Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model

M R BHAJANTRI^{1,2}, T I ELDHO^{1,*} and P B DEOLALIKAR²

¹Department of Civil Engineering, Indian Institute of Technology – Bombay, Mumbai 400 076, India

²Central Water & Power Research Station, Khadakwasla 411 024, Pune, India
e-mail: eldho@civil.iitb.ac.in

Abstract. Spillway flow, a classical problem of hydraulics, is generally a gravity-driven free surface flow. Spillway flows are essentially rapidly varying flows near the crest with pronounced curvature of the streamlines in the vertical direction. Two processes simultaneously occur in the flow over the crest, that is, formation and gradual thickening of the turbulent boundary layer along the profile, and gradual increase in the velocity and decrease in the depth of main flow. Spillway hydrodynamics can be obtained through physical modelling or numerical modelling. Physical modelling of spillways is expensive, cumbersome and time-consuming. The main difficulties in solving the spillway problem numerically are: rapidly varying flow, existence of both subcritical and supercritical flows, development of turbulent boundary layers, unknown free surface and air entrainment. Numerical simulation of such flows over spillways in all flow regimes is a challenging task. This paper describes a numerical model and its application to a case study to investigate the hydraulic characteristics of flow over spillway crest profiles by simulating the velocity distribution, pressure distribution and discharge characteristics. Results of the numerical modelling are compared with those from the physical modelling and found to be satisfactory.

Keywords. Spillway; hydrodynamics; numerical modelling; weakly compressible flow; free surface.

1. Introduction

The major driving force for construction of dams throughout the world is the need for reliable water supply, flood control, navigation, hydroelectric power generation and recreation. Water demand and consumption worldwide is expected to grow exponentially in the years to come. Hence, there is a need for technological interventions for harnessing, conservation and proper management of water resources. Spillways and other flood outlets are designed to safely

*For correspondence

A list of symbols is given at the end of the paper

convey floods to the watercourse downstream from the dam and to prevent overtopping of the dam. The selection and design of a particular type of spillway is based on the specific purpose of the project, hydrology, release requirements, topography, geology, dam safety and project economics. Provision of a hydraulically efficient and structurally strong spillway is very important for the safety of the dam, and life and property along the river down below. The Bureau of Indian Standards has published a code of practice (IS: 10137 1982) highlighting guidelines for the selection of spillways and energy dissipators. The Waterways Experiment Station (1973) and US Bureau of Reclamation (1977) have done extensive experimental work on hydraulic design of spillways.

Hydraulic models are used extensively to visualize and understand the complexity of hydraulic phenomena. Various hydraulic design aspects such as discharging capacity, pressures and water surface profiles and energy dissipation arrangement are considered, to evolve hydraulically efficient design of spillways. The width of the spillway plays a major role in the economic analysis and safety of the dam. Normally, a narrow spillway with higher discharge intensities would be less expensive than a wider spillway with moderate discharge intensities. This concept has resulted in large depths of overflow over the crest of the spillway. The major concern with dam safety worldwide is the provisions of adequate spillway flood capacity. Many dams were designed and built in the 1950s and 60s with limited hydrological information. Significant improvement made in the fields of meteorology and hydrology has updated the probable maximum outflow floods. As a result of these upward revisions, many dams have inadequate spillway capacities. Potential problems such as the generation of excessive negative pressure over a spillway crest under increased flood conditions could be encountered. Knowledge of spillway hydrodynamics is very important for safer design of spillways.

Spillway hydrodynamics can be obtained through physical modelling or numerical modelling. Physical modelling of spillways is expensive, cumbersome and time-consuming. The main difficulties in solving the spillway problem numerically are: rapidly varying flow, existence of both subcritical and supercritical flows, development of turbulent boundary layers, unknown free surface and air entrainment. In this paper, the hydrodynamics of spillway flow is discussed. With the help of a numerical model, an attempt is made to investigate hydraulic characteristics by simulating the velocity distribution, pressure distribution and discharge characteristics of a spillway.

2. Characteristics of flow over spillway

The conventional spillway, which is also called Ogee spillway, has four main parts; the upstream crest profile, downstream crest profile, the sloping face, and the energy dissipator at the toe. Upstream of the crest, the flow is subcritical (or gradually varying); the flow changes its state from subcritical to supercritical after the crest because the crest is followed by a steep sloping face. Spillway flows are essentially rapidly varying flows with pronounced curvature of the streamlines. Two processes simultaneously occur in the flow over the crest: formation and gradual thickening of the turbulent boundary layer along the profile and gradual increase in the velocity and decrease in the depth of main flow (figure 1).

At a certain point, called the point of inception, both the above thicknesses are equal and the flow is stated to be fully developed flow, and self-aeration of flow commences at this point. The point of inception, however, is realized for very small discharges for which the depths of flow are lower. Bulking of the flow due to air entrainment raises the free surface which is required to set the height of the sidewalls and to select the elevation of the trunnion axis for

Figure 1. Boundary layer development on the spillway face.

the radial gates or for any other structure in the vicinity of the water surface. Owing to the change of flow boundaries in a short distance, acceleration plays a dominant role in the flow as compared to shear resistance at the solid boundary. For spillway flow, vertical acceleration is significant, the flow velocity and pressure varies along the direction of flow as well as along the vertical direction.

3. Numerical modelling of flow over spillway

Physical models have been constructed in hydraulic laboratories to study the flow features of spillways but they are expensive and time-consuming. The physical model studies cannot handle many alternatives required to be studied simultaneously. The studies have to be conducted one after another for these modifications, which lead to very high costs and long durations of studies. Today, with the use of high-performance computers and more efficient CFD (computational fluid dynamics) codes, the behaviour of hydraulic structures can be investigated numerically in reasonable time and cost.

Since the last few decades, many researchers have tried to simulate spillway flows. Cassidy (1965) has calculated the coefficient of discharge and surface profile for flow over standard spillway profiles by using the relaxation technique in complex potential plane. Ikegawa & Washizu (1973) have studied spillway flow using the finite element method (FEM) by making considerable simplification of the basic problem. Most researchers have used either potential flow theory (Li *et al* 1989) or Reynolds-averaged Navier–Stokes (RANS) equations (Olsen & Kjellesvig 1998; Burgisser & Rutschmann 1999; Qun Chen *et al* 2002; Wei Wenli & Dai Huichao 2005) and available commercial codes (Bruce & Michael 2001; Jean & Mazen 2004). Unami *et al* (1999) developed depth-averaged 2-D numerical models of spillway flows using both FEM and FVM. Zhou & Bhajantri (1998) and Song & Zhou (1999) used space-averaged Navier–Stokes equations to develop 2-D and 3-D models respectively. Owing to the weak suitability of the finite difference method to curvilinear solid geometries, its application to the gravity-driven free surface flows with arbitrary curved solid boundaries is

strongly restricted. Meanwhile, finite volume method, finite element method and boundary element method that have excellent suitability to curved solid boundaries have been widely used. Many popular CFD codes use the finite volume method. While the finite difference and finite element methods start from the differential form of the governing equations, the finite volume method discretizes the Navier–Stokes equations directly in the integral form, ensuring the conservation of mass, momentum, and energy, both locally at the discrete cell level and globally over the entire flow domain. Conservation is important for capturing shocks and other flow discontinuities accurately in high-speed compressible flow simulations, and is the strength of the finite volume method. As both subcritical and super critical flows exist in spillway flow problems, numerical methods capable of capturing shock waves should be used.

Spillway flows are essentially rapidly varying flows near crests with pronounced curvature of the streamlines in the vertical direction. The flow over a spillway crest is characterized by the formation and gradual thickening of the turbulent boundary layer along the profile and gradual increase in accelerating flow followed by decrease in the depth of flow. Owing to the curvilinear nature of solid surface of spillway followed by steep slope downstream of crest, vertical acceleration plays a dominant role in the flow as compared to shear resistance at the solid boundary.

Most researchers have used either potential flow theory or RANS equations and available commercial codes. Very few people have used space-averaged Navier–Stokes equations. The comprehensive study of the flow over spillway requires a 3-dimensional model based on the complete form of Navier–Stokes equations. However the amount of computation required is probably still beyond the reach of present generation personal computers. However, a 2-dimensional (2-D) model provides good insight into the hydrodynamics of flow over a spillway for most design and analysis purposes. In the present study, a 2-D model is developed based on space-averaged weakly compressible flow equations for the hydrodynamics simulation of flow over a spillway using the finite volume method.

4. Mathematical formulation of flow over spillway

For modelling a flow that varies rapidly in the vertical direction and shows negligible flow variation in the lateral direction such as the flow over the spillway, we can use a 2-D vertical model. This model in its simplified form can be used as a width-averaged model, which is analogous to a physical 2-D sectional model.

4.1 Governing equations

Typical free surface flows in nature are of large Reynolds number and are characterized by large-scale turbulent mixing. These flows are of small Mach number and usually treated as incompressible flows. Under rapidly changing conditions, the first order of compressibility may not be negligible. In reality, fluids are compressible and the pressure is always related to density. Hence it is legitimate to use the compressible form of equations even for so called incompressible flow regimes (Chorin 1967). Compressibility is incorporated in order to make the problem more amenable to numerical solution. Song & Yuan (1988) developed the weakly compressible flow (also called compressible hydrodynamic) equations that are applicable and efficient for practical flows of small Mach number and large Reynolds number.

For general Mach number flows, including transient flows, the equation of continuity is (Song & Yuan 1988),

$$\frac{\partial p}{\partial t} + \rho_o a_o^2 \nabla \cdot \mathbf{V} = 0, \quad (1)$$

and the equations of motion can be written as,

$$\frac{\partial \mathbf{V}}{\partial t} + \mathbf{V} \cdot \nabla \mathbf{V} + \frac{1}{\rho_o} \nabla (p + g\rho_o y) = \frac{1}{\rho_o} \nabla \cdot \tau_{ij}, \quad (2)$$

where p = pressure, ρ_o = density of the fluid, a_o = speed of the sound, $\mathbf{V}$ = velocity vector.

In the above equation, τ_{ij} is the shear stress tensor, and gravity is assumed to act in the y -direction. The flow, which satisfies (1) and (2), is called weakly compressible flow. Neglecting the viscous terms in the equations of motion (2), the resulting equations of motion are called Euler equations.

$$\frac{\partial \mathbf{V}}{\partial t} + \mathbf{V} \cdot \nabla \mathbf{V} + \frac{1}{\rho_o} \nabla (p + g\rho_o y) = 0. \quad (3)$$

4.2 Boundary conditions of spillway flow

Boundary conditions specify the flow variables or their gradients on the boundaries of computational flow domain. The upstream boundary can be setup on a reservoir section at which the reservoir water level and the incoming discharge can be known. This section should be far away from the spillway to avoid the reflection effect. The boundary condition based on velocity distribution is:

$$u = u_o(x_o, y), \quad (4)$$

$$v = 0, \quad \frac{\partial p}{\partial x} = 0, \quad (5)$$

where, x is the flow direction and y is the vertical direction. u, v are the velocity components in x and y directions; u_o is the velocity component in the x direction at the upstream boundary. The value of u_o is determined by the given discharge and the water depth. The downstream boundary should be located based on the range of the interested domain. For study of the crest shape effect, the downstream condition has no effect on the upstream flow since the flow over the downstream slope of the spillway is supercritical. The downstream section can be chosen on a sloping section where the flow is fully developed so that zero gradients of velocity and pressure can be assumed.

There are three kinds of solid boundary conditions available: full-slip boundary condition, partial-slip boundary condition and no-slip boundary condition. Full-slip means that the tangential velocity at the inner grid = tangential velocity on the solid surface; while no-slip means tangential velocity on the solid surface = 0; for partial slip condition, a wall function should be used. Selection of the three alternative boundary conditions depends on the relative magnitude of the grid size and the boundary layer thickness. At the solid boundary of the spillway and reservoir, full-slip condition is used in the present case. There is no flow across solid boundaries.

The most difficult boundary to simulate is the time-varying free surface position. There are many factors affecting the free surface such as wind stress, the heat exchange between

water and air, the surface tension stress etc. Usually, free surface is simulated using kinematic and dynamic conditions. The kinematic condition is based on the idea that free surface is a material surface and has the form of,

$$\frac{\partial Z_f}{\partial t} + \mathbf{u} \bullet \nabla Z_f = v_f, \quad (6)$$

where Z_f is the free surface displacement along the normal direction; $\mathbf{u}$ is the velocity vector in the direction of flow and v_f is the free surface velocity in the vertical direction. This equation is solved using the Mac-Cormack scheme to modify the new free surface position during the iteration. The dynamic boundary condition, ignoring the surface tension effect, is the zero stress condition. For this model, the dynamic boundary condition, which is zero stress on the free surface, is simplified as follows:

$$\frac{\partial \mathbf{V}}{\partial \mathbf{n}} = 0, \quad (7)$$

where $\mathbf{n}$ represents the unit vector normal to the free surface.

5. Numerical solution

Inviscid weakly compressible flow equations can be written in a conservative form and as such they can be generalized through the use of an integral formulation. For the numerical solution of flow over spillway, a model is developed based on explicit finite volume (FVM) scheme. To apply a FVM scheme, it is convenient to first re-write the governing equations, (1) and (3), in a conservative form as follows:

$$\frac{\partial G}{\partial t} + \nabla \bullet \mathbf{F} = 0, \quad (8)$$

where G is the flow variable (p, u, v) and F is the flux vector.

$$\mathbf{F} = i E_1 + j E_2, \quad (9)$$

$$G = [p \ u \ v]^T \quad (10)$$

$$E_1 = \left[\rho_o a_o^2 u \quad u^2 + \frac{p}{\rho_o} \quad uv \right]^T, \quad (11)$$

$$E_2 = \left[\rho_o a_o^2 v \quad uv \quad v^2 + \frac{p}{\rho_o} \right]^T. \quad (12)$$

Equation (8) can be integrated over an arbitrary finite volume, and the volume integral changed to the surface integral by applying the divergence theorem. After averaging it over the volume we have,

$$\frac{\partial \bar{\mathbf{G}}}{\partial t} + \frac{1}{V} \iint_s \mathbf{n} \bullet \mathbf{F} ds = 0, \quad (13)$$

where, $\mathbf{G}$ is the volume-averaged value of G , V is the volume; $\mathbf{n}$ is the unit normal vector and s is the surface area of the control volume.

Equation (13) is solved by the MacCormack two-step explicit predictor – corrector scheme (MacCormack 1969). Since this is an explicit method, the computational time step has to be based on the numerical stability considerations. The stability of the computations is controlled by the Courant–Friedrichs–Lewy condition (Courant *et al* 1967):

$$\Delta t \leq \text{Min} \left\{ \frac{\text{Volume}}{(|u_i s_i| + a_0 |s_i|)} \right\}, \quad (14)$$

where, s is the surface area of the control volume. The minimum value of the time step Δt computed over the whole domain satisfying the conditions of equation (14) is taken as the computational time step. Non-dimensional numbers of Mach number and Courant number, corresponding to the values of 0.01 and 0.7 respectively, have been used in the present numerical case study.

5.1 Computational procedure

Based on the above formulation, a numerical model has been developed for the simulation of hydrodynamics of flow over a spillway. In the model developed, starting from the given initial conditions, the governing equations are solved for the velocity and pressure at all grid points for the next time step. The method of Thompson *et al* (1985) is used for the finite volume mesh generation. New free surface is computed and new mesh system is generated. According to the explicit time-marching method (MacCormack 1969), the flow at each time step is calculated until the steady state solution is reached.

6. Model application - Omkareshwar dam spillway

Here, as a case study, the hydrodynamics of the Omkareshwar dam spillway, constructed on the river Narmada in Madhya Pradesh, India is simulated using the developed numerical model. The numerical model results are compared with the physical model study results of the Omkareshwar dam spillway carried out at the Central Water Power Research Station, Pune. The design of the spillway structure has been optimized functionally and economically through physical model studies, keeping in view the techno-economic feasibility of the project. The scale of the physical model is chosen according to Froude's law keeping in view the availability of space, discharge and head.

The physical model studies for Omkareshwar dam spillway were conducted on a 2-D sectional model constructed to a geometrically similar scale of 1:50 at the Central Water and Power Research Station, Pune. The model was reproduced in a glass-sided flume so as to observe the flow conditions upstream and downstream of the spillway including the performance of the energy dissipator. One full span and two half spans on either side were reproduced. The entire model was coated with enamel paint so as to have a very smooth surface. 5 mm diameter piezometers were provided on the spillway surface along the centre of the span for measurement of pressure. 1.5 m wide sharp-crested Rehbock weir was used for measurement of discharge. Water levels were measured using pointer gauges of 0.1 mm least count. Hydraulic model studies were conducted for the following aspects as a part of client-sponsored applied research work: discharging capacity of the spillway; pressures and water surface profiles on the spillway surface and efficacy of the energy dissipation arrangement for entire range of discharges.

Figure 2. Single zone mesh system for the spillway.

6.1 Numerical modelling

Omkareshwar dam spillway being constructed on the river Narmada in Madhya Pradesh, consists of 23 spans, the width of each span being 20 m. The spillway is designed to pass the outflow flood of 88,315 m³/s. Parts of the prototype simulated in the numerical model include: part of the reservoir (58.61 m long and 39.144 m deep); upstream spillway crest profile (8.61 m long) with $y = [0.724(x + 8.61)^{1.85}/18.971] + 4.018 - 1.580(x + 8.61)^{0.625}$ and downstream Ogee crest profile (27.862 m long) conforming to $x^{1.85} = 21.85y$. Ogee crest profile of the spillway portion is considered up to downstream tangent point.

Single zone body-fitted mesh system was generated for the FVM modelling. As described earlier, the method of Thompson *et al* (1985) method is used for mesh generation. Figure 2 shows the mesh system. The flow domain is discretized into 18400 ($230 \times 80 = 18400$) quadrilateral cells.

6.2 Boundary and initial conditions

Numerical simulation was done for a discharge of 66236 m³/s, which is 75% of the design probable maximum flood of 88,315 m³/s. The corresponding discharge intensity works out to be 143.99 m³/s/m. Initially, a water depth of 17 m was assumed over the spillway crest. Remaining boundary conditions such as open boundaries at the upstream and downstream end, free surface and solid boundary were considered as described in the section on formulation of the free surface 2-D model.

6.3 Results and discussion

Figure 3 shows pressure distribution (in the form of contours) after convergence of the solution. The solution became stable after 6,00,000 time steps to simulate 3 minutes of real time. The simulated results were examined from time to time until a steady state solution was reached. The whole computation took about 12 hours on 1.0 GHz Pentium PC. The piezometric pressures and water surface profiles observed on the physical model were compared with the simulated pressures and water surface profiles. Good agreement was found between the calculated values and experimental data. Figure 4 shows the pressure distribution on the

Figure 3. Pressure distribution (in the form of contours).

solid surface after the solution has converged in comparison with the physical model results. It is quite evident from figures 3 and 4 that hydrostatic pressure prevails in the reservoir portion where the flow is slow and stable. Figure 5 shows the velocity distribution and observed water surface over the spillway crest. Figure 6 shows the streamline pattern.

The computed and experimental values of coefficient of discharge were 0.72 and 0.69. The calculated value of coefficient of discharge was higher than the observed value by 4%. The possible reason may be due to the omission of viscous terms and turbulence and assumption of well-guided straight flow and absence of losses due to end-contractions because of piers and abutments in the width-averaged 2-D numerical model. The numerical model indicated the location of the critical flow section near the spillway crest, where the non-dimensional hydraulic parameter, Froude number ($V/\sqrt{gd}$) was found to be unity, indicating change of flow regime from sub-critical to super-critical. As seen from figures 5 and 6, a mild separation zone was seen forming over the upstream crest profile in the numerical model. Figure 6 indicates that the parallel streamlines in the reservoir region become concentric as the flow approaches the spillway crest.

Figure 4. Pressure distribution on the spillway surface compared to the physical model results.

Figure 5. Velocity distribution for the flow over the spillway.

7. Concluding remarks

In this paper, a finite volume-based numerical model using weakly compressible flow equations has been presented to investigate the hydraulic characteristics of flow over spillway crest profile. The velocity distribution, pressure distribution and discharge characteristics of the chosen spillway were estimated and compared with existing physical model data. Reasonable agreement is observed with the numerical and physical model results, showing the applicability of the present model in the hydrodynamics simulation of real case study of spillway.

Figure 6. Streamline pattern for the flow over the spillway.

Following are the conclusions from the presented case study:

- The computed and experimental values of coefficient of discharge were 0.72 and 0.69, the computed value being 4% higher than the experimental value observed on physical model. The possible reasons for the higher computed value may be the omission of viscous terms and turbulence, which generally induce a damping effect in the flow, and the absence of losses due to end-contractions because of piers and abutments in the 2-D numerical model.
- As seen from the figures depicting pressure contours and streamlines, it is quite evident that hydrostatic pressure prevails in the reservoir portion and the spillway portion is subjected to non-hydrostatic pressure owing to rapidly varying accelerated flow.
- The numerical model indicated the location of the critical flow section (Froude number = 1) near the spillway crest, which agrees well with the established fact from the model studies.
- The upstream crest profile was not guiding the flow over the crest properly, as a result of which a mild separation zone was seen forming over the upstream crest profile in the numerical model.

The authors are grateful to Prof Charles C S Song, University of Minnesota, USA and Ms V M Bendre, Central Water and Power Research Station, Pune for facilities and training on development of the model described in this paper.

List of symbols

a_o	sound speed;
C	coefficient of discharge
d	depth of flow
E_1	flux at i surface;
E_2	flux at j surface;
F	sum of fluxes E_1 , and E_2 ;
g	acceleration due to gravity;
$\mathbf{G}$	vector expression of p , u , and v ;
H	head over spillway crest
$\mathbf{i}, \mathbf{j}$	unit vectors in x - and y - directions;
L	span width of spillway
$\hat{\mathbf{n}}$	unit vector normal to the free surface;
p	pressure;
Q	discharge passing over the spillway
s	surface area of the control volume;
s_i	surface area of the control volume at the i th grid cell
t	time;
u, v	velocity components in x and y directions;
u_o	velocity component in x direction at the upstream boundary;
u_i	velocity component in x direction at the i th grid cell;
V	volume of computational grid cell;
$\mathbf{V}$	velocity vector;
V_f	free surface velocity in the vertical direction;

x, y	Cartesian coordinates in two dimensions;
x_o	value of x at the upstream boundary;
Z_f	free surface elevation along the normal direction;
ρ_o	density of fluid;
Δt	time step;
τ_{ij}	shear stress tensor.

References

- Bruce M S, Michael C J 2001 Flow over Ogee spillway; physical and numerical model case study. *J. Hydrol. Eng. ASCE* 127: 640–649
- Burgisser M F, Rutschmann P 1999 Numerical solution of viscous 2-D vertical free surface flows: Flow over spillway crests. *Proc. 28th IAHR Congress*, Technical University, Graz, Austria
- Cassidy J J 1965 Irrotational flow over spillways of finite height. *J. Eng. Mech. Div., ASCE* 91(6): 155–173
- Chorin A T 1967 A numerical method for solving incompressible viscous flow problem. *J. Comput. Phy.* 2: 12–26
- Courant R, Friedrichs K O, Lewy H 1967 On the partial differential equations of mathematical physics. *IBM J. Res. Dev.* 11: 215–234
- Ikegawa M, Washizu K 1973 Finite element method applied to analysis of flow over a spillway crest. *J. Numer. Methods Eng.* 6: 179–189
- IS: 10137-1982 The guidelines for the selection of spillways and energy dissipators. Bureau of Indian Standards, New Delhi
- Jean C, Mazen T 2004 Computational modelling of flow over an Ogee spillway. *J. Comput. Struct.* 82: 1805–1812
- Li W, Xie Q, Chen C J 1989. Finite analytical solution of flow over spillway. *J. Eng. Mech. ASCE* 115: 2635–2647
- MacCormack R W 1969 Effect of viscosity in hypervelocity impact cratering. *AIAA paper*, 69–354
- Olsen N R, Kjellsvig H M 1998 3-D numerical flow modelling for estimation of spillway capacity. *J. Hydrol. Res.* 36: 775–784
- Qun Chen, Guangqing Dai, Haowu Liu 2002 Volume of fluid model for turbulence numerical simulation of stepped spillway overflow. *J. Hydrol. Eng. ASCE* 128: 683–688
- Song C C S, Fayi Zhou 1999. Simulation of free surface flow over spillway. *J. Hydrol. Eng. ASCE* 125: 959–967
- Song C C S, Yuan M 1988. A weakly compressible flow model and rapid convergence methods. *J. Fluid Eng. ASME* 110: 441–445
- Thompson J F, Warsi Z U A, Martin C W 1985 *Numerical grid generation - foundations and applications*, (Amsterdam: North – Holland)
- Unami K, Kawachi T, Munir Baber M, Itagaki H 1999 Two dimensional numerical model of spillway flow. *J. Hydrol. Eng. ASCE* 125: 369–375
- US Bureau of Reclamation (USBR) 1977 Design of small dams. US Govt. Printing Office, Washington DC
- Waterways Experiment Station 1973 Overflow spillway crest, hydraulic design criteria. Vicksburg, USA
- Wei Wenli, Dai Huichao 2005 Simulation of turbulence flows on concave surfaces of spillways including the effects of streamline curvature. *Proc. 31st IAHR Congress*, Seoul, Korea
- Zhou F, Bhajantri M R 1998 Numerical study of the effects of spillway crest shape on the distribution of pressure and discharge. *Proc. of 3rd International Conference on Hydro-Science and Engineering*, Berlin, Germany