Abstract

The desired outcome of flow characteristics from the model testing on spillway design process often cannot be measured on the specific situation, but it is possible to obtain much of this information through numerical modeling. The flow characteristics in a whole spillway has been simulated based on the VOF method multidimensional two-phase flow model and standard k-ε method by FLUNT software under the conditions of the checked flood level (650.39m) and the design flood level (653.36m). The numerical computation results of the surface elevation, pressure and flow velocity along the spillway in two schemes fit the experimental results well, and the difference of the average velocity between calculated and experiment results was less 6%. By model tests and numerical simulation, it found that the flow pattern and surface elevation of two holes scheme is complex caused by boundary conditions, but it is well after modifying to single hole. So the advanced CFD method is used to solve the design problems in a practical spillway design, and the calculation results can be used as the basis of the shape optimization.

© 2011 Published by Elsevier Ltd.

Keywords: spillway; numerical simulation; VOF; standard k-ε method; flow characteristic

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. So the spillway must be carefully designed to verify the flow characteristics. There is no adequate time and funds to do the hydraulic model test, so the numerical simulation is adopted to solve this problem.
With the development of numerical technique (CFD) and computer science, the 3-D numerical simulation of spillway flow has gotten 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 based on the VOF model by FLUENT software, to analyze 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[4,5,6]. The entire calculation domain is mainly related to two types of fluid, water and air. And the standard $k$-$\varepsilon$ turbulence model\(^6\) is used in the three-dimensional numerical simulation of spillway flow.

2.1. Basic Equation

The following is the basic equations:

Continuity equation:

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

Momentum equation:

$$\frac{\partial (\rho u_j)}{\partial t} + \frac{\partial (\rho u_j u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_j} + \frac{\partial}{\partial x_j}[(\mu + \mu_t)(\frac{\partial u_j}{\partial x_j} + \frac{\partial u_i}{\partial x_i})]$$  \hspace{1cm} (2)

$k$ equation:

$$\frac{\partial (\rho k)}{\partial t} + \frac{\partial (\rho u_j k)}{\partial x_j} = \frac{\partial}{\partial x_j}[(\mu + \mu_t)\frac{\partial k}{\partial x_j}] + G - \rho \varepsilon$$  \hspace{1cm} (3)

$\varepsilon$ equation:

$$\frac{\partial (\rho \varepsilon)}{\partial t} + \frac{\partial (\rho u_j \varepsilon)}{\partial x_j} = \frac{\partial}{\partial x_j}[(\mu + \mu_t)\frac{\partial \varepsilon}{\partial x_j}] + C_1 \frac{\varepsilon}{k} G - C_2 \frac{\varepsilon^3}{k}$$  \hspace{1cm} (4)

Where, \( t \) is the time; \( u_j \) is the velocity components; \( x_i \) is the coordinate components; \( \rho \) is the density; \( \mu \) is the molecular viscosity coefficient; \( P \) is the correct pressure; \( \mu_t \) is the turbulent viscosity coefficient, which can be derived from the turbulent kinetic energy \( k \) and turbulent dissipation rates \( \varepsilon; \mu_t = \rho C_1 k^2 / \varepsilon \).

Among them, the empirical constants, \( \sigma_k = 1.0, \sigma_\varepsilon = 1.3, C_1 = 1.44, C_2 = 1.92 \); and the turbulent kinetic energy source term \( G \) caused by the average velocity gradient can be defined by the following formula:

$$G = \mu_t (\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i})$$  \hspace{1cm} (5)

VOF 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: \( \alpha_w + \alpha_a = 1 \). Water-air interface can be determined by solving the following continuity equation:

$$\frac{\partial \alpha_i}{\partial t} + \mu_t \frac{\partial \alpha_i}{\partial x_i} = 0 \hspace{1cm} \text{where,} \hspace{1cm} \alpha_w \text{ is the fraction of water volume;} \hspace{1cm} \alpha_a \text{ is the fraction of air volume.}$$

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 pressure-velocity 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. The length of diversion channel is 71.7m with trapezoidal cross-section, and the wall is slope of 60°. The bottom elevation of the diversion channel is 642.0m with width 17.5m. Overflow weir, practical weir of hump type is adopted, weir height is 2.5m. There are two holes with width of 7.5m for each, the pier with thickness of 2.5m, the length of control section is 25.0m, and the top elevation is 657.00m. The total horizontal length of sluice channel is 250m. There are two sections, one with longitudinal slope \( i = 0.067 \) (horizontal length is 120m), the other one with longitudinal slope \( i = 0.2 \) (horizontal length of 130m). At the start of the small slope channel, there is the contraction section in plane (width from 17.5m to 10m, length is 50m). At the end of the steep channel, there is the trajectory bucket (horizontal length of 14.5m, the anti-arc radius is 22m, bucket top elevation is 607.49m, and lob angle is 39°) for energy dissipation.

Most domains are meshed in hexahedral grid, In order to ensure sufficient accuracy and stability requirements of VOF method, the unstructured hexahedral meshes are adopted in the control section. The calculated grid shows in Figure 1.

![Fig.1. The calculated grid of spillway (two holes scheme)](image)

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 the turbulent kinetic energy \( k \) and the dissipation rate \( \varepsilon \) can be calculated according to empirical formula. 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

By three-dimensional numerical calculation, the spillway surface elevation, pressure, flow velocity field and other flow characteristics obtained under the condition of the checked flood level (650.39m) and the design flood level (653.36m), are verified by the model tests. By model tests and numerical simulation, it found that flow pattern of original design with two holes is complex caused by boundary conditions, so the control section is modified to single-hole, and the width changes 14m, contraction section width changes 10m from 14m.
3.1. Surface Elevation

The surface elevation along the spillway calculated by the numerical simulation and measured by model test in the checked flood level and the design flood level conditions with two schemes are shown in figure 2. It can be seen that the simulated values of pressure fit the experimental results well. The flow pattern of the single hole scheme is better than the two holes scheme because of the effect of pier and bigger deflected angle in two holes scheme. Peak flow appears alternately in stake 0+075 m at some times.

![Fig.2. Comparison of computation and measurement of surface elevation](image)

3.2. Bottom Pressure

Figure 3 shows the pressure profiles on bottom along the spillway. It can be seen that most of the results agreed well with those from the experiment. But the difference was large in contraction section. The pressure is minimum near the top of weir and the stake 0+145m. There are negative pressures near the top of weir. The negative pressure has importance for the assessment of cavitations potential. In fact, the top of weir will be possibly destroyed because of the negative pressure. In laboratory, it is very difficult to measure the negative pressure without specific equipment. However, the numerical method in this paper can be easily used to simulate the negative pressure distribution.

![Fig.3 Comparison of computation and measurement of pressure on the bottom](image)

3.3. Cross-Section Velocity

The velocities were measured by Pitot tube in model test in some cross-sections. The average velocity
calculated by numerical simulation and measured by model test shows in Table 1. It shows that the velocity increase little with the water level increases. And the calculated value match well with experiment results because the difference was less 6%. The maximum velocity of downstream spillway is about 27.64m/s in checked flood level condition and 26.43m/s in design flood level condition. The results indicate that the change of velocity on the discharge capacity of two conditions can be negligible.

Table 1. Comparison of computation and measurement of velocity

<table>
<thead>
<tr>
<th>Stake</th>
<th>Design flood level</th>
<th>Checked flood level</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Experiment (m/s)</td>
<td>CFD (m/s)</td>
</tr>
<tr>
<td>0+025</td>
<td>11.46</td>
<td>11.26</td>
</tr>
<tr>
<td>0+055</td>
<td>12.63</td>
<td>12.50</td>
</tr>
<tr>
<td>0+085</td>
<td>13.10</td>
<td>12.95</td>
</tr>
<tr>
<td>0+145</td>
<td>15.87</td>
<td>16.44</td>
</tr>
<tr>
<td>0+175</td>
<td>19.90</td>
<td>18.99</td>
</tr>
<tr>
<td>0+205</td>
<td>22.40</td>
<td>22.04</td>
</tr>
<tr>
<td>0+235</td>
<td>25.24</td>
<td>24.10</td>
</tr>
<tr>
<td>0+265</td>
<td>26.43</td>
<td>25.96</td>
</tr>
</tbody>
</table>

4. Conclusion

The standard $k-\varepsilon$ turbulence model with VOF method can be applied to simulate the whole spillway in a practical project. The surface elevation, pressure distribution and velocity field are obtained to compare with the model test data. And the result shows there match well.

The flow pattern of two schemes were compared, the results show that the single hole scheme is better than the two holes scheme because of the affect of pier and bigger deflected angle in two holes scheme.

The numerical simulation can accurately compute the spillway 3-D flow field in different boundary conditions, 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.

References


