PHYSICAL AND NUMERICAL INVESTIGATION OF THE SKIMMING FLOW OVER A STEPPED SPILLWAY①

Rita Fernandes de Carvalho②, António Táboas Amador③

② Prof., University of Coimbra, Portugal, e-mail: ritalmfc@dec.uc.pt
③Prof., Technologic School of Barreiro - IPS, Portugal, e-mail: antonio.amador@estbarreiro.ips.pt

Abstract In the last two decades experimental research on stepped chute hydraulics has been very active. In this paper, the developing flow region over the stepped spillway was investigated using a physical and a numerical model. The numerical model is based on the 2D Navier-Stokes equations, where the free surface is represented using a refined Volume-Of-Fluid (VOF) algorithm, the internal obstacles are described by means of the Fractional Area-Volume Obstacle Representation (FAVOR) method. The numerical results were compared with experimental data obtained in a large scale model using the particle image velocimetry technique. The stepped channel is 0.50 m wide and 2.0 m high, with step height of 0.05 m and bottom inclination of 51.3º corresponding to a slope (V:H) of 1:0.8. At the upstream region of the chute the free surface is smooth and glassy and no air entrainment occurs, the numerical model is suitable for obtaining the characteristics and the influence of the presence of the steps. Typical flow phenomena in the cavity of the steps, in the shear layer and in the main flow are described. Velocity and turbulence intensities profiles are presented. The agreement between the numerical results and laboratory measurements was satisfactory.

Key words stepped spillways; turbulent flow; PIV; physical and numerical model; VOF

1 INTRODUCTION

The development of the roller compacted concrete (RCC) technology and the adoption of the stepped profile in spillways on RCC dams has opened new perspectives on the investigation of safer criteria to design these structures. The characterization of the flow is useful to preview the features of the flow, such as pressure profiles in the steps. The design discharges over a steeply sloping stepped spillway usually corresponds to skimming flow regime, characterized by a coherent stream that skims over the steps (Chanson, 2001). Inside the steps cavities there is a formation of recirculation vortices and these vortices and the external steps edges form a pseudo-bottom. At irregular time intervals momentum exchanges occurs between the main stream and the cavity flow (Matos, 1999).

Three zones of the flow over the spillway can be distinguished: at the upstream region of the chute the free surface is smooth and glassy and the flow is non-aerated next to the steps edges, flow turbulence is generated and starts the growth of the boundary layer (zone 1); after the inception point, where boundary layer equals the flow depth, free-surface instabilities as well as air concentration appear (zone 2); downstream the inception point, a rapidly varied flow region is observed (zone 3). Uniform flow is defined where equilibrium is reached, and the flow depth, velocity and air concentration values will not vary along the chute (Amador et al., 2006).

Several studies, mainly developed at laboratory using physical models, were done in the last decades. Experimental analysis is expensive, subject to scale effects and dependent on the equipment. For the visualization and

① This work is supported by IMAR.

characterization of the velocity field the use of Particle Image Velocimetry (PIV) is very interesting. This technique is a non intrusive measurement and permits obtaining some insight about the internal flow features that fill the cavities between the main flow and the faces of the steps (Amador, 2005).

The numerical modelling of such flows is still very scarce since it meets considerable difficulties, particularly the calculation of the instantaneous position and configuration of the free surface, the accurate calculation of the details of vortex shedding, turbulence effects, and air entrainment. An attempt for the description of the turbulence dynamics in strong, free hydraulic jumps was presented in Carvalho (2002).

In the present work a numerical study was conducted to study the nonaerated flow region on a stepped spillway. Numerical results can be compared with experimental data obtained in a large scale model using the PIV instrumentation (Amador 2005 and Amador et al. 2006).

The numerical model was based on 2D Navier-Stokes equations, where the free surface is represented using a refined Volume-Of-Fluid (VOF) algorithm, the internal obstacles are described by means of the Fractional Area-Volume Obstacle Representation (FAVOR) method. This work aimed to improve the understanding of the flow phenomena and also to evaluate numerical solutions.

2 EXPERIMENTAL SETUP AND INSTRUMENTATION

The stepped spillway model constructed at the Hydraulic Laboratory of A Coruna University is 0.5 m wide, 2.0 m high (from crest to toe), the step height (h) is 0.05 m and the bottom inclination is 51.3 deg., corresponding to a slope of 1:0.8 (v:h) typical of RCC dams. Figure 1 shows the view of the laboratory installation and Figure 2 illustrates its geometry. The tested unit water discharge (q) was 0.11 m^2/s, which corresponds to a Reynolds number (Re=q/ν) of 1.1×10^5.

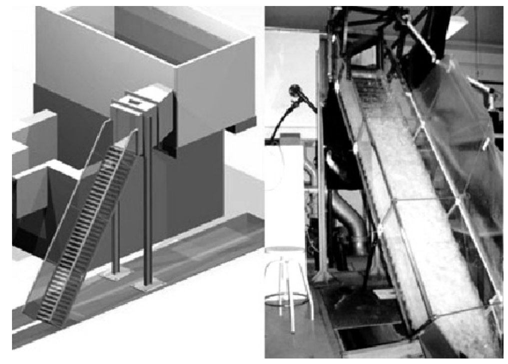


Fig. 1 Overview of the laboratory installation – stepped spillway (in Amador, 2005)

The PIV system used is composed of a double-pulsed Nd-Yag laser, with 400 mJ of energy per pulse, a 1024_1280 pixel resolution, high sensitive peltier-cooled, charge-coupled device CCD camera. A Nikkor 50 mm 2.8, together with a narrow bandwidth filter that passes the 532 nm light from the Nd-Yag laser, is mounted on the CCD camera. An electronic sequencer synchronizes the camera and laser pulses. The electromagnetic flowmeter installed in the supply conduit of the model had an accuracy of ±2%. More details can be found in Amador (2005) and Amador et al. (2006).

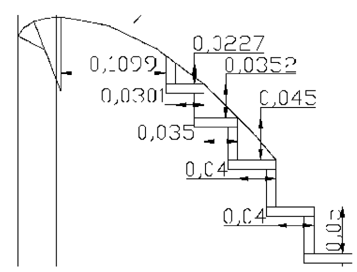


Fig. 2 Definition sketch of the stepped spillway

3 NUMERICAL SIMULATIONS

The numerical model is based on the Navier-Stokes equations governing the motion of the 2D incompressible flows in the plane $x-z$, in which the free surface is described using a refined Volume-Of-Fluid (VOF) method (Hirt & Nichols, 1981) and the internal obstacles are described by means of the Fractional Area-Volume Obstacle Representation (FAVOR) algorithm (Hirt & Sicilian, 1985). The mass and momentum conservation equations (Navier-Stokes equations) and the equation governing the time evolution of VOF-function $F = F(x,z,t)$, whose value is 1 for a point occupied by the fluid and zero elsewhere, are:

$$\nabla.(\theta \vec{u}) = 0 \quad (1)$$

$$\frac{\partial \vec{u}}{\partial t} + \frac{1}{\theta}\nabla.(\theta \vec{u} \otimes \vec{u}) = -\frac{1}{\rho}\nabla p + \\ + \frac{1}{\theta}\nabla.\left[\theta(\nu + \nu_T)\left\{\nabla \vec{u} + (\nabla \vec{u})^T\right\}\right] + \vec{g} \quad (2)$$

$$\frac{\partial F}{\partial t} + \frac{1}{\theta}\nabla.(\theta \vec{u}) = 0 \quad (3)$$

where $\vec{u} = (u(x,z,t), w(x,z,t))$ is the velocity vector, $p = p(x,z,t)$ is the pressure, ν is the coefficient of kinematic viscosity, ν_T is the eddy viscosity, $\vec{g} = (g_x, g_z)$ is the body force term and $\theta = \theta(x,z)$ is the porosity function whose value is "0" for a point inside an obstacle and "1" for a point that can be occupied by the fluid. If $\theta = \theta(x,z)$ = 1 for all points of the domain, equations (1) and (2) reduce to the usual Navier-Stokes equations.

The governing equations were discretized using a finite difference staggered grid system of rectangular cells (control volume) with variables width Δx_i and height Δz_j. The porosity function θ is defined at the horizontal and vertical faces of each control volume using the mesh-wide arrays AT and AR and at the centre of the control volume using another array AC, allowing the treatment of curved boundaries in cartesian meshes without stair-step instabilities.

The main computational cycle by which the dependent variables are updated from u^{n}, w^{n}, p^{n}, F^{n}, at time level $n\Delta t$ to u^{n+1}, w^{n+1}, p^{n+1}, F^{n+1}, at time $(n+1)\Delta t$ consists of several steps: Computing a provisional velocity field $\vec{u}^* = (u^*, w^*)$ using the discretized momentum equation with the advanced-time pressure p^{n+1} replaced by p^{n}. (it is important to treat the advection term accurately); calculate the advanced-time velocity $\vec{u}^{n+1} = (u^{n+1}, w^{n+1})$ and pressure p^{n+1} using a pressure-velocity iterative scheme based in a cell by cell pressure-velocity relaxation scheme so that $\vec{u}^{n+1}$ is discretely div-free; and calculate the advanced-time fluid configuration F^{n+1} in all cells of the domain using a special discretization of eq. (3) for maintaining the sharpness of the interface and conserving total fluid volume (Hirt & Nichols, 1981 and Rider & Kothe, 1998).

The computational domain of the flow was defined by a 2DV rectangle 0.625 m long (0.0724 to 0.675 m) and 0.8 m high (1.4 to 2.2 m), using a grid of approximately 240 x 320 cells with $\Delta x_{\min}$ = 0.0025 m and $\Delta z_{\min}$ = 0.0025 m. The acceleration due to gravity was set to g = 9.8 ms^{-2}. The geometry of the stepped spillway, including the ogee crest and steps, were represented by defining the fractional areas and volumes according to the FAVOR technique, so as to block the desired cells to the flow. The mathematical formulation for the ogee crest and steps positions are represented by Equations [4].a and [4].b, respectively.

$$\begin{cases} x^2 - 0.169x + z^2 - 3.931z = -3.870 \\ \quad for\ 0.06 < x < 0.0724 \\ x^2 - 0.200x + z^2 - 3.842z + 3.694 = 0 \\ \quad for\ 0.0724 < x < 0.1 \\ x^2 - 0.200x + 0.316z - 0.622 = 0 \\ \quad for\ 0.1 < x < 0.21 \end{cases} \quad (4a)$$

$$\begin{cases} 0.210 < x < 0.240 \quad \Lambda \quad z < 1.93694 \\ 0.240 < x < 0.275 \quad \Lambda \quad z < 1.90174 \\ 0.275 < x < 0.315 \quad \Lambda \quad z < 1.85674 \\ 0.315 < x < 0.355 \quad \Lambda \quad z < 1.80674 \\ 0.355 < x < 0.395 \quad \Lambda \quad z < 1.75674 \\ 0.395 < x < 0.435 \quad \Lambda \quad z < 1.70674 \\ 0.435 < x < 0.475 \quad \Lambda \quad z < 1.65674 \\ 0.475 < x < 0.515 \quad \Lambda \quad z < 1.60674 \\ 0.515 < x < 0.555 \quad \Lambda \quad z < 1.55674 \\ 0.555 < x < 0.595 \quad \Lambda \quad z < 1.50674 \\ 0.595 < x < 0.635 \quad \Lambda \quad z < 1.50674 \\ 0.635 < x < 0.675 \quad \Lambda \quad z < 1.45674 \\ 0.675 < x < 0.715 \quad \Lambda \quad z < 1.40674 \end{cases} \quad (4b)$$

In the laboratory installation, the flow over the spillway was $Q = 0.055$ m3/s or $q = 0.11$ m2/s ($b = 0.5$ m), $h_1 = 0.135$ m, being the level over the stepped spillway crest. For such conditions the flow at inflow boundary, upstream the spillway, was established by imposing fixed height, h_{in} = 0.135 m, and variable velocity in order to avoid instabilities. The velocity was established by the following conditions:

$$u_{1j} = u_2 + \left(u_{1jp} - u_2\right)\ a,\ a = \begin{cases} 0 & t \leqslant t_0 \\ \dfrac{t\text{-}t_0}{t_f\text{-}t_0}, & t_0 < t < t_f; \\ 1 & t \geqslant t_f. \end{cases}$$

$$w_{1j} = 0$$

$$F_{1j} = \begin{cases} 1 & z_{j-1} \leqslant h_1 \\ \left(h_1 - z_{j-1}\right)/dz_j & z_{j-1} > h_1 > z_j \\ 0 & z_j \geqslant h_1 \end{cases} \tag{5}$$

where u_1 = 0.0815 m/s and u_2 =0.815 m/s, t_0 = 0.0 and t_f = 0.08 s.

Different initial conditions were considered, such as an empty domain. In this case, in order to minimize the development of instabilities and to reach earlier a steady state solution, the computer simulations were done with a special initial condition. The free-surface was defined by a straight line parallel to the steps edges, and velocity equal to zero. This condition is not physically possible but after the first cycle, the output is similar to a physical behaviour of the flow. The formation and development of the flow over the step spillway was study by numerical model considering the coefficient of kinematic viscosity of ν = 1.15 x10^{-6} m^{2}s^{-1}, a very small mesh sizes and time increment Δt.

4 RESULTS AND DISCUSSION

The evolution of the flow over the spillway was reproduced in a realistic way. Figure 3 illustrates the flow over the stepped spillway and Figure 4 illustrate the details of the velocity field and free surface configuration over the step 34 and 35 (L/ks=13.04 and 10.98). The mean velocity field over the steps by PIV technique was obtained from 500 instantaneous velocity fields. The velocity vector spacing is 2.0 mm (i.e., 16 pixels).

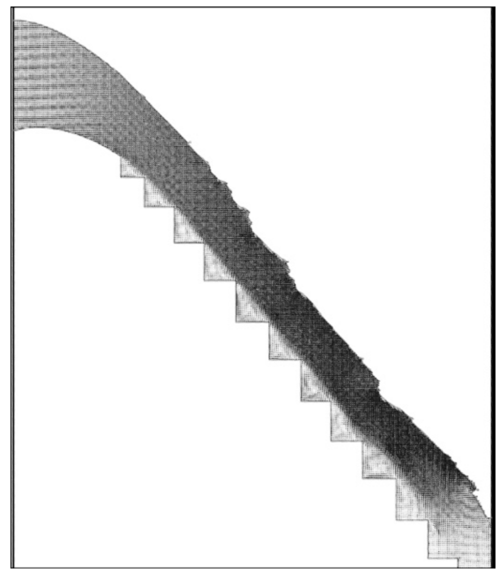
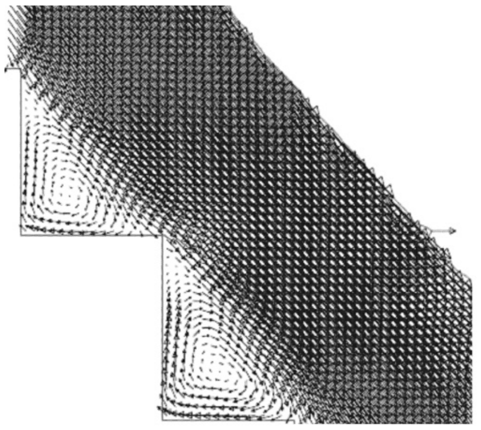


Fig. 3 Free surface and velocity field for the stepped spillway: Numerical simulation

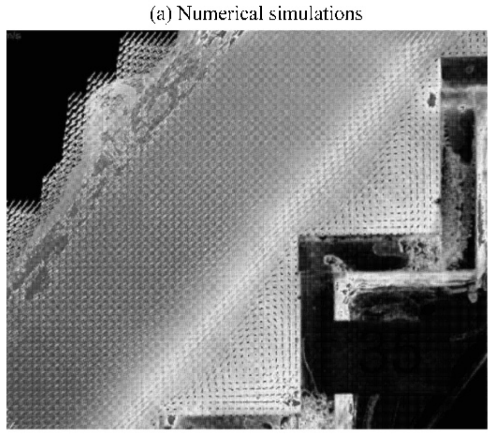
As the flow entered the domain at the initial section of the channel, it could move over the spillway and increased the velocity in the main stream. The formation of vortices in the cavities starts, as well as free surface corrugations. The dynamics and the unsteady position of the large-scale vortices were well reproduced, with a fair comparison with the vortex dynamics observed in the physical model. On the outflow boundary, the velocity was free to leave the domain.

From the Fig. 3 and 4, it can be observed that the numerical model predict with satisfactory accuracy the two kinds of flows, the recirculation vortices and the coherent stream over the pseudo-bottom. The streamlines shows a separation downstream external edges and in the cavity it can be observed the vortices with a horizontal axis. Particularly, in the main stream, the velocity is ten times the velocity in the vortices and between the two flows, where momentum exchanges occurs, there is a gradient.

Fig. 5 shows the mean velocity profiles obtained by the two methodologies, numerical model and PIV technique and Figure 6 shows the turbulence intensity profile. The mean velocity field



(a) Numerical simulations



(b) PIV

Fig. 4 Detail of free surface and velocity field for the stepped spillway: (a) Numerical simulation; (b) PIV data

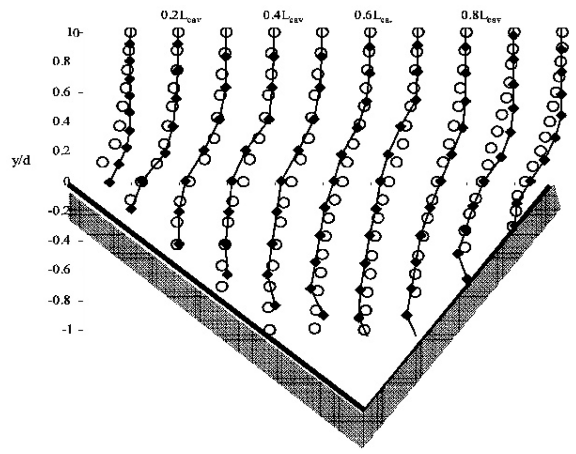
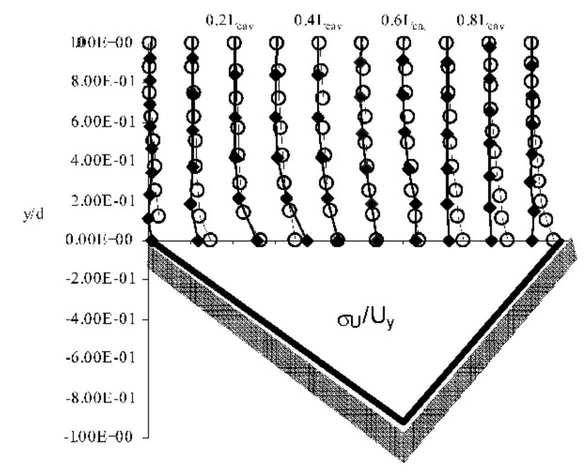
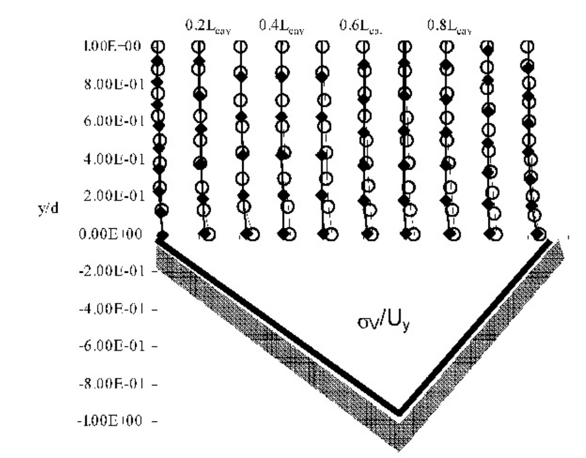


Fig. 5 Mean velocity profiles U/U_0 for the flow over the step 34 (L/ks=13.04) : Num. simulation (●) vs PIV data (○)

over the steps by numerical data was obtained by averaged about 400 files with Δt=0.01s. Regarding the turbulence intensities profiles it can be noticed an increase from free-surface to pseudo-bottom. The profiles obtained from the numerical simulation and the experimental data are in good agreement, mainly in the middle of the cavity. The greater differences are near the pseudo-bottom, particularly on the σ_U/U_y in the threes last profiles.



(a) σ_U/U_y



(b) σ_V/U_y

Fig. 6 Turbulence Intensities for the flow over the step 34 (L/ks=13.04) : Numerical simulation (●) vs PIV data (○)

5 CONCLUSIONS

The flow over a stepped spillway was investigated using a physical and a numerical model. Developing flow region over the stepped spillway data is presented, including velocity and turbulence intensities profiles.

Numerical simulations data are presented super imposed with experimental data obtained by the PIV technique in order to evaluate the capabilities of numerical simulations. The agreement between the numerical results and laboratory measurements was satisfactory. The numerical model is appropriate to evaluate velocities and turbulence intensities profiles.

REFERENCES

Amador, A. (2005). "Comportamiento Hidráulico de los Aliviaderos Escalonados en Presas de Hormigón Compactado". Phd thesis, Universitat Politècnica de Catalunya (UPC), Barcelona, Spain (in Spanish).

Amador, A., Sánchez-Juny, M. and Dolz, J (2006). "Characterization of the Nonaerated Flow Region in a Stepped Spillway by PIV". J. Fluids Eng., 128(6), 1266-1273.

Carvalho, R. F. (2002). "Acções hidrodinâmicas em estruturas hidráulicas: Modelação computacional do ressalto hidráulico", Tese de Doutoramento, Universidade de Coimbra, Portugal. (in Portuguese)

Chanson, H., (2001). The Hydraulics of Stepped Chutes and Spillways, Swets & Zeitlinger, Lisse, The Netherlands.

Hirt & Nichols, (1981). "VOF Method for the Dynamics of Free Boundaries". J. Comp. Phys. 39, 201-225.

Hirt, C. W. and Sicilian, J.(1985). "A porosity technique for the definition of obstacles in rectangular cell meshes", 4th Int. Conf. on Ship Hydrodynamics, Washington DC.

Matos, J., 1999. "Air Entrainment and Energy Dissipation on Flow Over Stepped Spillways," Ph.D. thesis, Technical University of Lisbon, IST. (in Portuguese)

Rider & Kothe, 1998. "Reconstructing Volume Tracking". J. Comp. Physics 141, 112-152.