FLOW SIMULATION IN A FLIP BUCKET SPILLWAY: DAKPOKEI RESERVOIR CASE STUDY

Vu Huy Cong*

The University of Danang - University of Science and Technology, Vietnam

*Corresponding author: vhcong@dut.udn.vn

(Received: April 16, 2025; Revised: June 11, 2025; Accepted: June 18, 2025)

DOI: 10.31130/ud-jst.2025.23(9B).506E

Abstract - Simulating the hydraulic behavior of an ogee spillway with a free jet energy dissipation form is a complex task due to the involvement of two phases: liquid (water) and gas (air). In this study, the 3D numerical model FLUENT was employed to simulate the hydraulic regime of such a spillway, specifically applied to the Dakpokei reservoir. The results indicate that the main hydraulic characteristics of the flow were successfully captured. The study also investigated the influence of surface roughness on the jet length. It was found that the jet length decreased from 50 meters to 32 meters as the roughness height increased from 0.3 mm to 5 mm. The hydraulic model was validated against experimental data, demonstrating high accuracy and confirming the reliability of the simulation approach.

Key words – Spillway; Flip Bucket; Fluent; roughness

1. Introduction

The spillway is one of the most critical components in irrigation and hydropower systems, playing a key role in flood regulation and ensuring the stability and safety of the reservoir. Primarily functioning to release flood and control the reservoir water level, the spillway acts as a “safety valve” for the entire structure. However, if the spillway is improperly designed or inefficiently operated, the consequences can be severe - particularly under extreme conditions such as prolonged heavy rainfall or floods exceeding the design frequency. A dam failure could result in significant damage to infrastructure and pose serious threats to lives and property in downstream areas.

Therefore, thorough research, analysis, and evaluation of the hydraulic behavior at the spillway are essential throughout all project phases - from design and construction to long-term operation. In particular, considering a range of possible scenarios is crucial for developing optimal solutions that ensure stable and efficient operation under various conditions. Investing in hydraulic simulations and testing hypothetical situations not only enhances design quality but also plays a vital role in risk prevention and in safeguarding both the structure and downstream communities.

In current practice, the design of spillway is often based on theoretical or semi-empirical formulas, depending on the classification of the structure, the design phase, and the complexity of each hydraulic system [1] - [4]. In cases requiring high accuracy or involving complex hydraulic conditions, physical modeling remains necessary to evaluate and verify design solutions [5]. However, experience has shown that theoretical formulas are limited in their ability to fully represent real-world conditions, potentially leading to calculation and design errors. While physical models offer high accuracy, they are time-consuming, labor-intensive, and costly - particularly when multiple scenarios need to be studied. As a result, the use of numerical models has become increasingly popular as an effective research tool. These models offer flexibility in simulating a wide range of operating conditions and serve as a valuable complement to traditional physical modeling.

Over the past several decades, mathematical modeling has been firmly established as an essential and effective tool for simulating hydrodynamic processes, significantly supporting the design, evaluation, and operational assessment of hydraulic structures. Hydraulic simulations using numerical models have been conducted in numerous studies. Harlow was among the pioneers in applying numerical models to simulate unsteady free-surface flows [6]. Subsequently, Savage extended the application of numerical modeling to analyze the overall hydraulic behavior of hydraulic structures [7], while Cook conducted three-dimensional simulations in the downstream area of dams to better understand the flow structures and spatial characteristics [8]. More recently, Vu has carried out simulations of flow over spillways with water jets [9], [10]. In addition, many other notable studies such as [1], [11] - [17] have further enriched the body of knowledge in this field. These studies reveal that the complexity of hydraulic modeling increases depending on the degree of multiphase interaction. In particular, simulating phenomena such as water splashing and breakup in air presents significantly greater challenges than modeling confined channel flows. With the advancement of computational technologies, incorporating recent research directions - such as multiphase interaction modeling, GPU-accelerated solvers, and AI-based turbulence modeling - can provide additional novelty and enhanced capabilities. Nevertheless, traditional modeling approaches remain effective and continue to offer high practical applicability, particularly in real-world spillway design scenarios.

In this study, the hydraulic behavior - particularly the jet trajectory - of the Dakpokei spillway is simulated using the three-dimensional numerical model Ansys Fluent. In addition, the research focuses on evaluating the influence of different surface roughness values on flow characteristics. The simulated water surface profiles are compared with experimental data to assess the model’s accuracy and its potential applicability in real-world engineering practice.

2. Data and Methodology

2.1. Parameters of the Dakpokei Spillway

The Dakpokei spillway features an ogee-shaped profile, as illustrated in Figure 1. Energy dissipation is facilitated by a ski-jump bucket, with a plunge pool located in the downstream section. The key structural and hydraulic parameters of the Dakpokei spillway are summarized in Table 1.

Table 1. *Parameters of the Dakpokei Spillway*

No.	Spillway	Unit	Value
1	Type of spillway		Ogee spillway, energy dissipation by jet plunge
2	Number of bays		3
3	Dimension of bays	m	10
4	Designed flood discharge Q_{tk} (P=1,0%)	m^3/s	222,3
5	Checked flood discharge Q_{kt} (P=0.2%)	m^3/s	332,7
6	Checked flood head H_{kt} (P=0.2%)	m	3,3

2.2. Fluent model

This study employs the student version of Fluent, a CFD program, for the investigation. The following are the equations for mass conservation:

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

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:

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

where p is the static pressure, $\overline{\overline{\tau}}$ 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 $\overline{\overline{\tau}}$, is given by:

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

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

2.3. Model setup

The equations above are solved using the "semi-implicit pressure linked equations" (SIMPLE) algorithm, an efficient method for solving pressure-velocity coupling in computational fluid dynamics. This method is used to enhance the convergence of the equations during the solution process, ensuring the accuracy and stability of the simulation results. The turbulence model applied in this study is the standard *k-ε* model, which is widely used for simulating hydrodynamic regimes, especially in open-channel flows and complex flow structures [18] - [20].

The computational model is depicted in the Figure 1. The size of the spillway in the simulation model is identical to that of the real structure, ensuring that the model accurately represents the physical dimensions of the spillway for the purpose of the hydraulic analysis.

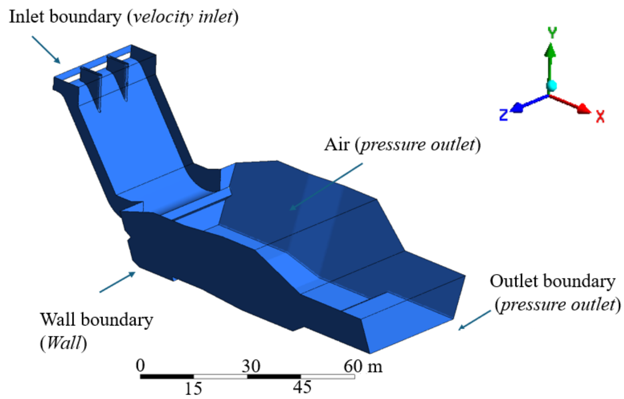

Figure 1. *Geometry of the model and boundary conditions*

The boundary conditions in the simulation are established to ensure the accuracy and stability of the model. Specifically, a no-slip condition is applied at the spillway sidewalls and bottom, where the velocity at the contact surface is set to zero, reflecting the real flow characteristics. At the inlet boundary, the flow velocity is defined along with the water column height to specify the discharge rate. For the outlet boundary condition, a pressure outlet type is employed to ensure dynamic equilibrium of the flow within the computational domain. This boundary condition has been successfully adopted in previous studies and has demonstrated reliable performance in similar hydraulic simulations [20], [21].

The top boundary, where the free surface interacts with the air, is set as a "pressure outlet" to accurately simulate the interaction between water and air. The surface tension coefficient between air and water is set to 0.072 N/m. Additionally, Table 2 presents other key parameters of the model.

Table 2. *Key Variables in the 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/s^2

The computational domain is discretized using a triangular mesh. Regions with complex geometry are meshed with higher resolution to capture detailed flow characteristics, while coarser mesh is applied near the sidewalls to optimize computational efficiency. A grid independence test was conducted using two mesh configurations: (1) a finer mesh with a cell size of 0.4 m near the spillway surface, and (2) a coarser mesh with a cell size of 0.6 m. Figure 2 illustrates the results obtained from both mesh configurations, showing no significant differences in water surface elevation. The Root Mean Square Error (RMSE) between the two cases was 0.02, indicating minimal sensitivity to mesh resolution. Based on these results, the coarser mesh configuration was selected for subsequent simulations. The computational mesh is illustrated in Figure 3. A time step of 0.01 s was adopted for all simulations.

During the simulation process in Fluent, the residuals for velocity components in the x, y, and z directions were maintained below 1×10^{-3}, while the residuals for turbulence parameters k and ε were controlled to be less than 1×10^{-5}. These convergence criteria indicate that the solution achieved a stable and reliable numerical state.

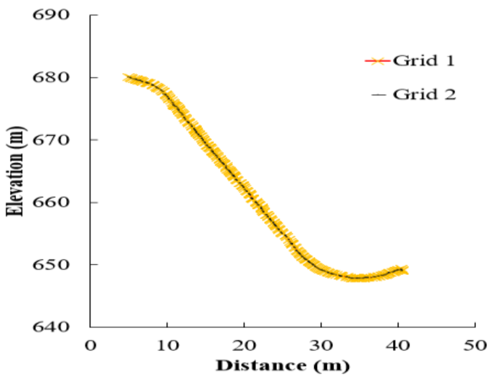

Figure 2. *Water surface elevation profiles corresponding to the two mesh configurations*

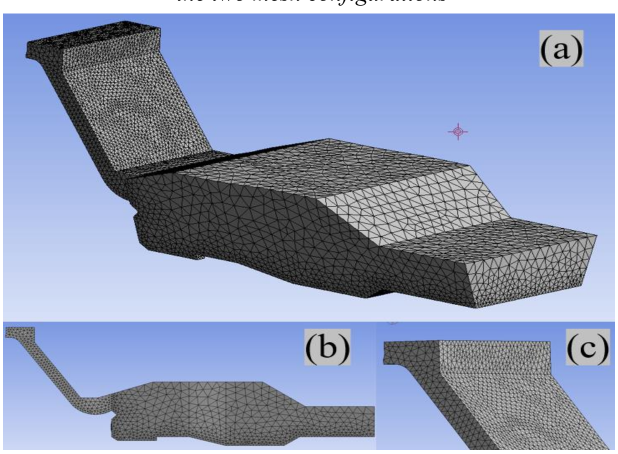

Figure 3. *Computational mesh: (a-b) Entire computational domain; (c) Detailed mesh in the critical region*

2.4. Model validation

In the experimental documentation conducted by the Design Consulting, Appraisal, and Construction Quality Inspection Office, it is clearly stated that the geometric conditions between the prototype structure and the physical model were maintained as equivalent. The model scale was determined based on the principle of Froude number similarity and in accordance with standard [5]. The geometric scale ratio, denoted as λ, was selected as 33. For surface roughness, the corresponding scale ratio is given by $\lambda_n = \lambda^{1/6}$. The roughness height is denoted as R (mm), while the Manning roughness coefficient is represented as n. The relationship between R and n can be referenced from [22].

In the physical model, the spillway surface was made of acrylic glass with a Manning's n of approximately 0.009, which is equivalent to the actual concrete spillway material with n ≈ 0.016. After the experiments were completed, the results were scaled up to prototype dimensions and published in the technical documentation. These results will be used in this study to validate the numerical model and assess its accuracy.

Figure 4 shows the water surface elevation over the spillway for the discharge case of Q = 250 m³/s. The results indicate that the simulated water surface closely aligns with the experimentally measured profile, demonstrating good agreement between the numerical model and physical observations. The RMSE coefficient between the simulated and measured water surface elevations was found to be 0.46.

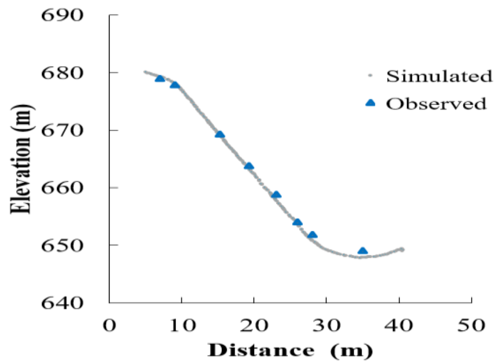

Figure 4. *Comparison of simulated and measured water surface profiles*

3. Results

3.1. Hydraulic flow regime over the spillway

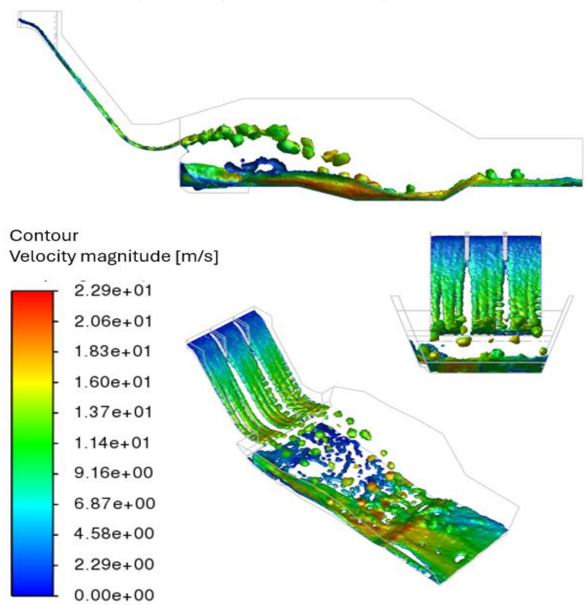

Figure 5. *Hydraulic Flow Regime over the Spillway, inlet velocity =1.5 m/s*

Figure 5 illustrates the simulated hydraulic behavior of flow over the spillway. The results indicate that the model effectively reproduces the hydraulic flow regime, particularly capturing the jet length of the water jet after leaving the spillway. As the flow passes over the crest, it adheres closely to the spillway surface and accelerates. At the crest, the flow velocity is approximately 1.5 m/s and continues to increase along the spillway. At the ski-jump bucket, the velocity reaches a maximum value of about 7 m/s.

The simulation results for different flow rate scenarios are presented in Figure 6.

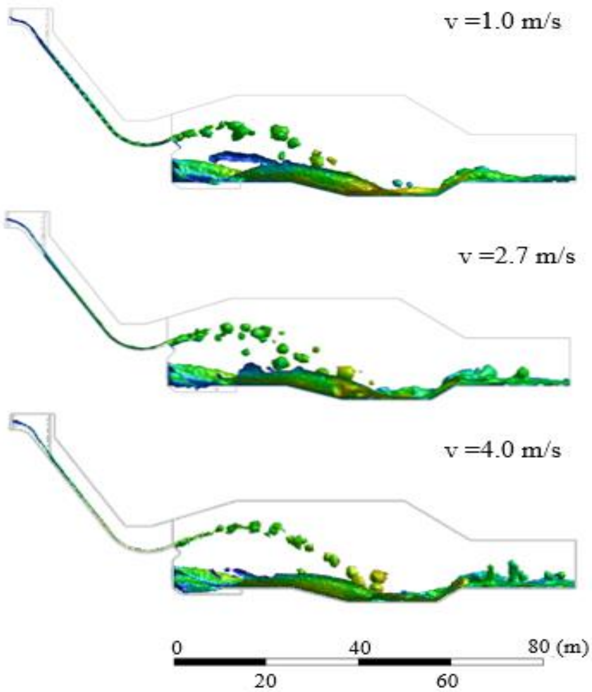

Figure 6. *Jet length at different inlet velocities*

Analysis shows that as the flow velocity at the spillway crest varies from 1 m/s to 4 m/s, the jet length increases accordingly. This is due to the higher kinetic energy of the flow, which allows the water to travel farther after passing over the spillway. All jet flows fall precisely within the location of the plunge pool, demonstrating the appropriateness of the energy dissipation structure design. The position and dimensions of the plunge pool are well-designed, ensuring effective energy dissipation, minimizing downstream erosion, and maintaining the overall safety of the structure.

3.2. Influence of roughness coefficient on jet length

The industry standard 14 TCN 198:2006 provides typical roughness values corresponding to different materials and construction methods. In this study, surface roughness values (R_s) were simulated as 0.3 mm, 2 mm, and 5 mm, corresponding to Manning's n values ranging from 0.011 to 0.017. The simulation results show that surface roughness significantly affects the flow characteristics over the spillway, particularly the jet length, which is a key factor in assessing the downstream flow impact. Figure 7 and Table 3 illustrate the effect of roughness coefficient on jet length. Specifically, as surface roughness increases from 0.3 mm to 2 mm, the jet length decreases from 50 m to 40 m. With a further increase in roughness to 5 mm, the jet length reduces to 32 m.

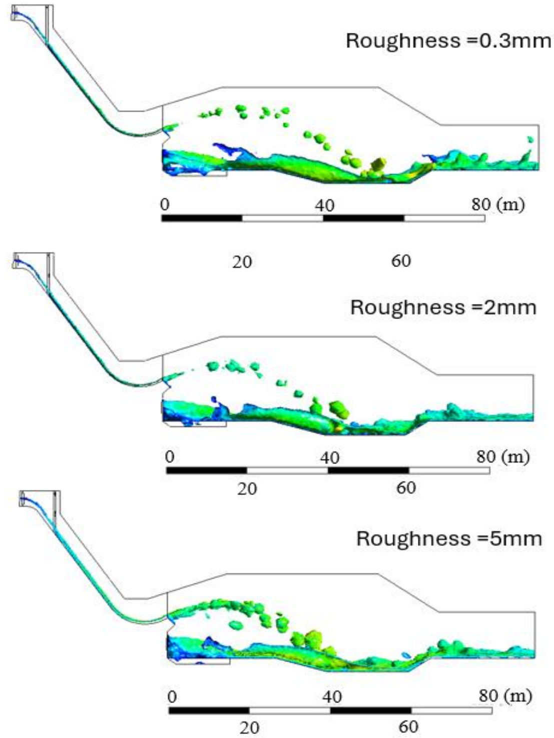

Figure 7. *Influence of roughness coefficient on jet length*

Table 3. *Jet length at different roughness coefficients*

No.	Head over the spillway (m)	Roughness (mm)	Jet length (m)
1	1.7	0.3	50
2	1.7	2	40
3	1.7	5	32

Within the scope of the above data, a non-dimensional function can be established to represent the relationship between Jet length (L), Head over the spillway (H), and Roughness (R_s). Through the process of analyzing and non-dimensionalizing the related quantities, this relationship is expressed as a function between two non-dimensional groups, L/H and R_s/H, specifically as given in below equation. The correlation coefficient, R^2, obtained from the above relationship is 0.96, indicating a very strong correlation between the analyzed dimensionless parameters.

$$\frac{L}{H}=8.01\left(\frac{R_s}{H}\right)^{-0.15}$$

This finding is crucial in designing and adjusting spillway surface materials to control the downstream flow range, ensuring the safety of the structure and the scour pool area. The equivalent roughness heights of 0.3 mm, 2 mm, and 5 mm correspond to different levels of construction quality: smooth concrete surface (good construction), average construction, and poor construction, respectively. Construction quality directly affects the surface roughness, which in turn influences the jet trajectory length of the water flowing over the spillway. Controlling the jet length is a critical factor in minimizing impacts on the downstream area. Therefore, the selection and control of construction methods play a vital role in ensuring hydraulic efficiency and structural safety. Moreover, poor construction quality may result in uneven or irregular areas on the spillway surface, causing local pressure drops that can lead to cavitation - a serious issue that can induce localized erosion and reduce both the discharge capacity and service life of the structure.

4. Conclusion

The study successfully developed and simulated a 3D hydraulic model for a spillway with energy dissipation through jetting. The simulation results indicate a significant increase in flow velocity as it passes over the crest. The flow velocity on the spillway reaches its maximum at the ski-jump bucket. The study also shows that the surface roughness coefficient significantly influences the flow characteristics, particularly the jet length. Specifically, as the roughness increases from 0.3 mm to 2 mm, the jet length decreases from 50 m to 40 m. With a further increase in roughness to 5 mm, the jet length reduces to 32 m. This relationship is also characterized using a dimensionless formulation. These findings provide a crucial basis for selecting appropriate spillway surface construction methods, enhancing energy dissipation efficiency, and ensuring the safety of the structure.

REFERENCES

[1] S. Gu, L. Ren, X. Wang, H. Xie, Y. Huang, J. Wei, and S. Shao, "SPHysics simulation of experimental spillway hydraulics", *Water*, vol. 9, no. 12, p. 973, 2017.

[2] Hydraulic structures − Hydraulic Calculation Process for Spillway, TCVN-9147:2012, 2012.

[3] Hydraulic structure - Calculation opening outlet and rock bed erosion by jetting dissipator, TCVN-8420:2010, 2010.

[4] Hydraulic structures – Spillway - Hydraulic calculation of Piano key weirs, TCVN-12262:2018, 2018.

[5] Hydraulics physical model test of water headworks, TCVN-8214:2009, 2009.

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

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

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

[9] S. T. Nguyen, H. C. Vu, and V. H. Nguyen, "Numerical simulation of flow over spillway: a case study of tahoet reservoir", *Journal of Construction Science and Technology- IBST*, vol. 03, pp. 35–43, 2023.

[10] H. C. Vu, "Effects of roughness on flow behavior of chute spillway", *Journal Science and Technology Water Resources - VAWR*, vol. 4, pp. 33–40, 2023.

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

[12] N. C. Thanh, "The application of turbulent model to simulate the free flow over spillway", *Journal of water resources & environmental engineering*, vol. 43, pp. 27–34, 2013.

[13] Đ. X. Khanh, L. T. T. Nga, and H. V. Hung, "The simulation of flow velocity and pressure on an ogee spillway using flow-3D", *Journal of water resources & environmental engineering*, vol. 61, pp. 99–106, 2018.

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

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

[16] I. S. Parvathi and A. Neelofer, "CFD analysis of flow over ogee spillway with varying upstream face slope", in *Journal of Physics: Conference Series*, vol. 2779, no. 1, p. 12024, 2024.

[17] H. Ibrahim and A. H. Kamel, "Flow Patterns Modeling over Spillway: Review Study", *Salud, Cienc. y Tecnol. Conf.*, no. 3, p. 858, 2024.

[18] A. I. Rajaa and A. H. Kamela, "Performance study of fluent-2D and flow-3D platforms in the CFD modeling of a flow pattern over ogee spillway", *Anbar J. Eng. Sci.*, vol. 11, no. 2, pp. 221–230, 2020.

[19] N. S. Ramya and N. S. Kumar, "Analysis and simulation of flow over stepped spillway using ANSYS-CFD", *Asian Res. J. Curr. Sci.*, vol. 5, no. 1, pp. 155–162, 2023.

[20] U. A. M. Alturfi, H. M. J. AL-Moadhen, and H. S. Mohammed, "Evaluation of computational fluid dynamic model in investigating the hydraulic performance of stepped spillway", *J Eng Appl Sci*, vol. 15, pp. 752–761, 2020.

[21] P. Zaid, "Hydraulic Performance of Moderate Stepped Spillway Using ANSYS-FLUENT Software", *Submitted to the Council of the College of Engineering at Salahaddin University-Erbil in Partial Fulfillment of the Requirements for the Degree of Master of Science in Water Resources Engineering*, 2023.

[22] *Handbook of Hydraulic Calculations,* Thuyloi University, p. 6, 1991.