Modeling of a Water Flow Over Stepped Spillways

Khadidja Kherbache, Xavier Chesneau, Stéphane Abide, Saâdia Benmamar

Université de Perpignan Via Domitia (UPVD), LAboratoire de Mathématiques et de Physique (LAMPS) EA4217, F- 66860, Perpignan, France

Laboratoire de Recherche des Sciences de l’eau, Ecole Nationale Polytechnique-10 Avenue Hassen Badi, 16000 Alger- Algérie.

Abstract

In this work, a numerical analysis of the flow over a stepped spillway is performed. The free surface flow is described by the Reynolds Averaged Navier-Stokes equation (RANS) coupled to k-ε model. These equations are solved using a commercial software based on the finite volume scheme and the VOF method. We analyzed the influence of the steps shape and water flow rate on velocity, turbulence dissipation rate and pressure distributions. In this, we found that the maximum values of the static pressure and the turbulence dissipation rate increase as the discharge increases and the main flow velocity is greater than the one of the secondary flow.

Nomenclature

- $C_\mu$, $C_{\varepsilon_1}$, $C_{\varepsilon_2}$: empirical constants for k –ε turbulence model;
- h: step height (m);
- $H_d$: design head of the spillway (m);
- $k$: turbulent kinetic energy ($m^2.s^{-2}$);
- L: step length (m);
- p: pressure (Pa);
- t: time (s);
- $\overline{U}$, $\overline{V}$: velocity component (m.s$^{-1}$);
- x , y: coordinate component (m);
- F: color function;
- $\varepsilon$: turbulent dissipation rate ($m^2.s^{-3}$);
- $\mu$: dynamic viscosity (kg.m$^{-1}$.s$^{-1}$);
- $\rho$: density (kg.m$^3$);
- $\sigma_k$ and $\sigma_\varepsilon$: turbulent Prandtl and Schmidt numbers and for k and $\varepsilon$.
- $q_m$ = unit discharge ($m^2.s^{-1}$)

subscript:
- a: air;
- t: turbulence;
- w: water;
1. Introduction

Stepped spillways are designed to have safely flood downstream a dam. Various studies have been carried out in order to analysis the effects of different parameters such as: discharge, number of steps, step height and inclination angle of the spillway on the flow parameters as the energy dissipation, the inception point, the air concentration, the velocity distributions, the pressure profiles, the turbulent kinetic energy and its dissipation rate [Benmamar, S. et al., 2003, Wu J., Zhang B., Ma F., 2013]. Most of these works showed that the skimming flow is characterized by counter cells located in the vicinity of the step corner interacting with the main flow [Bombardelli F. A., et al., 2010, Chen Q. et al., 2006, Cheng X.J., et al., 2004, Cheng X.J., et al., 2006, Chen Q. et al., 2002, Qian Z.D., et al., 2009]. The steps increase significantly the rate of energy dissipation of the flow and reduce the size of the required downstream energy dissipation basin [Bombardelli F. A. et al., 2010, Chanson H., 1993]. So, the knowledge of spillways hydrodynamics is necessary for safer design of spillways. The prediction of this flow is very difficult due to several phenomena such as turbulent boundary layers, free surface flow and complex geometry [Chen Q. et al., 2002, Chen Q. et al., 2004, Cheng X.J., et al., 2006]. These flows can be modeled using a two-dimensional [Benmamar, S., et al., 2003, Bombardelli F. A. et al., 2010, Chen Q. et al., 2002, Cheng X.J., et al., 2006, Meireles I., et al., 2009, Qian Z.D., 2009, Sabbagh-Yazdi S.R., et al., 2007] or three-dimensional [Chen Q. et al., 2002, Cheng X.J., et al., 2006, Meireles I., et al., 2009] models based on Reynolds Averaged Navier-Stokes equations (RANS) [Chen Q. et al., 2002, Meireles I., et al., 2009] or shallow water equations [Sabbagh-Yazdi S.R., et al., 2007] linked to turbulent models as standard k-ε [Bombardelli F. A et al., 2010, Chen Q. et al., 2002, Meireles I., et al., 2009], RNG k-ε [Bombardelli F. A et al., 2010, Chen Q. et al., 2004], Reynolds stress [Meireles I., et al., 2009], realizable k-ε [Cheng X.J., et al., 2006, Qian Z.D., et al., 2009], SST k-ω [Qian Z.D., et al., 2009], v2-f [Qian Z.D., et al., 2009] and LES [Qian Z.D., et al., 2009]. These equations are solved by using computational fluid dynamics (CFD) softwares as fluent [Cheng X.J., et al., 2006], FLOW-3D [2, Meireles I., et al., Sabbagh-Yazdi S.R., et al., 2007] or NASIR [Sabbagh-Yazdi S.R., et al., 2007].

In this section, we attend to report some papers about numerical and experimental studies. Concerning the numerical works, [Cheng et al., (2006)] analyzed the efficiency of two turbulent models; k-ε and RNG k-ε, the VOF and Mixture models for predicting the flow over a classical stepped spillways. It was found that the mixture model associated to RNG k-ε model allows the prediction of air entrainment phenomena and contra cells flow. Qian [Qian Z.D., et al., 2009] carried out a comparative analyze of the efficiency of turbulence model: the realizable k-ε, SST k-ω, v2-f and LES for the determination of flow properties. The realizable k-ε model associated to mixture model is more effective for the prediction of air entrainment and contra cells.
The experimental studies [Chanson H., 1993, Chanson H., 1994, Chanson H., 2001, Chen Q., Dai G., Liu H., 2002] revealed three types of flows, namely, nappe flow for small flow rates, transition flow for intermediate discharges and skimming flow for high flow rates. Most of these experimental works are focused on studies of air-water flows on stepped spillways with steep slope for which the steps are a flat horizontal plate and one the location of the inception point. This point, defined as the abscissa for which the boundary layer thickness is equal to the water depth is depending on discharge, step height, and inclination angle of the spillway. From this point air entrainment occurs and has an effect on the velocity and pressure fields downstream the flow. Upstream the inception point, the flow is non-aerated and downstream this point the flow is aerated [Bombardelli F. A., Meireles I., Matos J., 2010, Chanson H., 1993, Chanson H., 1994, Chanson H., 2001]. In the aerated flow, the flow velocity and the parietal stress are inferior to those of the non-aerated flow. It has been proved that the aeration of the flow reduces the cavitation problem [Chen Q., et al., 2002, Roshan R] characterized by a minimum pressure.

In order to analyze the effects of the discharge, the steps height on the location of the inception point, Wu et al. 2013 carried out an analytical and experimental study of a flow along a spillway. The analytical study is based on the boundary layer and the experimental one on the visualization of the flow. The experimental data of the inception point are expressed as the position of \( L/d = 13.80 q^{-0.15} \); where \( L \) is the distance of the inception point to the crest of the spillway, \( d \) is the liquid depth at the air entrainment position. The location of this point increases with the discharge and decreases as the step height increases.

Most searchers [Chanson H., at al. 1993, Chanson H., at al. 1994a, Chanson H., at al. 1994b, Chanson H., at al. 2001] have presented empirical relationships of \( d_c/h \) function of \( (h/l) \) where \( d_c \) is the critical depth; \( h \) and \( l \) respectively the height and the length of step. These relations are used to evaluate the range regime of the nappe flow and of the skimming flow. The energy dissipation of these flow regimes that the short spillway, nappe flows dissipate more kinetic energy than skimming flows [Chanson H., at al. 1994a].

In order to analyze the effect of the step shape, Felder [Felder S. Chanson H., 2013] performed a study about three type of step: flat horizontal step, pooled steps, and a combination of flat and pooled steps. Results show that a combination of flat and pooled steps leads to a stronger aerated flow and consequently to the largest average flow velocity in the case of the flat step. It will be noticed that the energy dissipation reached its maximal value in the case of the pooled.

R. Roshen [Roshan R., et al., 2010] carried out a study in order to analyze the effect of the numbers of steps (12 and 23), discharges and the flow regimes on the energy dissipation of the flow along a stepped spillway. The dissipation of energy is all the more great that the step numbers is low for the two spillway
configuration retained in this study. Moreover, it should be noted that the energy dissipation decreases with the step height [Roshan R., et al., 2010]. In order to analyze the influence of the dam slope on the energy dissipation rate Chen et al. 2010] carried out a comparative study of slope 1:0.7, 1:0.75, 1:0.8, 1:1 on the flow characteristics. It have been reported that the energy dissipation rate increases as the dam slope decreases. An analyze of the effect of the ogee at the spillway toe on the energy dissipation rate proved that the ogee increased turbulence dissipation rate [Chen et al. 2010]]. So, the ogee can improve the energy dissipation rate and consequently contribute to the reduction of the cavitation phenomenon [Chen et al. 2010]].

Despite several works that have been carried out on the spillways flow, it would appear, at our knowledge, that no one have been focused on the effect of the step shape on the flow characteristics.

So, this work involves a numerical analysis of a flow over a stepped spillway. Numerical modeling was performed using a Computational Fluid Dynamics (CFD) software (fluent) based on the finite volume scheme and the VOF method.

2. Physical model and mathematical formulation

2.1. Channel configuration

The geometry of the spillway model is shown in figure 1. It is a conventional spillway composed of 13 steps [Chen Q et al, 2002, Cheng X.J., et al., 2006]. The crest shape of the spillway is according to the standard spillway profile situated 78.9 cm above the toe [Chen Q et al, 2002, Cheng X.J., et al., 2006]. The design head Hd, equal to 9.7 cm, have a profile of type 1V:0.75H. The heights of steps from the crest are equal to 2; 2.4; 3; 4 and 5 cm, respectively. The three other steps are above the tangency point [Chen Q et al, 2002, Cheng X.J., et al., 2006]. Under the step No. 5, the step sizes are equal to h=6 cm and l=4.5 cm and the length of the channel upstream the crest equal to 25 cm.

We associate to this spillway a Cartesian reference (oxy), such as the axis [oy) is vertical and oriented to the opposite direction gravity and the axis [ox) is deviated to the left of the [oy). The origin O is located on the top of the crest.
2.2. Hypothesis

- We assume that the transfers are two dimensionnel, permanent and incompressible;
- The fluid properties are constant.

2.3. Governing equations

The water flow over the spillway is described by the Reynolds Averaged Navier-Stokes equations (RANS) writing in the (oxy) referential:

\[
\frac{\partial U}{\partial x} + \frac{\partial \bar{V}}{\partial y} = 0
\]

\[
\bar{U} \left( \frac{\partial \rho U}{\partial x} + \bar{V} \frac{\partial \rho U}{\partial y} \right) =
\]

\[
- \frac{\partial P}{\partial x} + 2 \frac{\partial}{\partial x} \left[ \mu + \mu_t \frac{\partial \bar{U}}{\partial x} \right] + \frac{\partial}{\partial y} \left[ (\mu + \mu_t) \frac{\partial \bar{U}}{\partial y} \right] + \frac{\partial}{\partial y} \left[ (\mu + \mu_t) \frac{\partial \bar{V}}{\partial x} \right]
\]

With: \( h \) and \( l \) respectively height and length of the step.
\[ \frac{-\bar{U}}{\partial \rho \bar{V}} \frac{\partial \rho \bar{V}}{\partial x} + \bar{V} \frac{\partial \rho \bar{V}}{\partial y} = \]

\[-\rho g - \frac{\partial p}{\partial y} + \frac{\partial}{\partial x} \left[ (\mu + \mu_t) \frac{\partial \bar{V}}{\partial x} \right] + 2 \frac{\partial}{\partial y} \left[ (\mu + \mu_t) \frac{\partial \bar{V}}{\partial y} \right] + \frac{\partial}{\partial x} \left[ (\mu + \mu_t) \frac{\partial \bar{U}}{\partial x} \right] \]

\[ \text{(3)} \]

Where:
\[ \rho \] - density of air-water; \( P \) - pressure; \( \bar{U}, \bar{V} \) - components of the velocity along \([x, y]\); \( g \) - Acceleration of gravity; \( \mu \) - dynamic viscosity; \( \mu_t \) - turbulence viscosity determined by the turbulent kinetic energy \( k \) and the turbulent dissipation rate \( \varepsilon \):
\[ \mu_t = \rho C_{\mu} \frac{k^2}{\varepsilon} \]

\[ \frac{-\bar{U}}{\partial \rho \bar{k}} \frac{\partial \rho \bar{k}}{\partial x} + \bar{V} \frac{\partial \rho \bar{k}}{\partial y} = \]

\[ \frac{\partial}{\partial x} \left[ (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial x} \right] + \frac{\partial}{\partial y} \left[ (\mu + \frac{\mu_t}{\sigma_k}) \frac{\partial k}{\partial y} \right] + \mu_t \left[ \left( \frac{\partial \bar{U}}{\partial y} + \frac{\partial \bar{V}}{\partial x} \right)^2 + 2 \left( \frac{\partial \bar{U}}{\partial x} \right)^2 \right] - \rho \varepsilon \]

\[ \frac{-\bar{U}}{\partial \rho \varepsilon} \frac{\partial \rho \varepsilon}{\partial x} + \bar{V} \frac{\partial \rho \varepsilon}{\partial y} = \]

\[ \frac{\partial}{\partial x} \left[ (\mu + \frac{\mu_t}{\sigma_{\varepsilon}}) \frac{\partial \varepsilon}{\partial x} \right] + \frac{\partial}{\partial y} \left[ (\mu + \frac{\mu_t}{\sigma_{\varepsilon}}) \frac{\partial \varepsilon}{\partial y} \right] + C_1 \frac{\varepsilon}{k} \mu_t \left[ \left( \frac{\partial \bar{U}}{\partial y} + \frac{\partial \bar{V}}{\partial x} \right)^2 + 2 \left( \frac{\partial \bar{U}}{\partial x} \right)^2 \right] - C_2 \frac{\rho}{k} \varepsilon \frac{\varepsilon^2}{k} \]

\[ \text{(5)} \]

\[ C_{\mu}, C_{\varepsilon_1}, C_{\varepsilon_2}, \sigma_k \] and \( \sigma_{\varepsilon} \) are empirical constants.

\[ C_{\mu} = 0.09; C_{\varepsilon_1} = 1.44; C_{\varepsilon_2} = 1.92; \sigma_k = 1 \text{ and } \sigma_{\varepsilon} = 1.3. \]

To equations (1-6), we associated the equation of the VOF method proposed by Hirt and Nichols [Hirt, C. W., Nichols, B. D., 1981].
\[ \bar{U} \frac{\partial F}{\partial x} + \bar{V} \frac{\partial F}{\partial y} = 0 \quad 0 \leq F \leq 1 \]

\[ \rho = \rho_w F + (1 - F) \rho_a \]

\[ \text{(7)} \]

\[ \text{(8)} \]
\[ \mu = \mu_w F + (1 - F) \mu_a \]  

(9)

where; \( \rho_a \) and \( \rho_w \) are respectively air and water density; \( \mu_a \) and \( \mu_w \) air and water dynamic viscosity, respectively.

1.1. Boundary conditions

- **Inlet boundary**: \( x = -L_r, -H_r \leq y \leq H_d \): \[ \overline{U}(x,y,t_0) = U_0 \]  
  \( H_d \leq y \leq H_f \): \[ P(x,y,t_0) = P_{atm} \]  

(10)

(11)

- **Outlet boundary**: \( x = L, -H \leq y \leq -H_1 \): \[ P(x,y,t_0) = P_{atm} \]  

(12)

- **Free surface**: \( -L_r \leq x \leq L_r, -H_1 \leq y \leq H_f \): \[ P(x,y,t_0) = P_{atm} \]  

(13)

- **Wall boundary conditions**: \( -L_r \leq x \leq L_r, -H \leq y \leq 0 \) m: \[ \overline{U}(x,y,t_0) = \overline{V}(x,y,t_0) = 0 \]  

(14)

2. Numerical methods

The equations (1-6) associated to the boundary conditions (10-14) are solved using Fluent Computational Fluid Dynamics (CFD) software. The equations (1-6) are discretized using an implicit method based on the finite volume method. Convective and diffusive terms are approximated using the second-order accurate upwind scheme. An unstructured mesh of quadrilateral cells is generated with a preprocessor. Computations have been performed using a grid size of 13230 corresponding to a mesh size \( \Delta x \) varying between 0.005 and 0.01 m and \( \Delta y \) between 0.005 and 0.01 m. Algebraic equations systems are resolved by a Gauss Seidel method. The PISO algorithm was also used for the pressure correction equation.

3. Grid sensibility analysis

To analyze the influence of the mesh on the results, we implemented calculations with four meshes 13562, 19060, 23371 and 44807 nodes respectively. The discrepancies of the maximum velocity for four meshes is reported in table 1. We note that the discrepancies are lower to 7.14%. Therefore, our choice fell on the fourth mesh to 44807 nodes.
Table 1: Effect of the mesh size on the maximum velocity.

<table>
<thead>
<tr>
<th>Number of nodes</th>
<th>13562</th>
<th>19060</th>
<th>23371</th>
<th>44807</th>
</tr>
</thead>
<tbody>
<tr>
<td>$V_{\text{max}}$ (m/s)</td>
<td>2.7</td>
<td>2.52</td>
<td>2.42</td>
<td>2.47</td>
</tr>
<tr>
<td>Errors (%)</td>
<td>0.00</td>
<td>7.14</td>
<td>4.13</td>
<td>2.06</td>
</tr>
</tbody>
</table>

4. Validation

In order to validate the numerical procedure, we applied the software to the more closed problem to the one studied in this paper [Chen Q at al., 2002]. The water depth is computed for a discharge value equal to 0.0068 m²/s. The comparison revealed good agreement as shown in Fig. 2.

5. Results and discussion

Computation were performed for five dynamic heights equal to 0.057, 0.067, 0.077, 0.087 and 0.097 m corresponding to unit discharge equal respectively to 1.7 10-3, 2.7 10-3, 3.8 10-3, 5.2 10-3 and 6.8 10-3 m³/s. Results are presented as velocity, static pressure and turbulent dissipation rate distribution.
Fig. 3 shows the velocity distribution of the flow in the region near the steps 9 to 11. As seen from this figure, the water flow is divided into two parts: The main flow (skimming flow); in which the water flows as a coherent stream over the outer edges of the steps; the fluid is accelerated by the gravity effect along the spillway. Consequently the velocity increases with the distance from the top of the spillway. So, higher velocities appearing in the mean flow increase from 2.04 m/s to 2.58 m/s. The secondary flow is described by counter cells located in the vicinity of the step corner where the smallest velocities values are observed in the domain surrounding the step corner. The minimum value of these velocities are situated in the center of this domain and increases from 1.97 $10^{-3}$ m/s to 3.86 $10^{-3}$ m/s as computation unit discharges increases from 1.7 $10^{-3}$ m$^2$/s to 6.8 $10^{-3}$ m$^2$/s.
Fig. 3. Velocity vectors (m/s) (a) \( q_m = 6.8 \times 10^{-3} \) m\(^2\)/s, (b) \( q_m = 5.2 \times 10^{-3} \) m\(^2\)/s, (c) \( q_m = 3.8 \times 10^{-3} \) m\(^2\)/s, (d) \( 2.7 \times 10^{-3} \) m\(^2\)/s and (e) \( 1.7 \times 10^{-3} \) m\(^2\)/s for steps 9 to 11.

In Fig. 4 is plotted the static pressure distribution. We note that the unit discharge have no one effect on the pressure pattern located in the vicinity of the horizontal plate of the step more precisely near its edge. The maximum value of the pressure is induced by the effect of the flow impacting the horizontal step. The minimum values located on the vertical step face, near its outer edge, is due to the flow separation as it can be seen in Fig.3. We note that the maximum value of the static pressure increases from \( 6.33 \times 10^2 \) Pa to \( 1.22 \times 10^3 \) Pa as the unit discharge increases for \( 1.7 \times 10^{-3} \) m\(^2\)/s to \( 6.8 \times 10^{-3} \) m\(^2\)/s.
Fig. 4. Static pressure pattern (Pa) (a) $q_m=6.8 \times 10^{-3}$ m$^2$/s, (b) $q_m=5.2 \times 10^{-3}$ m$^2$/s, (c) $q_m=3.8 \times 10^{-3}$ m$^2$/s, (d) $q_m=2.7 \times 10^{-3}$ m$^2$/s and (e) $q_m=1.7 \times 10^{-3}$ m$^2$/s for steps 9 to 11.

The Fig. 5 presents the turbulent dissipation rate (m$^2$/s$^3$) distribution. As it can be seen, the great values of the turbulent dissipation rate are situated on the crest of the step, more precisely in the domain of step where there is the interaction between the main flow and counter cells. The maximum values of the turbulence dissipation rate increases from $34.2$ m$^2$/s$^3$ to $61$ m$^2$/s$^3$ when of the unit discharge varying from $6.8 \times 10^{-3}$ m$^2$/s to $1.7 \times 10^{-3}$ m$^2$/s.
6. Conclusion

We performed a numerical study of the water flow over a stepped spillway. Transfers equations, based on Reynolds Averaged Navier-Stokes equation (RANS) coupled to k-ε model are solved using fluent software. Examination of the results led to the following conclusions.

The flow along the stepped spillway is divided into a homogeneous principal flow; in which the water flows as a coherent stream over the outer edges of the steps and a secondary flow. Maximum values of the pressure are localized on the vicinity of the horizontal part of the step.
The turbulent dissipation rate reaches its maximum value on the crest of step and increases with the distance along the spillway.

References


