Numerical simulation of hydraulic characteristics of surface jet on horizontal base plate with shallow depth

Zheng Yan¹, Zhao Shuxiang², Xu Hongchen¹, Pan Weiping¹ and Zhang Dong¹*

¹ Zhejiang Design Institute of water Conservancy & Hydro-electric Power, Hangzhou
² Lincang Xinhua Water Resources and Hydropower Development Co., Ltd., Yunnan
*Corresponding author’s e-mail: candyzhengyan123@163.com

Abstract. The VOF (volume of fluid) method was applied to track free water surface, and the Realizable turbulent model was used to close the two-phase flow time-averaged equation to numerically simulate the hydraulic characteristics of surface jet on horizontal base plate with shallow depth. The result showed that a big vortex exists in the flow field, and the greater the inlet flow velocity, the longer the length of vortex. The distribution laws of horizontal velocity and longitudinal velocity of a cross section are different in the backflow region and open channel flow region. The gradient decaying of the maximum velocity on cross-section was mainly affected by the inlet velocity.

1. Introduction
Jet flow generally refers to a fluid state in which a flow enters into another fluid at a high speed from the discharge outlet or nozzle, and mixes intensively with surrounding fluid. Because of the velocity difference between the jet flow and the surrounding fluid, entrainment and mixing are formed, which makes the cross-section of the jet expand continuously, the velocity of the jet decreases along the axis, and the concentration or temperature drops low. Jet has been applied in many fields, such as water conservancy, environmental protection, aviation, chemical engineering and machinery. Since the 1990s, jet research has become an important research object of fluid mechanics and environmental hydraulics. Based on experimental research and theoretical analysis, Ead et al. studied the hydraulic characteristics of wall jet and surface jet on smooth and rough floor, including the development of wall jet and the attenuation law of velocity along the way, as well as the length of vortex near the water surface and the drop value of water level near the gate [1-3]. Jiang Guoqiang et al. measured the flow field of slot turbulent jets in hydrodynamic environment under various nozzle lengths and velocity ratios [4-5]. Although there are many reports about jet flow, but few studies on wall jet Numerical Simulation.

2. Mathematical model

2.1 The Realizable ε turbulent model
In order to simulate the diffusion velocity of plane and Circular Jets more accurately in rotating flow calculation, boundary layer calculation with directional pressure gradient and separated flow calculation. The realizable K - ε model is selected for calculation, and the governing equation is as follows.

Continuity equation:
\[
\frac{\partial \rho}{\partial t} + \frac{\partial (\rho u_i)}{\partial x_i} = 0
\]
Momentum equation:
\[
\frac{\partial (\rho u_i)}{\partial t} + \frac{\partial (\rho u_i u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_i} \left[ \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right] - \frac{\partial}{\partial x_j} \left( \rho \frac{u_i u_j}{\Delta} \right) + \rho g_i
\]  
(2)

Reynolds stress equation:
\[
-\rho u_i' u_j' = \mu \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} (\rho k + \rho \varepsilon) \delta_{ij}
\]  
(3)

The turbulent kinetic energy $k$ and dissipation rate $\varepsilon$ are as follows:
\[
\frac{\partial k}{\partial t} + \frac{\partial }{\partial x_i} \left[ \rho u_i k \right] = \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial k}{\partial x_j} + \frac{\partial \varepsilon}{\partial x_j} \right) \right] + \rho C_\varepsilon \varepsilon \left( 1 + \frac{k}{\varepsilon} \right)
\]  
(4)

\[
\frac{\partial \varepsilon}{\partial t} + \frac{\partial (\varepsilon u_i)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ \mu \left( \frac{\partial \varepsilon}{\partial x_j} + \frac{\partial \varepsilon}{\partial x_j} \right) \right] + \rho C_\varepsilon \varepsilon (1 - \varepsilon) \left( 1 + \frac{k}{\varepsilon} \right)
\]  
(5)

The above equations constitute the closed equations for solving the time averaged flow field distribution. According to the working conditions, the corresponding boundary conditions are applied to form the solution of the equations.

2.2 The VOF (volume of fluid) method

VOF is an effective method to deal with complex free surface. The free surface (i=1, 2, 3) is tracked according to the volume ratio function $F_w (x_i, t)$ of fluid and mesh in mesh element. It track the change of fluid rather than the movement of particles on the free surface [6].

The $F_w$ in governing equations is as follows:
\[
\frac{\partial F_w}{\partial t} + \frac{\partial (u_i F_w)}{\partial x_i} = 0
\]  
(6)

$\rho$ and $\mu$ are the functions of $F_w (x_i, t)$;
\[
\rho = F_w \rho_w + (1 - F_w) \rho_a;
\]
\[
\mu = F_w \mu_w + (1 - F_w) \mu_a
\]  
(7)

$\rho_w$ and $\rho_a$ refer to the density of water and air, $\mu_w$ and $\mu_a$ refer to the turbulent viscosity of water and air. If the above equations (6), (7) and (8) are solved simultaneously with the basic equations (1) to (5) of the turbulence mathematical model, the unknown variables, such as pressure, velocity, turbulent kinetic energy, dissipation rate and water volume ratio function, can be obtained.

3. Calculation area and parameters

The surface jet on the horizontal bottom plate is shown in Fig. 1, where $B_0$ is the inlet height; $Z_0$ is the elevation of the lower edge of the intake from the bottom plate; $L_e$ is the vortex length; $v_1$ is the average velocity of inlet cross section; $v_2$ is the average velocity of the cross-section when the jet develops to the depth water; $Y_t$ is the tailwater depth.

The model test was carried out in a rectangular tank with a length of 7.6m, a width of 0.446m and a height of 0.6m. Two pumps were used to increase the water head, and the flow rate was measured by a magnetic flowmeter. The inlet is set at the water surface, and the upper edge of the inlet is flush with the water surface. Streamline devices are installed on the upper and lower edges of the inlet to produce jet in the tank. A gate at the end of the tank is used to control the water depth at the tail of the tank. After appropriate simplification, a two-dimensional model of surface jet on horizontal bottom plate is established, as shown in Fig. 2.

In this study, the surface jet under five different working conditions A, B, C, D and E are simulated and calculated. The influence of inlet height, average velocity of inlet section and tail water depth on the hydraulic characteristics of surface jet is comprehensively considered. Parameters of each working...
condition are shown in Table 1.

| Working conditions | B₀  cm | Z₀  cm | u₀  (m·s⁻¹) | F₀ | γ₁  cm |
|--------------------|-------|--------|--------------|----|-------|
| A                  | 1.0   | 50     | 0.75         | 2.4 | 51    |
| B                  | 1.0   | 50     | 1.50         | 4.8 | 51    |
| C                  | 1.0   | 50     | 2.25         | 7.2 | 51    |
| D                  | 1.0   | 30     | 1.50         | 4.8 | 31    |
| E                  | 2.0   | 50     | 2.25         | 5.1 | 52    |

4. Mesh Dividing
In order to simulate the free water surface, the whole model is divided into structured grids, and refined mesh were adopted near the water surface. The minimum height of the grid in the vertical direction is 2 mm, and the length of the grid in the horizontal direction is 1 cm.

![Figure 3. Grid of 2D simulation area.](image)

5. Boundary conditions
The initial values of turbulent kinetic energy k and dissipation rate ε are calculated by empirical formula. Pressure outlet is adopted at outlet boundary, and the turbulent flow at the outlet is considered to be relatively balanced, the velocity gradient is 0. Pressure inlet is adopted when the top surface contacts with air. The gate and bottom plate are walls, which are determined by standard wall function method.

6. Analysis and further discussion

6.1 Model Verification
Taking condition C as an example, the simulated values of horizontal velocity along the vertical lines of cross sections x = 0.5, 1.0 and 1.5 m are compared with the measured values. The results are shown in Fig. 5. It can be seen from the figure that there is a small difference between the simulated value and the measured values near the water surface, this may be due to the larger surface velocity and turbulence, which makes the measurement more difficult. The measured values are in good agreement, which proves the correctness and reliability of the simulation results.

![Figure 4. Comparison of horizontal velocity on different cross sections of condition C.](image)
Analyzing the horizontal velocity distribution in Fig. 4, the surface velocity is larger on the cross section of \( x = 0.5 \) and 1.0 m, and then rapidly drops to zero along the depth direction. The closer to the intake, the greater the maximum velocity on the cross-section is, and the faster the velocity decreases. And then it becomes negative along the depth direction caused by the vortex. Near the bottom plate, the horizontal velocity decreases rapidly to zero along the depth due to the stagnant of the side wall. It shows that the cross section at \( x = 0.5 \) and 1.0m is in the jet development zone. When \( x = 1.5 \) m, the horizontal velocity distribution along the water depth is more uniform, only slightly fluctuates near the water surface, and the velocity values are all positive, indicating that the flow at this place is close to the open channel.

6.2 Streamline distribution
The streamline of condition A, B and C is shown in Fig. 5. It can be seen from the figure that the streamline of the surface jet under different working conditions are similar. After the water is injected from the nozzle, the high-speed water flow is close to the water surface. Under it there is a large vortex. The vortex rotates clockwise, and the vortex almost fills the whole area. This area is the jet development area. After the vortex, the streamline diffuses rapidly and the streamline distribution is more uniform, which is similar to the open channel. The vortex size in the streamline diagram of three working conditions can be seen obviously. The larger the inlet velocity, the longer the vortex length.

![Figure 5. Streamline distribution of condition A, B and C.](image)

In order to compare the relationship between the vortex length and the hydraulic elements more accurately, the vortex length of each working condition is shown in Table 2. When other conditions are the same, the larger the inlet velocity, the longer the vortex length is compared with B and D conditions, it can be seen that other conditions are the same while the tail water depth is different, the vortex length changes little, which can be ignored. Comparing C and E, it can be seen that when other conditions are the same, the inlet width increases, but the vortex length decreases.

| Working conditions | A   | B   | C   | D   | E   |
|--------------------|-----|-----|-----|-----|-----|
| \( L_v / \text{m} \) | 0.65| 1.00| 1.30| 1.05| 1.15|

6.3 Maximum velocity decay characteristics along cross section
The maximum velocity decays along the cross section of each working condition is shown in Fig. 6.

The curve starts from \( v_1 \), at first there is a small distance along the path, then rapidly decreases along the way, and finally reduces to a minimum value, then it almost remains unchanged. It is due to the uniform distribution of velocity along the water depth after the jet is fully developed. It can be seen from Fig. 6 that except for condition A, the decay curve of maximum velocity in cross section of other working conditions has slight fluctuation, which may be caused by turbulence due to large velocity of surface jet.
By comparing the decay curves of A, B and C condition: When the tail water depth is same, the larger the inlet average velocity is, the longer distance along X-axis the maximum velocity decreases and the greater the decay gradient became. Comparing the curves of B and D condition, it can be seen that when the average inlet velocity is same, the decay curves of the maximum velocity along the cross-section are almost the same under different tail water depths, so the influence of different tail water depths on cross-sectional maximum velocity decreasing is very small. Compare the decay curves of C and E condition in the figure, the inlet width of the two working conditions is different while other conditions are the same. The curve of the two working conditions starts to decrease from the same speed, at the beginning stage, the curve of condition E is slightly higher than that of condition C, and is approximately parallel along the way. When \( x = 1.1 \)m, the two curves intersect, and then the curve of condition E drops below the curve of condition C and reaches the minimum value first. It can be seen that the maximum velocity of cross section changes first along the way, its change speed is slow. In addition, by comparing the maximum velocity decal distance along X axis of cross section in Fig. 6 with the corresponding vortex length in Table 2, it can be found that the maximum velocity decay distance of cross section is approximately equal to the length of reflux zone.

![Figure 6. Curve descent of maximum velocity in cross section under different working conditions.](image)

7. Conclusion
In this paper, the numerical simulation method is used to study the hydraulic characteristics of surface jet on horizontal base plate with shallow depth. By comparing the simulated and measured values, the results are in good agreement, which proves the correctness of the mathematical model and the reliability of the simulation results.

It is also found that the larger the inlet velocity, the larger the vortex length; the inlet width increases, while the vortex length decreases. In the jet development area and the open channel flow area, the distribution of horizontal velocity and longitudinal velocity is obviously different, which is mainly affected by the vortex. The decay length and gradient of the maximum velocity curve are mainly affected by the inlet velocity and inlet width.

References
[1] Ead S A, Rajaratnam N, Asce F. (2002) Plane turbulent wall jets in shallow tailwater. Journal of Engineering Mechanics, 128: 143–155.
[2] Ead S A, Rajaratnam N, Asce F. (2004) Plane turbulent wall jets on rough boundaries with limited tailwater. Journal of Engineering Mechanics, 130: 1245–1250.
[3] Ead S A, Rajaratnam N, Asce F. (2001) Plane turbulent surface jets in shallow tailwater. Journal of Fluids Engineering, 123: 121–127.
[4] Jiang Guoqiang, Li Wei. Shuilixu. (2004) Vortex structure of slot turbulent jets in cross-flow. Journal of Hydraulic Engineering, 35: 52–57.
[5] Jiang Guoqiang, Li Wei, Tao Jianhua. (2004) Hydraulic characteristics of slot turbulent jets in hydrodynamic environment. Journal of Fluids Engineering, 35: 51–55.
[6] Zhang Jan, Fang Jie, Fan Boqing. (2005) Review of theory and application of VOF method. Advances in Science and Technology of Water Resources, 25: 67–70.