

Experimental study on temperature spread of multi-angle inclined buoyant jet

S M Zhao¹, Z D Yang^{1,2}, Q L Zhang¹, J L Sun¹ and Y Wang¹

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

E-mail: yangzhendong@xaut.edu.cn

Abstract. The thermal discharge from power plants adversely affects its surrounding ecological environment, so it is of practical significance to study the diffusion characteristics of the thermal discharge as the global ecological environment problem becomes more and more serious. The change of the temperature and velocity of the buoyant jet with different inclination in the co-current environment were studied. The main conclusions are as follows: the jet velocity and temperature decay rapidly near the orifice, and the region with the largest change of velocity is also the region with the greatest dilution of temperature. The influence range is mainly within 80 times diameter of the nozzle. The smaller the jet angle is, the slower the change of velocity and temperature decay.

1. Introduction

After the thermal discharge from power plants into the water environment, it may mix and disperse in the receiving water which forms thermal pollution. Therefore, the study on the process and distribution of thermal water drainage in the receiving waters has been taken seriously by thermal pollution researchers [1]. It is of great theoretical and practical significance to study the flow field characteristics of turbulent buoyant jets [2,3], which has become one of the hot spots of current research.

Since the 1920s, many scholars have carried out a great deal of research work on the buoyant jet and have made great progress in the theory of the buoyant jet [4,5]. Zhang *et al* [6,7] discussed the research progress of numerical simulation of warm water drainage in recent years, and gave the main bibliography. Edmund A P [8] carried out co-current buoyancy jet test in the open channel and it was found that the width of the lateral expansion between rough bottom hot water and smooth hot water is of difference. Naudas-cher E and Zimmermann C [9] carried out a model test for the thermal discharge of a nuclear power plant on the Rhine. Due to the uncorrected roughness, it was found that the stratification in the vicinity of temperature diffusion and diffusion range did not correspond to the actual situation; On the smooth bed surface, the lateral expansion induced by the density lead to the rapid broadening of the thermal jets and the turbulence that occurred on the rough bed prevents the development of the delamination. The reason for the above-mentioned phenomenon can be attributed to the role of secondary flow of roughness bed surface. Huai W X [10] used the Reynolds stress model to simulate the secondary flow and the transference of contaminants in the double open channel. Huai W X [11] and others main studied the nozzle Froude number on the flow characteristics; on this basis, the numerical verification of the buoyancy jet shore occurred critical adsorption conditions. After a long period of research [12-14], a series of achievements under different floating conditions had been



studied.

The current study of thermal buoyant jet mainly focuses on the co-directional flow and vertical orifice discharge methods. However, few work has been done on the influence of the inclination angle on the thermal drainage admixture in the previous literature [15,16]. Therefore, the temperature and velocity attenuation characteristics were investigated with the different inclination angles in this paper, the dilution characteristics of the thermal pollution with different inclination angles of the orifice in the co-current ambient fluid were experimentally studied, and the thermal diffusion characteristics were obtained. The temperature of the thermal drainage in the experiment was close to the drainage temperature of the power plant, which was a typical positively buoyant jet. A straight channel with rectangular section was used in this paper. Although the cross-section shape and bed roughness are still very different from the real river channel, the available experimental data are provided for the simple empirical and integral models. The experiment gives an initial estimation of the discharge conditions in a short time and it is useful for the design purpose of discharge structures.

2. Experimental system

The mixing characteristics between the thermal discharge and the ambient water and the distribution of flow field and temperature field near the drainage port in the co-current orifice with different inclination are studied. The temperature of thermal discharge was 40°C, the temperature of ambient fluids was 20°C, the angles between ambient water flow direction and thermal discharge direction of the orifice were 10°, 20°, 30° respectively, the jet velocity of orifice flow was 1.2 m/s, and the velocity ratio R was 6.

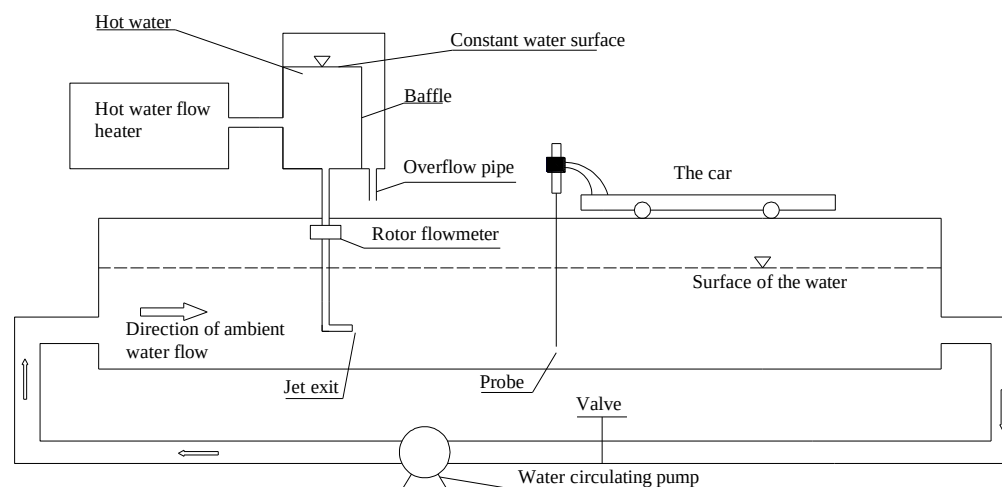


Figure 1. Test system diagram.

The experiments were performed in the test hall of Institute of Environmental Science, Xi'an University of Technology. Test system diagram is shown in figure 1. The test devices mainly include mobile devices, hot water tank device, flow rate measurement system and temperature measurement system.

The experiment was carried out in an poly methyl methacrylate (PMMA) sink, with the length of 30 m, the width of 0.8 m, and the depth of 0.5 m. The water flowed in a closed loop system. The sides of the sink were plexiglass walls for viewing and taking pictures. The bottom of the sink was a straight steel plate with white paint for visual observation of the jet flow. The left of sink was connected to the circulating pump, and the velocity of ambient water flow in the sink can be adjusted through the adjustment of the pump velocity. There was a water tank with hot water, with the size of 1.0 m*0.8 m*0.25 m. The hot water tank was above the sink. Hot water was supplied from the electric water heater

to the water tank through the hot water pipe. The bottom of the hot water tank was connected with a tube to provide jet power. The hot water was driven by gravity energy difference between the water tank and the environmental water body. There was a baffle inside the water tank. The excess water in the water tank was drained to the outside through the overflow pipe at the top of tank to ensure the water level inside the water tank. The baffle was to keep gravity energy difference constant, so as to ensure the stable outflow of the jet orifice. The jet pipe was submerged in the water, with a distance of 5 cm from the the bottom of the tank. The jet pipe was connected with the orifice through a rotor flow meter, which could record the jet orifice flow velocity accurately and adjust the velocity from the rotor flow meter finely.

Experimental section diagram was shown in figure 2. The coordinate origin was in the center of the jet orifice, the X-axis was horizontally arranged in line with the direction of the incoming flow, the Y-axis was perpendicular to the sink center line, and the Z-axis is perpendicular to the XY- plane with the direction upwards. The temperatures of fluid were gauged by 0.2-mm diameter standard thermocouples. The experimental data which were collected by data acquisition system were saved in a personal computer. All the experimental apparatus were calibrated before measurement. The max estimated uncertainties in the experiment were summarized in table 1. Experimental section structure is shown in figure 3.

Table 1. Uncertainties in measured and calculated parameters.

Parameter	Absolutely Uncertainty	Minimal measured Value	Maximal uncertainty
Ambient mass flow rate(kg/s)			1.0%
Jet flow rate (l/s)			1.0%
Fluid temperature (°C)	0.4	20	2.0%

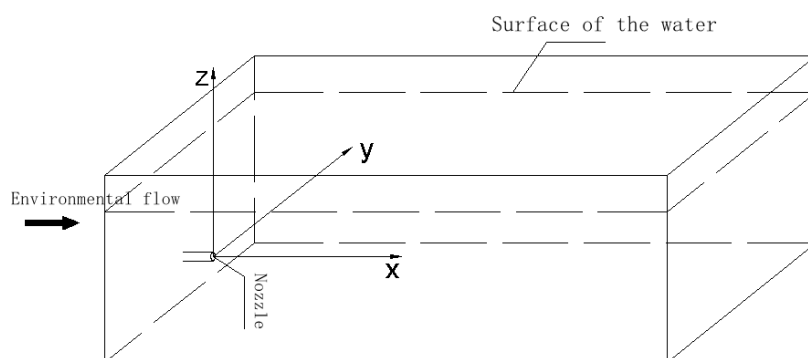


Figure 2. The test coordinate system.

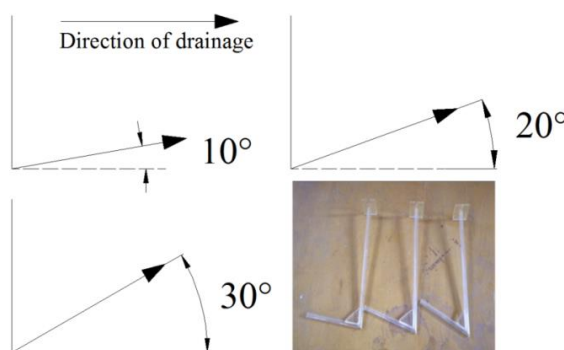


Figure 3. Test section physical model.

3. Results and discussions

3.1. The influences of jet angle on the distribution of velocity and temperature

The vertical distributions of flow velocity and temperature at different outflow angles at $X/D = 10$ and $X/D = 20$ cross sections are shown in figure 4. X/D is the ratio of the coordinate x of a point on the central axis to the diameter D of the jet orifice.

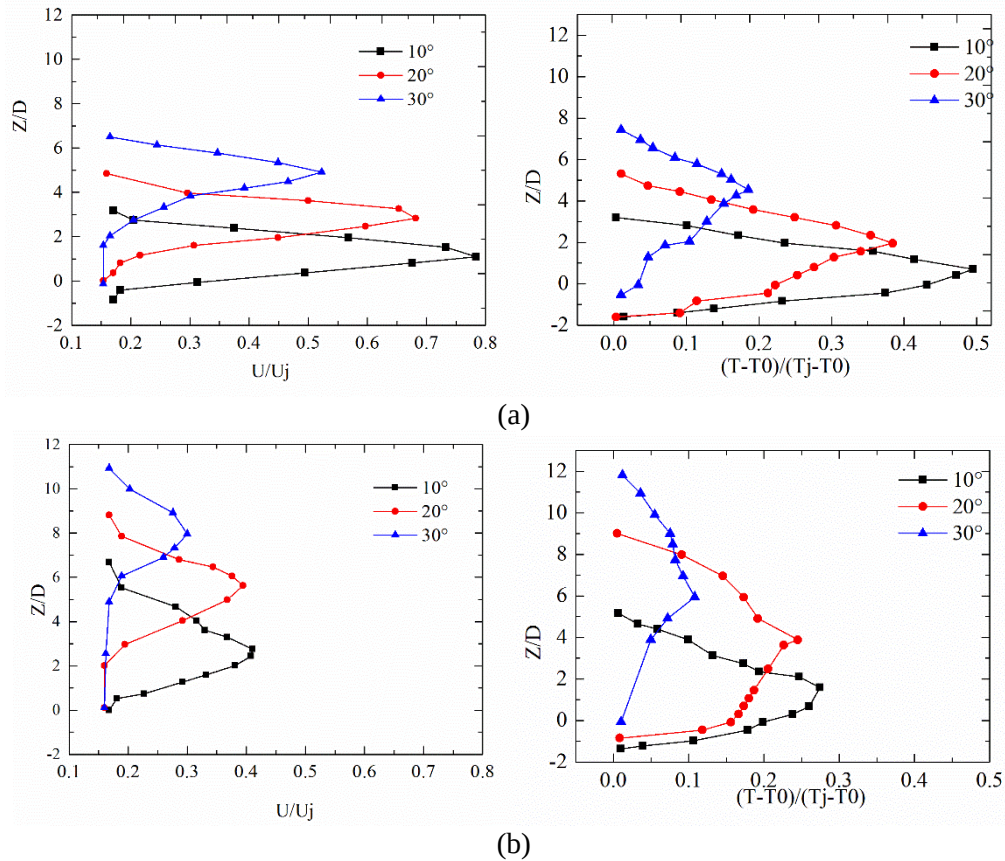


Figure 4. Effect of different outflow angles on cross-section velocity and temperature distribution. (a) Vertical velocity distribution and temperature at different outflow angles for $X / D = 10$ and (b) Vertical velocity distribution and temperature at different outflow angles for $X / D = 20$.

The Y-axis in the temperature change figure is the temperature dilution. The definition of temperature dilution is expressed as:

$$Rmm = \frac{\Delta T_m}{\Delta T_E} = \frac{T - T_0}{T_j - T_0} \quad (1)$$

Where, T is the temperature of the jet centerline at any point of the temperature, T_0 is the ambient temperature, T_j is the temperature of the jet outlet. The dimensionless velocity is expressed as the ratio of the velocity U at any point on the jet centerline to the velocity U_j at the jet orifice.

From the figure 4, the distribution trend of the two sections is similar, the temperature and the velocity are decaying faster and faster with the increase of the jet angle, and the larger the angle is, the larger the influence range in the longitudinal direction is, the more obviously the maximum cross-section temperature and velocity is. This shows that the larger the outflow angle is, the more fully the pollutants diffuse, and the affected area is larger, the buoyant jet easier to reach the water

surface.

The vertical velocity of 30° jet is 2.89 times bigger than that of 10° at jet opening. It can be found that vertical coordinates of maximum velocity of 30° is greater than that of 10° or 20° at the same section. The vertical coordinates of maximum temperature have a similar characteristics. Due to the impact of ambient water flow, the jet core couldn't linearly increase. Additionally, the vertical coordinates value of maximum temperature is greater than that of maximum velocity with the same section and jet angle.

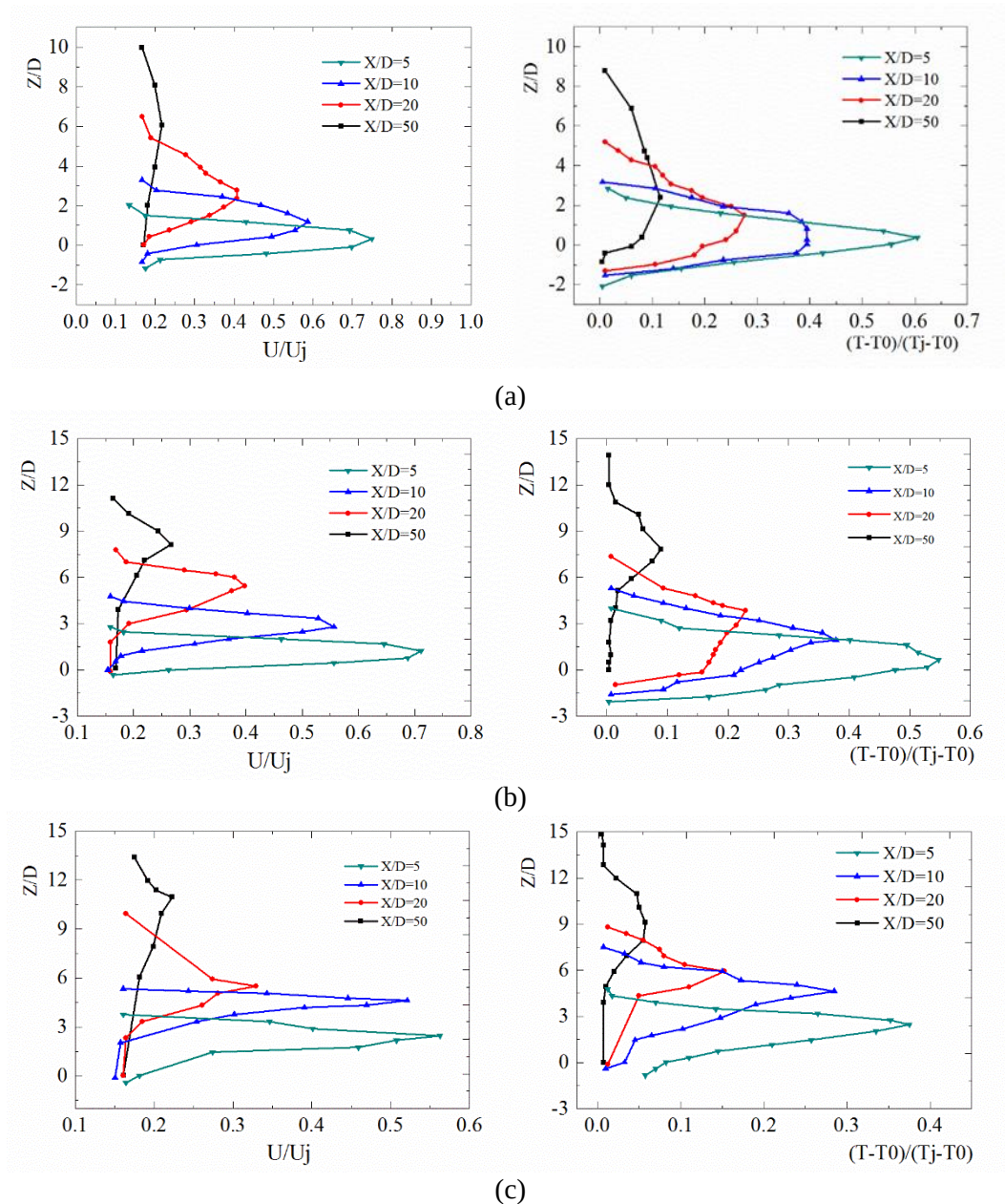


Figure 5. Temperature and velocity attenuation along azimuth at each inclination angle, (a) the variety of velocity and temperature with the jet of 10°, (b) the variety of velocity and temperature with the jet of 20° and (c) the variety of velocity and temperature with the jet of 30°.

3.2. The distribution of velocity and temperature along the Z direction on each characteristic cross section with the same outflow angle

The distributions of velocity along the Z direction at different positions on the axis of X at different angles are shown in figure 5. In the case of angular outflow, the jet lifts up obviously and the contour of the section velocity becomes a “sharp angle” shape, so the maximum of the velocity of each section is not on the horizontal axis. The initial momentum of the jet near the orifice is a main factor. However, outside the range of 20 times diameter, it can be seen that due to the decrease of the initial momentum of the jet, the mainstream area gradually disappears and the flow velocity profile become more uniform until the buoyant jet well mixed with the ambient fluid. When the jet angle is 10° or 20°, velocity distribution at the section $x/d=5, 10, 20, 50$ is approximately symmetrical. The velocity of angle 10° is greater than that of 20°, and the velocity of 20° is greater than that of 30° at the section $x/d=5$. At $x/d=50$, the discrepancy of velocity with the jet 10°, 20°, and 30° becomes smaller. It can be seen that the penetration distance of horizontal direction is not increased, even if the horizontal velocity of 10° is bigger than that of 30°. At $x/d=50$, the extension width of jet 10° in the vertical direction is significantly bigger than that of jet 20° and 30°, and the temperature at the center of the jet tail is higher than that of 20° and 30°. According to the temperature curves with the different jet angles, temperature spread is more unfavorable at 10°.

3.3. Comparison of flow field characteristics with different outflow angle

The jet trajectory is an important parameter to describe the jet, and the Macro characteristics of jet movement can be understood by studying the jet trajectory.

The trajectories of jet with different outflow angles are shown in figure 6. The jet trajectories with jet angle of 10°, 20° and 30° are more obviously where the distance is 5 times diameter of the downstream the jet. The jet trajectory uplift response amplitude when the jet angle increases from 10° to 20°. In the range of more than 5 times caliber, the magnitude of jet trajectory lift gradually decrease as the jet angle increases.

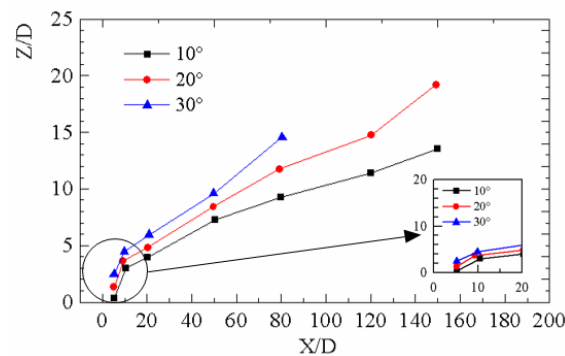


Figure 6. Jet trajectory at different outflow angles (relative position).

The non-dimensionalized form of velocity is:

$$\frac{U - U_0}{U_j - U_0} \quad (2)$$

Where, U is the velocity at any point on the axis of the jet, U_j is the flow velocity at the outlet of jet orifice, and U_0 is the flow velocity of ambient water. The attenuation of the velocity on the axis with different outflow angles is shown in figure 7. The jets flowing from the orifice interact with the ambient fluid. The turbulent kinetic energy near the orifice is very large, so the degree of entrainment is very high. The gradient of the velocity attenuation at the orifice is very large because the outflow angles are different. Therefore, the changes of the velocity attenuation zone are mainly concentrated in

the range of less than 45 times diameter of outlet. As the horizontal distance from the orifice increases, the turbulent kinetic energy of the jet decreases and the jet velocity decreases exponentially as the horizontal distance increases.

The attenuation of the temperature on the axis with different outflow angles is shown in figure 8. From the figure, we can see that the law of temperature attenuation along the central axis is very similar to the velocity attenuation, and the temperature decays rapidly near the orifice. With the increasing distance, the decay velocity decreases, the curve gradually develops into a horizontal line. Temperature decay mainly appeared within the range of less than 80 times diameter of outlet. Although the flow angle has changed from 10 to 30, the attenuation law has not changed, it can be seen from the change of the jet angle with the angle of 10° to 30° in the figure that the distance of the temperature decay is gradually shortened. The jet velocity and temperature decay rapidly near the orifice and the influence range of the jet is mainly within the 80 times aperture. The result is similar to Jiang's report [17], which experimentally investigated the mixing behavior of 45° inclined dense jets in unbounded co-flowing. Horizontal distance from the jet orifice is at the same location. The curve with a small outflow angle is always above the curve with a large outflow angle, which was obtained that the smaller the angle of the jet is, the more gradually the attenuation velocity and temperature change.

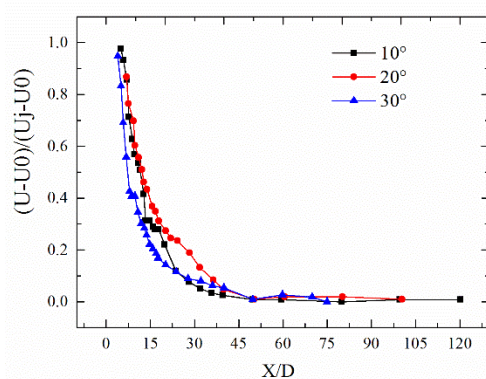


Figure 7. Jet velocity axis attenuation.

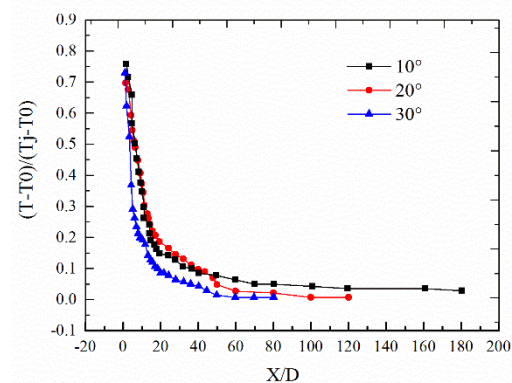


Figure 8. Jet temperature axis attenuation.

4. Conclusions

- Jet velocity and the temperature decay rapidly near the orifice outlet, and the area where the velocity of flow changes most is the area with the greatest temperature dilution.
- The larger the outflow angle is, the more obviously the trajectory of the jet rises. The lift of the jet trajectory gradually decreased in the range of 5 times diameter of outlet.
- The influence range of velocity is mainly within the range of 50 times diameter of outlet; the influence range of temperature is mainly within the range of 80 times diameter of outlet; the smaller the jet angle is, the slower the attenuation of velocity and temperature changing.

Acknowledgments

The research was supported by the National Science Foundation of China (51706180), the Natural Science Foundation of Shaanxi Province (2015JQ5181) and the Education scientific research project for the Education Department of Shaanxi Province (14JK1534).

References

- [1] Cheng Y L and Wu K 2016 Study on submerged type of thermal discharge in power plant by using numerical simulation *Journal of North China Electric Power University* **43** 1007-2691
- [2] Tatiana P T and leonadro M N 2009 Effects of a nuclear power plant thermal discharge on habitat complexity and fish community structure in ilha grande bay, brazil *Marine*

- Environmental Research* **68** 188-95
- [3] Poornima E H, Rajdurai M, Rao T S, *et al* 2005 Impact of thermal discharge from a tropical coastal power plant on phytoplankton *Journal of Thermal Biology* **30** 307-16
 - [4] Chu V H and Abdelwahed M S T 1990 Shore attachment of buoyant effluent in strong crossflow *Journal of Hydraulic Engineering* **116** 157-75
 - [5] Chen H Q, Xu Y L and He Y Y 2008 Progress and experience of 50 years' study on cooling water circulation of thermal nuclear power plants *Journal of China Institute of Water Resources and Hydropower Research* **6** 288-98
 - [6] Zhang C K, Yao J and Tao J F 2006 Advances in numerical simulation of thermal discharge of heat or nuclear power plants in coastal areas *Advances in Science and Technology of Water Resources* **20** 5-9
 - [7] Bao M X, Tao J F, Zhang C K and Yang T 2015 The analysis of river thermal discharge mixing and temperature ridge characteristics *Chinese Journal of Hydrodynamics* **30** 291-8
 - [8] Zhang S N 1988 *Ewironomental Hydraatics* (Nanjing: Hohai University Press) 102-4
 - [9] Prych E A 1970 Effects of density differences on lateral mixing in open channel flows (California Institute of Technology: WM Keck Laboratory of Hydraulics and Water Resources) pp 118-27
 - [10] Huai W X, Liang A G and Yang Z H 2008 Numerical simulation on solution transport in a unsymmetrical compound channel (II)-influence of the secondary current on solution transport *Journal of Basic Science and Engineering* **15** 445-9
 - [11] Huai W X, Chen X F and Chen D H 2001 Study on the 3D numerical modeling near-field behavior of waste water in side-discharge, 2, the predictionthe near-field behavior *Advances in Water Science* **12** 28-32
 - [12] Li Z W, Huai W X and Qian Z D 2009 Numerical simulation of turbulent radial jets in static ambient *Journal of Hydraulic Engineering* **11** 1320-5
 - [13] Hao R X, Zhou L X and Chen H Q 1999 3-D Numerical modeling of turbulent buoyant surface jet in cooling water projects *Journal of Hydrodynamics. Ser. A, Dec.* **14** 484-92
 - [14] Duan Y F, Zhao Y J, Ji P and Zhang B B 2017 Tank experiment of cooling water discharge and its numerical simulations using 2D, quasi-3D and 3D models *Journal of Hydroelectric Engineering* **36** 100-10
 - [15] Chen Y P, Tian W Q and Fang J Y 2016 Experimental study on hydrodynamic characteristics ofmultiple jets in wave environment *Advances in Water Science* **27** 569-78
 - [16] Ma R Q 2010 3-d Numerical simulation of thermal discharge in constant homogeneous water area (Taiyuan: Taiyuan University of Technology) pp 1-3
 - [17] Jiang M, Law W K and Zhang S 2017 Mixing behavior of 45° inclined dense jets in currents *Journal of Hydro-environment Research* **18** 37-48