Available online at www.sciencedirect.com
ScienceDirect
Journal of Hydrodynamics
2016,28(5):778-793
DOI: 10.1016/S1001-6058(16)60680-7
www.sciencedirect.com/
science/journal/10016058

# Modelling of a non-buoyant vertical jet in waves and currents*

Zhen-shan XU (徐振山)[1,2], Yong-ping CHEN (陈永平)[1,2], Jian-feng TAO (陶建峰)[1,2], Yi PAN (潘毅)[2], Chang-kuan ZHANG (张长宽)[2], Chi-Wei LI (李志伟)[3]

1. State Key Laboratory of Hydrology-Water Resources and Hydraulic Engineering, Hohai University, Nanjing 210098, China, E-mail: zsxu2006@ hhu.edu.cn
2. College of Harbor, Coastal and Offshore Engineering, Hohai University, Nanjing 210098, China
3. Department of Civil and Environmental Engineering, The Hong Kong Polytechnic University, Hong Kong, China

(Received February 7, 2015, Revised November 11, 2015)

**Abstract:** A generic numerical model using the large eddy simulation (LES) technique is developed to simulate a non-buoyant vertical jet in wave and/or current environments. The experimental data obtained in five different cases, i.e., one case of the jet in a wave only environment, two cases of the jet in a cross-flow only environment and two cases of the jet in a wave and cross-flow coexisting environment, are used to validate the model. The grid sensitivity tests are conducted based on four different grid systems and the results illustrate that the non-uniform grid system C (205×99×126 nodes with the minimum size of 1/10 jet diameter) is sufficiently fine for the modelling. The comparative study shows that the wave-current non-linear interaction should be taken into account at the inflow boundary while modelling the jet in wave and cross-flow coexisting environments. All numerical results agree well with the experimental data, showing that: (1) the jet under the influence of the wave action has a faster centerline velocity decay and a higher turbulence level than that in the stagnant ambience, meanwhile the "twin peaks" phenomenon exists on the cross-sectional velocity profiles, (2) the jet under a cross-flow scenario is deflected along the cross-flow with the node in the downstream, (3) the jet in wave and cross-flow coexisting environments has a flow structure of "effluent clouds", which enhances the mixing of the jet with surrounding waters.

**Key words:** large eddy simulation (LES), turbulent jet, wave, cross-flow, wave and cross-flow coexisting

## Introduction

The disposal of wastewater into coastal waters is one of common sewage treatment approaches in coastal cities. It is of importance to understand and predict the movement of the disposed wastewater, which is usually in the form of a turbulent jet, in order to make a more accurate assessment of the wastewater impact on the surrounding environment. In coastal waters, the jet is not only driven by its initial momentum, but also affected by the coastal hydrodynamics, such as tidal currents and/or waves. If the jet is vertically discharged into the ambience, the surrounding tidal currents could be approximated as a series of quasi-steady cross-flows.

Many experiments were carried out to investigate the vertical jet in either the cross-flow or the wave environment. A better understanding of the complex interaction mechanism between the jet and the surrounding waters was achieved by those studies. In the cross-flow environment, the jet is significantly deflected along the cross-flow direction, with the node in the downstream[1]. The flow from this node is supplied by the lateral flow, which is caused by the cross-flow passing over the jet. Apart from that, several large

---

* Project supported by the National Natural Science Foundation of China (Grant Nos. 51379072, 51109074), the Specialized Research Fund for the Doctoral Program of Higher Education (Grant No. 20120094110016), the 111 Project of the Ministry of Education and the State Administration of Foreign Experts Affairs, China (Grant No. B12032) and the Fundamental Research Funds for the Central Universities (Grant No. 2014B02514).

**Biography:** Zhen-shan XU (1988-), Male, Ph. D., Lecturer
**Corresponding author:** Yong-ping CHEN,
E-mail: ypchen@hhu.edu.cn

scale vortex structures, including the shear layer vortices, the horseshoe vortices, the wake vortices and the counter-rotating vortex pair (CVP) exist in the jet body when it is in the environment of a cross-flow[2]. Among those vortex structures, the CVP is considered as the most significant feature[3]. On the other hand, in the wave environment, the jet body sways to and fro with the pace of the wavy fluctuation, making the vortex structures not as clear as those in the cross-flow environment. The experimental measurements made by Chyan and Hwung[4] and Sharp et al.[5] show the existence of a "twin peak" distribution of the jet mean velocity and the concentration on the cross-sectional profiles when the jet is in a regular wave environment. Mossa[6,7] measured both the jet mean and turbulent velocities using an laser Doppler anemometer (LDA) system and found a larger lateral spreading and a higher turbulence level of the jet in the wave environment than those in a stagnant ambience. Recently, Hsiao et al.[8] adopted the particle image velocimetry (PIV) technique to measure the mean and turbulence structures of the jet in the wave environment and similar conclusions as Mossa's were made.

Although the interaction between the jet and the surrounding waters is 3-D, usually only the data on the jet symmetrical plane can be obtained due to the limited measurement techniques. However, the numerical model can provide an effective way to reproduce and predict the 3-D structure of the jet in various environments. For example, Yuan et al.[2] developed a large eddy simulation (LES) model for the jet in the cross-flow environment and four different vortex structures were clearly illustrated in three dimensions. Using a similar numerical model, Cavar and Meyer[9,10] revealed the originating, growing and shedding processes of the vortexes based on the 3-D proper orthogonal decomposition (POD) analysis of the modelling results. Chen et al.[11] and Lu et al.[12] applied the LES model to the simulation of a round jet in regular and random wave environments, and the numerical results confirmed the positive effect of the wave in both horizontal and transverse directions. In fact, numerical models could also generate some valuable results that can hardly be obtained from laboratory experiments. For instance, Muldoon and Acharya[13] reproduced the near field of a jet in the cross-flow environment using a direct numerical simulation (DNS) model. The particle traces originating from the jet orifice were plotted based on the numerical results and were used to visualize the jet flow structure and the spreading characteristics. However, most of various numerical models were developed either for the jet in a wave only environment, or for the jet in a cross-flow only environment, and a model that could be used to successfully simulate the jet in both wave and cross-flow environments is still desirable. In order to develop this kind of models, one problem that should be tackled is to simultaneously generate various dynamics over different temporal and spatial scales, including the jet, the wave and the cross-flow, using the same numerical scheme, another problem to be tackled is to validate the model with a lack of available experimental data. In this study, we first carry out a series of laboratory experiments to obtain some first-hand experimental data, and then develop a generic LES turbulence model with the same computational parameters, but under changeable boundary conditions, which are appropriate for modelling all kinds of dynamics. The robustness and the accuracy of the model are comprehensively examined by a comparison with the experimental data.

## 1. Model description

As the LES turbulence model was successfully applied in the simulation of the jet in various cross-flow environments[2,9,10] as well as that of the jet in various wave environments[11,12], it is chosen in this study to simulate the jet in waves and/or currents. For simplicity, only the non-buoyant jet is considered in this study.

### 1.1 *Governing equations*

Based on the principle of the LES theory, each flow variable $u$ can be decomposed into a large-scale component $\overline{u}$ and a sub-grid scale component $u'$, namely $u = \overline{u} + u'$. The spatially filtered Navier-Stokes equations can be written as:

$$\frac{\partial \overline{u}_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial \overline{u}_i}{\partial t} + \overline{u}_j \frac{\partial \overline{u}_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \overline{p}}{\partial x_i} + g_i + \nu \frac{\partial^2 \overline{u}_i}{\partial x_j \partial x_j} - \frac{\partial R_{ij}}{\partial x_j} \tag{2}$$

where $x_i\,(i = 1,2,3)$ are the spatial coordinates $x$, $y$, $z$ in horizontal, transverse and vertical directions, respectively, $u_i\,(i = 1,2,3)$ are the corresponding velocity components $u$, $v$ and $w$, $p$ is the pressure, $g_i$ is the acceleration due to gravity, $\nu$ is the kinematic viscosity, $\rho$ is the water density, $t$ is the time, $R_{ij}$ is the sub-grid scale stress tensor:

$$R_{ij} = \overline{u_i u_j} - \overline{u}_i \overline{u}_j \tag{3}$$

$$R_{ij} = \tau_{ij} + \frac{1}{3}\delta_{ij} R_{kk} \tag{4}$$

where $R_{kk}$ can be absorbed into the pressure term in Eq.(2), $\tau_{ij}$ is expressed by the Smagorinsky model: