OPTIMIZATION AND ANALYSIS OF FLOW CHARACTERISTICS IN A PRACTICAL SPILLWAY DESIGN BASED ON THE VOF MODEL

LI Jinping^{*}, LIU Fei[†], and YANG Jiandong[†]

*Water Resources and Hydropower Engineering Science State Key Laboratory, Wuhan University, Wuhan, Hubei, China e-mail: lukeping@whu.edu.cn

[†]Water Resources and Hydropower Engineering Science State Key Laboratory, Wuhan University, Wuhan, Hubei, China e-mail: liufei_cfd@126.com, jdyang@whu.edu.cn

Key words: Spillway, Numerical simulation, VOF, Optimization, Fluid Dynamics

Abstract: The spillway is an important structure in a hydropower station. It must have adequate discharge capacity to ensure the safety of the dam and the hydropower plant. For some reason, a spillway meets some design problems: there is a corner of 90-degree turn in the diversion channel just before the weir, and a fan-shaped layout of the water energy dissipater downstream to the weir deflecting to one side. There is no adequate time and funds to do the hydraulic model test, so the numerical simulation is adopted to solve this problem. Based on the Volume Of Fluid (VOF) multidimensional two-phase flow model, the numerical simulation of the whole spillway was carried out to research the discharge capability of the overflow, flow pattern in spillway, as well as the water jet nappe angle, the distance and the drop point of nappe. According to the numerical computation results of the pressure, the water depth and the flow velocity distribution along the spillway, the rationality of the spillway general layout is verified and some local structure shape and size of the spillway is optimized.

1 INTRODUCTION

The spillway is one of the most important hydraulic structures in the hydropower project to ensure the safety of hydraulic structures during the flood. In general, the spillway consists of diversion channel, overflow weir, chute (sluice channel), and energy dissipater. The main function of the structures is to guide the flow, control the flow, transport the flow and dissipate the flow energy efficiently. In a practical project, because of the complex topography of the spillway location, the composition structures of a flood spillway cannot be arranged in a straight line smoothly, which may disorder the flow pattern, and influence the discharge of the spillway over weir.

So the spillway must be carefully designed to verify the overflow discharge capacity, the water energy dissipation effect, and a series of features affecting its own security and stability during operations. At present, the physical model test is often used to be resolved the key problems in the spillway design, but it costs much time and lots of funds.

With the development of numerical technique (CFD) and computer science, the three-dimensional numerical simulation of spillway flow has made fruitful results^{1,2,3}. In this paper, the advanced CFD method is used to solve the design problems in a practical spillway design. The water flow in the whole spillway has been simulated by CFD based on the VOF model, to analyze discharge capacity, flow characteristics and flow pattern. The calculation results can be used as the basis of the shape optimization.

2 MATHEMATICAL MODEL

It is a free surface flow in the spillway primarily driven by gravity, and the flow is turbulent with great velocity in the chute. According to the literature, this kind of free surface flow can be simulated by the volume of fluid (VOF) method as water-air two-phase flow problems^{4,5}. The entire calculation domain is mainly related to two types of fluid, water and air. And the result of the standard $k - \varepsilon$ turbulence model is more consistent with the experimental value than the result of Reynolds stress model⁶. Therefore, the standard $k - \varepsilon$ turbulence model is used in the three-dimensional numerical simulation of spillway flow.

2.1 Basic Equation

The following is the basic equations: Continuity equation:

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \tag{1}$$

Momentum equation:

$$\frac{\partial \rho \mathbf{u}_i}{\partial t} + \frac{\partial}{\partial x_j} (\rho u_i u_j) = -\frac{\partial P}{\partial x_i} + \frac{\partial}{\partial x_j} \left[(\mu + \mu_t) (\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}) \right]$$
(2)

k equation:

$$\frac{\partial(\rho k)}{\partial t} + \frac{\partial(\rho u_i k)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[(\mu + \frac{\mu_i}{\sigma_k}) \frac{\partial k}{\partial x_i} \right] + G - \rho \varepsilon$$
(3)

 ε equation:

$$\frac{\partial(\rho\varepsilon)}{\partial t} + \frac{\partial(\rho u_i\varepsilon)}{\partial x_i} = \frac{\partial}{\partial x_i} \left[(\mu + \frac{\mu_t}{\sigma_{\varepsilon}}) \frac{\partial\varepsilon}{\partial x_i} \right] + C_{1\varepsilon} \frac{\varepsilon}{k} G - C_{2\varepsilon} \rho \frac{\varepsilon^2}{k}$$
(4)

Where, *t* is the time; u_i is the velocity components; x_i is the coordinate components; ρ is the density; μ is the molecular viscosity coefficient; *P* is the correct pressure; μ_i is the turbulent viscosity coefficient, which can be derived from the turbulent kinetic energy *k* and turbulent dissipation rates ε :

$$\mu_t = \rho C_\mu \frac{k^2}{\varepsilon} \tag{5}$$

Among them, the empirical constants, $\sigma_k = 1.0$, $\sigma_{\varepsilon} = 1.3$, $C_{1\varepsilon} = 1.44$, $C_{2\varepsilon} = 1.92$; And the turbulent kinetic energy source term *G* caused by the average velocity gradient can be defined by the following formula:

$$G = \mu_i \left(\frac{\partial \mu_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) \frac{\partial u_i}{\partial x_j}$$
(6)

VOF (The Volume of Fluid) model defines a volume of fluid fraction function F = F(x, y, z, t) to describe the changes of the free surface. The sum of volume fraction of water and air equals 1 in each calculation unit, namely:

$$\alpha_{w} + \alpha_{a} = 1 \tag{7}$$

Water-air interface can be determined by solving the following continuity equation:

$$\frac{\partial \alpha_w}{\partial t} + \mu_i \frac{\partial \alpha_w}{\partial x_i} = 0 \tag{8}$$

Where, α_w is the fraction of water volume; α_a is the fraction of air volume.

The unknowns and parameters shared by water and air in the flow field can be obtained from the weighted average of volume fraction.

The numerical model does not take into account the compressibility of water and air flow within the spillway. The SIMPLEC algorithm is used to solve the pressurevelocity coupled field. And the convergence can be speeded up if the appropriate relaxation coefficient is selected.

2.2 Computational Domain and Meshing

The computational domain includes the entire spillway, and a part of the downstream river. The specific arrangement of the spillway (Figure 1) is as the following:

Diversion channel, the length is 78.4m, and there is a 90° turn-over in plane. The bottom elevation of the curved diversion channel is 329.6m with width 32m, left side-wall is vertical and the right side-wall is slope of 45° .

Overflow weir, practical weir of hump type b (Figure 2) is adopted; weir top elevation is 333.0m. There are three holes with width of 9m for each, two piers with thickness of 2.5m.

Sluice channel, the total horizontal length of the chute is 187.2m. There are two sections, one with longitudinal slope i = 0.05 (horizontal length is 133.8m), the other one with longitudinal slope i = 0.4 (horizontal length of 35.4m), and the two sections are connected by a parabolic segment (horizontal length of 18.0m). At the start of the small slope channel, there is the contraction section in plane (width from 32m to 20m, length is 60m, and angle is 5.17°). At the end of the steep channel, there is the trajectory bucket (horizontal length of 14.92m, the anti-arc radius is 15m, bucket top elevation is 304.0m, and lob angle is 20°, Figure 3) for energy dissipation.

In order to ensure sufficient accuracy and stability requirements of VOF method, the unstructured hexahedral meshes are adopted, the grid size is from 0.1m to 0.6m. In the gate slots the size is 0.1m, in the weir the size is 0.4 m, and in the channel the size is

0.6m. The total number of grids is 1790140 and the number of nodes is 1889022. The computational domain shows in Figure 4.



Figure 4: The scope of numerical simulation domain

2.3 Boundary Condition

In the reservoir end, the boundary condition is pressure inlet. The water depth at the inlet is determined according to the flood level of reservoir (for example, the depth is 13.47m when the flood level is 343.07m). For the turbulent kinetic energy k and the dissipation rate ε can be calculated according to empirical formula. At the downstream river, and all of the air inlet or outlet condition, the pressure condition is adopted, and

the pressure value is the atmosphere pressure (1 atm). The side walls and floors of the spillway are set to no-slip wall boundary condition, and the wall function method is used to simulate the wall-nearby flow regime.

3 CALCULATION RESULTS AND ANALYSIS

3.1 Discharge Capacity

By three-dimensional numerical calculation, under the condition of the checked flood level (343.07m) of the reservoir, the overflow discharge is $1757m^3/s$. While the design discharge of the spillway is $1689m^3/s$ (calculated by empirical formula). The numerical result is larger than the design value by 4.03%. It shows that the discharge capacity is enough to ensure the security of the dam and hydropower station, and the curved diversion channel has little effect on the discharge capacity of the weir.

But the curved channel has effect on the uniformity of flow discharge through the weir. The numerical result shows that the discharge is different with each other among the three holes. The right one (face to the downstream) has the largest value of $593.1m^3/s$, the middle one has the medium value of $586.3m^3/s$, and the left one has the smallest value of $576.7m^3/s$. Because there is a 90° turn, the water goes to the right hand by centrifugal force. But, the difference between each hole is not eminent (figure 5). So the diversion channel does not need additional modification.



Figure 5: Streamline in the diversion channel and weir chamber

3.2 Distribution of Water Depth

The water depth along the spillway calculated by the numerical simulation is as figure 6. It shows that the numerical average depth is less than the design depth. So the design of side walls is safe.





In fact, according to the unsteady calculation result, the water surface in the spillway is constantly fluctuating. In some local areas, the surface fluctuation is even more dramatic. The water surface at the same section is not flat, that is to say, the water depth at the left and right side wall is not the same.

In the diversion channel section, the water depth at the right wall is greater than water depth at the left wall. The reason is that the left water have been driven to the right (significant lateral flow) by centrifugal force due to the sharp turn of the channel. At the end of the sharp turn, the water depth difference reaches 1.04m between the two sides of the wall. From the Figure 7, it can be clearly observed that there exists a surface concave at the left corner of the wall.

In the weir chamber, the water depth declines sharply with an average water depth down in 5.68 m. Due to the piers, there exists backwater in front of the pier, and the largest water level is higher than the average surface by 0.88m. At the end of pier, the phenomenon of water wing is significant due to the sudden expansion of water passage (Figure 8).



Figure 7: Free water surface in the diversion channel Figure 8: Water surface in the weir chamber

In the chute, there is also the lateral water surface fluctuation. While the hydraulic condition along chute is asymmetric, and there is a contraction cross-section in horizon, so the water surface in the chute is not flat. The water agitation between the left and the right makes the transverse wave in the chute.

From stake Y0+035.00 to Y0+057.00, the water surface fluctuation is relative large due to the influence of water wings. But the difference of water depth at cross-section is less than 0.3m. From stake Y0+060.00 to Y0+117.00, the water depth is getting larger for the cross-section contraction, and the difference of water depth at cross-section reaches 0.75m (Figure 9). From stake Y0+140.00 to Y0+220.00, the slope of the chute is getting from small slope to steep slope, and the difference of water depth at cross-section reaches 0.9m to 1.4m (Figure 10 and Figure 11).

In the trajectory bucket (Figure 12), the difference of water depth at cross-section is getting smaller, and at the end of the bucket the lateral distribution of water depth is almost getting uniform.



Figure 9: Water surfaces in chute from Y0+086.00 to Y0+127.20





a. water surface at right side







Figure 12: Water surfaces in the trajectory bucket

3.3 Flow Nappe Analysis

It can be observed from Figure 13 and Figure 14 that the placement of water nappe is on the slope and scattered to a large range. From Figure 15 to Figure 17, the maximum pressure value in the drop placement of lob flow is 12m water head. So, if the slope is designed to endure this pressure, there will be no question for the water dissipation.



Figure 13: Distribution of water scattered points in the slope



3.4 Analysis of Flow Patterns

Figure 18 and Figure 19 show the velocity distribution in the diversion channel. It can be observed that, the distribution of velocity is not uniform and there exists obviously a secondary flow. The velocity at the right side is low, and the left side velocity is high. The maximum lateral velocity in cross-section is 1.4m/s.



Figure 18: Velocity distribution at the end of the sharp turn of diversion channel (Y = 20)



Figure 19: Sectional velocity distribution at Y=20m

Figure 20 shows the pressure distribution on the wall boundary condition. The water pressure at the top of the hump weir can be mostly converted into the kinetic energy of water flow over weir, thus the pressure on the weir top is only 2m water head. At the tail of the pier there has shown a negative pressure of -3m water head.

Figure 21 shows there is no significant secondary flow in the weir chamber, and the flow pattern is good.



a. Front view

b. Back view

Figure 20: the pressure distribution at the weir bottom and the pier



Stream line in the left hole

Stream line in the weir chamber

Figure 21: The gate hole flow line distribution

Figure 22 shows the influence of the pier on the flow regime in the chute. With the distance from the piers getting long, the lateral flow velocity is gradually reduced, and the surface fluctuation is also gradually weakened. It is concluded that: the impact of water-wing is getting weakened along the way, and flow regime in the chute is gradually getting better. But the flow pattern immediately after the pier is not stable; to a certain extent the chute overflow capability will be influenced.



Figure 22: Velocity distribution of cross-section in some typical stake

4 CONCLUSIONS

- The standard $k \varepsilon$ turbulence model with the volume of fluid (VOF) method can be applied to simulate the whole spillway in a practical project. And the results can provide some detailed data for the design, to ensure the correctness of the design. It is convenient and economic against the physical model test.
- For the specific spillway design, the sharp turn of the diversion channel has little negative effect on the overflow discharge of the weir and the flow regime of the spillway. This conclusion simplifies the arrangement of the whole hydropower project.
- Although the sharp turn affects the uniform of the three weir chamber, the difference is not significant. And the stream-line, flow regime in the weir, even along the whole spillway is good.
- The trajectory bucket has a good role in the water energy dissipation. As the drop place of the nappe is scattered, and the shock wave pressure on the slope is too large. If the slope is designed specially, this energy dissipation measure is ok. In fact, during operation, the downstream water level will be high, so there will be a large plunge poll for the dissipation.
- It is necessary to further study the numerical simulation method, and to solve more and more problems in the practical projects.

REFERENCES

[1] CHEN Qun, DAI Guangqing. Three-dimensional numerical simulation of the stepped spillway overflow at the Yubeishan reservoir. *J. Hydroelectric Engineering*, **3**, pp. 62-72 (2002)

[2] SHA Haifei, ZHOU Hui, and WU Shiqiang et al. 3-D numerical simulation on discharge of spillway with muti-opening. *Water Resources and Hydropower Engineering*, **10**, pp. 42-46 (2005)

[3] LI Ling, CHEN Yongcan, LI Yonghong. Three-dimensional VOF model and its application to the water flow calculation in the spillway. *J. Hydroelectric Engineering*, **2**, pp. 83-87 (2007)

[4] Mashaye K F, Ashgriz N. Advection of axisymmetric interfaces by the volume-of-fluid method. *Int*. *J*. *Numer*.*Methods Fluids*, **20**, pp. 1337-1361 (1995)

[5] Hirt C W, Nichols B D. Volume of fluid (VOF) method for the dynamics of free boundaries. *J* . *Comput* . *Phys.*, **39**, pp.201-225 (1981)

[6] CHEN Qun. Numerical simulation and experimental study of stepped spillway. *Chengdu: Sichuan University Ph.D. thesis.* (2001)