NUMERICAL SIMULATION OF FLOW OVER SPILLWAY: A CASE STUDY OF TAHOET RESERVOIR

MÔ PHỎNG SỐ DÒNG CHẢY QUA TRÀN XẢ LŨ: ÁP DỤNG CHO TRÀN XẢ LŨ TA HOÉT

NGUYEN SY TRUONG[a], VU HUY CONG[b,*], NGUYEN VAN HUONG[b]
[a]Hydraulic Engineering Consultants Joint Stock company No.3
[b]University of Science and Technology - The university of Danang
[*]Corresponding author: *Email:* vhcong@dut.udn.vn
Received 20/9/2023, Revised 14/11/2023, Accepted 30/11/2023
https://doi.org/10.59382/j-ibst.2023.vi.vol3-5

Abstract: *The spillway is one of the most important construction components in the irrigation system; it is the key to ensuring the safety of the reservoir. In this study, the 3D-Fluent numerical model is used to simulate the hydraulic regime in the Ta Hoet spillway. The 8 simulation cases include different gate opening modes and variable flood flows. The change in velocity, water surface, and other hydraulic factors on the spillway have been studied in detail. The results also show that the direction of flow into the spillway is not straight but curved, resulting in different hydraulic regimes at the valve gates. The model results have been calibrated and show excellent accuracy when compared to the experimental data.*

Keywords: *spillway; Ansys Fluent; CFD; Ta Hoet*

Tóm tắt: *Tràn xả lũ là một trong những công trình quan trọng trong hệ thống đầu mối công trình thủy lợi, nó là chìa khóa đảm bảo sự an toàn cho công trình hồ chứa. Trong nghiên cứu này chế độ thủy lực trong tràn lũ Ta Hoét đã được mô phỏng bằng mô hình số 3D - Fluent. Có 8 kịch bản mô phỏng được thực hiện với các tần suất lũ khác nhau và các chế độ mở cửa van khác nhau. Sự thay đổi về vận tốc, đường mặt nước cùng các yếu tố thủy lực khác đã được mô phỏng chi tiết. Kết quả cũng cho thấy, hướng dòng chảy vào tràn bị uốn cong đã dẫn đến các chế độ thủy lực khác nhau tại các cửa van. Kết quả mô phỏng đã được kiểm định so sánh với kết quả thực nghiệm và cho thấy độ chính xác cao.*

Từ khóa: tràn xả lũ, Fluent, CFTD, Ta Hoét

1. Introduction

In the design, management, and operation of Vietnam's reservoir system, ensuring the safety of reservoirs and other structures like spillways, culverts, etc., is always crucial. These constructions' inaccurate design or inefficient operation will result in harm and unanticipated effects, particularly when a dam fails. Therefore, careful consideration must be given to the hydraulic regime of these structures in response to a variety of unfavourable situations at every stage of project design and operation. In current practice, the design work will apply theoretical formulas or semi-empirical formulas based on the project level, design stage, and complexity of the type of flood discharge project [1], [2], [3], [4]; or require physical model experiments [5]. Reality demonstrates that applying theoretical formulas might be constrained and result in inaccuracies while adopting experimental models will be financially expensive, particularly when many scenarios need to be developed. For a very long time, mathematical models have been a useful tool for simulating hydrodynamic issues and efficiently assisting design and operation testing. Numerous research have been conducted by scientists to simulate spillway flow.

Harlow used a numerical model to simulate the flow with a free surface over time [6]. Savage used numerical modeling to simulate the hydraulic regime [7] or Cook performed a 3-dimensional simulation for the entire downstream of the dam [8]. In addition, other studies can be mentioned, such as [9], [10], [1], [11], [12], [13]. The research mentioned above demonstrates the advantages of numerical models for simulating flow in hydraulic work, particularly when a variety of scenarios need to be generated. In this paper, the hydraulic regime on the Ta Hoet spillway was simulated using the Ansys Fluent 3D numerical model with 8 different cases linked to flood levels, number of gates opening, and opening degree. The simulation results are compared with experimental results obtained from published documents.

2. Methodology

2.1 Parameters of the Ta Hoet spillway

Ta Hoet spillway is a surface flood discharge project with three control gates. The shape of the spillway is shown in figure 1. There are baffle blocks to dissipate energy in the stilling basin located at the end of the chute spillway. Parameters of the Ta Hoet spillway are presented in Table 1.

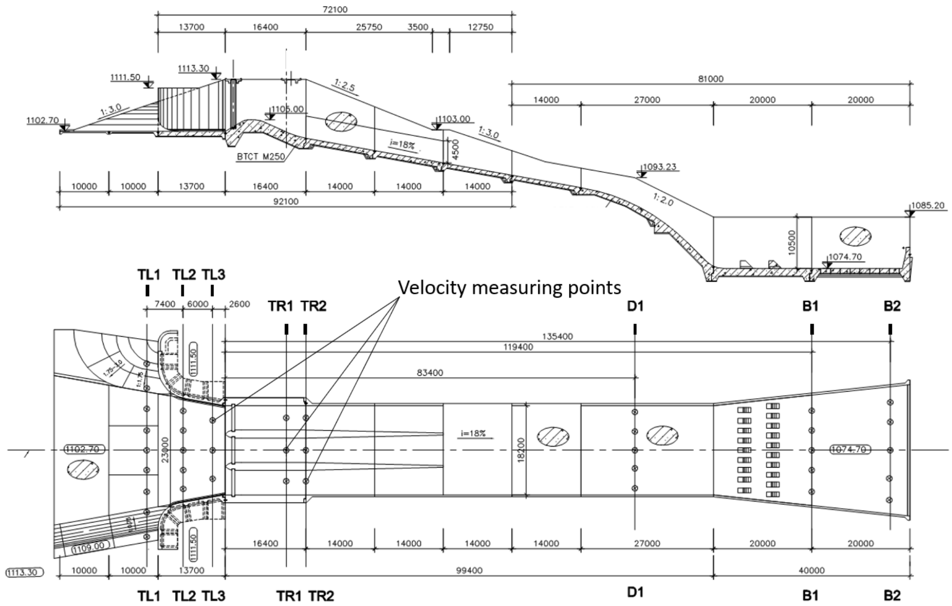

Figure 1. *Ta Hoet spillway*

Table 1. *Parameters of the Ta Hoet spillway*

No.	Spillway	Unit	Value
1	Crest level	m	+1.105,00
2	Number of gates		3
3	Dimension of gates (BxH)	m	5x5.5
5	Designed flood discharge Q_{tk} (P=1,0%)	m^3/s	430,0
6	Designed flood head H_t (P=1%)	m	6,50
7	Checked flood discharge Q_{kt} (P=0.2%)	m^3/s	565,0
8	Checked flood head H_{kt} (P=0.2%)	m	7,8
9	Length of chute	m	56,0
10	Width of chute	m	18,2
11	Slope of chute	%	18
12	Length of waterfall after leaving the chute	m	27,0
13	Form of energy dissipation		Bottom friction
14	Length of stilling basin	m	40,0

2.2 Experimental model

In this study, experimental data are supported by Hydraulic Engineering Consultants Joint Stock Company No. 3, which signed a contract with the experimental unit to use the experimental results. The name of the experimental document is "Summary Report on Hydraulic Experiment, Ta Hoet Reservoir, Lam Dong Province" from the Office of Consulting, Design Appraisal, and Construction Quality Inspection [14]. These data are used to evaluate the model's accuracy. The experimental layout is shown in Figure 2.

This experimental model guarantees that the model and the original have equivalent geometrical conditions. In particular, the spillway model complies with [5] and has a reduction ratio based on similar Froude coefficient (Fr) criteria. Regarding the size scale, the spillway model is scaled with a factor of λ= 30. Other scale parameters are specified in the document [14].

Scenarios are simulated with designed flood levels, checked flood levels, and normal water levels. The velocity was measured along the spillway at 8 cross-sections located at the spillway, on the chute, and in the stilling basin. There are 3 cross-sections upstream of the gate (TL1, TL2, and TL3), 2 cross-sections on the spillway (TR1 and TR2), 1 cross-section on the chute (D1), and 2 cross-sections on the stilling basin (B1 and B2). At each cross section, there are 3–9 points that measure the velocity at depths of 0.2H, 0.4H, and 0.8H. The locations of these points are shown in Figure 1. The laboratory unit estimated and presented the hydraulic regime results for the actual spillway size based on the physical model experimental results [14]. It should be emphasized that the authors did not do the scaling calculation based on the physical model findings, but instead used them. For details on setup, scaling factors, and data for the actual size of the spillway, see the documentation [14]. In order to validate the model, the authors will compare these findings with those from simulations.

Figure 2. *Experimental model of the Ta Hoet spillway and location of gates (from [14])*

2.3 Setup Ansys-Fluent model

Fluent, a CFD program, is used in this study's investigation. The Navier-Stokes equations in the assumption of a Newtonian and incompressible flow are solved using the finite volume method in the Fluent system. The following are the equations for mass conservation:

$$\frac{\partial \rho}{\partial t} + \nabla \cdot \left(\rho \vec{v} \right) = S_m \quad (1)$$

where ρ is the water's density, $\vec{v}$ is velocity, and S_m is sink or source of mass. In the present study, the S_m is zero in the current study.

The momentum equation is:

$$\frac{\partial}{\partial t}\left(\rho \vec{v} \right) + \nabla \cdot \left(\rho \vec{v}\vec{v} \right) = -\nabla p + \nabla \cdot \left(\bar{\bar{\tau}} \right) + \rho \vec{g} + \vec{F} \quad (2)$$

where p is the static pressure, $\bar{\bar{\tau}}$ is the stress tensor, and $\rho \vec{g}$ and $\vec{F}$ are the gravitational body force and external body force, respectively.

The stress tensor, denoted by $\bar{\bar{\tau}}$, is given by:

$$\bar{\bar{\tau}} = \mu \left[\left(\nabla \vec{v} + \nabla \vec{v}^{T} \right) - \frac{2}{3} \nabla \cdot \vec{v} I \right] \quad (3)$$

where μ and *I* are the dynamic viscosity and the unit tensor, respectively.

The turbulence model *k-ε* is used in this study. This model is widely applied when simulating hydrodynamic regimes. The configuration of the boundary conditions is shown in Figure 3, along with additional details. The size of the spillway in the simulation model is the same as the size of the real structure.

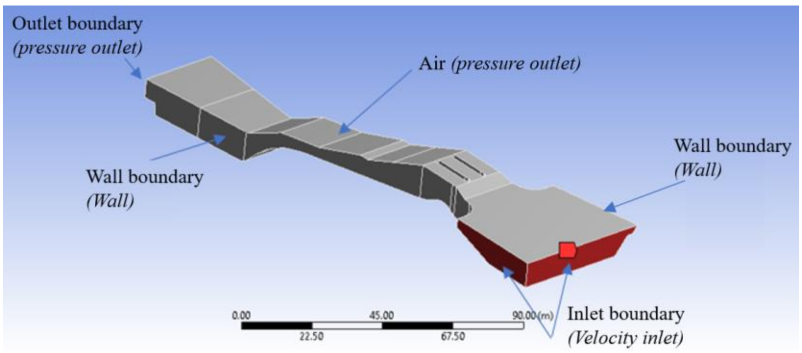

Figure 3. *Model geometry and boundary conditions*

According to Figure 3, a “*velocity inlet*” type is selected at the inlet boundary, allowing the velocity values and vector direction to be adjusted. The assigned velocity values correspond to the scenarios shown in Table 3. The "*wall*" type boundary with the "*No-slip*" option is used for the wing walls and chute slopes. A pressure outlet is set for outlet boundary condition. The upper boundary, where the open surface is in contact with the air, is a "*pressure outlet*" type. The surface tension coefficient between air and water is set equal to 0.072 n/m. The mesh elements of the model are triangular in shape. The total number of mesh elements for the model is around 326388, shown in Figure 4. Figure 4b displays specifics of the mesh's design surrounding spillway piers. The time step is 0.01s. Additionally, table 2 displays other key model parameters.

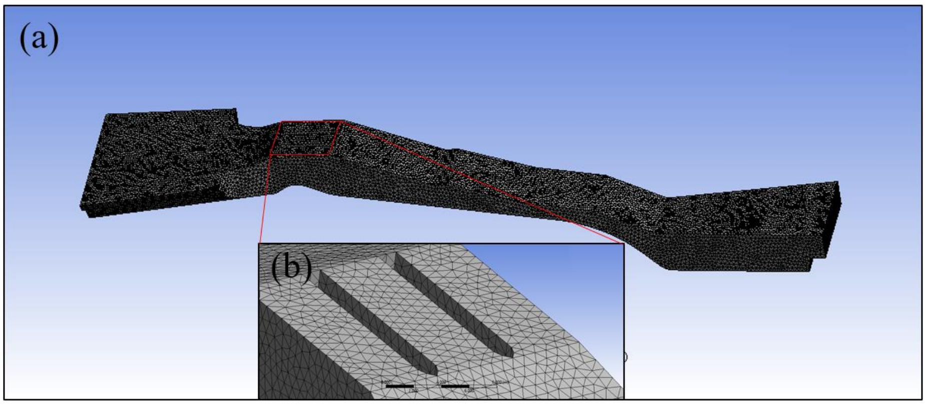

Figure 4. *Computational mesh: (a) Entire domain; (b) Around spillway piers*

Table 2. *Key variables in model setup*

Key variables	Values	Unit
Models	Multiphase/ VOF	
Materials	Water/air	
Turbulent Model	*k-e*	
Pressure velocity scheme	SIMPLE	
Surface tension Coef.	0.072	n/m
Time (steady/transient)	Transient	
Scheme (implicit/explicit)	Explicit	
Gravity	-9.81	m/s²

2.4 Simulation cases

The designed flood level (MNLKT = +1112.80m), the checked flood level (MNLTK = +1111.50m), and the normal water level rise (MNDBT = +1110.00 m) were simulated in the study. These cases corresponded to the number of gates opened and the opening degree at each gate was different. To verify the accuracy of the numerical model, TH1–TH3 were set up based on available experiments. To further study the hydraulic regime on the spillway when there is an operational issue, additional cases (TH4 –TH8) are simulated (see Table 3).

Table 3. *Simulation cases*

No.	Case	Experimental results	Water level	Discharge (m³/s)	Velocity at inlet (m/s)	Opening gates * /opening degree
1	TH1	Yes	MNLKT	565.00	0.78	All 3 gates
2	TH2	Yes	MNLTK	430.00	0.69	All 3 gates
3	TH3	Yes	MNDBT	300.00	0.60	All 3 gates
4	TH4	No	MNLTK	267.00	0.43	Gate 2
5	TH5	No	MNLTK	371.00	0.60	Gates 1, 3
6	TH6	No	MNLTK	371.00	0.60	Gates 2, 3
7	TH7	No	MNDBT	167.16	0.33	Opening degree, a=2m
8	TH8	No	MNDBT	222.31	0.44	Opening degree, a=3m

** Location of the gates, see figure 2.*

3. Results and Discussion

3.1 Compare with experimental results

The results of the simulation for the location of the hydraulic jump and the change in water level coincide with the experiment results. Figure 5 shows the flow velocity increasing on the chute. As the chute comes to an end, the flow velocity increases before reducing in the stilling basin. Hydraulic jump happens near the head of the stilling basin, which is perfectly consistent with the experimental results (see Figure 5b, c).

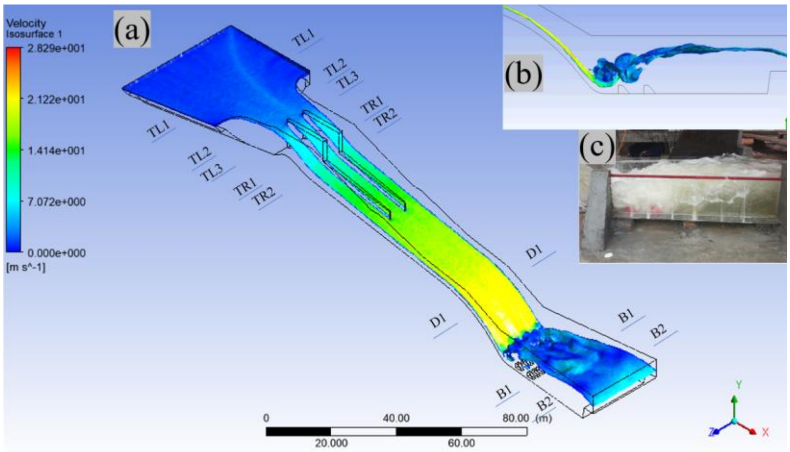

Figure 5. *Water level on the spillway and location of hydraulic jump (case TH1)*

Figure 6 shows the results of a comparison of the maximum velocity between the model and experiment at 8 locations along the spillway corresponding to 3 cases. The data is shown in Table 4.

Table 4. *The maximum velocity of the model and experiment*

Monitoring points	TH1_TN	TH2_TN	TH3_TN	TH1_MP	TH2_MP	TH3_MP
TL1	2.94	2.52	3.25	3.6	2.56	2.36
TL2	4.14	3.32	2.68	4.21	3.16	2.77
TL3	4.63	3.52	3.02	4.22	3.76	2.99
TR1	11.39	10.31	9.32	10.22	9.73	9.3
TR2	12.22	12.15	11.11	11.43	11.52	11.19
D1	16.1	13.68	14.13	19.46	18.68	18.13
B1	7.75	6.8	5.93	8.42	7.34	6.63
B2	6.7	7.49	6.06	6.61	5.55	5.51

For TH1 (see Figure 6a and Table 4), the velocity between the experimental model and the numerical model has high similarity at most locations including the inlet location, near the gates, and the stilling basin location, except for the location at the end of the chute. At cross sections TL1, TR2, and B1, the velocity variations from the experimental data are 0.66 m/s, 0.07 m/s, and 0.09 m/s, respectively. The results also show high precision for TH2 and TH3 (Figure 6b, c), particularly at the inlet position, near the gates, and the stilling basin location.

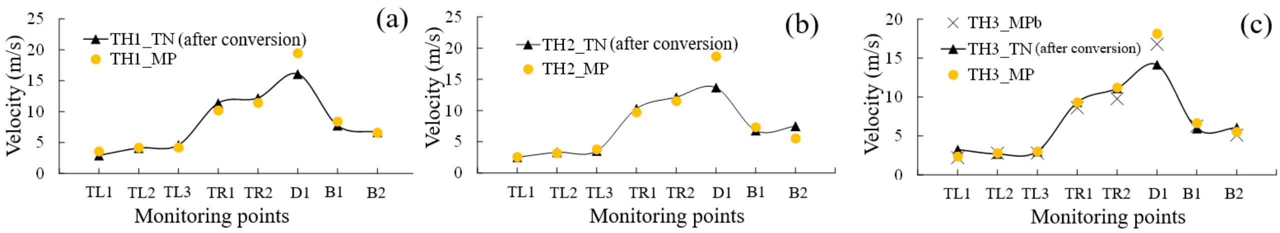

Figure 6. *Results of comparing the maximum velocity between the model and experiment (a): TH1; (b) TH2; (c) TH3*

Two mesh resolutions with a total of 326388 and 36559 grid cells were used to test the grid independence. The simulation results for the two grid resolutions above are represented in Figure 6c by the symbols TH3_MP and TH3_MPb, respectively. These test cases are simulated with normal water level rise (TH3). The results indicate that for 326388 grid cells and 36559 grid cells, the average velocity value for all cross-sections differs from the experimental value by 0.795 m/s and 0.925 m/s, respectively. This demonstrates that the mesh resolution's impact in the above two options is stable. In addition, observing the scaled residuals of the x_, y_, z_velocity, k, epsilon parameter values on the simulation screen reveals that they reach a value of 10^{-4} and remain steady over time. This demonstrates that the model has attained a state of stability. As a results, the proposed grid, which has 326388 grid cells, is then chosen to simulate the following cases. The results additionally show that the boundary conditions and model parameters above can be trusted to simulate further cases.

3.2 Simulation results for cases TH4 -TH8

Cases TH4-TH6

The ability to discharge water decreases in situations when 2 gates (TH4) or 1 gate (TH5) are jammed. The water level is high (MNLTK) and continues to rise, causing an overflow on the top of the gate as shown in Figure 7. Operation is not guaranteed in either of the situations mentioned above. This is consistent with reality when the ability to discharge water drops sharply in the case of MNLTK.

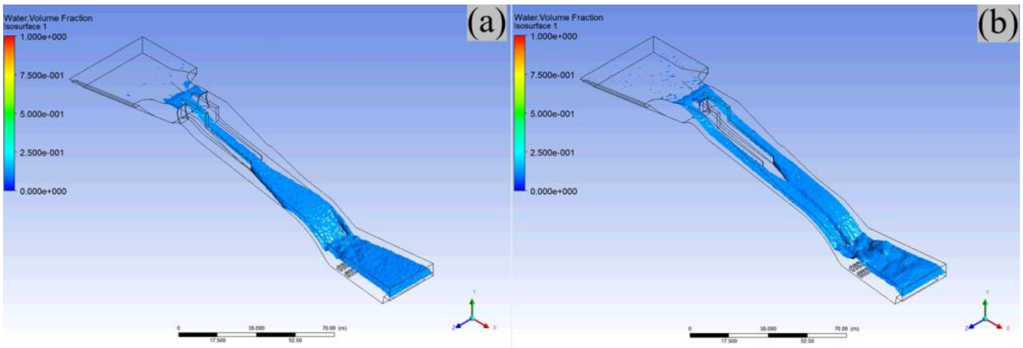

Figure 7. *Water surface corresponding to case TH4 (a), and case TH5 (b)*

The pathlines and velocity distributions along the spillway and gate area for cases TH4 and TH5 are shown in Figures 8 and 9, respectively.

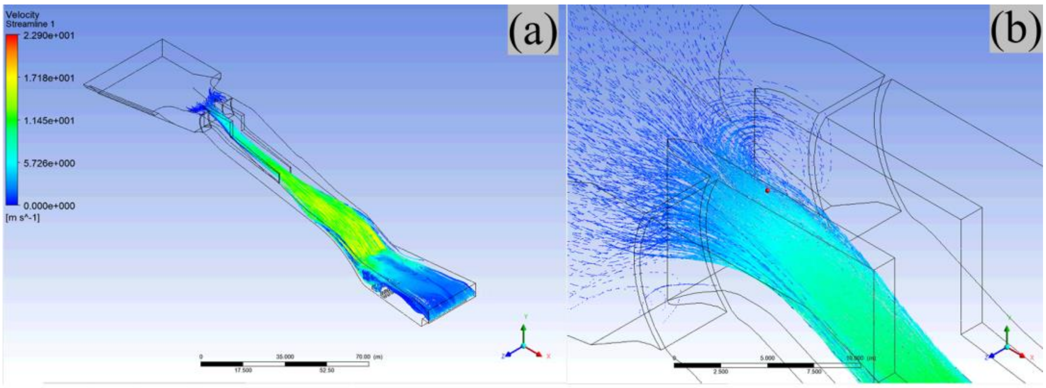

Figure 8. *The pathlines and velocity vectors for case TH4*

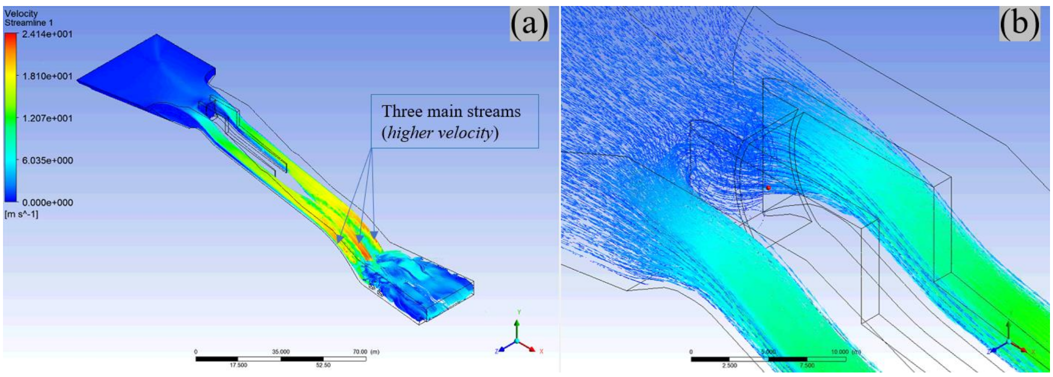

Figure 9. *The pathlines and velocity vectors for case TH5*

Vortices may be observed in the area at the closed gate in both of these cases because the flow is diverted toward the open gate. The flow is coiling at the two side gates in Figure 8b and at the centre gate in Figure 9b. Figure 8a further illustrates the phenomena of the flow from the two gates being separated by the chute's guide walls for a considerable distance.

The velocity distribution along the length of the spillway in both of these cases follows the general rule, that is, after passing the gates, the velocity gradually increases and at the end of the chute, the velocity reaches its maximum value. After passing through the stilling basin, the speed decreases significantly. This also proves that the designed stilling basin works well. An interesting thing can be found in case TH5 (see figure 9a), at the end of the chute, the flow is distributed into three main streams, where the velocity is higher, and they alternate with two minor streams, where the velocity is lower. For case TH6, the Gate 1 is stuck, the flow after passing through the flow direction wall is deflected to the right and there is no flow distribution like case TH5 (Figure 10a).

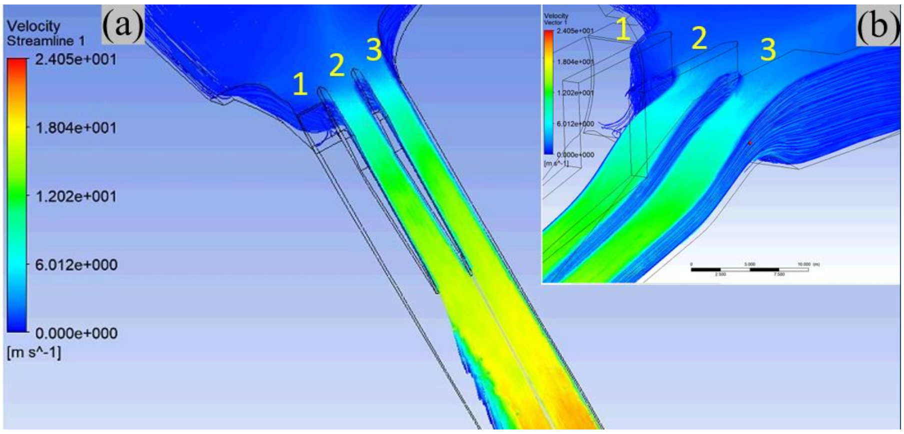

Figure 10. *The pathlines for case TH6*

Cases TH7 and TH8 (Gate opening degree: 2m and 3m)

The gates can be opened to various degrees during operation based on the reservoir's water level and the amount of discharge. The scenario where the gates are opened with a= 2m (TH7) is depicted in Figure 11. In comparison to other situations, the discharge through the spillway is lower, and the velocity along the chute is also lower. The primary and minor flow distribution (shown in figure 9a) cannot be found in this case. Additionally, the flow in the stilling basin is less disrupted, and the height of the disturbed water is also lower than in the earlier case.

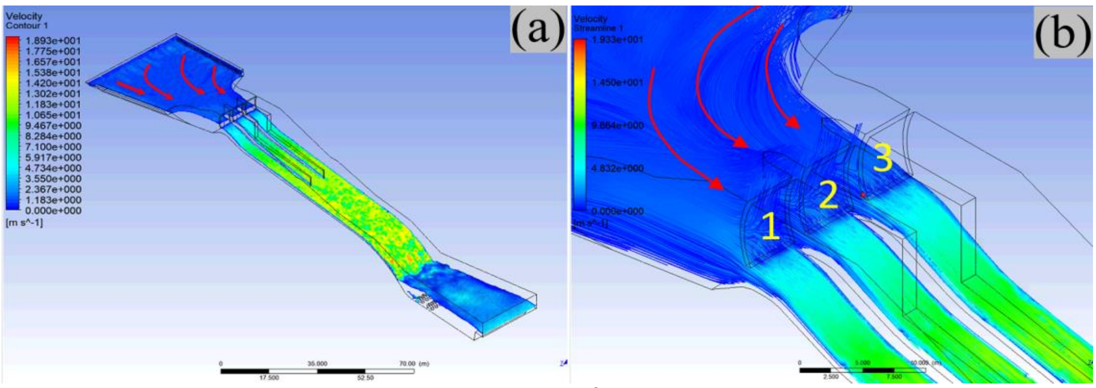

Figure 11. *Velocity distribution: (a) on the water surface; (b) in the area around the gates (TH7)*

The flow characteristics near the gate are shown in Figures 11b and 12. As it passes through the gate, the flow vertically narrows. Small velocities are present in the area in front of the gates, and accelerating velocities are found in the area behind the gates. It's interesting to note that while the phenomenon of water climbing up the gate occurs at Gates 1 and 2, it does not occur at Gate 3. This can be explained by the flow being redirected at upstream (see Figure 12a). The main direction of flow is concentrated at Gates 1 and 2, and less at Gate 3. This is completely consistent with the actual topography of the spillway, which produces such directional flow. The author also displayed boundary conditions in figure 3, where the input boundaries are placed in the middle and one side (inlet), while the remaining boundary is a hard boundary (wall).

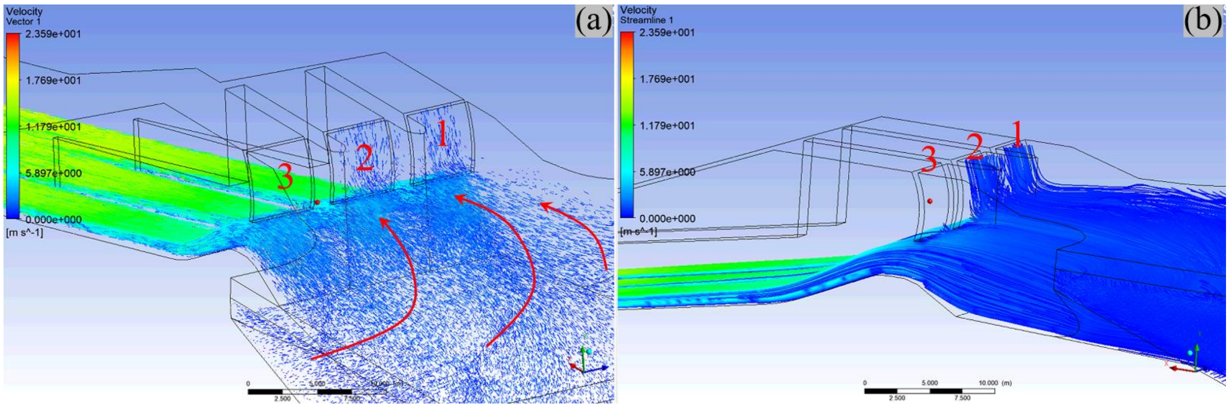

Figure 12. *Flow characteristics in the area in front and behind the gates for TH8: (a) velocity vector; (b) pathline*

4. Conclusions

In this study, the flow through the Ta Hoet spillway was simulated using the Ansys Fluent 3D numerical model. The simulation results were compared with the experimental model. Changes in velocity, water surface, and other hydraulic factors have been simulated in detail. Some results achieved are as follows:

+ Both the location of the water jumping in the stilling basin and the change in velocity over the length of the chute are clearly displayed. Behind the gates, the flow velocity gradually increases until it reaches its peak value at the end of the chute, where it then begins to significantly decrease in the stilling basin. In some cases, the water level will exceed the wall of the stilling basin;

+ The hydraulic regime on the spillway is altered when one or more gates are opened, resulting in the formation of vortices at the closed gates. When two gates are opened, the flow is divided into three main streams with higher velocities that alternate with two minor streams at the end of the chute with lower velocities;

+ The flow in the upstream area is strongly deflected when entering the overflow area due to terrain. As a result, the velocity distribution at the gates could become uneven. Gate 3 has a weaker flow than Gates 1 and 2.

REFERENCES

[1] S. Gu *et al.* (2017), *"SPHysics simulation of experimental spillway hydraulics," Water*, vol. 9, no. 12, p. 973.

[2] TCVN-9147:2012, *"Tiêu chuẩn quốc gia TCVN 9147:2012 về Công trình thủy lợi - Quy trình tính toán thủy lực đập tràn,"* Viện khoa học Thủy lợi Việt Nam, p. https://thuvienphapluat.vn/TCVN/Xay-dung/TCVN-9147.

[3] TCVN-8420:2010, *"Tiêu chuẩn TCVN 8420:2010 Tính thủy lực công trình xả kiểu hở," https//luatvietnam.vn/xay-dung/tieu-chuan-tcvn-8420-2010-tinh-thuy-luc-cong-trinh-xa-kieu-ho-164531-d3.html (accessed Jul. 07, 2023).*

[4] TCVN-12262:2018, *"Tiêu chuẩn Quốc gia TCVN 12262:2018 Công trình thủy lợi - Tính toán thủy lực tràn dạng phím piano".*

[5] TCVN-8214:2009, "TCVN 8214:2009 Thí nghiệm mô hình thủy lực công trình thủy điện, thủy lợi".

[6] F. H. Harlow and J. E. Welch (1965), *"Numerical calculation of time-dependent viscous incompressible flow of fluid with free surface",* Phys. fluids, vol. 8, no. 12, pp. 2182–2189.

[7] B. M. Savage and M. C. Johnson (2001), *"Flow over ogee spillway: Physical and numerical model case study",* J. Hydraul. Eng., vol. 127, no. 8, pp. 640–649.

[8] C. B. Cook, M. C. Richmond, J. A. Serkowski, and L. L. Ebner (2002), *"Free-surface computational fluid dynamics modeling of a spillway and tailrace: Case study of the Dalles project"*.Citeseer.

[9] M. Tabbara, J. Chatila, and R. Awwad (2005), *"Computational simulation of flow over stepped spillways",* Comput. Struct., vol. 83, no. 27, pp. 2215–2224.

[10] N. C. Thành (2013), *"Ứng dụng mô hình dòng chảy rối trong tính toán dòng chảy tự do qua đập tràn",* Tạp chí Khoa học Kỹ thuật Thủy lợi và Môi trường, tập 43, pp. 27–34.

[11] Đ. X. Khánh, L. T. T. Nga và H. V. Hùng (2018), *"Ứng dụng phần mềm Flow-3D tính toán vận tốc và áp suất trên đập tràn thực dụng mặt cắt hình cong"*, *Tạp chí Khoa học Kỹ thuật Thủy lợi và Môi trường*, tập 61, pp. 99–106.

[12] S. Damarnegara, W. Wardoyo, R. Perkins, and E. Vincens (2020), *"Computational fluid dynamics (CFD) simulation on the hydraulics of a spillway",* in *IOP Conference Series: Earth and Environmental Science*, vol. 437, no. 1, p. 12007.

[13] S. Gu, W. Zheng, H. Wu, C. Chen, and S. Shao (2022), *"DualSPHysics simulations of spillway hydraulics: A comparison between single-and two-phase modelling approaches"*, J. Hydraul. Res., vol. 60, no. 5, pp. 835–852.

[14] Liên danh Công ty cổ phần tư vấn xây dựng Thủy lợi 3 (HEC3), Công ty TNHH tư vấn xây dựng 2H Việt Nam và Văn phòng Tư vấn Thẩm định thiết kế và Giám định chất lượng Công trình, "Báo cáo kết quả thí nghiệm mô hình thủy lực hồ chứa nước Ta Hoét," 2021.