design effluent discharges in open waters more efficiently. This requires an understanding of the problem and knowledge of the tools needed to address the questions surrounding it.

The term “numerical modeling” is still a general term and vague to some extents. There are different numerical modeling techniques that try to solve the equations of transport for effluent discharges. These models may use either simplified or complex sets of partial differential equations (PDEs) such as mass conservation, momentum conservation, and transport equations. There are two main methods available in solving effluent discharge problems numerically that are briefly summarized below.

1.1.1 Integral models

Jet integral models, according to Robinson et al. (2015), solve mass and momentum conservation equations based on the assumptions that the velocity profiles of jets have no radial variation, and that the jet profile is axisymmetric and Gaussian. In other words, complex governing equations of flow hydrodynamics are integrated over the cross section, assuming a Gaussian cross-sectional distribution. These models simplify the PDEs to ordinary differential equations (ODEs) that can be easily solved using numerical integration of differential equations such as explicit and implicit numerical methods. Explicit methods calculate the state of a system at a later time using the state of the system at the current time ($S(t + \Delta t) = F(S(t))$, while implicit methods use both current and future states of the system to find a solution ($F(S(t), S(t + \Delta t)) = 0$). In the 1950s and 1960s, first-order jet integral models were proposed by Morton et al. (1956) and Fan (1967) based on the jet entrainment closure approach and by Abraham (1963) and Turner (1969) based on the jet diffusion approach. Wang and Law (2002), Yannopoulos (2006), and Jirka (2004) developed second-order jet integral models. Since the turbulent mixing of effluent discharges are complex, as are their numerical solutions, integral models rely on experiments to derive the coefficients for their simplified analytical methods.

According to Robinson et al. (2015), the integral models are less reliable when there is any of the following: (i) the discharge’s initial momentum and buoyancy acting in opposite directions, resulting in instabilities on the edge, as observed in the inner half (lower half) of inclined dense jets; (ii) noticeable interaction between the mean flow and the jet, (iii) an unsteady mean ambient flow; (iv) a significant effect due to horizontal or lateral boundaries; (v) an unstable near-field area, with a re-entrainment of concentrated effluent into the jet; or, (vi) a large re-entrainment of concentrated effluent from mid- and far-fields into the near-field jet due to tidal cycles.

The most popular integral models in effluent discharge modeling are: CORMIX, VISUAL PLUMES, and VISJET. These models have been reviewed by Palomar et al. (2012) in detail, and the following provides a summary of that study.

CORMIX (Cornell Mixing Zone Expert System) software (Doneker and Jirka, 2001) is a commercial model that was developed in the 1980s at Cornell University (USA) as a project funded by the Environmental Protection Agency (EPA). Supported by the EPA, it became one of the most popular programs for discharge modeling. CORMIX is an expert system for predicting the discharge trajectory and dilution into water bodies in steady-state without considering time series data. CORMIX can simulate the disposal of effluents with positive, negative, and neutral buoyancy under different discharge and ambient conditions. The subsystems CORMIX 1, 2, and 3 are based on dimensional analysis of the processes, while the CORJET model is based on the integration of differential equations.

VISUAL PLUMES by Frick (2004) is a free access software developed by the EPA, which includes several models to simulate positively, negatively, and neutrally buoyant effluents discharged into receiving water bodies. VISUAL PLUMES considers the effluent properties, the discharge configuration, and the ambient conditions (temperature, salinity, and currents whose intensity and direction can be variable through the water column). It is limited to near-field region modeling and does not simulate the interaction of the flow with boundaries. It can consider time series data, simulating discharges in scenarios which change over time.

VISJET (Innovative Modeling and Visualization Technology for Environmental Impact Assessment) software (Cheung et al., 2000) is a commercial model developed by the University of Hong Kong, which can simulate positively and negatively buoyant discharges. VISJET considers the effluent properties, the discharge configuration, and the ambient conditions (temperature, salinity, and currents whose intensity and direction can be variable through the water column). It is limited to near-field region modeling and does not simulate the interaction of the flow with boundaries.

1.1.2 Computational fluid dynamics (CFD) models

Effluent discharge modeling by CFD tools is not perfect, but it is an improvement over the parameter-based jet integral models. Issues that remain with CFD tools include the following: (i) accuracy, (ii) stability, (iii) computational time, (iv) complicated codes that require expert knowledge to use them efficiently and accurately, and (v) simulations that need calibrating and validating.

Turbulent flow models are often resolved with a turbulence model to parameterize unresolved mixing and dispersion scales. One should apply turbulence models with caution, as they sometimes provide stable but unrealistic solutions, such as when they are applied to physical scenarios for which they have not been validated for.

When using a CFD model, it can be a challenge to create and resolve the mesh and to define appropriate boundary conditions (e.g., intensity and turbulence dissipation rate). A high-mesh resolution is often needed for a stable solution, even when the turbulence model is a good match. This means that CFD modeling is computationally expensive. Even with current computing systems, accurate CFD models for near-field dispersion and mixing might need simulation times of several days or weeks. This is much more expensive compared to the integral models that can produce results on the order of minutes and seconds. There is a balance between model stability, numerical diffusion, mass and momentum conservation, boundedness, and computational cost. These choices can significantly influence the estimation of modeled concentration.

However, once built, calibrated, and validated, CFD models can produce high-resolution three-dimensional images of jet mixing and dynamics. CFD models are free from some of the assumptions that restrict integral models. Since CFD models do not require the assumption of a steady-state condition or self-similarity in the jet profile, they can include a variety of external effects such as the presence of surface waves and encompass a wide range of boundary conditions to allow users to directly simulate the boundary interaction.

CFD modeling of jet discharges has been approached in a variety of ways, including both hydrostatic and nonhydrostatic approaches to the Reynolds-averaged Navier-Stokes (RANS) and the Large Eddy Simulations (LES) models. Both models have functioned well over the past decade to simulate effluent discharges. RANS models are based on a time-averaging method and result in a time-averaged mean velocity field, which is averaged over a longer time period than the time constant of the velocity fluctuations, and results in a constant mean velocity without fluctuation for time-dependent variations. LES is based on filtering instead of averaging. A filter size is identified, and flow scales equal to or larger than this size are calculated exactly, and scales smaller than the filter size are modeled. The smaller the filter size, the more concise is the calculated time variation resolution of the velocity vectors. RANS models are more numerically efficient than LES models, while providing enough detail for engineering applications. Thus, they have become the most prevalent CFD models used for the design of outfall systems.

The Direct Numerical Simulation (DNS) method is less applicable to engineering problems, functioning more as a research tool. It is CPU-intensive, as it attempts to resolve Navier-Stokes equations with no approximation of the turbulence and requires a very fine numerical resolution to capture all the turbulence details. It basically resolves entire turbulence scales temporally and spatially. Mesh systems should be very fine to resolve all the spatial scales (Kolmogorov, 1941).

Table 1.1 (after Zhao, Chen and Lee, 2011) summarizes the existing modeling packages (commonly used in the academia and industry) for the simulation of jet and plume mixing.

1.2 Application of outfalls

Outfalls have been used for many years. Initially, they have been used as a means of transporting the effluents to the discharge point, in the absence of environmental regulations. In the modern era, outfalls are used in both inland and coastal waters more carefully, and as a system that increases the dilution of discharged effluents to meet the environmental regulations in both near-field and far-field mixing zones. In other words, outfalls are not simply a method of transport, they represent a sustainable technology to preserve the environment while meeting its main objectives. More restrictive regulations have been developed throughout the years and the design of outfall systems has become more complex.

It is noteworthy that stormwater outfalls that discharge non-impacted waters into river or marine environments are not part of what is discussed here. In this book, we primarily refer to the outfalls that transport and discharge effluents with elevated temperature, salinity, and other chemicals. For instance, outfalls used in desalination plants, nuclear power plants, and wastewater treatment plants are among those mostly studied with respect to mixing problems.

Table 1.1 Existing modeling packages for simulation of jet and plume mixing

Models	*Mathematical approaches for jet/plume mixing*	*Availability*	*Major functionalities and capabilities*
CORMIX	Empirical solutions; Eulerian jet integral method	Commercial package	Prediction of jet and (or) plume geometry and dilution in the near field; single or multiple jets
VISJET	Lagrangian jet integral method	Commercial package	
Visual PLUMES	Empirical solutions; Eulerian and Lagrangian jet integral methods	Free package	
NRFIELD	Empirical solutions	Free package	Prediction of jet and (or) plume geometry and dilution in the near field of multiport diffusers
Sophisticated Multidisciplinary Models			
OpenFOAM	FVM; RWPT method	Free package	Predictions of ocean hydrodynamics; pollutant fate and transport in the near and far fields; water quality; sediment processes
MIKE21/3	FVM; RWPT method	Commercial package	
Delft3D	FDM; RWPT method	Free package	
ANSYS CFX	FVM; RWPT method	Commercial package	
ANSYS Fluent	FVM; RWPT method	Commercial package	
FLOW-3D	FDM; RWPT method	Commercial package	
TELEMAC-2D/3D	FEM; RWPT method	Free package	
EFDC	FDM; RWPT method	Commercial package	Predictions of ocean hydrodynamics; Pollutant dispersion in the far field; Suspended sediment transport
HydroQual–ECOMSED	FDM; RWPT method	Free package	Predictions of ocean hydrodynamics; Pollutant fate and transport in the far field; Sediment processes

Note: FVM: Finite Volume Method, FDM: Finite Difference Method, RWPT: Random Walk Particle Tracking.

Although outfalls are essential to human needs, they impact the environment they discharge into. The direct discharge of wastewater into lakes, rivers, and seas can increase turbidity and change the ambient temperature. Salinity is also a major public and scientific concern. Coastal waters receive concentrated salt brine as discharges from seawater desalination plants, chemical wastes from biofouling (e.g., chlorine), and fertilizers. The water bodies that receive the industrial discharges are often very sensitive environments, and designing outfalls to disperse effluent and reduce the concentration of effluents is essential in helping to protect the receiving water bodies.

Hopner and Windelberg (1996) noted that certain coastal ecological zones are particularly vulnerable to effluent discharges, including salt marshes, mangrove forests, coral reefs, and other low-energy intertidal areas. The Persian Gulf and the Red Sea are particularly sensitive to effluent due to their low hydro-dynamism. Local fisheries, tourism industries, and other economic concerns are affected by the health of coastal environments (Figure 1.1).

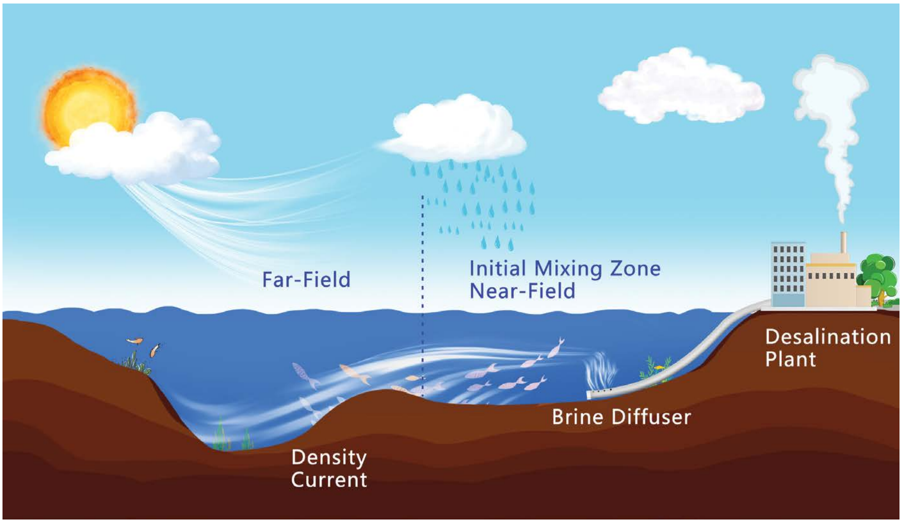


Figure 1.1 An overview of the effluent discharge in a water balance concept.

Based on the above paragraphs, it is clear that outfalls are not only a technical and economic concern. It goes beyond that as it affects the socio-environmental aspects of our lives. It is important that both designers and regulators pay attentions to each single outfall system being designed and constructed, as each outfall has its own challenges and unique characteristics.

A key item to note when reviewing the applications of outfalls for effluent discharges is the discharge objectives that would be the basis for the process leading to the water quality regulations, discharge limits, and design criteria. Figure 1.2 shows the general flowchart of environmental evaluations when designing the proper outfall system for a specific site.

The performance of an ocean or inland effluent discharge outfall is dependent on several factors such as the outfall configuration, topographic and bathymetric conditions of the discharge area, receiving water hydrodynamics, etc. This is the main reason that each outfall should be looked at as a unique design with unique characteristics. The following sections summarize key characteristics of effluent discharges.

1.3 Different outfall configurations

The effluents produced by industrial plants (e.g., brine produced by a desalination plant) can be disposed of in several ways such as discharge in inland open waters, injection into ground wells, discharge into a large evaporation pond, or discharge into coastal waters. There are two general methods of discharging effluents: surface discharge through open channels and submerged discharges through pipes extending into ambient waters. These two methods are illustrated in Figure 1.3. Both these types of discharges intend to increase the dilution and mitigate the environmental impacts.

The selection of the type of outfall is site specific and depends on several parameters such as desalination technology, plant operation and production rate, costs, and environmental considerations. Moreover, the characteristics of discharge (such as density)

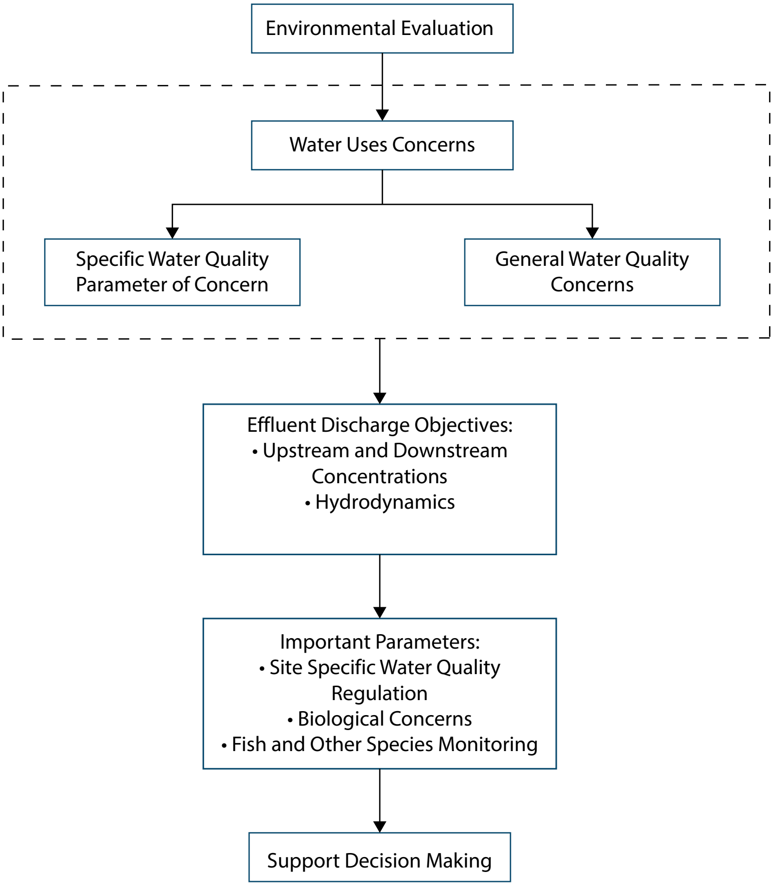


Figure 1.2 Environmental evaluations to preserve aquatic environments from effluent discharge.

and receiving ambient waters (such as density, ambient currents, and buoyancy) influence the type of outfall to be selected.

Surface discharges through open channels are often the most economic options available due to easier construction and maintenance. They are still constructed and used, despite the fact that their efficiency is lower than submerged discharge outfalls in terms of mixing and entrainment (dilution). Surface discharges are excellent options for plants that discharge effluents at a high flow rate, where conveyance through long pipes is extremely difficult due to pipe size and head losses associated with the conduits. Figure 1.4 illustrates the various surface discharge configurations.

On the other hand, submerged discharges are very popular due to their higher efficiency in reaching the required dilution. Therefore, many industrial plants (e.g., mining) will consider the discharge requirements at the very initial stages of the design, and will thus use submerged outfalls in order to meet the regulatory requirements for their permitting purposes. Submerged outfalls often use a pipe near the seabed to discharge effluents. The submerged discharges have been well studied both experimentally and numerically in past decades and our understanding of their mechanisms is relatively well established. It is noteworthy that outfall often refers to the pipe that transports the effluents from

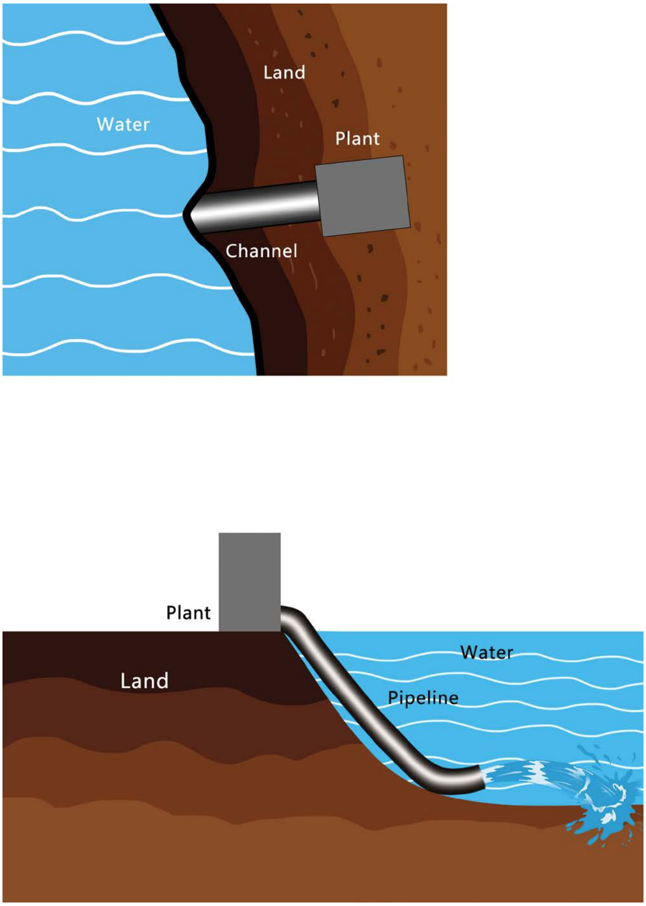
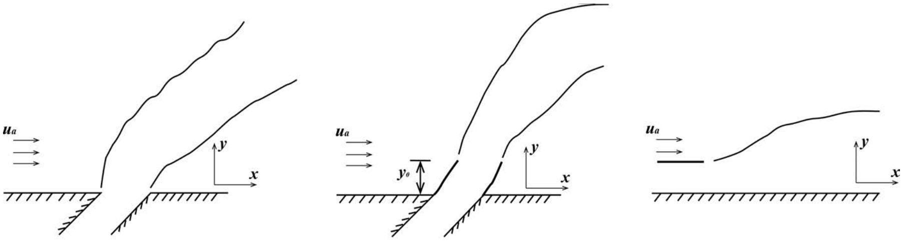


Figure 1.3 Schematic representation of brine discharge systems.

upland into the water. At the discharge location, it can be discharged through the same pipe (i.e., one discharge nozzle) or through a series of nozzles (diffusers). If a single nozzle is used to discharge the effluent into the receiving water, it is called a "single-port diffuser." However, in many cases, the series of nozzles are attached to the discharge point of the outfall, which is called a multiport diffuser (referring to several ports/nozzles to discharge effluents). In this case, the total discharge head



discharge flush with bank

protruding discharge

coflowing along bank

Figure 1.4 Discharge configurations of surface channel relative to bank/shoreline.

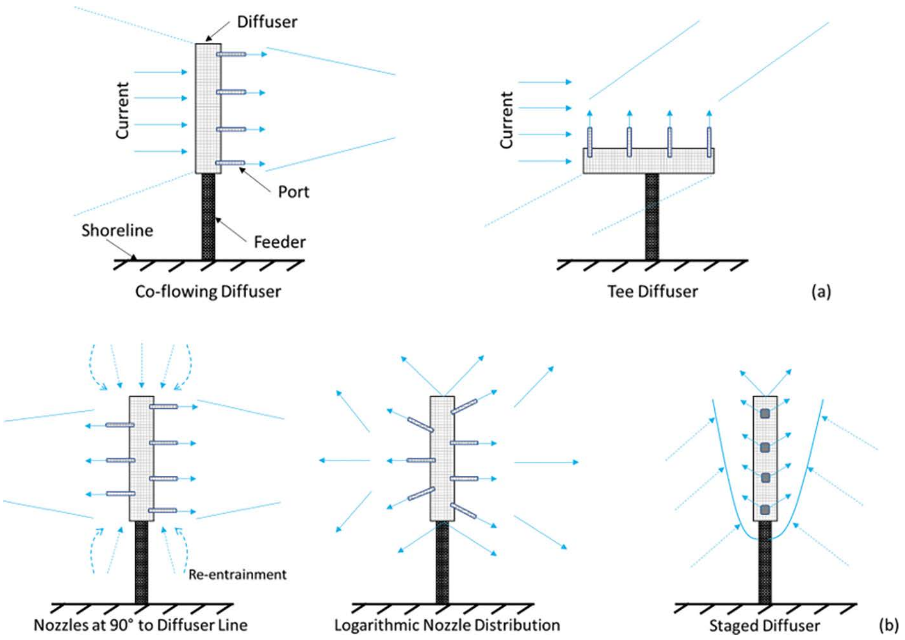


Figure 1.5 Multiport diffusers: (a) Unidirectional diffusers with cross-flow; (b) Alternating diffuser.

is distributed between the number of ports. The configuration of nozzles can vary depending on the ambient condition and design considerations. Figure 1.5 shows five different configurations of multiport diffusers that are commonly used. Another emerging multiport diffuser configuration is the rosette outfall configuration, as shown in Figure 1.6. Unlike the single-port discharges, multiport effluent discharges have been less studied, both experimentally and numerically.

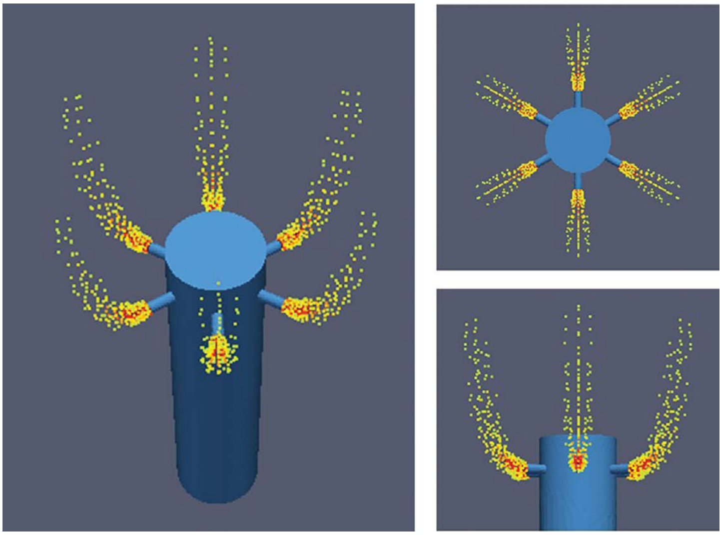


Figure 1.6 Rosette jet, top view.

Introduction

Abraham, G. Jet Diffusion in Stagnant Ambient Fluid. Ph.D. Thesis, TU Delft, Delft, The Netherlands, 1963.

Cheung, S.K.B. , Leung, D.Y.L. , Wang, W. , Lee, J.H.W. , and Cheung, V. VISJET – A computer ocean outfall modeling system. *Proceedings of the International Conference on Computer Graphics* , IEEE Computer Society, 2000.

Doneker, R.L. and Jirka, G.H. CORMIX-GI systems for mixing zone analysis of brine wastewater disposal. Desalination 2001, 139, 263–274.

Fan, L.N. Turbulent Buoyant Jets into Stratified or Flowing Ambient Fluids. Ph.D. Thesis, California Institute of Technology, Pasadena, CA, 1967.

Frick, W.E. Visual Plumes mixing zone modeling software. Environ. Model. Softw. 2004, 19, 645–654. http://www.epa.gov/ceampubl/swater/vplume/.

Hashim, A. and Hajjaj, M. Impact of desalination plants fluid effluents on the integrity of seawater, with the Arabian Gulf in perspective. Desalination 2005, 182, 373–393.

Hopner, T. and Windelberg, J. Elements of environmental impact studies on costal desalination plants. Desalination 1996, 108, 11–18.

Jirka, G.H. Integral model for turbulent buoyant jets in unbounded stratified flows, Part 1: The single round jet. *Environ. Fluid Mech*. 2004, 4(2004), 1–56.

Kolmogorov, A.N. The local structure of turbulence in incompressible viscous fluid for very large Reynolds number (in Russian). Cr. Acad. Sci. URSS 1941, 30, 301–305.

Lattemann, S. and Höpner, T. Environmental impact and impact assessment of seawater desalination. *J. Desal*. 2008, 220, 1–15.

Morton, B.R. , Taylor, G.I. , and Turner, J.S. Turbulent gravitational convection from maintained and instantaneous sources. *Proc. R. Soc. Lond. Ser. A Math. Phys. Sci*. 1956, 234, 1–23.

Palomar, P. , Lara, J.L. , Losada, I.J. , Rodrigo, M. , and Alvarez, A. Near-field brine discharge modeling part 1: Analysis of combined tools. J. Desalination 2012, 290, 14–27.

Robinson, D. , Wood, M. , Piggott, M. , and Gorman, G. CFD modeling of marine discharge mixing and dispersion. *J. Appl. Water Eng. Res*. 2015, 4, 152–162.

Turner, J.S. Buoyant plumes and thermals. *Annu. Rev. Fluid Mech*. 1969, 1, 29–44.

Wang, H. and Law, A.W.-K. Second-order integral model for a round turbulent buoyant jet. *J. Fluid Mech*. 2002, 459, 397–428.

Yannopoulos, P.C. An improved integral model for plane and round turbulent buoyant jets. *J. Fluid Mech*. 2006, 547, 267–296.

Zhao, L. , Chen, Z. , and Lee, K. Modeling the dispersion of wastewater discharges from offshore outfalls: A review. *Environ. Rev*. 2011, 19, 107–120.

An introduction to numerical modeling

Daly, B.J. and Harlow, F.H. Transport equations in turbulence. Phys. Fluid. 1970, 13, 2634–2649.

Davidson, P.A. Turbulence: An introduction for scientists and engineers. Oxford University Press, 2015.

Deardorff, J.W. A numerical study of three-dimensional turbulent channel flow at large Reynolds numbers. J. Fluid Mech. 1970, 41, 453–480.

Deltares *Delft3D-FLOW Simulation of Multi-Dimensional Hydrodynamic Flows and Transport Phenomena Including Sediments, User Manual*. Deltares Delft, 2014.

DHI *MIKE 3 Flow Model FM; Hydrodynamic Module Scientific Documentation; Module-User Guide*. Danish Hydraulic Institute, 2017.

Durbin, P.A. Separated flow computations with the k-epsilon-v-squared model. AIAA J. 1995, 33, 659–664. doi:10.2514/3.12628.

Elder, J. The dispersion of marked fluid in turbulent shear flow. J. Fluid Mech. 1959, 5, 544–560

Harlow F.H. and Welch J.E. Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface, Physics Fluids 1965, 8, 2182.

Hervouet, J.-M. TELEMAC, a hydroinformatic system. La houille blanche 1999, 3–4, 21–28.

Heyrani, M. , Mohammadian, A. , Nistor, I. , and Dursun, O.F. Numerical modeling of venturi flume. Hydrology 2021, 8, 27.

Hirt, C.W. , & Nichols, B.D. Volume of fluid (VOF) method for the dynamics of free boundaries. J. Comput. Physics 1981, 39(1), 201–225.

Kolmogorov, A.N. Equations of turbulent motion in an incompressible fluid. Proc. SSSR Acad. Sci. 1941, 30, 299–303.

Kumar, R. , and Dewan, A. Assessment of buoyancy-corrected turbulence models for thermal plumes. Eng. Appl. Comput. Fluid Mech. 2013, 7, 239–249.

Launder, B.E. , and Spalding, D.B. The numerical computation of turbulent flows. Comput. Methods Appl. Mech. Eng. 1974, 3, 269–289.

Menter, F. , Zonal two equation k-ω turbulence models for aerodynamic flows. *In* 23rd fluid dynamics, plasmadynamics, and lasers conference. 1993.

Nelson, K. and Fringer, O. Sediment dynamics in wind wave-dominated shallow-water environments. J. Geophys. Res. Oceans 2018, 123, 6996–7015.

Prandtl, L. 7. Bericht über Untersuchungen zur ausgebildeten Turbulenz. ZAMM-J. Appl. Math. Mech. 1925, 5, 136–139.

Rodi, W. Turbulence models and their application in hydraulics. CRC Press, 1993.

Rodi, W. Turbulence modeling and simulation in hydraulics: a historical review. J.Hydraul. Eng. 2017, 143, 03117001.

Rodi, W. , Constantinescu, G. , and Stoesser, T. Large-Eddy simulation in hydraulics, 1st edn. CRC Press, 2013.

Shih, T.H. , Liou, W.W. , Shabbir, A. , Yang, Z. and Zhu, J. , *A new k-ε eddy viscosity model for high reynolds number turbulent flows* . Computers & fluids, 1995, 24(3), 227–238.

Smagorinsky, J. General circulation experiments with the primitive equations: I. The basic experiment. Mon. Weather Rev. 1963, 91, 99–164.

Spalart, P.R. Comments on the feasibility of LES for wings, and on a hybrid RANS/LES approach. *Proceedings of First AFOSR International Conference on DNS/LES* . Greyden Press, 1997.

Spalart, P.R. Detached-Eddy simulation. Ann. Rev. Fluid Mech. 2008, 41, 181–202. doi: 10.1146/annurev.fluid.010908.165130.

Van Maele, K. and Merci, B. Application of two buoyancy-modified k-ε turbulence models to different types of buoyant plumes. Fire Safety J. 2006, 41, 122–138.

Warner, J.C. , Sherwood, C.R. , Signell, R.P. , Harris, C.K. , and Arango, H.G. Development of a three-dimensional, regional, coupled wave, current, and sediment-transport model. Comput. Geosci. 2008, 34, 1284–1306.

Wilcox, D.C. Reassessment of the scale-determining equation for advanced turbulence models. AIAA *J.* 1988, 46(11), 1299–1310.

Worthy, J. , Sanderson, V. , and Rubini, P. Comparison of modified k-ε turbulence models for buoyant plumes. Numer. Heat Tr. B: Fund. 2001, 39, 151–165.

Yan, X. and Mohammadian, A. Numerical modeling of vertical buoyant jets subjected to lateral confinement. J. Hydraul. Eng. 2017, 143, 04017016.

Yakhot, VS ., Orszag, SA ., Thangam, S. , Gatski, TB ., Speziale, C. Development of turbulence models for shear flows by a double expansion technique. Physics of Fluids A: Fluid Dynamics. 1992, 4(7), 1510–1520.

An introduction to OpenFOAM

Gildeh, H.K. , Mohammadian, A. , and Nistor, I. Inclined dense effluent discharge modelling in shallow waters. J. Environ. Fluid Mech. 2021, 21, 955–998. doi:10.1007/s10652-021-09805-6.

Issa, R.I. Solution of the implicitly discretized fluid flow equations by operator-splitting. J. Comput. Phys. 1985, 62, 40–65.

Jasak, H. Error analysis and estimation for the finite volume method with applications to fluid flows. Ph.D. Thesis, Imperial College of Science, Technology and Medicine, 1996.

Jasak, H. , Weller, H. , and Nordin, N. In cylinder CFD simulation using a C++ object-oriented toolkit. *SAE Technical Papers*, 2004.

Juretic, F. Error analysis in finite volume. Ph.D. thesis, Imperial College of Science, Technology and Medicine, 2004.

Millero, F.J. and Poisson, A. International one-atmosphere equation of state of sea water. J. Deep-Sea Res. 1981, 28A(6), 625–629.

Applications

Abou-Elhaggag, M.E. , Elgamal, M. , and Farouk, M.I. Experimental and numerical investigation of desalination plant outfalls in limited disposal areas. J. Environ. Prot. 2011, 2, 828–839.

Ajeel Fenjan, S. , Bonakdari, H. , Gholami, A. , and Akhtari, A.A. Flow variables prediction using experimental, computational fluid dynamic and artificial neural network models in a sharp bend. Int. J. Eng. Trans. A Basics 2016, 29(1), 14–22.

Alfaifi, H. , Mohammadian, A. , Gildeh, H.K. , and Gharavi, A. Experimental and numerical study of the characteristics of thermal and nonthermal offset buoyant jets discharged into stagnant water. Desalin. Water Treat. 2019, 141, 171–186.

Bemporad, G.A. Simulation of round buoyant jet in stratified flowing environment. J. Hydraul. Eng. 1994, 120, 529–543.

Bloomfield, L.J. and Kerr, R. Inclined turbulent fountains. J. Fluid Mech. 2002, 451, 283–294.

Chan, S.N. and Lee, J.H.W. Particle tracking modeling of sediment-laden jets. Adv. Geosci. 2014, 39, 107–114, https://doi.org/10.5194/adgeo-39-107-2014.

Chen, C.J. and Rodi, W. A Review of Experimental Data of Vertical Turbulent Buoyant Jets. Pergamon Press: Tarrytown, NY, USA, 1980.

Cipollina, A. , Brucato, A. , Grisafi, F. , and Nicosia, S. Bench scale investigation of inclined dense jets. J. *Hydraul. Eng.* 2005, 131, 1017–1022.

Dejoan, A. , Santiago, J.L. , Pinelli, A. , and Martilli, A. Comparison between LES and RANS computations for the study of contaminant dispersion in the MUST field experiment. Am. Meterol. Soc. 2007. submitted.

Duraisamy, K. , Zhang, Z.J. , and Singh, A.P. *New Approaches in Turbulence and Transition Modeling Using Data-driven Techniques* . AIAA Aerospace Sciences Meeting, 2015.

El-Amin, M.F. , Sun, S. , Heidemann, W. , and Muller-Steinhagen, H. Analysis of a turbulent buoyant confined jet modeled using realizable k-ϵ model. Heat Mass Transf. 2010, 46, 943–960.

Fletcher, D.F. , McCaughey, M. , and Hall, R.W. Numerical simulation of a laminar jet flow: a comparison of three CFD models. Comput. Phys. Commun. 1993, 78(1–2), 113–120. https://doi.org/10.1016/0010-4655(93)90147-5.

Germano, M. , Piomelli, U. , Moin, P. , and Cabot, W.H. A dynamic subgrid-scale eddy viscosity model. Phys. Fluids A: Fluid Dyn. 1991, 3, 1760–1765.

Ghaisas, N.S. , Shetty, D.A. , and Frankel S.H. Large eddy simulation of turbulent horizontal buoyant jets. J. Turbulence. 2015, 16(8), 772–808. DOI: 10.1080/14685248.2015.1008007

Gildeh, H.K. Numerical Modeling of Thermal/Saline Discharges in Coastal Waters. Master's Thesis, University of Ottawa, Ottawa, ON, Canada, 2013.

Gildeh, H.K. , Mohammadian, A. , Nistor, I. , and Qiblawey, H. Numerical modeling of turbulent buoyant wall jets in stationary ambient water. J. Hydraul. Eng. 2014, 140(6), 04014012.

Gildeh, H.K. , Mohammadian, A. , Nistor, I. , and Qiblawey, H. Numerical modeling of 30_ and 45_ inclined dense turbulent jets in stationary ambient. Environ. Fluid Mech. 2015, 15, 537–562.

Gildeh, H.K. , Mohammadian, A. , Nistor, I. , Qiblawey, H. , and Yan, X. CFD modeling and analysis of the behavior of 30° and 45° inclined dense jets—New numerical insights. J. Appl. Water Eng. Res. 2016, 4, 112–127.

Gildeh, H.K. , Mohammadian, A. , and Nistor, I. Inclined dense effluent discharge modelling in shallow waters. Environ. Fluid Mech. 2021, 21, 955–987. https://doi.org/10.1007/s10652-021-09805-6

Gildeh, H.K. , Mohammadian, A. , and Nistor, I. Vertical dense effluent discharge modelling in shallow waters. Water 2022, 14, 2312. https://doi.org/10.3390/w14152312.

Gullbrand, J. and Chow, F. Investigation of numerical errors, subfilter-scale models, and subgrid-scale models in turbulent channel flow simulations. In Proceedings of the Summer Program; Center for Turbulence Research; NASA Ames/Stanford University: Stanford, CA, USA, 2002, pp. 87–104.

Huai, W. , Li, Z.-W. , Qian, Z.-D. , Zeng, Y.-H. , Han, J. and Peng, W.-Q. Numerical simulation of horizontal buoyant wall jet. J. Hydrodyn. 2010, 22, 58–65.

Jiang, B. , Law A.W.K. , and Lee J.H.W. Mixing of 30° and 45° inclined dense jets in shallow coastal waters. J. Hydraul. Eng. 2014, 140(3), 241–253.

Jiang, M. and Law, A.W.K. Mixing of swirling inclined dense jets—A numerical study. J. Hydro-Environ. Res. 2018, 21, 118–130.

Karimpour, F. Turbulence Modelling of Stably Stratified Wall-Bounded Flows. Ph.D. Thesis, Colorado State University, Fort Collins, CO, USA, 2014.

Kikkert, G.A. , Davidson, M. , and Nokes, R.I. Inclined negatively buoyant discharges. J. Hydraul. Eng. 2007, 133, 545–554.

Kochkov, D. , Smith, J.A. , Alieva, A. , Wang, Q. , Brenner, M.P. , and Hoyer, S. Machine learning accelerated computational fluid dynamics. PNAS 2021. doi: 10.1073/pnas.2101784118.

Launder, A. and Spalding, D.B. The numerical computation of turbulent flows. Comput. Methods Appl. Mech. Eng. 1974, 3(2), 269–289.

Launder, B.E. and Sharma B.I. Application of the energy-dissipation model of turbulence to the calculation of flow near a spinning disc. Lett. Heat Mass Transfer. 1974, 1(2), 131–137.

Law, A.W.-K. and Herlina, H. An experimental study on turbulent circular wall jets. J. Hydraul. Eng. 2002, 128, 161–174.

Lee, A.W.-T. and Lee, J.H.-W. E_ect of lateral confinement on initial dilution of vertical round buoyant jet. J. Hydraul. Eng. 1998, 124, 263–279.

Li, Z.-W. , Huai, W. , and Han, J. Large eddy simulation of the interaction between wall jet and O_set jet. J. Hydrodyn. 2011, 23, 544–553.

Lou, Y. , He, Z. , Jiang, H. , and Han, X. Numerical simulation of two coalescing turbulent forced plumes in linearly stratified fluids. Phys. Fluids 2019, 31(3), 037111.

Lui, H.F.S. and Wolf, W.R. Construction of reduced-order models for fluid flows using deep feed-forward neural networks. J. Fluid Mech. 2019, 872, 963–994.

Mohan, A.T. and Gaitonde, D.V. A deep learning based approach to reduced order modeling for turbulent flow control using LSTM neural networks. *arXiv* 2018. DOI: 10.48550/arXiv.1804.09269.

Mohammadaliha, N. et al. Numerical investigation of nozzle geometry effect on turbulent 3-D water offset jet flows. J. Appl. Fluid Mech. 2016, 9, 2083–2095.

Mohammadian, A. , Gildeh, H.K. , and Nistor, I. CFD modeling of effluent discharges: A review of past numerical studies. Water 2020, 12(3), 856.

Nemlioglu, S. and Roberts, P.J. Experiments on dense jets using three-dimensional laser-induced fluorescence (3DLIF). In Proceedings of the 4th International Conference on Marine Waster Water Discharges & Coastal Environment (MWWD), Antalya, Turkey, 6–10 November 2006.

Novati, G. , de Laroussilhe, H. L. , and Koumoutsakos, P. Automating turbulence modelling by multi-agent reinforcement learning. Nat. Mach. Intell. 2021, 3(1), 87–96.

Oliver, C.J. , Davidson, M.J. , and Nokes, R.I. K-ϵ predictions of the initial mixing of desalination discharges. Environ. Fuild Dyn. 2008, 8, 617–625.

Oliver, C.J. , Davidson, M.J. , and Nokes, R.I. Behavior of dense discharges beyond the return point. J. Hydraul. Eng. 2013, 139, 1304–1308.

Papakonstantis, I.G. , Christodoulou, G.C. , and Papanicolaou, P. Inclined negatively buoyant jets 1: Geometrical characteristics. J. Hydraul. Res. 2011a, 49, 3–12.

Papakonstantis, I.G. , Christodoulou, G.C. , and Papanicolaou, P. Inclined negatively buoyant jets 2: Concentration measurements. J. Hydraul. Res. 2011b, 49, 13–22.

Piggott, M. , Gorman, G.J. , Pain, C.C. , Allison, P.A. , Candy, A.S. , Martin, B.T. and Wells, M.R. A new computational framework for multi-scale ocean modelling based on adapting unstructured meshes. Int. J. Numer. Methods Fluids 2008, 56, 1003–1015.

Ramezani, M. , Abessi, O. and Firoozjaee, A.R. Effect of proximity to bed on 30° and 45° inclined dense jets: A numerical study. Environ. Process. 2021, 8, 1141–1164. https://doi.org/10.1007/s40710-021-00533-z

Roberts, P.J.W. , Ferrier, A. , and Daviero, G. Mixing in inclined dense jets. J. Hydraul. Eng. 1997, 123, 693–699.

Savage, SB . and Sobey, RJ . Horizontal momentum jets in rotating basins. J. Fluid Mech. 1975, 71, 4, 755–768. https://doi.org/10.1017/S0022112075002832.

Shao, D. Desalination Discharge in Shallow Coastal Waters. Ph.D. Thesis, Nanyang Technological University, Singapore, 2009.

Shao, D. and Law, A.W.K. Integral modelling of horizontal buoyant jets with asymmetrical cross sections. In Proceedings of the 7th International Symposium on Environmental Hydraulics, Singapore, 7–9 January 2014.

Sharp, J.J. The use of a buoyant wall jet to improve the dilution of a submerged outfall. Proc. Inst. Civ. Eng. 1975, 59, 527–534.

Sharp, J. J. , and Vyas, B. D. (1977). The buoyant wall jet. Proc. Inst. Civ. Eng. Part 2 Res. Theory, 63(3), 593–611.

Shih, T.H. , Liou, W.W. , Shabbir, A. , Yang, Z. , and Zhu, J. A new k–ϵ eddy-viscosity model for high Reynolds number turbulent flows model development and validation. Comput. Fluids 1995, 24, 227–238.

Smagorinsky, J. General circulation experiments with the primitive equations. Mon. Weather Rev. 1963, 91, 99–152.

Tracey, B.D. , Duraisamy, K. , and Alonso, J.J. A *Machine Learning Strategy to Assist Turbulence Model Development* . 53rd AIAA Aerospace Sciences Meeting, 2015.

Vafeiadou, P. , Papakonstantis, I. , and Christodoulou, G. Numerical simulation of inclined negatively buoyant jets. In Proceedings of the 9th International Conference on Environmental Science and Technology, Rhodes, Greece, 1–3 September 2005.

Verhoff, A. The Two-Dimensional Turbulent Wall Jet with and without an External Stream, Rep. No. 626, Princeton University: Princeton, NJ, USA, 1963.

Yan, X. and Mohammadian, A. Numerical Modeling of vertical buoyant jets subjected to lateral confinement. J. Hydraul. Eng. 2017, 143(7), 04017016.

Yan, X. and Mohammadian, A. Numerical modeling of multiple inclined dense jets discharged from moderately spaced ports. Water 2019, 11(10), 2077. DOI: 10.3390/w11102077.

Yan, X. , Mohammadian, A. , and Chen, X. Numerical modeling of inclined plane jets in a linearly stratified environment. Alexandria Eng. J. 2020, 59(3). doi:10.1016/j.aej.2020.05.023.

Yang, Z. and Shih, T.H. New time scale based k-ε model for near-wall turbulence. AIAA Journal 1993, 31(7), 1191–1198.

Zhang, S. , Law, A.W.K. , and Zhao, B. Large eddy simulations of turbulent circular wall jets. Int. J. Heat Mass Transf. 2015, 80, 72–84. https://doi.org/10.1016/j.ijheatmasstransfer.2014.08.082.

Zhang, S. , Jiang, B. , Law, A.W.-K. , and Zhao, B. Large eddy simulations of 45° inclined dense jets. Environ. Fluid Mech. 2016, 16, 101–121.

Zhang, S. , Law, A.W.K. , and Jiang, M. Large eddy simulations of 45° and 60° inclined dense jets with bottom impact. J. Hydro Environ. Res. 2017, 15, 54–66.

Zhang, Z. , Guo, Y. , Zeng, J. , Zheng, J. , and Wu, X. Numerical simulation of vertical buoyant wall jet discharged into a linearly stratified environment. J. Hydraul. Eng. 2018, 144(7), 06018009.

Zhao, Y. , Akolekar, H.D. , Weatheritt, J. , Michelassi, V. , and Sandberg, R.D. RANS turbulence model development using CFD-driven machine learning. J. Comput. Phys. 2020, 411, 1–19.