Numerical study on hydraulic characteristics of improved X-shaped FGP

Yang Xue1, *, Lingjun Xu1, CaiPin Wang2
1 Powerchina Huadong Engineering Corporation Limited, Hangzhou, China
2 Huoshan County Water Affairs Bureau, Lu’an, China
*Corresponding author e-mail: xue_y3@ecidi.com

Abstract. The three dimensional numerical simulation was carried out for flood discharge and energy dissipation process of five improved X-shaped FGPs in hydropower project by using coupled Level set and VOF model and RNG k-ε turbulent model. The change laws of flow pattern, water surface profile and velocity distribution of X-shaped FGP and lock chamber were gained. The results of numerical simulation were compared with the model test data and good agreement was found. It validated the numerical simulation method is reliable, It can provide strong references for the design of energy dissipation and shape optimization of practical project, the results from computation can be supplement for experiment.

1. Introduction

X-shaped flaring gate piers were widely applied in high-head large single-width power stations. The water flow shrink laterally in the gate chamber under the action of FGP, the water was divided into upper and lower parts, after exiting the gate, the lower water flow spread laterally along the stepped dam surface, the upper water flow then stretch longitudinally in the air as a jet stream and formed water fin, which increased the energy dissipation space and Intensified the turbulence of the water flow. So it has its unique advantages in energy dissipation effect. In recent years, many scholars have carried out research on the design and improvement of the flaring gate piers. Due to the complex shape and hydraulic elements such as free water surface under its action are much more complicated than conventional hydraulic structures, the design and optimization of the flaring gate pier mainly rely on model experiments at present.

Numerical simulation is an important research approach to hydraulic problems, and some scholars have also achieved certain results in numerical simulation of the hydraulic characteristics of the flaring gate piers. However, the numerical simulation technology has the shortcomings of low accuracy while simulating the flow field characteristics of hydraulic structures, and there are certain errors between the simulation results and the model experiments. On the basis of summarizing previous experience, the three dimensional numerical simulation was carried out for flood discharge and energy dissipation process of five improved X-shaped FGPs in hydropower project by using coupled Level set and VOF model and RNG k-ε turbulent model. The reliability of the numerical simulation method is verified through model experiments, and it is expected to provide a reference for the numerical simulation of hydraulic structures with complex free surfaces.
2. Control equation and discrete domain

In this paper, combined with the actual project of a hydropower station with improved X-shaped FGP, numerical simulation was carried out by using coupled Level set and VOF model and RNG $k$-$\varepsilon$ turbulent model, the control equation equations are as follows:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0$$  \hspace{1cm} (1)

$$\frac{\partial u_i}{\partial t} + \frac{\partial u_i u_j}{\partial x_j} = -\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j} \left[ \left( \mu + 0.0845 \rho \frac{k^2}{\varepsilon} \right) \left( \frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) \right]$$  \hspace{1cm} (2)

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ 1.39 \left( \mu + 0.0845 \rho \frac{k^2}{\varepsilon} \right) \frac{\partial k}{\partial x_j} \right] + G_k + \rho \varepsilon$$  \hspace{1cm} (3)

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_i \varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_j} \left[ 1.39 \left( \mu + 0.0845 \rho \frac{k^2}{\varepsilon} \right) \frac{\partial \varepsilon}{\partial x_j} \right] + 1.42 G_k \frac{\varepsilon}{k} - 1.68 \rho \frac{\varepsilon^2}{k}$$  \hspace{1cm} (4)

$u_i$ is the velocity component in the $x_i$ direction, the index value of $i$ and $j$ range from 1 to 3, $\rho$ is the fluid density, $p$ is the pressure, $\mu$ is the fluid viscosity, $k$ is the turbulent kinetic energy, $\varepsilon$ is the dissipation rate, and $G_k$ is the production term of the turbulent kinetic energy caused by the average velocity gradient.

In order to solve the complex free water surface effectively, the CLSVOF method was adopted in the calculation. This new coupling method was used first in the VOF to construct the geometric reconstruction interface, then calculated the $\phi$ function in the Level set, and then initialized the $\phi$ function again, and the normal vector and curvature of the intersecting interface were no longer used F function, but calculated by continuous $\phi$ function. In this way, the CLSVOF method can avoid the problem of non-conservation of physical quantities in the Level set method, and can also made up for the accuracy of the interface normal and curvature in the VOF method. It combined the advantages of the two while avoiding their shortcomings, and achieved good result.

The calculation area in this paper includes the reservoir, the gate chamber section, the platform stage and the stilling pool section. Due to the complex body shape, the grids in the calculation domain are combined with different density according to the different areas. The number of grids is 1.2 million. The calculation area is 113m high, 108m wide and 270m long.

There are five spillway holes in improved X-shaped FGP, according to the symmetry of the flow, the two sides of the simulation calculation are defined as symmetrical boundary conditions. The upstream is given the reservoir water level, the upper part of the reservoir and the water tongue are defined as pressure inlet. The outlet are defined as pressure outlet with limited water level. All walls are defined as non-slip boundary conditions. The viscous bottom layer is treated by wall function method. The whole field is a transient simulation. The inlet and outlet flow error is within 5% When the calculation is considered stable.
3. Analysis of simulation calculation and model experiment results

The process of water flowing through the chamber and flying in the air is truly reproduced through CLSVOF numerical simulation. Observing from the flow direction of the water, it can be found that the forward flow at the inlet of the chamber is relatively symmetric before the water flow entering the chamber, and the water level fluctuates little with a tiny bump at the head of the FGP. After entering the chamber, the water surface began to appear different height with low left and high right, As the water flows in, the height difference between the left and right increase increase in the same pace. After traveling to 1/4 the length of the lock chamber, the height difference between the two began to gradually decrease. After entering the middle of the chamber, the height difference between the left and right sides is 0, the water surface began to appear high left and low right after the middle of the chamber, which was exactly opposite with before. After that, the height difference between the two sides gradually increased and remained until the water flowed out of FGP.

The water flow in the chamber is divided into upper and lower parts by the FGP, an inflection point of water flow begins to appear at the conical surface, then the water surface starts to climb along the wall and rises upward, appearing as a convex line. The upper water flow behind the FGP develops vertically and horizontally under the action of FGP. The curvature of the upper water surface changes greatly, and reaches the highest point after the water flow provokes a certain angle, Then it begins to fall under the action of gravity, showing a hydraulic phenomenon like a tall and thin rooster tail. Due to the difference between the water velocity and the air resistance, the separation will occur During the fall. After passing through the conical surface, the upper horizontal water flow is stretched horizontally to a very small area, and is connected with the vertical water flow, The lower water flow spreads horizontally along the lower step surface after flowing through the FGP in the form of a water skiing flow.

Figure 3 shows the velocity vector distribution diagram behind the improved X-shaped FGP. It can be seen that after the water flows out of the chamber, a large amount of gas is entrained, and the water-gas two-phase turbulence is strong.
Figure 2. Flow pattern of improved FGP flow field

Figure 3. Vector distribution after FGP

Figure 4 shows the comparison results of the water surface line when the flow rate is 12766 m$^3$/s and 15554 m$^3$/s in the entire calculation area. It can be seen from the figure that with the increase of the flow, the water surface line in the chamber under the large flow condition is lower than that in the small flow condition. After the water flows out of the FGP, the greater the flow, the greater the height of the parabolic jet, and the closer it will drop. There are three intersections of the water surface lines of the two conditions. The first intersection appears at the entrance of the chamber, the second intersection is located behind the FGP, and the third intersection appears during the free fall of the water tongue. Figure 5 shows that: the greater the water flow, the lower the water surface in the chamber, the greater the height of the water projected behind the FGP, The more gas is mixed in the air. The closer drop point indicates that the design of the stilling basin can be shortened accordingly. Compared with other energy dissipaters, the advantage of the improved X-shaped FGP is also to reduce the cost of the design of the stilling basin.

![Figure 4. The comparison of water surface profile](image)

4. Conclusion
Reasonable division of the calculation area grid and the use of CLSVOF numerical simulation methods are the key elements for the successful completion of the three-dimensional flow field calculations in this paper. The CLSVOF numerical simulation method combines the advantages of the Levelset method and the VOF method, which Inject new vitality into the simulation calculation of incompressible two-phase flow interface.
The flow pattern of the improved X-shaped FGP flow field and the water surface line and velocity changes in the chamber are obtained through numerical simulation. It is concluded that the water surface in the lock chamber presents two twisting states with different twisting directions, the first half of the left is low and the right is high, the middle is level, and the second half is left high and right low. This makes up for the lack of observation in the model experiment to a certain extent.

With the increase of the flow, the flow velocity of the water in the chamber increases, and the water surface line decreases with the increase of the flow. The height of the water shot behind the FGP increases, and the drop point is closer.

References
[1] Xiaohong Nan, Zongxiang Liang, Hansheng Liu. Application and research of a new type of wide-tail pier in Suofengying Hydropower Station [J]. Journal of Hydraulic Engineering, 2003(8): 49-57.
[2] Guangtong He, Xiankang Zeng, Zufà Li. Optimal design of new energy dissipater structure for Shuidong Hydropower Station [J]. Hydropower, 1994(9): 26-28.
[3] Chengjin Yin, Zongxiang Liang, Zhi Zheng, etc. Experimental study on the pressure characteristics of the stilling pool floor combined with the wide tail pier [J]. Journal of Hydroelectric Engineering, 2011, 30(4): 103-108.
[4] Hongqin Zhang, Baoping Sun, Wei He, etc. Application of X-shaped flared pier in Ludila Hydropower Station [J]. Northwest Hydropower, 2007(4): 40-45.
[5] Yaohua Hu, Chao Wu, Ting Zhang. Analysis of influencing factors of stepped aerated cavity of X-shaped flared pier [J]. Journal of Sichuan University (Engineering Science Edition), 2007, 39(3): 24-29.
[6] Ting Zhang, Chao Wu, Hong Lu, etc. Numerical simulation of three-dimensional flow field of combined energy dissipation of X-shaped flared pier and stepped overflow dam [J]. Journal of Hydraulic Engineering, 2004, (8): 15-20.
[7] Haifei Sha, Hui Zhou, Shiqiang Wu, et al. Two-dimensional numerical simulation of two-phase flow of flood and gas in dam body [J]. Research and Progress in Hydrodynamics, 2007, 22(3): 311-316.