Middle-East Journal of Scientific Research 15 (3): 446-451, 2013
ISSN 1990-9233

DOI: 10.5829/idosi.mejsr.2013.15.3.2749

Submerged Nozzle Performance on Kinetic Energy Dissipation in Static Ambient Fluid

[1]Payam Taheri, [2]Habib Mousavi Jahromi, [3]Houshang Hassoni Zadeh and [4]Heidarali Kashkooli

[1]Department of Irrigation Eng. Science and Research Branch,
Islamic Azad University,Khuozestan, Ahwaz, Iran
[2]Faculty of Water Science engineering, Shahid Chamran University, Ahwaz, Iran
[3]Department of Water Engineering, Shoushtar Branch, Islamic Azad University, Shoushtar, Iran
[4]Department of Irrigation Eng, Science and Research Branch,
Islamic Azad University, Khuozestan, Ahwaz, Iran

Abstract: Behavior of turbulent jets such as free jets (All hydraulic jets that are not affected by solid surface) is relying on jet parameters, environmental and geometric parameters. Submerged jets can be used for washing sediments accumulated in the reservoir floor, intakes and pumps suction basin. In addition, it can be used to dilute pollutants to ambient flow. With the purpose of improving the performance of hydraulic jets, experimental works and numerical analysis of submersible water jet in static ambient fluid are carried out. Laboratory observations in pervious researches showed that the flow velocity near the nozzle has sharp changes but these changes in a certain distance suddenly decreases, this behavior can also be observed in the numerical model output. Also, the comparison between experimental and numerical results has been performed. Comparison showed good agreement between them and the maximum accuracy was observed in near field (before jet establishment) of the jet. With increasing distance from the nozzle and in a certain region, fluctuations in the Reynolds number increases, since this region, Froude number dependence of flow behavior is suddenly reduced.

Key words: Submerged jet · Comsol Multiphysics · Nozzle Froude number · Near Field

INTRODUCTION

Behavior of turbulent jet is relying on jet parameters, environmental and geometric parameters. In this study, effect of jet parameter such as inertial angle on the generating turbulence of the hydraulic jet is considered. Submerged jets can be used for washing sediments accumulated in the reservoir floor, intakes and pumps suction basin. In addition, it can be used to dilute pollutants to ambient flow. Thus, when a pollutant is pumped into receiving water or a river, there is usually a legal or administrative requirement that significant dilution be achieved as rapidly as possible. One of the many techniques for achieving rapid dilution is the turbulent jet, which entrains and mixes large volumes of receiving water with the pollutant discharge.

By this definition, the simple jet is a turbulent flow pattern generated by a continuous source of momentum. A closely related phenomenon is the plume; however, the plume has no initial momentum and is generally considered as a buoyancy drive phenomenon [1].

The fluid mechanics community has investigated turbulent buoyant jets intensively for many years. Some researchers have focused on the physical properties only but others also have examined the environmental effects of the jet such as dispersion of pollutants, sediment management, etc. Following a brief history of studies on hydraulic jet such as wall jet, round jet, buoyant pool, free and submerged jump are presented.

Rajaratnam *et al* [2-6] has a lot of research in the field of turbulent wall jet, free and submerged hydraulic jump and degradation effects of wall jet. Hogg *et al* [7] studied the erosion pattern based on Gaussian distribution of shear stress, their model was able to predict water and wind erosion with sufficient accuracy. Peiqing *et al* and also Chen *et al* [8, 9] examined the energy dissipation in plunge pool and also predicted the pool efficiency.

Corresponding Author: Payam Taheri, Department of Irrigation Eng. Science and Research Branch, Islamic Azad University, Khuozestan, Ahwaz, Iran.

Fig. 1: Experimental setup (left) and Longitudinal section of a jet nozzle (right).

Karim *et al* [10] examined the ability of Fluent software to predict the wall jet behavior with different methods for turbulence modeling. Aderibigbe *et al* [11] and Mazurek *et al* [12] and Xue *et al* [13] focused on wall jet erosion effects. Adduce *et al* [14] investigated the scour by turbulent wall jet and also they measured the erosion development by time. Sequeiros *et al* [15] examined the scour of cohesive soil in combined sewer over flow storage by using wall and round jet. Ahadiyan *et al* [16, 17] focused on hydraulic properties of single jet with different nozzle's shape. Jafarinia *et al* [18] studied and extracted the relationships for degradation effects of hydraulic jet at downstream of siphon spillway. Soleimani *et al* [19] built a laboratory model for their experiments. They studied the hydraulic behavior of single and multi jets on a removable bed.

In submerged hydraulic jet flow field two different regions, near field and far field can be distinguished. The near field of an isolated jet comprises the region where the jet flow gets established as it is discharged into the ambient fluid. It includes the developing wall boundary layer and the expansion of the flow into the ambient fluid associated with the entrainment of ambient water, which causes a down slope decay of flow velocity and wall shear stress. The far field of an isolated jet discharge is the region where buoyancy effects, generated by the sediment originally suspended in the near field, constitute the main driving force of the flow and the velocity and shear stress induced by the initial wall jet are mostly dissipated.

Experimental Setup: The experimental tests were performed in a flume (Figure 1) that was developed by Soleimani (2012) in Hydraulic Laboratory of K.W.P co. The flume has 1.5m wide, 1m deep and 6m long. The supply pipe and nozzle placed at the upstream side and a morning glory spillway with 8cm in diameter and 0.7m in high located at the other side. The spillway was adjusted in order to have constant submergence depth in all experiments. The supply pipe connected to the array of two centrifugal Pumps with flow rate and pressure capacity between 20 to 100 lit/min and 25 to 35m respectively. To ensure the cyclic behavior of the flow, the spillway outlet pipe connected to the pump suction tank. In order to flow establishment, when the flume was filled with water, pumps are turned on and immediately the overflow outlet valve opened. The flow regulated by a control valve and measured with a Rotameter device. Velocity values were recorded at 5 cm intervals a long the central axis of nozzle by the EM flow meter.

Modeling Approach and Comsol Software: The concept of the Finite Element Method can be traced back to the technique used in stress calculations whereby a structure was divided into small sub-structure of various shapes. The structure is then re-assembled after each element has been analyzed. In the past few decades, the Finite Element Method (FEM) has been developed into a key indispensable technology in the modeling and simulation of various engineering systems. In the development of an advanced engineering system, engineers have to go through a very rigorous process of modeling, simulation, visualization, analysis, designing, prototyping, testing and finally, construction. As such, techniques related to modeling and simulation in a rapid and effective way play an increasingly important role in building advanced engineering systems and therefore the application of the FEM has multiplied rapidly. COMSOL is a FEA software package used for analyzing coupled phenomena in physics and engineering applications. This package solves arbitrary systems of partial differential equations (PDEs) and particularly useful for modeling processes involving transport phenomena [20, 21]. This software can be used in many branches of engineering such as Solid and Fluid Mechanics, Electronic and Electromagnetic,

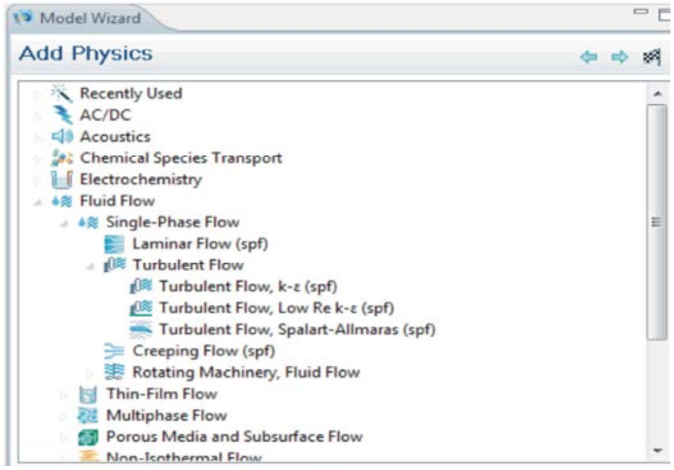

Fig. 2: Comsol model wizard

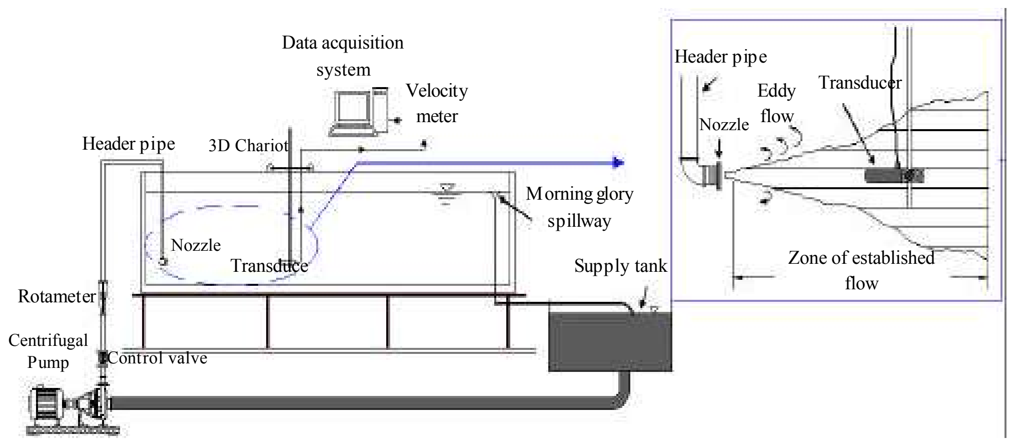

Fig. 3: Flume details plan.

Acoustics, Heat transfer and basic science such as mathematics, chemistry and physics. Comsol is an integrated environment for solving systems of time dependent or stationary second order in space partial differential equations. The software has many advantages including: powerful graphical user interface (GUI), solving multi and zero dimensional (without geometry) problems, user friendly, no need to other software, having a library of materials. But Comsol requires a lot of memory that this is the main disadvantage.

Dynamic Equations: The governing equations for incompressible, turbulent flow are the continuity equation for mass conservation and RANS equations for momentum transport.

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

$$\rho \frac{\partial \overline{u_i}}{\partial t} + \rho \overline{u_j} \frac{\partial \overline{u_i}}{\partial x_j} = \rho \overline{F_i} \frac{\partial \overline{p}}{\partial x_j} + \frac{\partial}{\partial x_j}\left(\mu \frac{\partial \overline{u_i}}{\partial x_j} - \rho \overline{u_i^{'} u_j^{'}} \right) \tag{2}$$

Where $\overline{u_i}$ is the mean velocity component, $u_i^{'}$ the fluctuating velocity component, P the pressure and F is the volumetric force. RNG k?ε turbulence model and wall function are suitable for prediction of the maneuvering hydrodynamics of full-ship model [22]. Turbulent kinetic energy (k) transport equation is:

$$\frac{\partial}{\partial_j}(\rho k)\frac{\partial}{\partial x_j}(\rho k u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_k}\right)\frac{\partial k}{\partial x_j}\right] + G_k - \rho\varepsilon \tag{3}$$

And the equation of dissipation of turbulent energy (ε) is:

$$\frac{\partial}{\partial_j}(\rho \varepsilon)\frac{\partial}{\partial x_j}(\rho \varepsilon u_i) = \frac{\partial}{\partial x_j}\left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon}\right)\frac{\partial \varepsilon}{\partial x_j}\right] + C_{1\varepsilon}\frac{\varepsilon}{k} - G_k - C_{2\varepsilon}\rho\frac{\varepsilon}{k} - R_\varepsilon \tag{4}$$

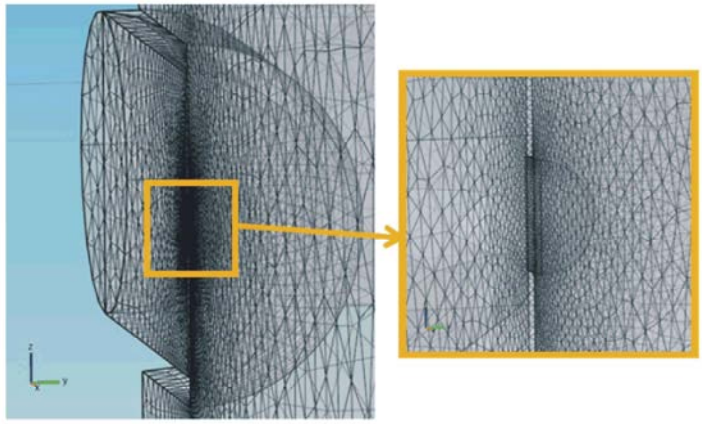

Fig. 4: Tetrahedral scheme and the mesh density.

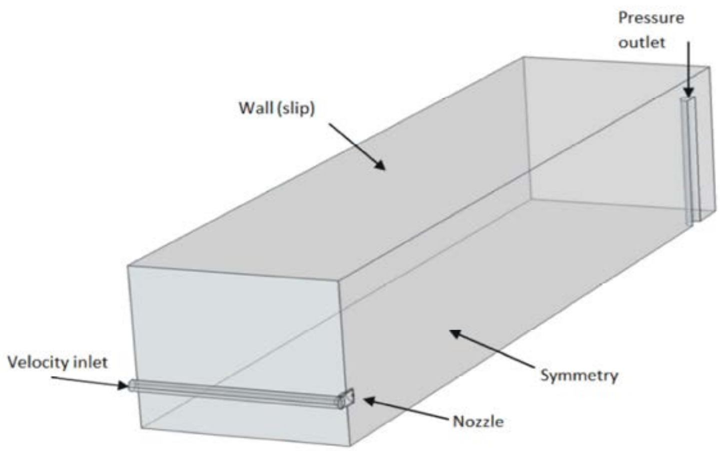

Fig. 5: Boundary conditions.

Boundary Conditions and Mesh Convergence: The near field domain has a tighter mesh compared to the rest of the cell, as this is where the kinetic gradient is much greater than the other regions. Several meshes were simulated and considered. The cell averaged diameters vary between 1mm to 8cm. The eventual mesh has between 5×10^6 to 3×10^6 cells (Figure 2). The boundary conditions for the model are as follows (Figure 3).

RESULTS

The velocity, turbulent intensity at nozzle outlet and the Grid cell size can play an important role in energy dissipation that should be considered in the modeling. The average time required for convergence was about 25hours. After this time, the relative equilibrium of velocity and pressure is established. Modeling for the internal angles of 90°, 45° and 30° and discharges 0.97, 0.78 and 0.55 lit/s were performed. Numerical results and experimental data are presented in the following figures. According to the following graphs, the accuracy decreases with increasing flow rate or increasing distance from the jet. However, due to reduction of turbulence intensity in nozzle outlet with decreasing internal angle of the jet, the accuracy increases.

Laboratory observations showed that the flow velocity near the nozzle has sharp changes but these changes in a certain distance suddenly decreases, this behavior can also be observed in the numerical model output. These observations also showed that the velocity near the nozzle is greatly dependent on the nozzle Froude number. With increasing distance from the nozzle and in a certain region, fluctuations in the Reynolds number increases, since this region, Froude number dependence of flow behavior is suddenly reduced. In other words, to show changes in velocity along the nozzle axis, two relations can be found that in one of them, the effect of Froude number is very high and in the other one, this effect can be neglected.

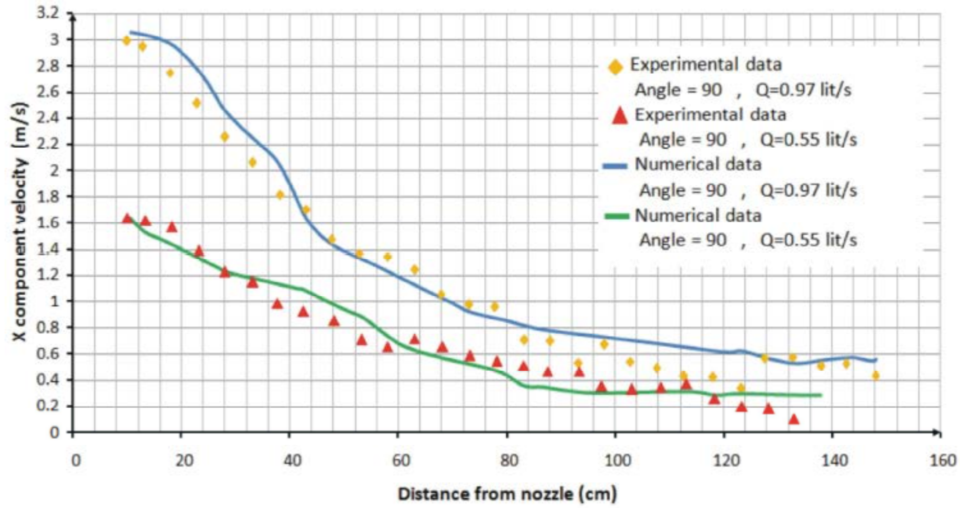

Fig. 6: Numerical results and experimental data (Internal angle= 90°).

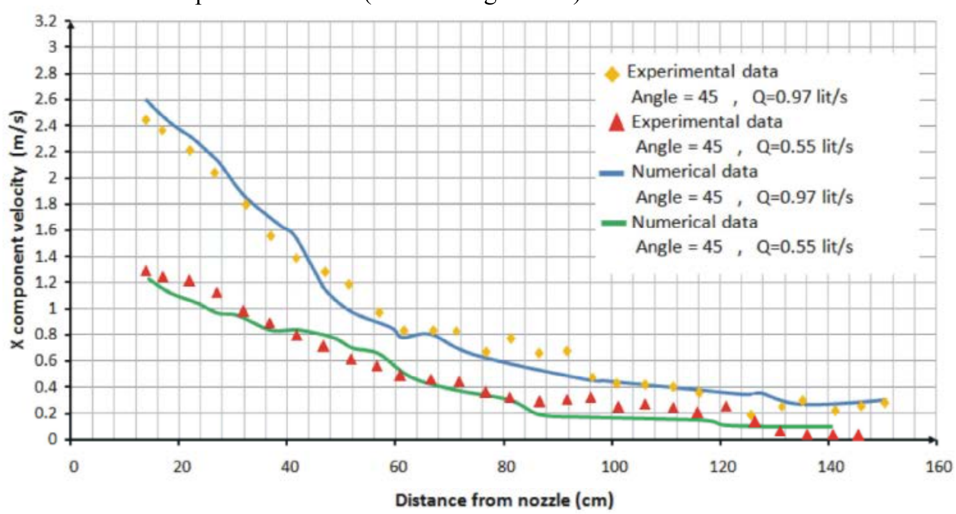

Fig. 7: Numerical results and experimental data (Internal angle= 45°).

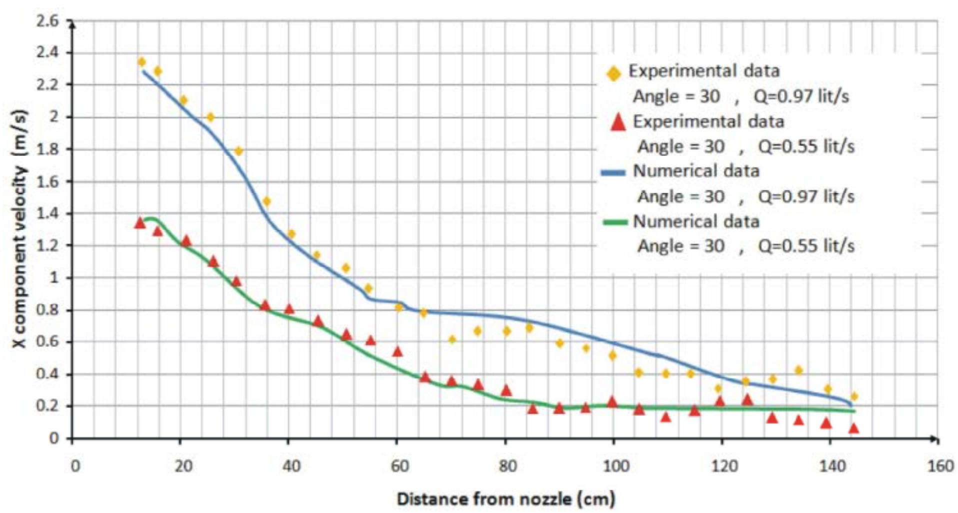

Fig. 8: Numerical results and experimental data (Internal angle= 30°).

CONCLUSION

In this paper, the behavior of the submerged jet has been studied. Comsol software was used for simulation and then the experimental and numerical results were compared for different conditions. Simulation results for the internal angles (90°,45°,30°) and discharges (0.97, 0.78, 0.55lit/s) were evaluated. This comparison showed that, the turbulent intensity at nozzle outlet can play an important role in calibration of model. The accuracy decreases with increasing flow rate, increasing distance from the jet or decreasing internal angle.

REFERENCES

1. White, F., 1991. Viscose Fluid Flow. McGraw-Hill, Inc.
2. Rajaratnam, N., 1965. Submerged hydraulic jump, J. Hydr. D i v. ASCE, 91: HY4.
3. Rajaratnam, N. and B.S. Pani, 1974. Three dimensional turbulent wall jets, J. Hydr. Div. ASCE, 100: HYl.
4. Wu, S. and N. Rajaratnam, 1995. "Free jumps, submerged jumps and wall jets". Journal of hydraulic research, ASCE, 33: 2.
5. Rajaramam, N., 1976. Turbulent Jets, Amsterdam: Elsevier Sc. Publ. Co.
6. Rajaratnam, N., 1977. "Erosion by circular turbulent wall-jets". J. Hydr. Res., 15(3): 277-289.
7. Hogg, A., H. Huppert and B. Dade, 1997. "Erosion by planar turbulent wall jets", J. Fluid Mech., 338: 317-340.
8. Peiqing, L., G. Jizhang and L. Yongemi, 1998. "Experimental investigation of submerged impinging jets in a plunge pool downstream of large dams", Science in China (Series E). Vol. 41, No.4.
9. Chen, J.Y., 2001. "Characteristics of check dam scour hole by free over-fall flow". Journal of the Chinese Institute of Engineers, 24(6): 673-680.
10. Karim, O.A. and K.H.M. Ali, 2000. "Prediction of flow patterns in local scour holes caused by turbulent water jets". J. Hydr. Res., 38: 4.
11. Aderibigbe, O., 1998. "Effect of sediment gradation on erosion by plane turbulent wall jets." J. Hyd. Eng. ASCE, 124(10): 1034-1042.
12. Mazurek, K.A., 2003. "Scour of a cohesive soil by submerged plane turbulent wall jets". Journal of Hydraulic Research, 41(2): 195-206.
13. Xue, W.Y., W. Huai and Z. Qian, 2010. "Numerical simulation of sediment erosion by submerged plane turbulent jets". 9thInternational conference on hydrodynamics.
14. Adduce, C. and P. Mele, 2004. "Local scour by submerged turbulent jets". Advances in hydro-science and -engineering, Vol VI.
15. Sequeiros, O., Y. Nino and M. Gacia, 2005. "Sedimentation Management in Combined Sewer Over flow Storage Reservoirs Using Water Jets". CIVIL ENGINEERING STUDIES Hydraulic Engineering Series, pp: 76.
16. Ahadiyan, J. and S.H. Musavi Jahromi, 2008. "Investigation of variation of efflux momentum in shallow receiving water by using FLOW-3D". Proceeding of the International Symposium of Water Resource Management, Tabriz, Iran, (ISWRMT'), pp: 551-557.
17. Ahadiyan, J. and S.H. Musavi Jahromi, 2009. "Effect of jet hydraulic properties on geometry of trajectory in circular buoyant jet in the static ambient flow". Journal of Applied Sciences, 9(21): 3843-3849.
18. Jafarinia, R., S.H. Musavi Jahromi and M. Shafai Bajestan, 2010. "Prediction of scour dimensions downstream of siphon spillways". World Applied Science Journal, 9(7).
19. Soleimani Babarsad, M. Musavi Jahromi, S.H. Kashkuli, H. Samani, H. Sedghi, M. (2012). "Study of inner Convergence Angle on Maximum velocity and effective length in Submersible jet".Journal of Archives Des Sciences, Vol. 65, No. 7; Jul 2012.
20. Abdulnaser, S., 2009. "Computatinal fluid dynamics". Copyright © 2009 Abdulnaser sayma & ventus publishing ApS.
21. Pryor, R., 2011. "Multiphysics modeling using Comsol (A first principles approach)". Copyright © 2011 Jones and Bartlett publishers, LLC.
22. Yang, C.F., B.S. Wu and H.C. Shen, 2006. "The Analysis of Experiment Validation for Full-Ship Maneuverability Hydrodynamic Forces Prediction". Journal of Ship Mechanics.