*E-proceedings of the 36th IAHR World Congress*
*28 June – 3 July, 2015, The Hague, the Netherlands*

# NUMERICAL SIMULATION OF A NEUTRALLY BUOYANT ROUND JET IN A WAVE ENVIRONMENT

YEN-LUNG CHEN[1], SHIH-CHUN HSIAO[2], YU-CHENG HOU[3], HAN-LUN WU[4] & YUAN-CHIEH WU[5]

*[1]PhD student, Department of Hydraulics and Ocean Engineering, National Cheng Kung University, Taiwan*
*n88021121@mail.ncku.edu.tw*

*[2]Professor, Department of Hydraulics and Ocean Engineering, National Cheng Kung University, Taiwan*
*scshiao@mail.ncku.edu.tw*

*[3]Master, Department of Hydraulics and Ocean Engineering, National Cheng Kung University, Taiwan*
*ychou@iner.gov.tw*

*[4]PhD, Department of Hydraulics and Ocean Engineering, National Cheng Kung University, Taiwan*
*bluebox627@gmail.com*

*[5]Associate Research Fellow, Institute of Nuclear Energy Research, Atomic Energy Council, Executive Yuan, Taiwan*
*ycwu@iner.gov.tw*

## ABSTRACT

This paper aims to simulate a neutrally buoyant round jet discharged at the mid-depth in the opposite direction to the wave propagation using a three-dimensional numerical model named *FLOW-3D*. The present numerical model solves the Reynolds-averaged Navier-Stokes (RANS) equations combined with standard $k-\varepsilon$ turbulence model, and the free surface deformation is tracked using volume of fluid method (VOF). The round jet is modelled by a horizontal pipe and mass source. The capability of numerical model simulating the round jet in a stagnant environment is firstly validated by an empirical formula. Comparisons between the empirical formula and present numerical results show good agreements. Then, a neutrally buoyant round jet in a wave environment is investigated. The decay of jet centerline velocity is compared with available experimental data. Finally, the variation of the jet cross-sectional profile under different wave height is discussed. The results show that the ratio of the jet vertical width (z-direction) to its horizontal width (y-direction) is insensitive to wave periods, but it increases in the potential core and decreases in the near field with the increase of the wave height.

*Keywords: Water waves; neutrally buoyant round jet; $k-\varepsilon$ model.*

## 1. INTRODUCTION

Discharging jet into coastal water is a common practice in many communal and industrial applications. For instance, the sewage and cooling water are discharged through the outfalls located along a coastline. Most investigations focused on the problem of a jet in a stagnant or steady flow ambient. Detailed flow properties of a round jet in such flows have been extensively studied by laboratory experiments (Cowen et al., 2001; Hussein et al., 1994; Mi et al., 2007; Quinn, 2005). For a jet under the wave environment, some studies have reported that the wave field causes the jet centerline velocity to decrease and the jet width to increase (Chang et al., 2009; Hsiao et al., 2011; Mori and Chang, 2003; Ryu et al., 2005). Furthermore, Chang et al. (2009) observed that the ratio of the jet vertical width (z-direction) to its horizontal width (y-direction) in the potential core region is different from in the near field region. Hsiao et al. (2011) investigated the effect of wave height and phase on the turbulence property by measuring the velocity field.

Due to the limitation of the measurement techniques, the complete three dimensional (3D) flow field is hard to obtain. In the recent years, numerical modeling based on the Navier-Stokes equations serves an alternative to investigate the problem and provides high quality results. AshforthFrost and Jambunathan (1996) investigated the turbulence kinetic energy of a wall jet under Reynolds number 23,000 by the standard $k-\varepsilon$ turbulence model. Aziz and Khan (2011); Aziz et al. (2008) found that the standard $k-\varepsilon$ turbulence model performs equally well or better than the renormalized group $k-\varepsilon$ turbulence model. Chen et al. (2008) studied a vertical round jet discharged in a random wave environment by a 3D large eddy simulation (LES) model. However, they did not consider the jet discharged in a direction opposite to the wave propagation direction.

The aim of this paper is to study the cross-sectional profile of a horizontal round jet in an opposite water waves to further understand the kinematic characteristic of jet under water waves. A 3D numerical model called *FLOW-3D* is used to simulate the neutrally buoyant round jet in a wave environment. The model solved Reynolds-averaged Navier-Stokes (RNAS) equations with standard $k-\varepsilon$ turbulence model. The setup of numerical tank is based on the previous physical experiment (Hsiao et al., 2011). The effect of wave height and wave period on the jet cross-sectional profile is discussed in this paper for the lack of 3D velocity field in the experiment.

## 2. NUMERICAL MODEL

A well-known computational fluid dynamics code, *FLOW-3D* solves Navier-Stokes type equations embedded with various turbulence models. In this paper, RANS equations are solved with standard $k-\varepsilon$ turbulence model and volume of fluid (VOF) method is applied to track the free surface elevation (Hirt and Nichols, 1981). The interface between fluid and solid boundaries is simulated with the fractional area-volume obstacle representation (FAVOR) method. This method computes open area and volume in each cell to define the area that is occupied by obstacle. The continuity and momentum equation are described as:

$$\frac{\partial\langle u_i\rangle}{\partial x_i}=S(x_i) \qquad [1]$$

$$\frac{\partial\langle u_i\rangle}{\partial t}+\langle u_j\rangle\frac{\partial\langle u_i\rangle}{\partial x_j}=-\frac{1}{\rho}\frac{\partial\langle p\rangle}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\nu\frac{\partial\langle u_i\rangle}{\partial x_j}-\overline{u_i'u_j'}\right] \qquad [2]$$

where $x_i$ represents coordinate directions ($i=1,2,3$ for $x,y,z$ directions, respectively), $\langle u_i\rangle$ and $\langle p\rangle$ are ensemble mean velocity component and mean fluid pressure, $t$ is the time, $\rho$ is the fluid density, $\nu$ is the kinematic viscosity, $g$ is the acceleration due to gravity, and $S(x_i)$ is the mass source term. The Reynolds stress term, $-\overline{u_i'u_j'}$, is modeled by the standard $k-\varepsilon$ turbulence model. For more detailed information on the numerical model, readers can refer to its well-written manual (Flow Science, Inc., 2012). Generally, a round jet is generated by specifying the jet velocity at inflow boundary in the aforementioned studies (Aziz and Khan, 2011; Aziz et al., 2008; Chen et al., 2008). To prevent the wave reflection from the end of wave tank, the jet is composed of a horizontal pipe and mass source and the numerical wave tank is lengthened. The jet discharge ($Q$) is controlled by the mass source term, it can be calculated as shown in Eq. [3], and the mass source term can be described as Eq. [4] for a constant source function over a source region with volume ($V$).

$$\int_{\Omega} S(x_i)d\Omega=Q \qquad [3]$$

$$S(x_i)=\frac{Q}{V} \qquad [4]$$

## 3. Numerical validation

In this study, the experimental data reported by Hsiao et al. (2011) is used to evaluate the capability of numerical model simulating a neutrally buoyant round jet in a stagnant/wave environment. Figure 1 shows a schematic diagram of the main observation area in numerical wave tank, which is 150.00 cm long, 33.00 cm width and up to 40.00 cm deep. The lengthen area is used to avoid the reflected wave from the boundary. The jet discharged horizontally at the water depth $h=33.00\,\text{cm}$ from a tube with an inner diameter $D=0.62\,\text{cm}$. The jet exit velocity $u_j$ and Reynolds number $\text{Re}=u_jD/\nu$ with $\nu=10^{-2}\,\text{cm}^2/\text{sec}$ respectively are $123.00\,\text{cm/sec}$ and 7,626. The length of jet pipe is 13.00 cm. The grid sizes in the y- and z-direction are in the range of $D/7-2D$, and is in the range of $D/3-2D$ in the x-direction.

Firstly, the jet discharged in a stagnant environment is simulated. Figure 2 (a) shows the comparison between the empirical one-seventh power-law velocity profile (Eq. [5]) and numerical result inside the jet pipe. The horizontal velocity is normalized by the jet centerline horizontal velocity ($u_c$), $R$ is the radius of the pipe and $r$ is the distance measured from the centerline. Overall, the results show in a good agreement with empirical formula. Figure 2 (b) shows the normalized horizontal velocity profiles in the near field region and the corresponding virtual origins ($x_0=-4D$) is used. The normalized horizontal velocities show a Gaussian distribution and consist with the empirical formula reported by Hussein et al. (1994).

$$\frac{u}{u_c}=\left(1-\frac{r}{R}\right)^{1/7} \qquad [5]$$

Next, the water waves are added to the numerical wave tank for modeling the jet discharged in a wave environment. The water depth is kept at 33.00 cm. The mid-depth regular waves with the wave height and the wave period are 3.00 cm and 1.00 sec, respectively. The centerline horizontal velocity in wave environment is defined as the maximum horizontal velocity at each vertical cross section. The decay of centerline horizontal velocity is shown in figure 3. The circle and cross separately represent the experimental data (Hsiao et al., 2011) and numerical result in this study. The centerline velocity decay well corresponds to the experimental data even though the discrepancy of its location increases with the increase of distance from the entrance of the jet in the near field region. Overall, the results show in a good agreement with experimental data.

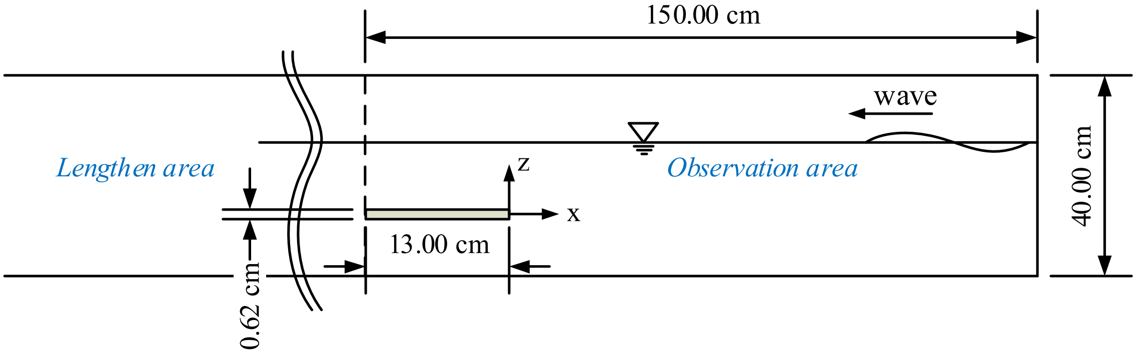


Figure 1. An illustrative sketch of numerical wave tank (side view).

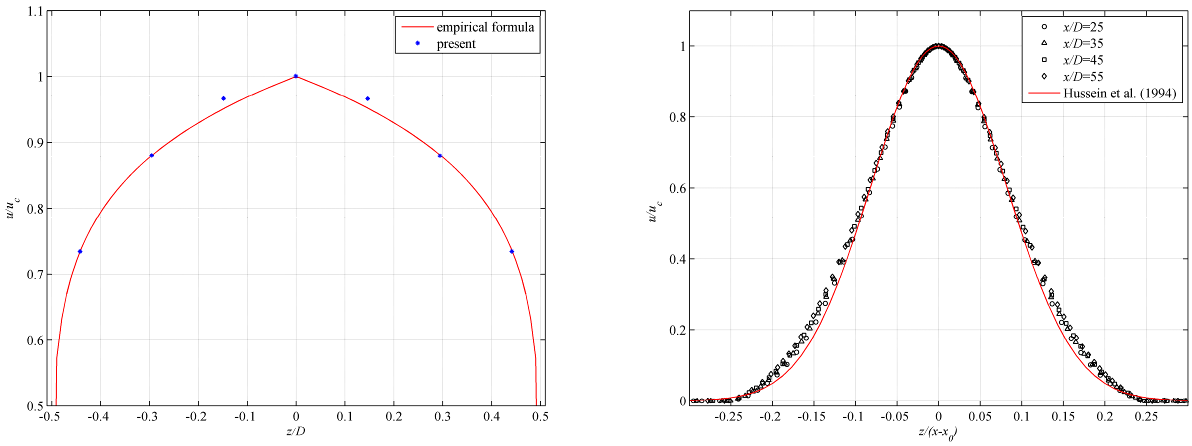


Figure 2. Cross sectional velocity profile on the inside of the jet pipe (left) and in the near field region (right).

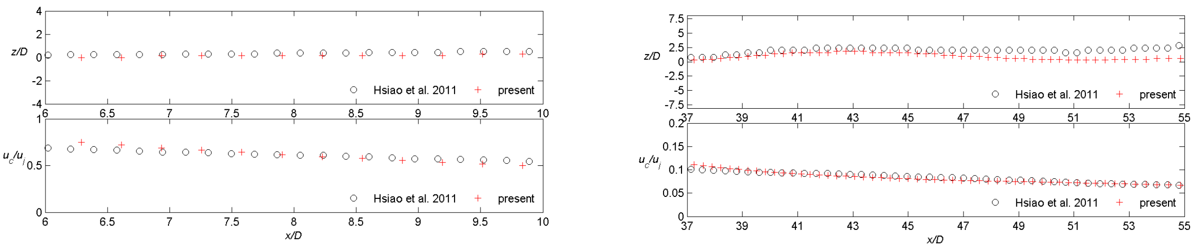


Figure 3. The jet centerline velocity decay under regular waves in the potential core region (left) and near field region (right).

## 4. RESULTS AND DISCUSSION

The previous section showed that *FLOW-3D* model is capable of modeling jet-wave interaction. For understanding the kinematic characteristic of jet under water waves, the horizontal width and vertical width of jet are used to describe the jet cross-sectional profile. To find the width of jet, the horizontal velocity contour in x-direction in the cross section is used and the horizontal/vertical width is defined as the distance along the horizontal/vertical direction between the locations of maximum horizontal velocity to the half of maximum horizontal velocity. Chang et al. (2009) found that the ratio of the jet vertical width to its horizontal width is less than one in the potential core region and greater than one in the near field region at a particular phase. In this section, different wave heights and wave periods are used to study the variation of jet cross-sectional profile at different phase.

Figure 4 shows that the degree of jet cross-sectional profile's deformation increase with the nonlinearity in the potential core region and more violent in the near field region. Even though the ratios vary significantly at different wave phases, the values are mostly less than one in the potential core region and mostly greater than one in the near field region. Interestingly, the ratio decreases with the increase of nonlinearity in the potential core region while it increases in the near field region. In contrast, the ratio is insensitive to wave periods both in the potential core region and near field region, as shown in figure 5.

## 5. CONCLUSIONS

The kinematics of a neutrally buoyant round jet in wave environment was numerically investigated. The jet was successfully modeled by a horizontal pipe which was described by FAVOR method and mass source. The agreement between experimental data, empirical formulas, and numerical results for the jet in a stagnant/wave environment were reasonably well. From the variation of the ratio of the jet vertical width to its horizontal width in different wave nonlinearity, it found that the ratio was insensitive to wave periods, but decrease with the increase of nonlinearity in the potential core region and increase in the near field region. The ratios were mostly less than one in the potential core region and mostly greater than one in the near field region. It was consistent with Chang et al.'s observation from physical experiment.

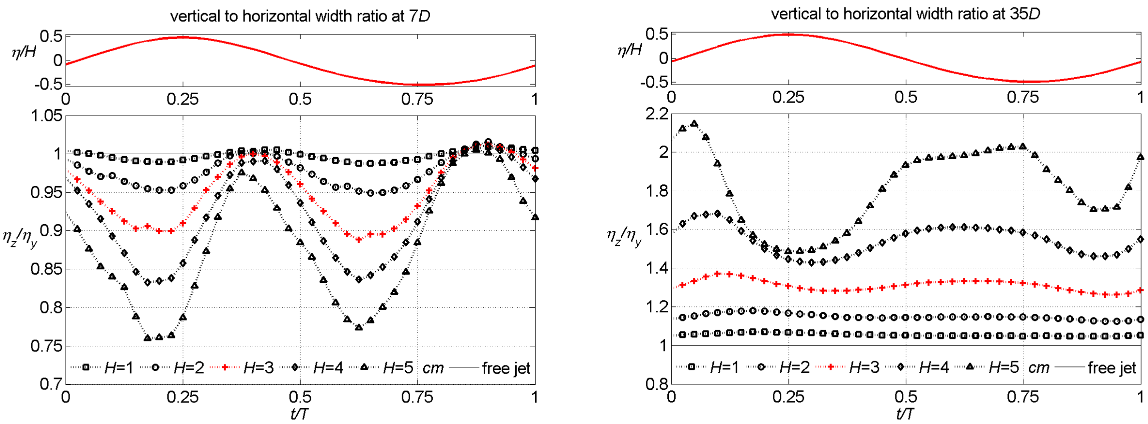


Figure 4. The variation of the ratio of the jet vertical width to its horizontal width under different wave height.

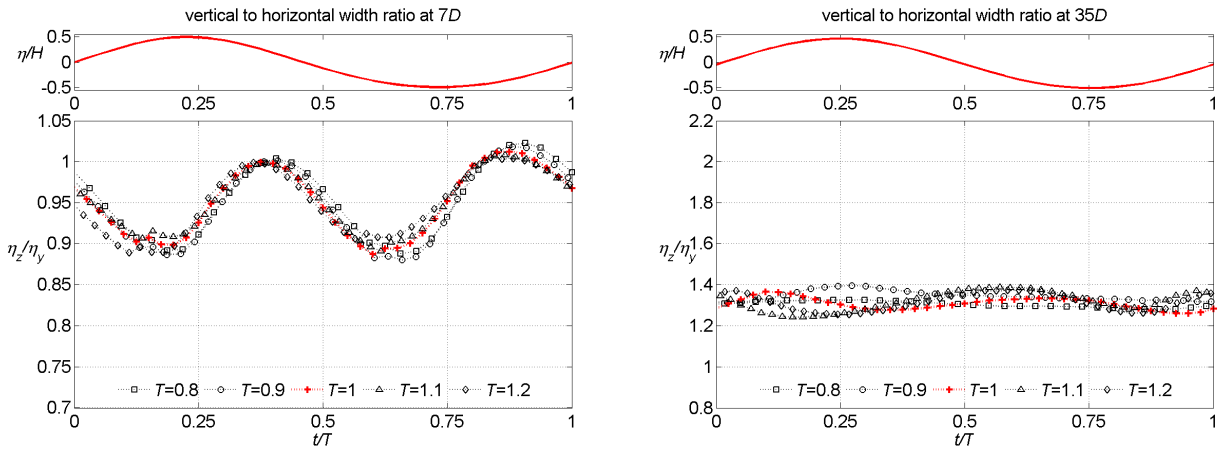


Figure 5. The variation of the ratio of the jet vertical width to its horizontal width under different wave period.

## ACKNOWLEDGMENTS

This research was financially supported by the National Science Council of Taiwan under grants NSC 101-2628-E-006-015-MY3, MOST 103-2221-E-006-229-MY3, and by the Institute of Nuclear Energy Research, Atomic Energy Council, Executive Yuan, Taiwan, under grants of Project no. 1042001INER034 and 1042001INER035.

## REFERENCES

AshforthFrost, S. and Jambunathan, K., (1996). Numerical prediction of semi-confined jet impingement and comparison with experimental data. Int J Numer Meth Fl, 23(3): 295-306.

Aziz, T.N. and Khan, A.A., 2011. Simulation of vertical plane turbulent jet in shallow water. Advances in Civil Engineering, 2011: 1-10.

Aziz, T.N., Raiford, J.P. and Khan, A.A., (2008). Numerical simulation of turbulent jets. Engineering Applications of Computational Fluid Mechanics, 2(2): 234-243.

Chang, K.A., Ryu, Y. and Mori, N., (2009). Parameterization of neutrally buoyant horizontal round jet in wave environment. J Waterw Port C-Asce, 135(3): 100-107.

Chen, Y.P., Li, C.W. and Zhang, C.K., (2008). Numerical modeling of a round jet discharged into random waves. Ocean Eng, 35(1): 77-89.

Cowen, E.A., Chang, K.A. and Liao, Q., (2001). A single-camera coupled ptv-lif technique. Experiments in Fluids, 31(1): 63-73.

Hirt, C.W. and Nichols, B.D., (1981). Volume of fluid method for dynamics of free boundaries. J Comput Phys, 39(1): 201-225.

Hsiao, S.C., Hsu, T.W., Lin, J.F. and Chang, K.A., (2011). Mean and turbulence properties of a neutrally buoyant round jet in a wave environment. J Waterw Port C-Asce, 137(3): 109-122.

Hussein, H.J., Capp, S.P. and George, W.K., 1994. Velocity-measurements in a high-reynolds-number, momentum-conserving, axisymmetrical, turbulent jet. Journal of Fluid Mechanics, 258: 31-75.

Mi, J., Kalt, P., Nathan, G.J. and Wong, C.Y., (2007). Piv measurements of a turbulent jet issuing from round sharp-edged plate. Experiments in Fluids, 42(4): 625-637.

Mori, N. and Chang, K.A., (2003). Experimental study of a horizontal jet in a wavy environment. J Eng Mech-Asce, 129(10): 1149-1155.

Quinn, W.R., (2005). Measurements in the near flow field of an isosceles triangular turbulent free jet. Experiments in Fluids, 39(1): 111-126.

Ryu, Y., Chang, K.A. and Mori, N., (2005). Dispersion of neutrally buoyant horizontal round jet in wave environment. J Hydraul Eng-Asce, 131(12): 1088-1097.

Flow Science, Inc (2012). FLOW-3D User Manual v10.1.