Establishment and application of three-dimensional realistic river terrain in the numerical modeling of flow over spillways

Lan Qi, Hui Chen, Xiao Wang, Wencai Fei and Donghai Liu

ABSTRACT

We present an integrated three-dimensional (3D) spillway model where the realistic and complicated river terrain is implemented by the platform CATIA (Computer Aided Three Dimensional Interactive Application). This integrated 3D spillway model allows for complicated topographic and geomorphic conditions and describes the spatial distribution of the spillway dam (upstream reservoir, downstream river channel and the spillway dam itself) precisely, thus making it a real alternative to the physical model. Furthermore, this model provides the premise and possibility of a full-scale simulation of the spillway flow, that is, it can not only be used to study the hydraulics on the spillway face, but also can be used to study the hydraulics along the downstream river channel and estimate the scour problem associated with both the spillway flow and downstream river channel. In this model, turbulence was simulated using RNG $k−\varepsilon$ equations. The flow velocity and surface pressure from the numerical model were verified by the data from experiments. Moreover, the river flow was studied and flow velocities downstream were obtained. The scour formed downstream of a ski-jump was also studied in this study on the location and shape of a scour hole. In all, this study provides new approaches for solving relevant hydraulic engineering problems.

Key words | 3D numerical simulation, realistic river terrain, scour depth, spillway dams flow, turbulent model

INTRODUCTION

The spillway dam is both a water-retaining and discharge structure, which requires a sufficient discharge capacity to ensure smooth flow through the dam surface (Kermani et al. 2015). The vibration and cavitation of the dam must be avoided. In addition, for the safety of the dam, the river-bed scour by the discharge flow must be controlled. When planning and designing a spillway dam in practice, there are two methods without exception: one is the physical model method, the traditional method, and the other is the mathematical model method (Zhang et al. 2016). Computational fluid dynamics (CFD), which is a type of mathematical model and able to solve diverse fluid flow problems cost-effectively, has captured considerable attention of many scholars and has been increasingly applied in a wide range of hydraulic research and applications (Chanel & Doering 2008; Kirkgoz et al. 2009; Rostami et al. 2012).

There is plenty of literature about the flow over spillway dams using different CFD approaches. For example, Kim & Park (2005) and Savage & Johnson (2001) carried out numerical simulation for the flow over an ogee spillway using Flow-3D software. Chatila & Tabbara (2004) and Tabbara et al. (2005) used ADINA to model the flow over a spillway. More recently, new approaches have also been applied to simulate flows over a spillway, such as smoothed particle
hydrodynamics (Husain et al. 2014). However, these studies focused on two-dimensional (2D) models and failed to capture the three-dimensional (3D) flow structures. Dargahi (2006), Mu et al. (2012) and Rahimzadeh et al. (2012) simulated the 3D flows over a spillway using Fluent software. Actually, the downstream scour created by water jets also plays an essential role in a spillway dam design, since excessive local scour may progressively threaten the dam’s safety. De Cesare et al. (2011) and Epeley-Chauvin et al. (2015) simulated the 3D scouring process generated by turbulent water jets using Flow 3D software.

These stated models have achieved significant developments in theories and computational methods of spillway flow and greatly improved the efficiency and accuracy of the simulation. However, most studies centered on the flow characteristics over the separate spillway and neglected the hydraulics and scour problems in the downstream river channel, which have been proved to be very important for dam stability and river slope protection. This neglect could greatly simplify the model. Furthermore, few studies consider both the surface flows over the spillway dam and the local scour induced by water jets. With more and more 3D digital modeling developed, a 3D visualization model of hydraulic engineering has become the research hotspot. Motivated by these, in the current research, we present a 3D spillway model with realistic river terrain which was performed using an available 3D modeling software (CATIA). The 3D flow structure and downstream scour were then studied using Flow 3D software.

The main contributions of this work are as follows:

1. As the precision of models is a central issue of modeling, the realistic and complicated river terrain was implemented and integrated in a 3D spillway model for the numerical simulation of turbulent flow and the whole modeling process was presented in detail. This spillway model considering river terrain enables the actual project to be described well, including the upstream reservoir, downstream river channel and the spillway dam itself.

2. This model provides the premise and possibility of a full-scale simulation of the spillway flow. Not only hydraulic parameters on the spillway face were studied, but also the river flow characteristics and the estimation of scour below the spillway dam were analyzed. The latter fully considered the effect of the released flow on the river channel downstream; as a result, the simulation results are of greater guiding significance and this study is more valuable in engineering application.

3. The simulation results were compared with that of the physical model and satisfactory agreement demonstrates the model’s reliability.

METHODOLOGY

Turbulent model

The RNG $k - \varepsilon$ turbulence model has known advantages when the flow has a high strain ratio and large streamline curvature and has been proved to be better suited to the detailed separated areas than the standard $k - \varepsilon$ model (Hargreaves et al. 2007; Rahimzadeh et al. 2012). This is why we chose this model in the study. The governing equations include the continuity equation, the momentum equations, and the $k$ and $\varepsilon$ equations:

$$\nabla (AU) = 0$$

$$\frac{\partial U}{\partial t} + \frac{1}{V_F} U \cdot \nabla (AU) = -\frac{1}{\rho} \nabla P + F$$

$$\frac{\partial k}{\partial t} + \frac{1}{V_F} U \cdot \nabla (Ak) = P_T + G_T + Diff_T - \varepsilon$$

$$\frac{\partial \varepsilon}{\partial t} + \frac{1}{V_F} U \cdot \nabla (A\varepsilon) = \frac{CDIS1 \cdot \varepsilon}{k} (P_T + CDIS3 \cdot G_T) + Diff - CDIS2 \frac{\varepsilon^2}{k}$$

where, $U$ is the fluid velocity; $A$ and $V_F$ are the area and volume fractional, respectively; $\rho$ is the fluid density; $P$ is the fluid pressure; $F$ includes body and viscous forces. The term $k$ is the turbulent energy; $G_T$ is the turbulence production by fluid flotation; $P_T$ is the turbulence produced by the velocity gradient; $\varepsilon$ is the turbulence dissipation rate; $Diff_T$ and $Diff$ are the turbulence diffusion terms; $CDIS1$, $CDIS2$ and $CDIS3$ are experience coefficients without dimension.
Sediment scour model

This model enables simulation of the behavior of sediment particles and estimation of sediment erosion, advection and deposition. The momentum equation used to express the sediment motion and fluid-sediment mixture behavior can be expressed as (Flow Science Inc. 2012):

\[
\frac{\partial U_{s,i}}{\partial t} + \mathbf{U} \cdot \nabla U_{s,i} = - \frac{1}{\rho_{s,i}} \nabla P + \frac{K_{i}}{\bar{f}_{s,i} \rho_{s,i}} U_{t,i} \tag{5}
\]

\[
\frac{\partial \mathbf{U}}{\partial t} + \mathbf{U} \cdot \nabla \mathbf{U} = - \frac{1}{\bar{\rho}} \nabla P + \mathbf{F} \tag{6}
\]

where, \( U_{s,i} \) is the velocity of sediment species \( i \); \( \rho_{s,i} \) is the sediment density; \( f_{s,i} \) is the volume fraction of sediment species \( i \), i.e. the ratio of sediment volume to total volume; \( K_{i} \) is the drag function; \( U_{t,i} \) and \( \mathbf{U} \) are the relative velocity and the mean velocity respectively, which can be written as:

\[
U_{t,i} = U_{s,i} - U \tag{7}
\]

\[
\mathbf{U} = \left( 1 - \sum_{j=1}^{N} f_{s,j} \right) U + \sum_{j=1}^{N} f_{s,j} U_{s,j} \tag{8}
\]

where, \( N \) is the total number of sediment species. Subtracting Equation (6) from Equation (5) gives:

\[
\frac{\partial U_{\text{drift},i}}{\partial t} + \mathbf{U} \cdot \nabla U_{\text{drift},i} = \left( \frac{1}{\bar{\rho}} - \frac{1}{\rho_{s,i}} \right) \nabla P - \frac{K_{i}}{\bar{f}_{s,i} \rho_{s,i}} U_{t,i} \tag{9}
\]

where, \( U_{\text{drift},i} = U_{s,i} - \mathbf{U} \) is the drift velocity, which can be solved simultaneously by Equations (7) and (8):

\[
U_{\text{drift},i} = (1 - f_{s,i}) U_{t,i} - \sum_{j=1}^{N} f_{s,j} U_{t,j} \tag{10}
\]

And the mixture density \( \bar{\rho} \) is

\[
\bar{\rho} = \sum_{i=1}^{N} f_{s,i} \rho_{s,i} + \left( 1 - \sum_{i=1}^{N} f_{s,i} \right) \rho \tag{11}
\]

Assuming that the motion of the sediment is nearly steady at the scale of the computational time and neglecting the advection term in Equation (9), we have

\[
U_{t,i} = \frac{\nabla P}{\bar{\rho} K_{i} (\rho_{s,i} - \bar{\rho})} f_{s,i} \tag{12}
\]

For most problems, the ratio of pressure gradient to mixture density is typically treated as equal to the acceleration of gravity \( g \), then Equation (12) can be simplified as:

\[
U_{t,i} = \frac{g}{K_{i}} (\rho_{s,i} - \bar{\rho}) f_{s,i} \tag{13}
\]

A reliable formula for the drag function \( K_{i} \) is

\[
K_{i} = \frac{3}{4} f_{s,i} \left( C_{D,i} |U_{t,i}| + 24 \frac{\mu}{\rho d_{s,i}} \right) \tag{14}
\]

where, \( d_{s,i} \) and \( C_{D,i} \) are the diameter and the drag coefficient of sediment species \( i \), respectively; \( \mu \) is the fluid viscosity.

Finally, the drift velocity is computed from Equations (10), (13) and (14).

The entrainment lift velocity of sediment is then computed based on an empirical model proposed by Mastbergen & Van den Berg (2003):

\[
U_{\text{lift},i} = a_{i} n_{s} d_{s,i}^{0.5} (\theta_{i} - \theta_{cr,i})^{1.5} \sqrt{\frac{||g|| d_{s,i} (\rho_{s,i} - \bar{\rho})}{\rho}} \tag{15}
\]

where, \( a_{i} \) is the entrainment parameter, whose recommended value is 0.018 (Mastbergen & Van den Berg 2003); \( n_{s} \) is the outward pointing normal to the packed bed interface; \( ||g|| \) is the magnitude of the gravitational vector; \( d_{s,i} \) is the particle diameter of sediment species \( i \); \( \theta_{cr,i} \) is the critical Shields parameter; \( d_{s} \) is the dimensionless mean particle diameter and \( \theta_{i} \) is the local Shields number:

\[
d_{s} = d_{s,50} \left[ \frac{\rho (\rho_{s,i} - \bar{\rho}) ||g||}{\mu^{2}} \right]^{1/3} \tag{16}
\]

\[
\theta_{i} = \frac{\tau}{||g|| d_{s,i} (\rho_{s,i} - \bar{\rho})} \tag{17}
\]
where, $\mu$ is the viscosity; $\tau$ is the local shear stress. The critical Shields parameter $\theta_{cr,i}$ is computed based on the Shield-Rouse equation (Guo 2002):

$$\theta_{cr,i} = \frac{0.1}{R_i^{(2/3)}} + 0.054 \left[ 1 - \exp\left(-\frac{R_i^{0.52}}{10}\right) \right]$$

(18)

where, $R_i$ is the dimensionless parameter and can be obtained from:

$$R_i = \frac{\sqrt{0.1(\rho_{s,i} - \rho)g|\psi|d_{50}}}{\mu}$$

(19)

The critical Shields parameter is then modified by the effects of armoring (Kleinhans 2002) and further modified for sloping surfaces to include the angle of repose:

$$\theta'_{cr,i} = \frac{1.666667}{\log_{10}(19(d_{50,i}/d_{50}))^2}$$

(20)

$$\theta'_{cr,i} = \frac{\cos \psi \sin \beta + \sqrt{\cos^2 \beta \tan^2 \varphi_i - \sin^2 \psi \sin^2 \beta}}{\tan \varphi_i}$$

(21)

where, $\beta$ is the computed angle of the packed interface normal relative to the gravitational vector; $\varphi_i$ is the user-defined angle of repose for sediment species $i$ and $\psi$ is the angle between the flow and the upslope direction. $d_{50}$ is the local mean particle size.

**3D MATHEMATICAL MODEL WITH REALISTIC TERRAIN**

**Generation of 3D spillway dam with realistic river terrain**

In this work, flow over Yujiahe spillway dam, located at Enshi City in China, was studied, the 3D spillway modeling was generated using CATIA, and the main procedures were as follows:

1. Convert the topographic contour data into point cloud data, which can then be imported into CATIA and modified by filtering and deleting the wrong points (Figure 1(a)).
2. Mesh on the surface ground and the shaped surface can be generated, as is shown in Figure 1(b) and 1(c), respectively.
3. Turn into the Component Design Module and draw lines along the boundary of the generated shaped surface, then the geologic model is integrally generated after stretching and segmentation, as is shown in Figure 1(d). The 3D river terrain is finally modeled as shown in Figure 1(e).
4. The 3D spillway dam was built by importing the 2D AutoCAD drawing of the spillway dam into Sketcher, thus the modeled 3D spillway model integrated with 3D river terrain and 3D dam is shown in Figure 1(f).

This modeling method describes the actual situation furthest, thus making the numerical model a real alternative to the physical model shown in Figure 1(h). The final 3D spillway dam is shown in Figure 1(g), where the red region was excavated for the subsequent erosion analysis.

**Grid sensitivity analysis**

As we all know, the simulation results are sensitive to the grid quality. Therefore, the grid sizes that meet the demands of calculation precision and efficiency should be decided properly. Motivated by this, the sensitivity analysis of grid size and quality was studied (Savage & Johnson 2001; Aydin & Ozturk 2009; Savage et al. 2016).

The physical model of this spillway was built in Wuhan University with a scale of 1:60. It consists of the upstream, with a length of 166.7 cm (equal to 100 m of the prototype), the whole spillway dam structure, and the downstream of the dam axis with a length of 433.3 cm (equal to 260 m of the prototype). Therefore, the entire model is spatially discretized in three blocks with regular cubic grids (blocks 1, 2 and 3), as shown in Figure 2(g). As block 2 contains the overflow dam section and the flow structure in this region is complicated by strong turbulence, the sensitivity analysis mainly focuses on grid size in this region. The sensitivity analysis was conducted by five meshes with different cell sizes of 3.3 cm, 2.6 cm, 2.0 cm, 1.5 cm and 1.2 cm.
Flow3D utilizes a porosity technique of the FAVOR (Fractional Area/Volume Obstacle Representation) method to model complex geometric regions. This method is used to calculate the fluid fraction in each cell by the portion of volume or area occupied by the obstacles, thus describing the geometry by grids without any distortion. It is a quick and effective way to judge how well a computational grid resolves the geometry. The spillway models built at different grid sizes are presented in Figure 2(b)–2(f). Figure 2(b) shows the spillway model at a grid size of 3.3 cm, the main structure of the spillway is built; however, some detailed features are still not presented. With the reduction of grid size, the spillway model becomes much more detailed with a smoother surface. The spillway models in Figure 2(e) and 2(f) are basically consistent with the CATIA geometric model (Figure 2(a)) when the grid size is reduced to 1.5 cm or 1.2 cm. This great fitness indicates that grid quality at a grid size of 1.5 cm has satisfied the requirement of precision.

In order to further check for grid convergence, numerical simulation schemes of the above spillway models discretized at different grid sizes were conducted and typical hydraulic parameters (flow regimes, velocities and pressures) were obtained to perform the following comparisons. The flow volume into the upstream boundary was set as Q of 59.9 L/s (equal to 1,669 m^3/s of the prototype, i.e., check flood) and a water elevation of 8.303 m (equal to 498.15 m of the prototype) was specified at the downstream boundary.

Figure 3 shows the flow regimes of the spillway models respectively. By comparison, flow tends to be more continuous and smooth with the grid size reducing. Flow along the spillway model gridded at 3.3 cm occurs with distortion with flow break. This discontinuous phenomenon disappears until the grid size is reduced to 2.0 cm, while flow is not smooth. When the grid size was further reduced to 1.5 cm, flow was not only continuous but also smooth. However,
when the grid size was further reduced to 1.2 cm, the flow regime had no significant difference to that of the spillway model gridded at 1.5 cm.

Based on the research requirement, eight measuring points on the spillway dam surface were designed for the experiment, and the corresponding arrangement is shown in Figure 4(a). As measuring point 1 is located at the inlet and the maximum velocity occurs at measuring point 8, these two measuring points are selected to assess the sensitivity of grid sizes to velocities. Similarly, measuring point 1 and measuring point 7, with the maximum pressure, are selected to assess the sensitivity of grid sizes to pressures. Figure 4(b) and 4(c) plot the velocity and pressure as a function of grid size. It can be seen that both curves of velocity and pressure change obviously when the grid size ranges from 3.3 cm to 2.0 cm, and both parameters reach steady levels without significant change when the grid size is reduced to 1.5 cm.

As discussed above, when the grid size is reduced to 1.5 cm and 1.2 cm, the model reconstruction agrees well with the actual project and satisfactory simulation results are achieved. The computation time would grow considerably when the grid size is reduced from 1.5 cm to 1.2 cm. Therefore, the grid size of the whole model was decided as 1.5 cm and the total quantity of grid elements in the domain is about 8 million. According to the physical model, three simulation schemes were performed and one (flow discharge of 59.9 L/s and downstream water elevation of 8.303 m) was taken as the example to be presented here.

The flow volume into the upstream boundary is set as $Q$ of 59.9 L/s and water elevation of 8.303 m is specified at the
downstream boundary. The lateral and top boundaries are set as symmetric and the bottom boundary is defined as a solid wall. For the roughness of the solid boundary, Flow-3D uses Nikuradse sand-grain-equivalent roughness length instead of Manning’s roughness coefficient; 1.25 mm is chosen as the equivalent roughness length of the spillway and that of the river channel with gravel bed is taken as 6 mm. In the initial condition, the flow velocity is set to be zero everywhere and the water level in the entire domain is the same as specified at the downstream boundary. The pressure variation in the vertical direction satisfies the hydrostatic condition.

**RESULTS AND DISCUSSION**

**Pressure on spillway dam**

Table 1 lists the pressure variations obtained from experiment and simulation. A majority of the numerical
results match the experiment results and the discrepancies are all within 10%. Part of the discrepancy may be from the measurement error because of the violent turbulent flow and complex pressure distribution. The minimum pressure occurs near the top of the weir (points 1 and 2) and the maximum is near the bottom of the weir (point 7). Despite negative pressures at point 1 and point 2 being obtained, they are within the specified range of 1,000 Pa (equal to 0.06 MPa of the prototype) when the gate is fully open under the condition of check flood level.

Velocity distribution

Table 1 also summarizes the simulated and measured velocities at the measuring points. It can be seen that the velocity increases from the weir crest to the flip bucket and reaches the maximum at the end of the flip
bucket due to gravity. Both of the simulated and the measured results present this tendency and the differences between the two are not significant, less than \( \pm 5.1\% \). These demonstrate the numerical model is valid in the test case.

### Estimation of scour below spillway dam and downstream river velocity distribution

The domain for the 3D scour simulation is the red region in Figure 1(g). The median particle diameter is set to 2.0 cm; the repose angle is 33°; and the critical Shields number is 0.042.

Figure 5(a) and 5(b) show the 3D form of the riverbed impinged by water jets. The original riverbed was changed with varying degrees of scour due to the turbulent flow. The 3D shapes of the scour hole are diverse, with a plane shape of irregular ellipse or half oval. As can be seen from Figure 5(a), the location of the scour hole tends to the left bank of the river. This tendency is due to the water jets impinging on the river bed closely to the left bank. Therefore, proper protection measures should be taken to strengthen the anti-scouring ability of the left side slope in practice. According to statistics of the maximum scour depth, the simulated values were 26.2 cm. With regard to the maximum scour depth comparison to the measured values of 26.0 cm, there are no significant differences between the simulated and the measured results. As far as the scour depth is concerned, the simulated results are basically consistent with the experiments. Moreover, five cross-sections were set for measuring flow velocity and the corresponding arrangement is shown in Figure 5(c), where both the simulated and the measured velocities are also labeled with values and the blue arrows show the flow direction. It can be seen that the simulated values agreed well with the measured values. This satisfactory agreement demonstrates the accuracy and reliability of the model. In addition, we can find that the velocity increased when flow passed through the river bend (Section 3), especially at the left bank, where the velocity reached 1.2 m/s which equates to more than 9 m/s of the prototype model. This increasing flow velocity may result in side slope erosion, thus threatening the slope stability. Therefore, these results provide a scientific basis for protecting slopes from streams eroding.

### CONCLUSIONS

(1) The realistic and complicated river terrain was implemented and integrated in the 3D Yujiahe spillway model based on CATIA. This integrated modeling enables description of the interactions between the spillway flow and the river terrain in the simulation, thus a full-scale simulation can be performed over the whole process of water discharge from upstream reservoir to downstream river channel. What’s more, informative and satisfactory results make the model calibration more efficient and accurate, and provide a powerful modeling approach for relevant research. Grid sensitivity to the simulation results was conducted and thus the conclusion is drawn that grid...
quality and size do have great impact on the results, and a grid size of 1.5 cm was decided on in our study.

(2) The model performance was demonstrated by comparing the simulation results with measurements, in the aspects of velocity and pressure distribution. The model results agree well with the measurements. These satisfactory agreements also suggest that the method presented in this paper, a smooth integration between CATIA and Flow3D, achieves an efficient coupling for 3D numerical simulation of spillway flow.

(3) The estimation of scour below the spillway dam has also been performed to validate the model performance. The 3D shape of the scour hole and the maximum scour depth are all in good agreement with available experimental data. Moreover, the velocity distribution along the river channel downstream was obtained and can be used for side slope protection.

ACKNOWLEDGEMENTS

The authors gratefully acknowledge the support of the State Key Laboratory of Hydraulic Engineering Simulation and Safety (Tianjin University) and the State Key Laboratory of
Water Resources and Hydropower Engineering (Wuhan University). In addition, this work was supported by the National Natural Science Foundation of China under Grant No. 51479132.

REFERENCES


First received 12 December 2016; accepted in revised form 16 May 2017. Available online 31 May 2017.