EXPERIMENTAL STUDY AND NUMERICAL SIMULATION FOR THE FLOW THROUGH A VERTICAL GATED SPILLWAY

Nguyễn Công Thành[1]

Abstract: *The flow through vertical gated spillways is a rapidly varied flow with highly curvilinear streamlines. The behavior of this flow can be investigated numerically in the reasonable time and expense. In this study, some hydraulic characteristics of this type of flow were evaluated by using a commercial computational fluid dynamics (CFD) packages, namely, the Ansys/Fluent. Two turbulence models, namely, the Realizable* $k-\varepsilon$ *and Reynolds stress model (RSM) were applied to simulate this type of flow for both 2-D and 3-D models. The experimental data, which were conducted by the authors in the Hydraulic Structures Laboratory at the National University of Civil Engineering (NUCE), Vietnam, were used to validate the numerical simulation. The results show that these turbulence models could be applied to the analysis of the hydraulic performances for the flow in this field. The water surface profiles and the discharge were predicted within an accuracy range of* $+0.48 \sim -7.29\%$*. Additionally, some 3-D vertical gated spillway models were performed to investigate the effect of side contraction on the discharge capacity due to piers and abutments. These results indicate that there is a good approximation for the effective length formula while using the 2-D extrapolation compared to 3-D spillway models.*

Keywords: CFD, turbulence models, Ogee-spillway, vertical gate, discharge.

1. INTRODUCTION

A spillway structure equipped with gates is often called "gated spillway". This is a type of orifice spillway when the gate is partial opening for flood release or during supplied water purposes. In the past, hydraulic characteristics of the free flow over spillways have been historically obtained by using physical models and theoretical methods. One of the most popular methodologies dated back to the middle of the twentieth century were produced by Russian's researches (Slisskii, 1986) or North-American institutions (USACE, 1992)

With the development of computer power and success in research of fluid dynamics, the CFD model has been studied and developed during recent decades. To date, CFD modeling has been generally used as a valuable tool in the optimization phase of hydraulic projects prior to the commissioning of physical model study (Bomnac, 2014). Recently, some works even have shown that numerical modeling can be suitably substituted for the physical modeling of gated spillways in hydraulic applications (Bhosekar, et al, 2011). However, these works did not consider the flow characteristics of a standard vertical gated spillway, which is designed with the Waterways Experiment Station (WES) crest profile. Moreover, the effect of the side contraction on the discharge capacity due to the piers and abutments does not investigate for gated spillways scheme.

A comparative study of two turbulence models (Realizable $k-\varepsilon$ and Reynolds stress model) was carried out to analyze some hydraulic characteristics of the flow through a standard vertical gated spillway. Firstly, two-dimensional (2-D) models, which were integrated in commercials CFD software, namely, Ansys/Fluent, were implemented. The experimental data was conducted in a laboratory flume and used to validate the numerical simulation. Secondly, some three-dimensional (3-D) models of a vertical gated spillway were

[1] *Đại học Xây dựng.*

implemented to investigate the effect of the side contraction on the discharge capacity based on the 2-D model simulation and an empirical formula for the effective crest length.

2. PHYSICAL MODEL

2.1 Description

These experiments were conducted by the author in the Hydraulic Structures Laboratory at the National University of Civil Engineering (NUCE), Hanoi, Vietnam. The spillway model was designed according to the WES profile $z = 0.5x^{1.85} / H_d^{0.85}$ (USACE, 1992) for the design head of H_d = 0.2 m. The model was 0.4 m wide (L_m) and 0.35 m high (P). The vertical gate was made of steel and was positioned on the crest of the spillway and finished with sharp-edged. Model of the spillway was constructed by Plexiglas and installed in a flume at a distance of 3 m from the flume inlet. The flume consisted of a steel frame with transparent Plexiglas sides, 0.4 m in width, 6 m in length, and 0.8 m in depth. In addition, the bottom of the flume was made of stainless steel with a horizontal slope. The laboratory flume scheme and the vertical gated spillway model are shown in Fig. 1. These characteristics of the assembled data are summarized in Table 1 and the remaining variable parameters are referred to Fig. 2.

Table1. Experimental cases and parameters

Test cases	G_o (m)	E (m)	V_o (m/s)	H_o / G_o	F_r (upstream)	R_e
E1	0.08	0.551	0.191	2.54	0.082	1.04e+5
E2	0.08	0.672	0.204	4.05	0.079	1.36e+5
E3	0.09	0.505	0.191	1.74	0.086	9.55e+4
E4	0.09	0.600	0.223	2.81	0.092	1.33e+5
E5	0.10	0.533	0.222	1.86	0.097	1.17e+5
E6	0.10	0.593	0.242	2.46	0.100	1.42e+5

Fig.1 The sketch of experimental set-up

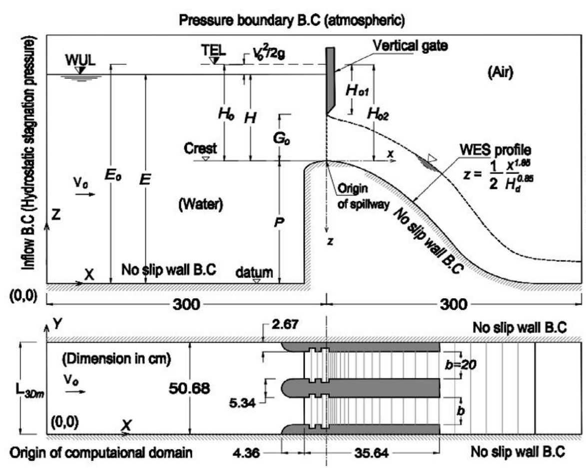

Fig. 2 The scheme of Computational domain and B.Cs for the modeling (Dimensions in cm)

2.2 Experiment procedure

The water surface from upstream to downstream was measured at the centerline of the flume with a point gauge with an accuracy of ± 0.1mm. Furthermore, the WUL (water upstream level) of model was measured at position of 1.5 m from the origin of the spillway toward upstream. The crest pressure was measured by static tube that was a piezometer board with glass tubes vented to the atmosphere. The average pressure head measured on the piezometer board was readable to within ± 1mm. Value of gate opening G_o was measured with a meter fixed on the gate with an accuracy of ± 1mm. In addition, the discharge in the flume was also measured by a rectangular sharp-crested weir located in the gathering tank. The relative uncertainty in the discharged measurement was about 3%.

The experiments were carried out for the various gate opening G_o (m), and the water head H (m). In order to prevent the tailwater from interfering with the spillway pressure taps, free discharge from the spillway toe to the flume was allowed. After the flow upstream conditions of the spillway models were achieved, the water surface profile, crest pressures, and discharge were measured carefully for each gate opening case in the steady-state and free controlled flow.

3. NUMERICAL METHODOLOGY

3.1 Governing Equation

The Ansys/Fluent packages solve the conservative form of time-dependent Reynolds-averaged Navier-Stokes equations (RANS) over a grid system. Generally, the governing RANS and continuity equations for incompressible, constant-property flows are written as follows (Wilcox, 2007):

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = -\frac{1}{\rho}\frac{\partial p}{\partial x_i} + \frac{\partial}{\partial x_j}\left(2\nu S_{ij} + \tau_{ij}\right) + g_i \tag{2}$$

in which u_i are the velocity (m/s) in the x_i directions that are x, y, z-directions in the Cartesian coordinates; t is the time (s); p is the pressure (Pascal); ρ, ν are the density and kinematic viscosity, respectively; g_i is the gravitational force in the subscript directions; $S_{ij} = \frac{1}{2}\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)$ is the strain-rate (1/s); τ_{ij} represents the Reynolds stresses tensor (m^2/s^2).

To numerically solve the rapidly varying flow through the gated spillways, Volume of fluid (VOF) (Hirt and Nichols, 1981) method would be used in this study. The shape of the free surface is determined by computing the fraction of each near-interface cell of a fixed grid that is partially filled.

To closure the RANS equations, the Reynolds stresses tensor τ_{ij} must be known. The solution for the Reynolds stresses τ_{ij} based on some hypotheses, such as linear or non-linear eddy-viscosity hypothesis, Reynolds transport equation, etc. This solution has been obtained by many researchers. For instance, the $k-\varepsilon$ model, that uses the linear eddy-viscosity hypothesis, for the determination of τ_{ij} is written as follows:

$$\tau_{ij} = 2\mu_T S_{ij} - \frac{2}{3}k\delta_{ij} \tag{3}$$

Where μ_T is the eddy viscosity (m^2/s), k is the turbulent kinetic energy (m^2/s^2), ε is the turbulence dissipation (m^2/s^3), δ_{ij} is the Kronecker symbol. On the other hand, the RSM closes the RANS equations by solving transport partial differential equations for the Reynolds stresses together with an equation for the dissipation rate. In this study, Reynolds stress model and Realizable $k-\varepsilon$ model were used to simulate the flow through the vertical gated spillway. More details about these models can be found in Ansys theory guide (2009). The reason for the choice of such turbulence models is due to these models could predict properly the characteristics of rapidly varied flow with highly curvilinear streamlines (Tadayon, R. and A. Ramamurthy, 2009).

3.2 Numerical Model Implementation

The computational domain for the numerical modeling is shown in Fig. 2. Dimensions of the modeling region are 6 m in length and 0.7 m in height. In order to ensure that the numerical results could be compared precisely to the physical model, the width of the numerical modeling, firstly, was taken to L_m= 0.4 m (Y – direction) for 2-D model (Ansys/Fluent). In addition to investigate the effect of the side contraction on the discharge capacity due to the piers and abutments, secondly, three 3-D spillway modeling would be performed. The total width of 3-D models, which includes two-half abutment and a pier, is equal to B_{3Dm} = 0.5068 m.

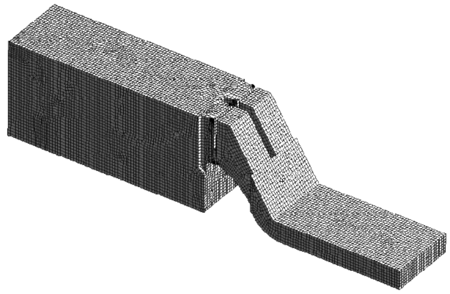

Fig. 3 3-D Mesh for simulation purpose

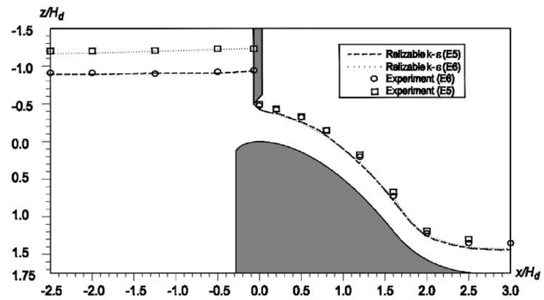

Fig. 4 Water surface profile (E5-E6 cases-Realizable k-ε model)

The mesh sizes were the unstructured quadrilateral (in 2-D model), and irregular hexahedral mesh (in 3-D model). Both of these mesh sizes Δh are approximately Δh = (0.02 – 0.05)H_d. This mesh spacing is appropriate for the size of current computational domain according to the research of Kim and Park (2005) or Thanh N.C and Wang.L.L (2014).

3.3 Boundary conditions

Boundary conditions (BCs) of given flow are shown in Fig. 2. The boundaries of the mesh and their coordinate directions were set as follows: X_{min} – hydrostatic stagnation pressure condition with total energy E_o with a hydrostatic pressure distribution ($p_{bcs} = E_o$ with $V_o = 0$); X_{max} - outflow; Y_{min} and Y_{max} - free slip/symmetry for 2-D model and no slip wall for 3-D model; Z_{min} – wall with no slip condition; Z_{max} - pressure boundary with a gauge pressure equal to zero (atmospheric). The spillway and the vertical gate obstacle boundary were modeled as a surface with no slip condition. With this configuration, the flow moves from left to right between the no slip floor.

The surface roughness height of the spillway model was selected as 0.05 mm in these numerical modeling. Kim and Park (2005) indicated that the effect of the small roughness height (0.05 mm) on the numerical simulation is insignificant compared with the hydraulically smooth surface. Therefore, the numerical modeling of hydraulically smooth surface would be suitable in this study due to the spillway model was fabricated by Plexiglas material.

4. RESULTS AND DISCUSSION

4.1 Discharge flow-rate comparison

The discharge flow-rates from the numerical simulation and the physical model in six test cases are summarized in Table 2. In order to quantify the deviation, it was appropriate to compute the relative difference (*%Diff*) between computed (X_{com}) and measured values (X_{meas}). The value of *%Diff* has the following expression:

$$\%Diff = \frac{X_{meas} - X_{com}}{X_{meas}} 100\% \quad (4)$$

Additionally, an empirical discharge formula through a standard vertical gated spillway with WES profile, which is suggested by USACE (1992), is presented as follows:

$$\frac{Q_g}{Q_d} = \frac{C_{dg}}{C_d}\left(\frac{H_o^{3/2} - H_{o1}^{3/2}}{H_o^{3/2}}\right) \quad (5)$$

Table 2. Comparison between simulated and measured discharge (m3/s)

Test case	E1	E2	E3	E4	E5	E6
Measured data	0.042	0.055	0.039	0.054	0.047	0.057
Equation (5)	0.0425	0.059	0.038	0.056	0.048	0.060
%Diff	-1.09%	-7.25%	+1.96%	-4.31%	-0.47%	-4.52%
Realizable k- ε model	0.0418	0.054	0.0401	0.054	0.049	0.059
%Diff	+0.48%	+0.82%	-3.99%	-1.54%	-4.16%	-1.85%
RSM model	0.043	0.056	0.0403	0.054	0.050	0.059
%Diff	-1.62%	-2.10%	-4.41%	-0.95%	-4.67%	-1.98%

According to Table 2, the differences between the physical and the numerical models are not significant and they are in a range of +0.48% ~ -7.29%. Furthermore, the Realizable k-ε model's results predict more accurately than the other models (*%Diff* = +0.48 ~ -4.16%).

The results of Eq. (5) were also compared to the experimental data and presented in Table 2. In Eq. (5), Q_g is the gate-controlled discharge (m^3/s) at total head H_o(m) and gate opening G_o(m). H_{o1} is the water head from the TEL to gate lip (m). C_{dg} is the discharge coefficient for vertical gate opening. Q_d is the free-flow discharge at the total head H_o(m^3/s). C_d is the discharge coefficient and equal to 0.495 for the free-flow (USACE, 1992). The other parameters are shown in Fig. 2. Following to USACE (1992), the relation C_{dg}/C_d varied slightly with the discharge ratio Q_g/Q_d, but could be assumed as unity. As shown in Table 2, the discharge predicted by Eq. (5) is close with the measured data.

4.2 Water surface profiles comparison

Regarding the water surface profile, Fig. 4 shows a good agreement between the experimental results and the prediction of the E5 and E6 cases on the upstream of the vertical gate using the Realizable k-ε model. However, the predicted water depth is under-estimated as compared to the measured data on the downstream of the gate. The maximum value of *%Diff* between the physical and the numerical models is approximately +7.2% in these cases. Moreover, the predictions of the RSM model were essentially within ±3% of the Realizable k-ε model prediction. For clarity, only the Realizable k-ε data are shown in Fig. 4. It is clear that the predicted water depths are lower than the measured data in all test cases. The reason for this deviation is that the effect of side friction in numerical modeling may be smaller than in the experiment that was conducted in the laboratory flume. Apart from the influence of friction on the side wall, the occurrence of the aeration phenomenon (white flow) also affects the precision of water surface profile measurements in the laboratory.

4.3 The effect of side contraction on the discharge capacity due to pier and abutment

In the 2-D scheme, numerical models reproduce a unit discharge without considering the side contractions at all. In order to apply 2-D results to an actual gated spillway with a side contraction, an effective length has to be considered. In general, there is no specific expression for calculating the effective length in gated spillways. The empirical formula for the effective length L_{eff} (m), which is commonly used for the free-flow Ogee crests, could be found in USACE (1992) as follows:

$$L_{eff} = \lfloor L_m - 2(NK_p + K_a)H_o \rfloor \qquad (6)$$

Consequently, the actual discharge of gated spillway is determined by a following formula:

$$Q_r = q_{2D} L_{eff} \qquad (7)$$

Where Q_r is the actual discharge (m^3/s) that considers the effect of side contraction, and q_{2D} is the unit discharge (m^3/s/m) that is deduced from the 2-D simulation and the Realizable k–ε model; $L_m = b(N+1)$ is the net length of crest (m); b=0.2 is the width of a bay (m); N=1 is the number of the piers (as shown in Fig. 2). In relation to the geometry of the pier, the abutments and the criteria of the USACE (1992), the values of K_p and K_a are taken to be equal to 0.01 and 0.1, respectively. To compare the predicted discharge between the 2-D and 3-D models, three 3-D spillway models (see Fig.3) were performed to obtain the discharge capacity Q_{3D} (m^3/s) by using the Realizable k–ε model.

Table 3. Discharge in 2-D and versus 3-D numerical modeling (Realizable k-ε model)

Test case	q_{2D} (m^3/s/m)	L_{eff} (m)	Q_r (m^3/s)	Q_{3D} (m^3/s)	$\%Diff_{2D}$
E1	0.105	0.355	0.037	0.038	–2.7%
E2	0.136	0.329	0.045	0.046	–2.22%
E3	0.100	0.365	0.037	0.037	–0.5%
E4	0.136	0.344	0.047	0.048	–2.13%
E5	0.123	0.359	0.044	0.045	–2.27%
E6	0.146	0.346	0.051	0.052	–1.96%

As shown in Table 3, the 3-D simulation (Q_{3D}) match the predictions of the extrapolations of 2-D results (Q_r) within difference $\%Diff_{2D}$ from −0.5% ~ −2.7% following Eq. (7). This suggests that the expression for calculating the effective length developed for the free-flow ogee crests is a relatively good approximation to account for the side contractions for flow through the vertical gated spillways based on numerical modeling results.

4.4 Crest pressure distribution assessment

The negative pressure values over the computational domain on the crest spillway behind the vertical gate are shown in Fig. 5 for two cases, namely, the E1 and E5 cases. The number bar and the color indicate the magnitude of the gauge pressure value (Pa). As can be seen from Fig. 4, the negative pressure appears in these cases when the total head is still less than or equal to the design head ($H_o \le H_d$).The reason for appearance of negative pressure on the spillway crest behind the gate opening is due to the increase of the velocity of the flow under orifice flow condition. As a result, when the spillways usually work under the gate opening regime, a flatter downstream profile could be applied instead of WES profile, e.g., parabolic profile. This could help reducing the negative pressure on the spillway surface.

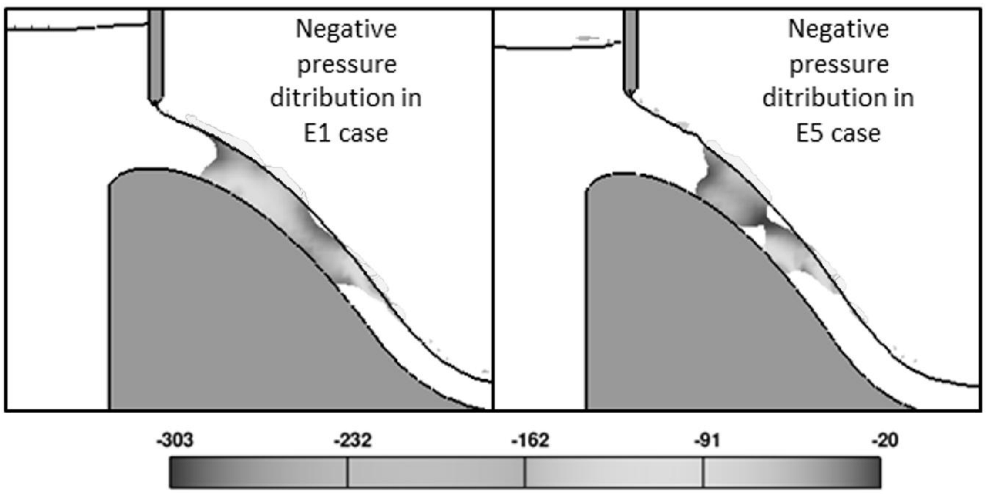

Fig. 4 Negative crest pressure value over the computational domain (Pascal)

Table 4 presents the absolute gauge pressure difference (cm) of water level for the physical and numerical model at a given x/H_d position. In which, x is the horizontal distance from the crest axis (m) (Fig. 2). The absolute pressure difference is defined as $\Delta h_p = h_{pex} - h_{pnu}$, where h_{pex} is the pressure head (cm) in the physical model and h_{pnu} is the calculated pressure head (cm) from the numerical model.

Table 4. Difference between simulated and measured crest pressure head Δh_p (cm) using Realizable k-ε model

x/H_d	0.0	0.2	0.5	0.8	1.2
h_{pex_E1}	2.90	0.80	-0.80	-0.40	0.30
h_{pnu_E1}	2.60	0.40	-1.00	-0.56	0.40
Δh_{p_E1}	0.30	0.40	0.20	0.16	-0.10
h_{pex_E2}	2.20	-1.40	-4.00	-2.05	-0.70
h_{pnu_E2}	2.03	-1.52	-4.22	-2.15	-0.85
Δh_{p_E2}	0.17	0.12	0.22	0.1	0.15
h_{pex_E3}	3.60	1.61	1.00	0.12	0.08
h_{pnu_E3}	3.48	1.28	0.60	0.10	0.05
Δh_{p_E3}	0.12	0.33	0.40	0.02	0.03
h_{pex_E4}	2.50	-1.19	-3.20	-1.88	-0.59
h_{pnu_E4}	2.10	-1.50	-3.50	-2.00	-0.58
Δh_{p_E4}	0.40	0.31	0.30	0.12	-0.01
h_{pex_E5}	3.70	1.11	-0.92	-0.38	0.08
h_{pnu_E5}	3.20	0.80	-1.20	-0.50	0.04
Δh_{p_E5}	0.50	0.31	0.28	0.12	0.04

h_{pex_E6}	2.70	-0.89	-3.00	-2.08	-0.59
h_{pnu_E6}	2.30	-0.90	-3.40	-1.92	-0.70
Δh_{p_E6}	0.40	0.01	0.40	-0.16	0.11

5. SUMMARY AND CONCLUSIONS

In this study, the flow characteristics of a standard vertical gated spillway, such as the discharge, water surface profile, crest pressures, and the effect of side contraction on the discharge capacity are investigated by using a commercial CFD software packages, namely, the Ansys/Fluent. Two turbulence models, namely, the $k-\varepsilon$, Realizable $k-\varepsilon$ and Reynolds stress model along with VOF scheme, which were integrated in these packages, were applied in the numerical modeling. The experimental data was conducted in the laboratory flume and used to validate the numerical simulations. The important results comprise the following: 1) Both the numerical results and the empirical formula predicted well the discharge and water surface profile in comparison with the experimental data. The range of relative difference is −7.29% ~ +0.48 for the discharge and −7.2% ~ +0.8 for the water surface profile. 2) The formulation for computing the effective length in free-flow spillways can be a good approximation for calculating the actual discharge of the vertical gated spillway basing on the 2-D numerical modeling. 3) The simulated crest pressures distribution is smaller than the experimental data in almost cases. Furthermore, the negative pressure appears in both numerical and physical models even if the total head is less than or equal to the design head ($H_o \le H_d$). A further research is necessary to improve the confidence as using the CFD models in the flow simulation of gated spillways.

REFERENCES

Slisskii, S,(1986), *Hydraulics structures with high head*, Moscow Construction publisher. 304, Moscow (in Russian).

USACE, (1992), *EM 1110-2-1603 Hydraulic Design of Spillway*, Washington DC, US.

Bombac, M., et al, (2014), *Numerical and physical model study of a vertical slot fishway,* Journal of Hydrology and Hydromechanics. 62(2), 150-159.

Bhosekar, V., V. Jothiprakash, and P. Deolalikar, (2011), *Orifice Spillway Aerator: Hydraulic Design,* Journal of Hydraulic Engineering. 138(6), 563-572.

Wilcox, D.C, (2007), *Turbulence Modeling for CFD* 3rd ed, DCW Industries Inc. 600, California.

Hirt, C.W. and B.D. Nichols, (1981), *Volume of fluid (VOF) method for the dynamics of free boundaries,* Journal of Computational Physics. 39(1), 201-225.

Ansys, I, (2009), *Ansys Theory Guide*, Ansys sofware V12.1.

Tadayon, R. and A. Ramamurthy, (2009), *Turbulence Modeling of Flows over Circular Spillways,* Journal of Irrigation and Drainage Engineering. 135(4), 493-498.

Kim, D. and J. Park, (2005), *Analysis of flow structure over ogee-spillway in consideration of scale and roughness effects by using CFD model,* KSCE Journal of Civil Engineering. 9(2), 161-169,.

Nguyen C.T. and Wang L.L, (2014), *Physical and Numerical Model of Flow through the Spillways with a Breast Wall,* KSCE Journal of Civil Engineering. 19(7), 2317-2324.

Tóm tắt*:*

THÍ NGHIỆM MÔ HÌNH THỦY LỰC VÀ MÔ PHỎNG SỐ DÒNG CHẢY QUA ĐẬP TRÀN CÓ SỬ DỤNG CỬA VAN PHẲNG

Dòng chảy qua đập tràn mặt cắt thực dụng sử dụng cửa van phẳng là dòng biến đổi gấp và có độ cong lớn. Trong nghiên cứu này, một số đặc trưng của dòng chảy qua đập tràn mặt cắt thực dụng

sử dụng cửa van phẳng sẽ được nghiên cứu bằng phần mềm tính toán động lực học dòng chảy (CFD), Ansys/Fluent, kết hợp với hai mô hình dòng chảy rối, cụ thể là Realizable $k-\varepsilon$ *và mô hình ứng suất rối Reynolds. Thí nghiệm mô hình thủy lực sẽ được sử dụng để đánh giá mức độ chính xác của những mô hình tính toán trên. Kết quả so sánh đường mặt nước và lưu lượng tháo giữa thí nghiệm và mô phỏng số (mô hình 2 –D) có sai số trong khoảng +0.48∼-7.29%. Ngoài ra, mô hình 3-D cũng được mô phỏng để xét đến mức độ thu hẹp của dòng chảy qua cửa van khi kể đến ảnh hưởng của trụ pin và trụ biên. Kết quả chỉ ra rằng việc sử dụng công thức tính toán chiều dài ngưỡng hiệu quả ap dụng cho tràn tự do là chấp nhận được trong trường hợp này.*
Từ khóa: CFD, mô hình dòng chảy rối, đập tràn mặt cắt thực dụng, cửa van phẳng, lưu lượng.

BBT nhận bài: 25/2/2016
Phản biện xong: 06/4/2016