Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2017,29(1):172-175
DOI: 10.1016/S1001-6058(16)60729-1
www.sciencedirect.com/
science/journal/l0016058

Numerical study of the flow and dilution behaviors of round buoyant jet in counterflow*

Meng Gao (高猛), Wen-xin Huai (槐文信), Yi-tian Li (李义天), Wei-jie Wang (王伟杰)
State Key Laboratory of Water Resources and Hydropower Engineering Science, Wuhan University, Wuhan 430072, China, E-mail: gm60024@whu.edu.cn

(Received October 17, 2016, Revised December 9, 2016)

Abstract: Pollutants are usually discharged into the receiving water bodies in the form of a turbulent jet or plume, and the presence of a counterflow enhances the initial dilution of the jet effluent. To understand the behaviors of jets in actual situations, a round buoyant jet issued horizontally into a uniform counterflow is simulated for different combinations of densimetric Froude number and jet-to-current velocity ratio. A two-phase mixture model is used to simulate this flow, and the renormalization group k - ε model is used to address the flow turbulence. The inter-phase interactions are described in terms of the relative slip velocity between phases. The jet features, including the trajectory of the jet centerline and the decay of the centerline velocity and the concentration, are investigated. The length scale analysis reveals the relationships between the distance and the centerline dilutions, and different flow mechanisms are revealed before and after the penetration point.

Key words: Buoyant jet, counterflow, mixture model, velocity decay, concentration dilution

The wastewater from industrial areas and cities or the thermal effluents from the cooling systems of power plants are usually discharged into the receiving water bodies via outfalls in the form of turbulent jets or plumes. The behavior of these jets completely differs from that in a stagnant ambient water because the presence of a moving ambient water can significantly change their flow structure and mixing properties. When the jet and the main flow directions are opposite to each other, a turbulent jet is formed in the counterflow. The presence of a counterflow enhances the mixing efficiency of the jet, thereby making this flow configuration an interesting issue for many engineering applications, especially, for environmental engineering.

* Project supported by the National Natural Science Foundation of China (Grant Nos. 11672213, 51439007 and 11372232).
Biography: Meng Gao (1989-), Male, Ph. D. Candidate
Corresponding author: Wen-xin Huai,
E-mail: wxhuai@whu.edu.cn

Some experimental studies examined the behavior of a non-buoyant jet in a counterflow using laser-based flow measurement techniques, including the laser-induced fluorescence (LIF) and the laser Doppler anemometry[1]. Numerical simulations were also performed to investigate turbulent round jets in a uniform counterflow[2]. Due to the merging of buoyant jets, the flow and mixing fields of these jets are generally complex and difficult to predict. Lee[3] investigated the global spreading patterns of a round buoyant jet in a counterflow by conducting LIF measurements in a laboratory. These measurements offer abundant information about the concentration field.

In this paper, a round buoyant jet in a uniform counterflow is simulated using a two-phase mixture model. Several flow cases with a densimetric Froude number (Fr) ranging from 3 to 10 and a velocity ratio (R) ranging from 5 to 15 are considered in the simulation, focusing on the central vertical plane, on which the jet behavior obviously changes along with the jet trajectory. The flow behaviors, including the jet centerline velocity and dilution, are investigated via a length scale analysis.

The simulation is performed using a two-phase mixture model. The wastewater jet flow is assumed to comprise the water and the separate, interpenetrating particle phases. α_s and α_f denote the volume fractions of the particle and fluid phases, respectively. The mixture model allows these two phases to move at different velocities with slip velocities. A Schiller-Naumann drag model is used to describe the interaction between these phases[4,5].

The continuity equation of the mixture takes the following form

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \boldsymbol{u}_m) = 0 \tag{1}$$

where ρ_m and $\boldsymbol{u}_m$ denote the mixture density and the mass-averaged velocity, respectively. These variables can be determined as $\rho_m = \sum_{i=1}^{n} \alpha_i \rho_i$ and $\boldsymbol{u}_m = \frac{1}{\rho_m} \cdot \sum_{i=1}^{n} \alpha_i \rho_i \boldsymbol{u}_i$, where α_i is the volume fraction of phase i.

The momentum equation for the mixture can be obtained by taking the sum of the individual momentum equations for all phases expressed as

$$\frac{\partial}{\partial t} \rho_m \boldsymbol{u}_m + \nabla \cdot (\rho_m \boldsymbol{u}_m \boldsymbol{u}_m) = -\nabla p + \nabla \cdot [\mu_m (\nabla \boldsymbol{u}_m + \nabla \boldsymbol{u}_m^{\mathrm{T}})] + \nabla \cdot \left(\sum_{i=1}^{n} \alpha_i \rho_i \boldsymbol{u}_{dr,i} \boldsymbol{u}_{dr,i} \right) + \rho_m \boldsymbol{g} + \boldsymbol{F} \tag{2}$$

where n is the number of phases $(n = 2)$, $\boldsymbol{F}$ is the body force, μ_m is the viscosity of the mixture $(\mu_m = \sum_{i=1}^{n} \alpha_i \mu_i)$, and $\boldsymbol{u}_{dr,i}$ is the drift velocity for the secondary phase $(\boldsymbol{u}_{dr,i} = \boldsymbol{u}_i - \boldsymbol{u}_m)$. The drift velocity for the secondary phase and the relative velocity $\boldsymbol{u}_{sf}$ are related as $\boldsymbol{u}_{dr,s} = (1 - \alpha_s) \boldsymbol{u}_{sf}$, where $\boldsymbol{u}_{sf}$ is the velocity of the secondary phase relative to the velocity of the primary phase defined as

$$\boldsymbol{u}_{sf} = \frac{\tau_p (\rho_s - \rho_m)}{f_{\mathrm{drag}} \rho_s} \left[\boldsymbol{g} - (\boldsymbol{u}_m \cdot \nabla \boldsymbol{u}_m) - \frac{\partial \boldsymbol{u}_m}{\partial t} \right] \tag{3}$$

where τ_p is the particle relative time, which is determined as $\tau_p = \rho_s d_s^2 / (18 \mu_m)$, where d_s is the diameter of the particles of the secondary phase and f_{drag} is the default drag force. If $Re \le 1\,000$, then $f_{\mathrm{drag}} = 1 + 0.15 Re^{0.687}$. By contrast, if $Re > 1\,000$, then $f_{\mathrm{drag}} = 0.0183 Re$, where Re is the Reynolds number.

The renormalization group k-ε model is used to close the governing equations. The turbulent kinetic energy k and its rate of dissipation ε are expressed as follows

$$\frac{\partial}{\partial t}(\rho_m k) + \nabla \cdot (\rho_m \boldsymbol{u}_m k) = \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_k} \nabla k \right) + G_{k,m} - \rho_m \varepsilon \tag{4}$$

$$\frac{\partial}{\partial t}(\rho_m \varepsilon) + \nabla \cdot (\rho_m \boldsymbol{u}_m \varepsilon) = \nabla \cdot \left(\frac{\mu_{t,m}}{\sigma_\varepsilon} \nabla \varepsilon \right) + \frac{\varepsilon}{k} (C_{1\varepsilon}^* G_{k,m} - C_{2\varepsilon} \rho_m \varepsilon) \tag{5}$$

where $\mu_{t,m}$ is the turbulent viscosity $(\mu_{t,m} = \rho_m C_\mu k^2 / \varepsilon)$, $G_{k,m}$ is the production term of the turbulence kinetic energy $(G_{k,m} = \mu_{t,m} [\nabla \boldsymbol{u}_m + (\nabla \boldsymbol{u}_m)^{\mathrm{T}}] : \nabla \boldsymbol{u}_m)$, and $C_{1\varepsilon}^*$ is expressed as $C_{1\varepsilon}^* = C_{1\varepsilon} - \eta (1 - \eta / \eta_0) / (1 + \beta \eta^3)$. The model constants $C_{1\varepsilon}$, $C_{2\varepsilon}$, C_μ, η_0, β, σ_μ and σ_ε take the following default values: $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.68$, $C_\mu = 0.0845$, $\eta_0 = 4.377$, $\beta = 0.012$, $\sigma_\mu = 1.0$ and $\sigma_\varepsilon = 1.3$.

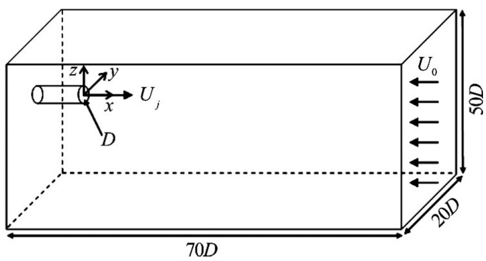

Fig.1 Schematic diagram of the flow configuration

A round buoyant jet with initial velocity U_j, density ρ_0, Reynolds number Re_0 and nozzle diameter D is discharged into the ambient water with an infinite depth, velocity U_0, and density ρ_a. The numerical domain has the dimension of $70D \times 20D \times 50D$ in the axial (x), lateral (y), and vertical (z) directions, as shown in Fig.1.

Similar to previous experiments, salt water is used as the negatively buoyant jet effluent. The salt has a density of 1 035.1 kg/m^3, while the water has a den-