

PAPER • OPEN ACCESS

Numerical simulation on the characteristics of thermal buoyant jets with the different flow profiles in crossflow

To cite this article: Y X Liu *et al* 2018 *IOP Conf. Ser.: Earth Environ. Sci.* **191** 012062

View the [article online](#) for updates and enhancements.

You may also like

- [An optically accessible secondary combustion zone for the transverse injection of reacting jets into a high-speed, vitiated crossflow within a staged, gas turbine model combustor](#)
N S Rodrigues, C T McDonald, O O Busari et al.
- [Optimal perturbations in viscous channel flow with crossflow](#)
Cheng Chen, Jiang-Tao Huang and Wei-Guo Zhang
- [Improved nonlinear parabolized stability equations approach for hypersonic boundary layers](#)
Shaoxian Ma, , Yi Duan et al.



ECS
The
Electrochemical
Society
Advancing solid state &
electrochemical science & technology

DISCOVER
how sustainability
intersects with
electrochemistry & solid
state science research

Numerical simulation on the characteristics of thermal buoyant jets with the different flow profiles in crossflow

Y X Liu¹, Q L Zhang^{1,2}, J L Sun¹, Y Wang¹, Z D Yang¹ and C Wang¹

¹State Key Laboratory of Eco-hydraulics in Northwest Arid Region of China, Xi'an University of Technology, Shanxi Xi'an 710048, China

E-mail: zqling0712@163.com

Abstract. In this paper, the numerical simulation on the characteristics of thermal buoyant jets of the cross-flow equal area of the round hole and square hole thermal jets is carried out, which based on the SIMPLE algorithm and Fluent's RSM model. According to the calculation results, the variation characteristics of velocity and temperature of single-hole jets with the different flow profiles in crossflow are analyzed, and the decay laws of velocity and temperature in heat jet are obtained. The results show that the closer to the jet exit, the more rapidly the speed change, and there is a clear jet mainstream area, but the speed fluctuations gradually weaken with the flow to the downstream. The dimensionless temperature difference of the trajectory of thermal buoyant jets decreases gradually along the direction of incoming flow, and the trajectory centre of the thermal buoyant jets moves away from the outlet. Compared with the square hole, the round hole jet impacts on the incoming flow strongly, causes more intense mixing with the surrounding fluid, and increases velocity and temperature attenuate at faster rate.

1. Introduction

After the thermal drainage discharged by the fire and the nuclear power plant into the environment water body, it may mix and disperse with the receiving water which forms a hot water thermal pollution area. Therefore, more attention has been paid by thermal pollution researchers to the study on the process and distribution of thermal drainage in the receiving waters. Study of flow characteristics of turbulent floating jets has important theoretical and practical value, and hence become one of the major subject of current research.

With the rapid development of turbulence theory, numerical calculation and computer technology, great progress has been made in the research of buoyancy jet characteristics by the continuous efforts of many scholars. Zhang *et al* [1,2] carried out numerical simulation and experimental research on the formation of thermal wake by cooling water discharged from underwater vehicles in temperature stratified environment. Then the rising law and temperature distribution of the thermal wake in linear temperature stratified seawater were obtained. Using the laser-induced fluorescence technique, Huang [3], Smith and Munga [4] studied the single-hole and multi-hole jets discharged from the bottom of the flume. The results showed the bifurcation phenomenon and the horseshoe vortex structure. Huai Wenxin *et al* [5] analyzed the turbulent jet characteristics of a single hole in the crossflow and found the jet bifurcation phenomenon. Larsen *et al* [6] studied the characteristics of the floating jet in the cross flow and compared the k- ϵ model with the integral model. El-Amin *et al* [7] studied the characteristics of temperature field, velocity field and turbulence intensity in a jet of fluid with a linear



change of jet fluid temperature over a time period. Zhang *et al* [8] used the three-dimensional turbulent model and its hybrid finite analytical solution method to simulate the characteristics of helium jets in a cross-flow environment. Chen *et al* [9] utilized the Realizable k- ε turbulence model of Fluent and SIMPLE Consistent method to investigate the effect of spacing of nozzles and initial velocity of thermal buoyant jet on flow and temperature fields.

In spite of the development of jet research towards diversification, there are relatively few studies on the influence of different cross-sectional shapes on the dilution characteristics of thermal jet flow field. Therefore, the authors used the RSM model in Fluent software to solve the three-dimensional Navier-Stokes equation considering buoyancy. The characteristics of the flow and the temperature distribution of the single-hole jet and the square-hole jet under the same working conditions in the environment are provided, which mainly focuses on the influence of different outlet shapes on crossflow, in order to offer a theoretical basis for the design of the sewage discharge diffuser.

2. Mathematical model and calculation method

The jet in the flowing environment is a complex turbulent flow with a complicated phenomenon of random pulsating flow. The authors assume that the thermal buoyant jets with constant and sufficient development are discharged into an infinite water body, which is an incompressible fluid, and the density of the water body changes with temperature. According to the basic governing equations of fluid motion and the Reynolds stress model, the three-dimensional mathematical model of thermal jet in flowing environment is obtained as follows:

$$\text{Continuity equation: } \frac{\partial(\rho u_i)}{\partial x_i} = 0 \quad (1)$$

$$\text{Momentum equation: } \frac{\partial(\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[\gamma \frac{\partial u_i}{\partial x_j} - \rho \overline{u'_i u'_j} \right] + \rho g_i \quad (2)$$

$$\text{Temperature equation: } c_p \frac{\partial(\rho u_j T)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\lambda \frac{\partial T}{\partial x_j} - c_p \rho \overline{u'_i T'} \right] \quad (3)$$

$$\text{k equation: } \frac{\partial(\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \frac{\partial k}{\partial x_j} \right] + \mu_t \overline{u'_i u'_j} \frac{\partial u_i}{\partial x_j} - \rho \varepsilon \quad (4)$$

$$\varepsilon \text{ equation: } \frac{\partial(\rho u_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j} \left[\left(\mu + \frac{\mu_t}{\sigma_\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + \rho \left(c_{1\varepsilon} \frac{\pi}{\varepsilon} - c_{2\varepsilon} \right) \frac{\varepsilon^2}{k} \quad (5)$$

Reynolds stress transport equation:

$$\begin{aligned} \frac{\partial(\rho u_k \overline{u'_i u'_j})}{\partial x_k} = & \frac{\partial}{\partial x_k} \left[\frac{\mu_t}{\sigma_k} \frac{\partial \overline{u'_i u'_j}}{\partial x_k} + \mu \frac{\partial \overline{u'_i u'_j}}{\partial x_k} \right] - \rho \left[\overline{u'_i u'_k} \frac{\partial u_j}{\partial x_k} + \overline{u'_j u'_k} \frac{\partial u_i}{\partial x_k} \right] - C_1 \rho \frac{\varepsilon}{k} \left[\overline{u'_i u'_j} - \frac{2}{3} k \delta_{ij} \right] - \\ & C_2 \left[P_{ij} - \frac{2}{3} P_k \delta_{ij} \right] - C_3 \left[-\rho \beta (g_i \overline{u'_j T'} + g_j \overline{u'_i T'}) - \frac{2}{3} \delta_{ij} \right] \left(-\rho \overline{u'_i u'_k} \frac{\partial u_j}{\partial x_k} \right) - \frac{2}{3} \rho \varepsilon \delta_{ij} \end{aligned} \quad (6)$$

Where, ρ is the density of the fluid, kg/m³; u_i ($i=1,2,3$) is the velocity component of x, y, z, m/s; p is static pressure, Pa; μ is the viscosity coefficient, N·s/m²; T is the temperature of the fluid, K; g_i is the gravity acceleration component, m/s²; c_p is the constant pressure specific heat, J/(kg·K); k is turbulent kinetic energy; ε is the turbulent energy dissipation rate; μ_t is the turbulent viscosity coefficient; β is

the thermal expansion coefficient.

Fluent software is often used to calculate the fluid flow and heat transfer problems. In this paper, the governing equations are discretized by the finite volume method. The second order upwind scheme is adopted. The SIMPLE algorithm is used to solve the Navier-Stokes equations considering gravity effects. The energy equation converges to 10^{-6} , and the rest of the equations have an accuracy of 10^{-4} . In fluent software, the Prandtl number default setting for the RSM model is 0.85 as a default constant and C_{1e} , C_{2e} , σ_k , σ_ϵ are 1.44, 1.92, 1.0, 1.3 respectively.

3. Meshing and boundary conditions

Research object is shown in figure 1, and the temperature of the thermal jet is higher than the ambient water. In figure 1, x is the direction of flow of ambient water, y is the width direction of the water, and z is the depth direction of the water. The diameter of the jet outlet is D , the water depth is H , the ambient water temperature is T_0 , the ambient velocity is u_0 , the jet velocity is u_j along positive z -axis direction, and outlet temperature is T_j . The u_j is the characteristic velocity, the jet flow exit diameter D is the characteristic length, and the ambient temperature T_0 is the characteristic temperature.

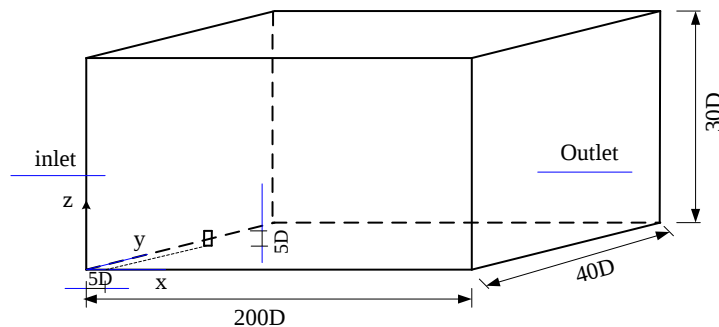


Figure 1. Schematic diagram of three-dimensional heat flux in mobile environment.

The calculation area is a cuboid area with a length of 2000 mm, along the x direction, a width of 400 mm, defined in y direction and a depth of 300 mm, defined along the z direction. The shapes of the jet outlets are round and square, and the outflow cross-sectional area is equal respectively, all of which are round areas with a diameter of 10 mm. The jet outlet center coordinate is (50,200,0). The direction of the outflow of heat jet is along the positive z -axis, and the direction of the ambient jet is along the positive x -axis. The calculation area is divided by Gambit with the tetrahedron mesh. The calculation grids for round and square orifice are about 0.7 and 1.3 million respectively and these meshes are then further used to solve in fluent.

In the present paper, u_0 , v_0 , w_0 are the velocity component of the incoming flow at the inlet of the region; T_0 is the temperature of the incoming flow at the boundary of the inlet of the region; u , v , w are the flow velocity component at any point in the flow field of the calculation area and T is the temperature of the fluid in the calculation area. R is defined as the ratio of the velocity of ambient and jet exit as u_j/u_0 , which is an important physical quantity of the jet.

The boundary conditions for mathematical models are as follows: the environmental inflow boundary adopts speed entrance, as $u=u_0$, $v=w=0$, and the temperature of the environmental fluid is $T=T_0$; the speed entrance of jet inflow is $u=v=0$, $w=u_j$, and the temperature of the jet fluid is $T=T_j$; the outlet boundary is to set free flow, and because the calculation area is large enough, this outlet boundary condition has little effect on the flow near the jet outlet; the boundary of non-sliding wall surfaces is used, and near-wall areas are treated with the standard wall surface function method; the free liquid surface uses the rigid cover assumption.

4. Grid independence

During the simulation calculation process, the total number of grids in the model calculation area and

the number of grids at the inlet of the thermal jet have a certain influence on the accuracy of the calculation results. The larger the number of grids is, the denser the grid is and the higher the calculation accuracy is. However, the characteristics of the flow field under different grid numbers are basically the same, and the final outflow is not very sensitive to the number of grids. The larger the number of grid is, the longer the calculation time is. Therefore, the speed and efficiency of the calculation and the effect on the calculation result should be comprehensively considered during the calculation process, which is shown in table 1.

Table 1. Grid independence.

Number of grids (million)	Jet inlet mesh size (mm)	Jet velocity (kg/s)	Total outflow (kg/s)	Relative error (%)
20.5	0.2	0.042095	11.948496	0.000137591
39.3	0.25	0.0437400	11.95014	0
67	0.3	0.0437400	11.95014	0.000122425
136.8	0.45	0.0452024	11.951603	

By mesh-independent analysis, when the total grid reaches 200,000, the round hole jet flow and the total outflow flow reach a stable value; when the total grid reaches 670,000, the change has been close to the minimum. Considering comprehensively, this paper adopts 670,000 grid division.

5. Results and discussion

The velocity u_0 at the calculation area is 0.1m/s and the flow temperature T_0 is 293.15 K. The outlet velocity of the thermal jet u_j is 0.6 m/s with temperature 313.15 K, and the flow velocity ratio R is 6. The dimensionless temperature θ is defined as $(T-T_0)/(T_j-T_0)$. The above conditions are studied numerically and the simulation results are discussed.

5.1. Flow field characteristics analysis

Figure 2 to figure 4 shows the dimensionless velocity distribution at the different distances along the x-axis and the z-axis with the different outlet shapes at the center of the thermal jet outlet symmetry.

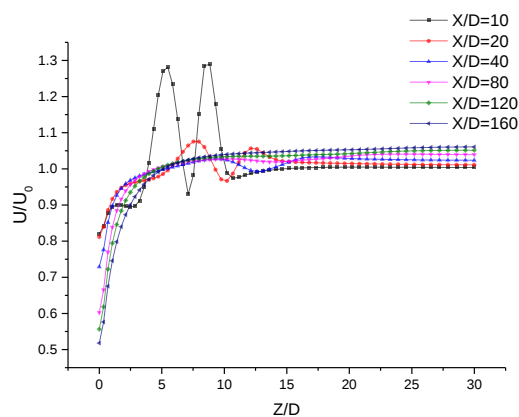


Figure 2. Round orifice at different distances along the z-axis velocity distribution.

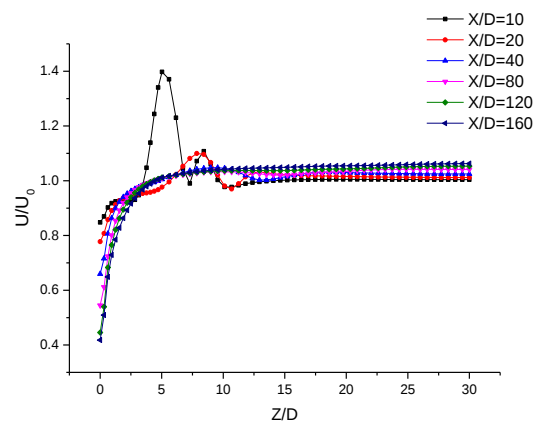


Figure 3. Square orifice at different distances along the z axis velocity distribution.

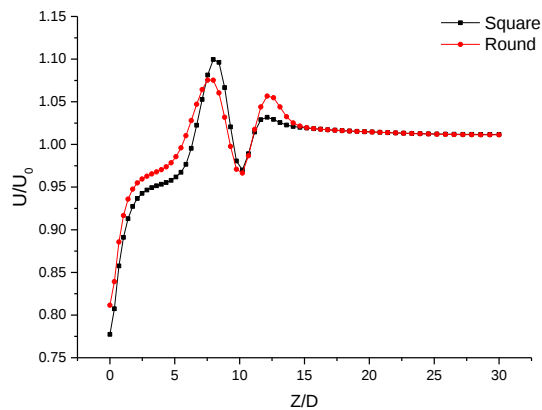


Figure 4. Distribution of different outflow sections along the z-axis.

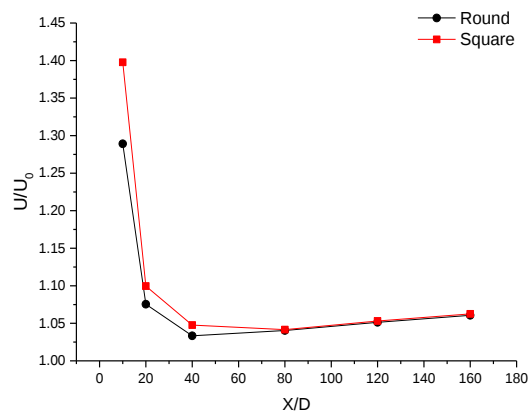


Figure 5. Jet trajectory center velocity distribution.

From figures 2 and 3, it can be seen that the closer from the jet exit in the x direction is, the more rapidly the dimensionless velocity variation on the vertical direction is, and there is an obvious jet mainstream area. The fluctuation of the dimensionless velocity decreases gradually along with the flow moving downstream. The troughs of the dimensionless velocity of the jet trajectory gradually move toward the positive z axis with the incoming flow develops, and decrease gradually. The width of the trajectory of the thermal jet is about 10 times the diameter of the jet outlet. Figure 4 shows the distribution of the round hole jet and the square hole jet along the z-axis at $X/D = 20$. It can be seen that the trough values are basically the same, but the peak values are varied considerably due to the generation of vertical bifurcation by the jet flow. Therefore, the round orifice jet flow velocity is smaller than that of the square jet at $Z/D = 8$. However, the round orifice jet flow velocity is higher than the square orifice flow velocity at $Z/D = 12$.

Figure 5 shows the dimensionless velocity distributions at the center of the trajectory of the thermal jet under different outflow profiles when the jet T_j is 313.15 K. The distribution of outflow velocity at the round orifice is obviously lower than the square orifice along the distance of $X/D < 80$. As the incoming flow developing downstream, the center velocity of the jet trajectory gradually converges with the incoming velocity. In the area closer to the exit of the jet, the jet of round orifice impacts more strongly on the incoming flow than the exit of the square, so the velocity field changed more drastically.

5.2. Temperature field characteristics analysis

Figures 6 and 7 show that the distribution of the dimensionless temperature difference along the z-axis at different distances of x direction on the horizontal center of the outlet, when the flow rate ratio R is 6, and the heat jet outlet temperature is 313.15 K. It can be seen that with the development of the jet along the direction, the maximum dimensionless temperature difference gradually decreases and the temperature of the jet trajectory gradually converges with the incoming temperature. The center of the jet locus moves away from the jet outlet in the positive direction of the z-axis. The width of the dimensionless temperature difference along the z-axis gradually increases. The dimensionless temperature difference of the trajectory of the thermal jet $\Theta > 0.005$ is about 10 times the jet outlet diameter at $X/D = 20$ and it is about 25 times at $X/D = 160$.

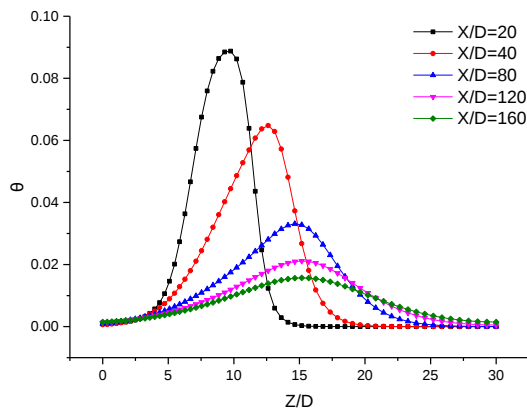


Figure 6. Distribution of temperature along the z-axis at different distances of a round jet.

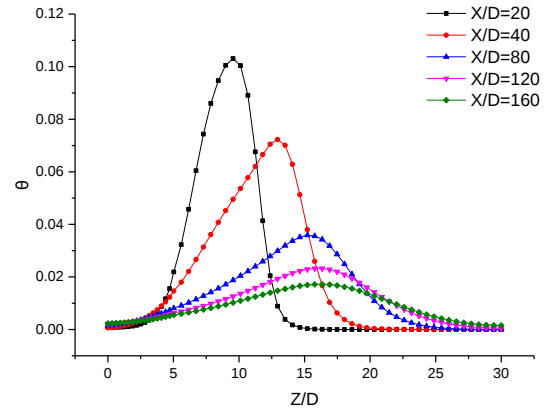


Figure 7. Distribution of temperature along the z-axis at different distances of a square jet.

Figure 8 shows the distribution of dimensionless temperature difference Θ along the z-axis of different outflow sections at the center of the jet outlet. Compared with the square jet, the maximum dimensionless temperature difference of the jet trajectory of the round jet is smaller, and the trajectory width of the thermal jet is larger. This is because the round jet impacted the environment more strongly, and the perturbation is aggravated. The heat transfer capacity between the thermal fluid and the surrounding fluid is enhanced, the thermal diffusion is accelerated, and the maximum Θ of the jet trajectory is reduced.

Figure 9 is the maximum dimensionless temperature difference of the thermal jet trajectory along the x-axis distribution curve. It can be seen that the maximum dimensionless temperature difference of the outflow of the round hole is 0.08875 and the square hole is 0.10307 at $X/D = 20$. The maximum dimensionless temperature difference is 0.0157 for the round hole outflow, and it is 0.01721 at $X/D = 160$ for the square hole outflow. The maximum dimensionless temperature difference of the trajectories along the x direction decreases obviously. Before $X/D=120$, the temperature decreases rapidly, then the temperature decreases slowly. It is due to the fact that the impact of the jet velocity along the x-direction on the fluid area is gradually weakened and the thermal diffusion is gradually slowed down. In contrast, the round orifice jets attenuate faster than the square orifice.

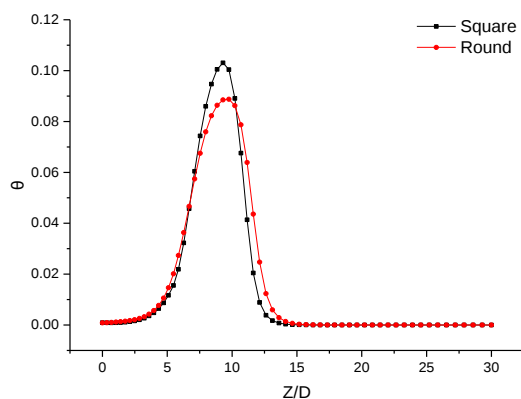


Figure 8. Distribution of Θ in different outflow sections along the z-axis.

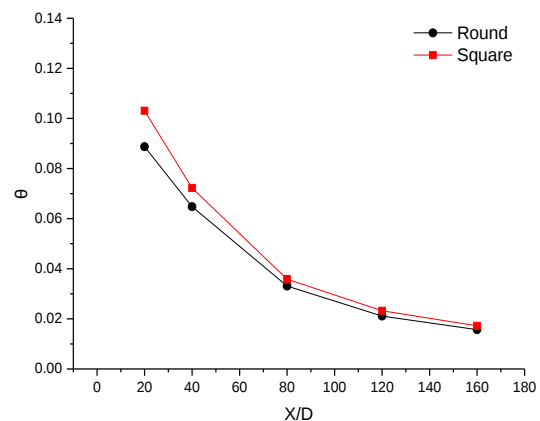


Figure 9. The distribution of Θ at the center of jet trajectory.

6. Conclusion

The characteristics of the round hole jets and the square hole jets are compared and analyzed through

the numerical calculation. The diffusion law and the temperature distribution of the three-dimensional jets are obtained as follows:

- The closer the distance from the jet exit is, the more severely the dimensionless speed change is, and there is an obvious jet mainstream area. As the flow moves downstream, the fluctuation of the dimensionless speed gradually weakens, whose trough gradually shifts to the positive direction of the axis with the development of incoming flow, and gradually reduces. Along the flow direction distance $X/D < 80$, the velocity distribution of the circular orifice is significantly lower than that of the square orifice.
- At the same velocity and temperature of the exit of the thermal jet, the maximum dimensionless temperature difference of the trajectory along the x direction decreases gradually. With the increase of X/D , the center of the trajectory of the heat jet gradually moves away from the jet outlet in the positive direction of the z -axis, and the dimensionless temperature difference of the trajectory of the heat jet gradually increases along the z -axis.
- As compared with the square hole, the maximum dimensionless velocity and the maximum dimensionless temperature difference for the round hole are smaller in the same section. The results show that the impact of the round jet on the incoming flow is stronger, resulting in more intense mixing of the thermal jet and the surrounding fluid, faster thermal diffusion, faster jet velocity and temperature dominance in the initial jet phase and correspondingly faster attenuation.

Acknowledgments

The research was supported by the Major Basic Research Projects in Natural Science of Shaanxi Province (2017ZDJC-20), the Natural Science Foundation of Shaanxi Province (2015JQ5181) and the Education scientific research project for the Education Department of Shaanxi Province (14JK1534).

References

- [1] Zhang J, Chen X, Yang L, Yuan J T and Zhang X H 2009 Study of temperature characteristic of cooling water discharged by underwater vehicle *Journal of Ship Mechanics* **13** 533-9
- [2] Zhang J, Yang L, Yuan J T, Chen X, Liu H K and Zhang X H 2008 Experimental research of underwater vehicle thermal wakes *Journal of Experiments in Fluid Mechanics* **2008** 7-13
- [3] Huang Z L, Li Y L and Yu C Z 1994 Study on the measurement of jet concentration field in cross-flow by planar laser-induced fluorescence *J HYDRAUL ENG-ASCE* **1994** 1-7
- [4] Smitthm M 1998 Mixing in the jet in crossflow *J FLUID MECH* **357** 83-122
- [5] Huai W X, Li W and Peng W Q 1998 Calculation and characteristic analysis of turbulent jet with single circular hole in cross flow *J HYDRAUL ENG-ASCE* **1998** 8-15
- [6] Larsen T, Petersen O and Chen H B 2010 Numerical experiment on turbulent buoyant jets in flowing ambients T Larsen, O Petersen and C Haibo *Numerical Experiments on Turbulent Buoyant Jets in Flowing Ambients Direct and Large-Eddy Simulation VII* (Netherlands: Springer) **13** 339-46
- [7] El-Amin M F, Abdulmajeed A G, Amgad S and Sun S Y 2015 Numerical simulation and analysis of confined turbulent buoyant jet with variable source *J HYDRODYN* **27** 955-68
- [8] Zhang X Y, Li W and Li C C 2002 Numerical simulation for the flow field of vertical round turbulent jet entering the cross current *J HYDRAUL ENG-ASCE* **33** 32-8
- [9] Chen J, Wang Y and Sun J L 2017 Study on numerical simulation of two-nozzle thermal buoyant jet in cross-flow environment *Journal of Water Resources & Water Engineering* **28** 168-73