JEE Journal of Ecological Engineering

Journal of Ecological Engineering 2023, 24(11), 274–286
https://doi.org/10.12911/22998993/171704
ISSN 2299–8993, License CC-BY 4.0

Received: 2023.07.12
Accepted: 2023.08.30
Published: 2023.09.19

CFD Analysis of the Effects of Compound Downstream Slope on Flow Over the Spillway

Udai A. Jahad[1], Ali Chabuk[1], Mohammed A. Alabas[2], Ammar S. Mahmoud[2], Nadhir Al-Ansari[3*], Jan Laue[3]

[1] Department of Environment Engineering, College of Engineering, University of Babylon, Babylon 51001, Iraq

[2] Department of Civil Engineering, College of Engineering, University of Babylon, Babylon 51001, Iraq

[3] Department of Civil Environmental and Natural Resources Engineering, Lulea University of Technology, SE-971 87 Lulea, Sweden

* Corresponding author's e-mail: nadhir.alansari@ltu.se

ABSTRACT

The volume of the stilling basin can be reduced, energy can be dissipated, and floods can be contained with the help of spillways. The aim of this Computational Fluid Dynamics (CFD) study is to investigate how compound slopes change water flows through spillways. To measure turbulence, the Realizable k-ε model was used, and the multiphase volume of fluid (VOF) method was utilized to determine where air and water meet. Five models of spillways with different slopes (normal slope (MS1) = 30°, compound slope(MS2 and MS3) = 20°/39°, and compound slope (MS4 and MS5) = 39°/20°) were modelled and simulated using the ANSYS Fluent software to determine their flow characteristics. Numerical simulation results were compared to experimental results, and it was found that the CFD model captured the key flow aspects accurately. The numerical model carefully observes the several flow patterns (nappe, transition, and skimming) that emerged owing to variations in slope and geometry. When it comes to dissipating energy, models with a compound slope (39°/20°) do the best. When compared to the normal slope model (30°) with a step size of 10, the increase in energy dissipation is 14%. According to the findings, the TKE (turbulent kinetic energy) was elevated by the compound slope. The results of this research show that the spillway can be operated effectively and reliably under a wide range of flow conditions, fulfilling an important goal of the project.

Keywords: compound slope, volume of fuid; spillway; CFD; energy dissipation.

INTRODUCTION

Spillways are a popular and practical choice for many hydraulic structures due to their efficient air entrainment and energy dissipation abilities. Chanson (2022) highlights that these spillways can be identified by their step configuration, which acts as a source of resistance, dissipating energy and slowing the velocity of the flow. As well as this, researchers have suggested that stepped spillways are the most efficient way to control air entrainment and water erosion. Additionally, stepped spillways also have the potential to reduce the formation of whirlpools and turbulent areas in the water flow body. The benefits of stepped spillways make them ideal for many hydraulic structures and make them invaluable when it comes to water management.

The studies conducted by (Wan et al., 2019, Ghaderi et al., 2021, and Ma et al., 2022), have established the importance of step height and chute channel angles in pooled stepped spillways. Moreover, (Baylar et al., 2009) examined the impact of channel slopes on the effectiveness of stepped spillways' aeration. The results showed that for stepped cascade aerators, increasing channel slope led to higher aeration efficiency. This is a substantial increase, and suggests that raising the channel slope could be a good approach to enhance the effectiveness of aeration in stepped spillways. According to Bung (2011), stepped spillways, as opposed to other kinds of hydraulic structures, have a stronger effect on re-aeration and energy dissipation processes. The stepped geometry of stepped spillways, which consists of a sequence of steps that give higher abrasion from a wider variety of flow depths, distinguishes them from other types of spillways. Due to this, it was determined that modifications to the spillway's features, such as slope, the height of macro-roughness, and discharge, had a substantial impact on the flow's velocity and depth. The effectiveness of energy dissipation on stepped spillways with a nappe flow regime was examined in a study by Hamedi et al. (2014). There were many various models for stepped spillways that were examined, including slanted steps with varying end sills. The variables having the largest impact were the height, thickness, and upper angle of end sills. The hybrid model was found to maximize energy dissipation through experimentation.

The subsequent explorations bolstered these conclusions by investigating the flow characteristics over the stepped spillways. To further extend the findings, (Felder, 2013), it was found that the uniform and non-uniform stepped spillways both dissipate energy differently. Also, the two different types of structures almost dissipated energy at the same rate. This result shows that stepped spillways do not necessarily dissipate more energy when the step heights are varied. Roushangar et al. (2014) conducted a study, revealing that stepped spillways are an effective energy-dissipating mechanism. The research highlighted the significant energy loss that occurs in stepped spillways. Therefore, stepped spillways are an essential tool for controlling the energy of overflowing flows and preventing damage. Additionally, the study also revealed that, when compared to traditional straight chutes; stepped spillways were more effective at dissipating energy. This research points to the advantageous status of stepped spillways for managing water flow and emphasizes the benefits of their implementation. Felder and Chanson (2015) have drawn attention to the fact that stepped spillway flows exhibit particularly potent energy dissipation and robust free-surface aeration downstream of the point of air entrainment in comparison to other flow diversion configurations. Geometry, boundary conditions, and turbulence are only a few of the variables that have an impact on the stepped designs' flow behaviour and pattern. Torabi et al. (2018) found that, in regards to the nappe flow regime, energy dissipation was elevated by 15–20% when steps were constructed with roughness elements, as opposed to smooth steps. Thus, the continued research surrounding stepped spillways has similarly indicated that they are a reliable and effective method for dissipating excess flow energy.

Furthermore, Zhou et al. (2020) investigated the ability of stepped spillways to dissipate energy and contrasted their findings with those of smooth spillways. The stepped spillways clearly have better energy dissipation benefits when in a nappe flow regime as opposed to the skimming flow regime. The study's findings were quite positive and suggested that the stepped spillway would make an efficient energy dissipation system. The performance of stepped spillways may be affected by changes in shape or flow intensity. Further study may be conducted to see how this mechanism might be used to manage water resources. According to Salmasi and Abraham (2022), the effects of the spillway's slope and step count on the rate of energy loss are not as important as first thought. However, their research revealed that an increase in the slope and quantity of steps of a stepped spillway will lead to an increase in energy dissipation while maintaining a constant discharge rate. Furthermore, they recommended that, as each stepped spillway has the ability to function differently, the implications should be investigated and evaluated on a case-by-case basis. Their findings have been crucial in enabling civil engineers to create effective spillways that maximize energy dissipation.

Moreover, numerical modelling studies have revealed important information about stepped spillways. Research has shown that by using successive tiers to moderate the difference between the water level and the outflow, energy loss can be minimized and efficient hydraulic performance can be gained. Modelling also enables engineers to identify the optimum design for spillways by gauging their performance in different scenarios, for example, by considering changes in design parameters such as slope angle and step length. Having the ability to accurately simulate the overspill and the loss of momentum due to the steps has allowed engineers to streamline the design process, as well as account for variables such as flow rate and velocity. Ultimately, this leads to more effective restoration projects, better hydraulic performance, and lower energy loss.

Castillo et al. (2014), have proposed various turbulence closure models as a means of resolving attempts to simulate turbulent flows with the Reynolds-Averaged Navier-Stokes equations. These models can range from simple eddy viscosity models, such as the Boussinesq assumption, to complex versions, such as the two-equation models. The effectiveness of the turbulence closure models is heavily dependent on the accuracy of the numerical scheme employed in the simulations. Hamedi et al. (2016) conducted a study to evaluate flow characteristics and energy dissipation for stepped spillways with normal end sills. Using a combination of the experimental approach and the finite volume method with the standard k-ε model, the research team analysed the results and compared them with numerical approaches. The study revealed a reasonable agreement between the experimental and numerical approaches, both for flow characteristics and energy dissipation. This indicates the reliability of the numerical method to accurately compute the characteristics of stepped spillways. This research provides a strong foundation for the idea that numerical tools can be effectively used to calculate the performance of stepped spillways with normal end sills. Dong et al., (2019) made use of state-of-the-art multiphase flow Computational Fluid Dynamics (CFD) models, with the goal of improving the operation of several dams with stepped spillways in China. Using the powerful Reynolds-averaged Navier–Stokes equation, highly accurate simulations of the velocity profile were made possible. The predicted results of these simulations showed a strong agreement with the corresponding experimental measurements, providing insight into how the dams can modify and improve their operation in order to meet modern requirements for water storage and energy generation. This shows great promise for CFD modelling to improve the management of dam systems globally. In addition, Lebdiri et al. (2020) analysed the effects of stepped spillways on the rate of energy dissipation and shear stress levels along the flow walls. To undertake their research, Ansys-Fluent was used to calculate the turbulent k-ε as well as the Volume of Fluid (VOF) in order to solve Navier-Stokes equations. Through their work, the authors found the CFD model to be most beneficial in understanding the internal hydraulic variations that cannot be seen physically. What's more, the results demonstrated the potential of using the CFD approach to investigate hydraulic problems. This indicates the significant role of the CFD model in hydraulic engineering, providing innovative ways to find solutions for hydraulic problems. Yalcin et al. (2023) built on their earlier research to evaluate the suitability of turbulence models for CFD engineering problems. The application of turbulence models to the investigation of the unsteady flow behaviours of stepped spillways was the main focus of the study. The authors created a simulation model employing the k-large eddy simulation (LES), and renormalization group (RNG) turbulence models in order to find this out. The results of the simulations demonstrated that the k-model provides the best agreement for temporal change of pressure, while LES is the most capable for the prediction of velocities, water surface, and energy dissipation rate. Additionally, the RNG turbulence model's optimization near the flow exit improved the simulations' accuracy even more. In conclusion, the study indicates that different turbulence models offer varying levels of accuracy for different types of CFD engineering challenges. Additionally, it has been observed that modifications to the design of stepped spillways will have an immense effect on how water is managed. Consequently, stepped spillways can be seen as a great solution as they are both highly efficient in terms of turbulence and energy dissipation, and their modifications can be used.

The goal of this CFD investigation is to better understand how compound slopes influence flow characteristics through stepped spillways. Using ANSYS Fluent software, various stepped spillway models with varying elevations are modelled and simulated. The Volume of Fluid (VOF) approach is used to discern the boundary between air and water, while the Realizable k-ε model is used to assess disturbances in the flow of stepped spillways. Both of these methods are referred to as the volume of fluid (VOF). All models are simulated with varying discharge rates to investigate the relationship between slope variation and the change in energy dissipation.

MATERIAL AND METHODS

Computational Fluid Dynamics, or CFD for short, is a powerful subject in fluid dynamics that examines a broad variety of different complex flow systems by combining numerical computation with a variety of different data structures. At the core of this method are the Navier-Stoscke equations. Different numerical methods can be used to solve these equations, such as finite element, finite difference, or finite volume. The finite volume (FV) method is preferred by most solvers due to its high accuracy and efficiency. In CFD, the system is usually modelled as a connected set of small-volume elements called finite volumes, which are not directly linked to any physical boundaries. This unstructured mesh formulation is particularly useful when studying solid structures and other complicated systems. The software Fluent is the leader in CFD software that uses the finite volume method to discretise the domain of interest. Fluent is equipped with an array of features to reduce the solution time and memory requirements of the finite volume algorithms.

This study focuses on the numerical simulation of five spillways with compound slopes using Fluent software. To precisely represent the boundary layer, the multiphase volume of fluid (VOF) method was combined with a realizable k-ε turbulent model. The velocity-pressure coupling was modelled with the SIMPLE algorithm, momentum was modelled with an upwind scheme, and the dissipation rate of turbulent kinetic energy was modelled with first-order upwind. The base stepped spillway model was composed of ten steps with a slope of 30° (the slope is the $\tan^{-1}$ spillway height (hd) to spillway length (li)), while the spillway models with a compound slope varied in 20°/39° and 39°/20°. Also, the step number is 10 for MS1, 5/5 for both MS2 and MS3, and 9/9 for both MS4 and MS5. As explanation, in the case 9/9 for the compound slope 39°/20°, it is to show a 9 steps for the part with slope 39° and 9 steps for the part with slope 20°. The models height was 0.3 m, the channel width was 0.5 m, and the broad crest was 0.1 m in length. By applying variations to the height and length of the steps, different slopes were achieved for the spillways, as seen in Figure 1. The details of all five stepped spillway models are illustrated in Table 1. Through this numerical model, not only the relative velocity, pressure, turbulence kinetic energy, and turbulent velocity can be achieved for stepped spillways of different slope ratios, but also insight into the behaviours of the slopes and the turbulence generated can be gained. This could be useful in analysing the flow characteristics of stepped spillways, and in designing or improving their hydraulic performance.

The experimental work was conducted at Deakin University in Australia. A 7.0 m long, 0.5 m wide, and 0.6 m deep flume was used to conduct the laboratory experiments. The discharge rate (Q) in the flume was measured using a flowmeter which had an accuracy of ±1 L/s. In rectangular channel q = Q/b where b is the width of the channel. In order to measure the velocity (V0 and V1) at points 0 and 1 respectively (Figure 2a), a Prandtl-Pitot tube was used. This tube had a plastic, incurved shape with a diameter of 3 mm and was equipped with a digital gauge system to facilitate accurate readings. In addition, the Prandtl-Pitot tube was installed at an inclination angle of 30° to minimise the instrument error to ±0.2 mm. Moreover, the tube was remotely supported by a horizontal beam to provide stability and limit vibration during high flow. All the measurement data obtained from the Prandtl-Pitot tube was used to ascertain the velocity distributions of non-aerated flows (Jahad et al., 2022).

The geometry and mesh

There are two basic meshing methods: unstructured mesh and structured mesh, both of which can be used to describe the domain of geometry involved in the flow of air and water. A static-structured rectangular hexahedral mesh was ultimately chosen as the optimal solution for the scenarios in this study. This mesh was further modified with the addition of inflation layers, which are critical elements for a successful CFD simulation. In order to achieve the desired results, an extremely fine mesh was utilized; as illustrated in Figure 2, the MS1 model featured a total of 330617 elements and 330197 nodes. Accurately

Table 1. The dimensions of the models used in CFD

Model#	θ°	Model height (hd) mm	Crest length (lc) mm	Step length (l) mm	Step height (h) mm	Step number (N)
MS1	30	300	100	50	30	10
MS2	39/20	300	100	80/40	30	5/5
MS3	20/39	300	100	40/80	30	5/5
MS4	39/20	300	100	30	30	9/9
MS5	20/39	300	100	30	30	9/9

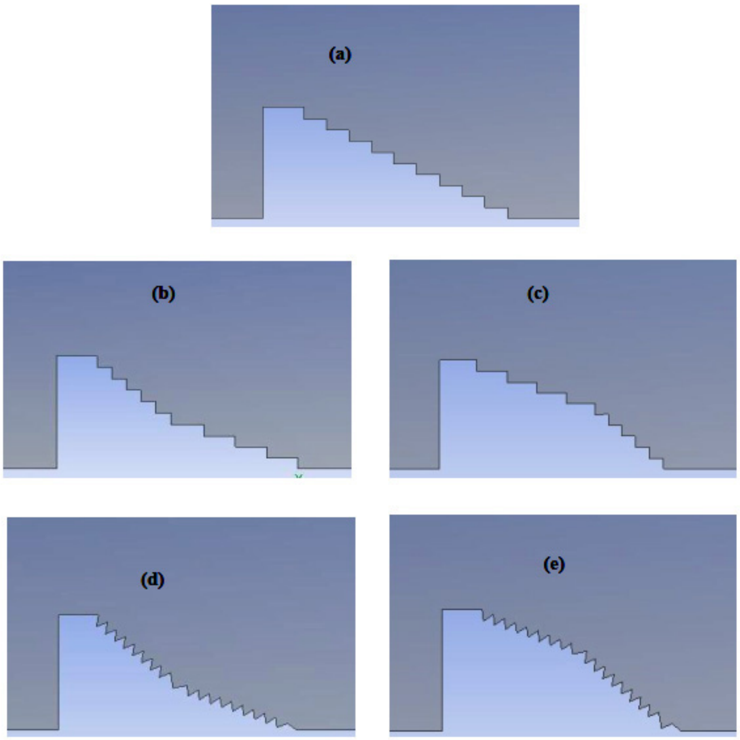

Figure 1. Geometry of the model, (a) MS1, (b) MS2, (c) MS3, (d) MS4, and (e) MS5

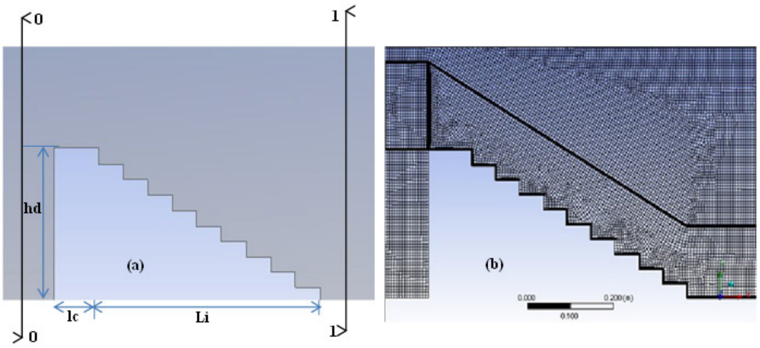

Figure 2. MS1 model details (a) The geometry (b)The mesh and inflation layers

mapping out the physical properties of the environment in this manner marks an important step in ensuring reliable results.

Boundary Conditions

The water velocity over the stepped spillway is controlled by the inlet velocity. Depending on the discharge rate, the velocity of the stepped spillway will vary. The higher discharge will result in a greater amount of velocity being applied at the velocity inlet. The value of the backwater flow from the outlet is set to zero for the pressure outlet condition at the outlet of stepped spillways, where there is no backwater flow. The walls of the stepped spillway are considered stationary, nonslip walls. Consequently, these are considered permanent walls.

VOF Method

The VOF is a multiphase method established by Hirt and Nichols in 1981. It is designed to calculate the volume of two fluids, not multiple fluids. Therefore, in order to accurately calculate the volume fraction of each fluid, a volume fraction variable must be introduced. This variable is denoted as βw for the volume fraction of water, and βa for the volume fraction of air. These two variables can be used to calculate the total volume fraction of both fluids throughout the domain, which must be equal to 1. Equation (1) allows us to make this calculation. Furthermore, the VOF method allows us to understand the interdependent relationship between the two fluids and quickly calculate the flow rate and velocity of each fluid. This is beneficial for understanding a variety of engineering problems, such as simulating pipeline systems, or for monitoring temperature during water-cooling processes. Additionally, researching the multiphase model can enlighten us on the behaviour of fluids under various conditions.

$$\beta_w + \beta_a = 0 \qquad (1)$$

VOF is an important tool used in computational fluid dynamics (CFD) to simulate and analyse fluid flows. VOF is a variety of equations used to measure the ratio of air and water present at each point in a given space. It does this by tracking the volume percentage of each element in the space, based on its fractional concentrations. By using this information to solve the continuity equation, the interface between air and water can be identified. Additionally, VOF can also be used to track other elements with varied densities, such as mixtures between air, water, and other types of fluids. As such, it is a highly versatile tool and is one of the most important simulations used in modelling multiphase flows.

$$\frac{\partial \rho}{\partial t} + \frac{\partial \rho u_i}{\partial x_i} = 0 \qquad (2)$$

$$\begin{gathered}\frac{\partial \rho u_i}{\partial x_i} + \frac{\partial}{\partial x_i}\left(\rho u_i u_j\right) = -\frac{\partial p}{\partial x_i} + \\ + \frac{\partial}{\partial x_j}\{\mu\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} - \frac{2}{3}\delta_{ij}\,\frac{\partial u_k}{\partial x_k}\right)\} + \\ + \frac{\partial}{\partial x_j}\left(-\rho u_i' u_j'\right)\end{gathered} \qquad (3)$$

where: ρ is the density of the fluid,
μ is molecular viscosity,
u_i is the velocity component,
x_i is the coordinate component,
t is the time, and
p is the pressure.

The deviatoric stress component in Equation. (3) can be expressed as:

$$\rho u_i' u_j' = \mu_t\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right) - \frac{2}{3}(\rho k)\delta_{ij} \qquad (4)$$

where: μ_t is the turbulent viscosity and the stress tensors $\delta_{ij} = 1$ when i = j and $\delta_{ij} = 0$ when i≠ j, using the k-ε model for turbulence closure:

$$\begin{gathered}\frac{\partial(\rho k)}{\partial t} + div(\rho k U) = \\ = div\{\left(\frac{\mu_t}{\sigma_k}\right) gradk\} + 2\mu_t E_{ij} E_{ij} - \rho\varepsilon\end{gathered} \qquad (5)$$

$$\begin{gathered}\frac{\partial(\rho\varepsilon)}{\partial t} + div(\rho\varepsilon U) = \\ = div\{\left(\frac{\mu_t}{\sigma_\varepsilon}\right) grad\varepsilon\} + \\ + C_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t E_{ij} E_{ij} - \ C_{2\varepsilon}\rho\frac{\varepsilon^2}{k}\end{gathered} \qquad (6)$$

$$\mu_t = \rho\, C_\mu \frac{k^2}{\varepsilon} \qquad (7)$$

Turbulence model

The Realizable k-ε model, which was developed by (Shih et al., 1994), is a powerful and comprehensive modelling approach that surpasses the capabilities of the standard k-ε model. This advanced model offers more customizable features, such as the variable for uniformity co-efficient, known as Cμ, which takes Reynolds stresses and physical components of turbulent flow into account when calculating turbulent viscosity. Furthermore, the dissipation rate equation in the realizable k-ε model is unique due to its combination of production and destruction terms. This distinctive feature adds to the accuracy of the model, specifically when dealing with twists, turns, and vortices in the flow. In fact, the model has been used to examine turbulence, rotation, and such phenomena that occur in stepped spillways. The primary equations used to define turbulent kinetic energy and dissipation rate are as follows:

$$\frac{\partial(\rho k)}{\partial t} + div(\rho k U) = = div\{\left(\frac{\mu_t}{\sigma_k}\right) gradk\} + 2\mu_t E_{ij} E_{ij} - \rho\varepsilon \tag{8}$$

$$\frac{\partial(\rho\varepsilon)}{\partial t} + div(\rho\varepsilon U) = = div\{\left(\frac{\mu_t}{\sigma_\varepsilon}\right) grad\varepsilon\} + + C_{1\varepsilon}\frac{\varepsilon}{k} 2\mu_t E_{ij} E_{ij} - C_{2\varepsilon}\rho\frac{\varepsilon^2}{k} \tag{9}$$

$$\mu_t = \rho\, C_\mu \frac{k^2}{\varepsilon} \tag{10}$$

where: $Cu = 0.09$,
$\sigma_k = 1.0$,
$\sigma_\varepsilon = 1.3$,
$C_{1\varepsilon} = 1.44$,
$C_{2\varepsilon} = 1.92$.

RESULTS AND DISCUSSION

A series of tests by ANSYS fluent were conducted on the stepped spillway models to better understand the energy dissipation and turbulent Kinetic Energy (TKE) of these structures. The stepped spillways models had varying angles from a traditional slope of 30° to compound slopes of 20°/39° and 39°/20° with different step shapes. Utilizing the Volume of Fluid (VOF) and k-ε realizable models, flow simulations were conducted at a range of discharge rates from 4.4×10^{-3} to 55.56×10^{-3} m^3/s. The simulations studied the effects of the various slopes and discharge rates on the energy dissipation and TKE of the flow.

CFD model validation

To obtain CFD results, the experimental method was utilized to ascertain the velocity downstream of the stepped spillway. Initially, the devised solvers for stepped spillways were being evaluated and required verification. To prove that the CFD model was correct, it was used with experiments based on equations 9 and 10's root mean square error (RMSE) and mean absolute percentage error (MAPE) criteria. Data acquired from the CFD simulations were compared to the experimental results in order to make judgments of accuracy. This method of comparison can provide confidence for the CFD simulations and make it possible for engineers to use them without worrying about their validity.

$$\text{RMSE} = \sqrt{\frac{1}{n}\sum_{1}^{n}(V1_{\text{exp}} - V1_{CFD})^2} \tag{11}$$

$$\text{MAPE} = 100 * \frac{1}{n}\sum_{1}^{n}\left|\frac{\text{V1}_{\text{exp}} - \text{V1}_{\text{CFD}}}{\text{V1}_{\text{exp}}}\right| \tag{12}$$

Similarly, the correlation coefficient R was used to determine the precision of the experiment and CFD results. Equation 13 served as the basis for determining R.

$$\text{R} = \frac{\text{C}_{\text{xjyj}}}{\sigma_{\text{xj}}\ \sigma_{\text{yj}}} \tag{13}$$

Where: "*xi*" and "*y*j" are defined as the CFD and experimental output results, respectively; C_{xjyj} is the covariance between the CFD (xj) and the physical model results (yj); σxj and σyj are the standard deviations of the CFD and the physical model results, respectively. Table 2 illustrates a comparison between the experimental and CFD results, with velocity of flow, V1, being used as a variable for analysis. Statistical values such as RMSE, R, and MAPE were used to assess the accuracy of the results, where zero or close-to-zero values indicate a more precise model. It was found that the MAPE and RMSE for the five values of the discharges were 4.32 and 0.21, respectively, with a value of R equalling 0.996. Clearly, the findings reveal that the RMSE was closest to zero, indicating the strongest agreement between the two models. Overall, this study revealed the strength of the CFD model in comparison to the experimental data, demonstrating its capability in providing reliable results in a short period of time.

Slope effect on flow pattern

The downstream slope of the spillway has significant impacts on the flow in terms of

Table 2. Results of experimental and CFD statistical calculations using the MS1 model with five different discharges

Model	Q m^3/s	Experimental data	CFD data	The relative error, %	RMSE	MAPE, %	Correlation (R)
		V_1, m/s	V_1, m/s				
MS1	0.0044	0.147	0.155	5.03	0.021	4.32	0.996
	0.0100	0.297	0.312	5.05			
	0.0250	0.505	0.492	2.57			
	0.0390	0.615	0.638	3.74			
	0.0555	0.671	0.706	5.22			

hydraulics. The downstream slope of the spillway plays a major role in governing the hydraulic behaviour of the flowing water. Merging water and the adjacent topography to create a smooth, uniform flow is essential for the successful operation of a spillway. The slope influences the amount of energy lost along the flow path, so it has to be considered carefully during design. Thus, the selection of the ideal downstream slope is paramount to ensure the safety of the spillway. The results from the CFD model run predicting flow patterns under different flow conditions are shown in Table 3 and Figures 3 and 4 with the same flow conditions. In general, the longitudinal water surface profiles and flow characteristics predicted by the model and observed in previous experimental studies are consistent.

Nappe flow occurred for $q \approx 0.009$ $(m^3/s)/m$ as shown in Figure 3. Plunging flow is modelled to cause waves and disturbances on the free surface. In Models MS2 and MS4, the Nappe flow and Transition flow are clearer than the others. Observed that there is plunging flow in the MS4 model. Significant quantities of air are entrained by the water flow. This manifests as white water regions in the experiments and a low water-to-air ratio ($w/a < 1$) in the model and is observed on every step, intensifying as the flow progresses down the steps. In addition to accurately predicting the transition to the Transition regime, the numerical model depicted in Table 3 for the MS2 and MS4 models also accurately predicts the progression to the Transition regime. Table 3 shows the flow pattern changes with the slope. The models MS2 and MS4 have transition flow for the $q = 0.050$ $(m^3/s)/m$ and the other models have Skimming flow. Additionally, for larger discharges ($q \geq 0.111$ $(m^3/s)/m$) where the flow surface appears relatively undisturbed, the model predicts skimming flow, notably for the Step model. For the MS2 and MS4 geometries, some waviness is seen, due to the obstruction to the flow by the slope and steps changes. Models' results are shown in Figure 4. Except for some slight waviness of the surface, the numerical water surface profiles are relatively undisturbed, a feature of the skimming flow regime. Air entrainment is also predicted by the model to be less significant, which is

Table 3. Flow pattern details of CFD data for used models

Model#	θ°	q(m^3/s)/m	Vo, m/s	V1, m/s	CFD flow type
MS1	30	0.009	0.028	0.147	Nappe flow
		0.050	0.133	0.505	Skimming flow
		0.111	0.257	0.671	Skimming flow
MS2	39/20	0.009	0.028	0140	Nappe flow
		0.050	0.133	0.535	Transition flow
		0.111	0.257	0.665	Skimming flow
MS3	20/39	0.009	0.028	0.150	Nappe flow
		0.050	0.133	0.537	Skimming flow
		0.111	0.257	0.673	Skimming flow
MS4	39/20	0.009	0.028	0.322	Nappe flow
		0.050	0.133	0.539	Transition flow
		0.111	0.257	0.675	Skimming flow
MS5	20/39	0.009	0.028	0.378	Nappe flow
		0.050	0.133	0.550	Skimming flow
		0.111	0.257	0.690	Skimming flow

confirmed by W-VOF. Also, Figure 5 presents the relationship between the relative velocity along the spillway at predicted skimming flow regime which sapporting the results in Figure 4.

Slope effect on energy dissipation

Figure 6 displays the effects of various slope angles (30°, 39°/20°, and 20°/39°) on energy dissipation. The base model in this study is MS1. The minimum and maximum dissipated energy are 43% and 63% respectively, for the flow over MS1. While models MS3 and MS5 with compound slope (39°/20°) display accept improved when compare with MS1. The improvements in energy dissipation on average are 4% and 6% for MS3 and MS5 respectively. The models with compound slope (MS2 and MS4) resulted in the greatest energy dissipation. For the flow over MS2 and MS4, the energy dissipation at low discharge is 79% and 84% respectively. Also, the energy dissipation at high discharge is 47% and 49% for the same model. The high energy dissipation in low discharges basically comes from the Nappe flow with geometry effects (Jahad et al., 2022). Moreover, the energy dissipation increases due to increased turbulence, the flow velocity, and the changes in slope. Despite the close similarity between MS2 and MS4 in the shape, the efficiency of MS4 shows better performance because of the steps direction and the steps number. In order to gain more insight into the effect of different slopes, further studies should be conducted on spillways of various heights. Additionally, monitoring the turbulent flow and hydraulic pressure at each step may provide additional insight into the effect of slope on energy dissipation.

Slope effect on turbulent kinetic energy

Turbulent kinetic energy, or TKE, is a measure of the chaotic movement of energy in a turbulent (fluid) flow. In the context of fluid flow over a spillway, TKE is a measure of the chaotic and random motion of water molecules as they cascade over the spillway. This chaotic motion is responsible for the dynamic turbulence of the stream and can contribute to the erosion of the spillway surface. Moreover, TKE can increase the flow rate of a spillway, which can result in greater pressure stress on downstream structures. This is why studying the impact of TKE on the flow over a spillway is so important. According to Valero and Bung (2018) TKE

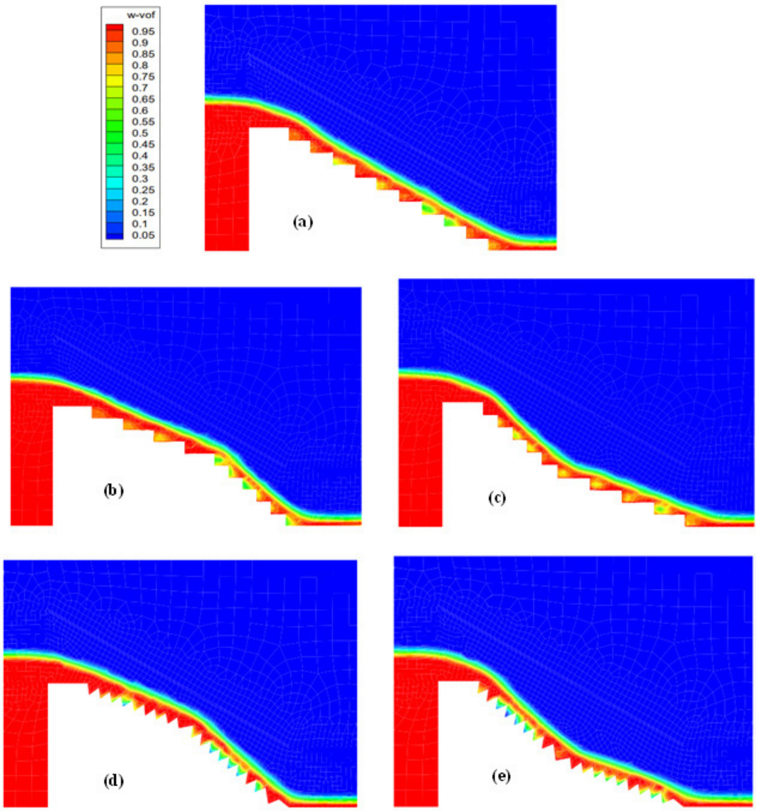

Figure 3. Nappe flow displayed by using w-vof on; (a) MS1, (b) MS2, (c) MS3, (d) MS4, and (e) MS5

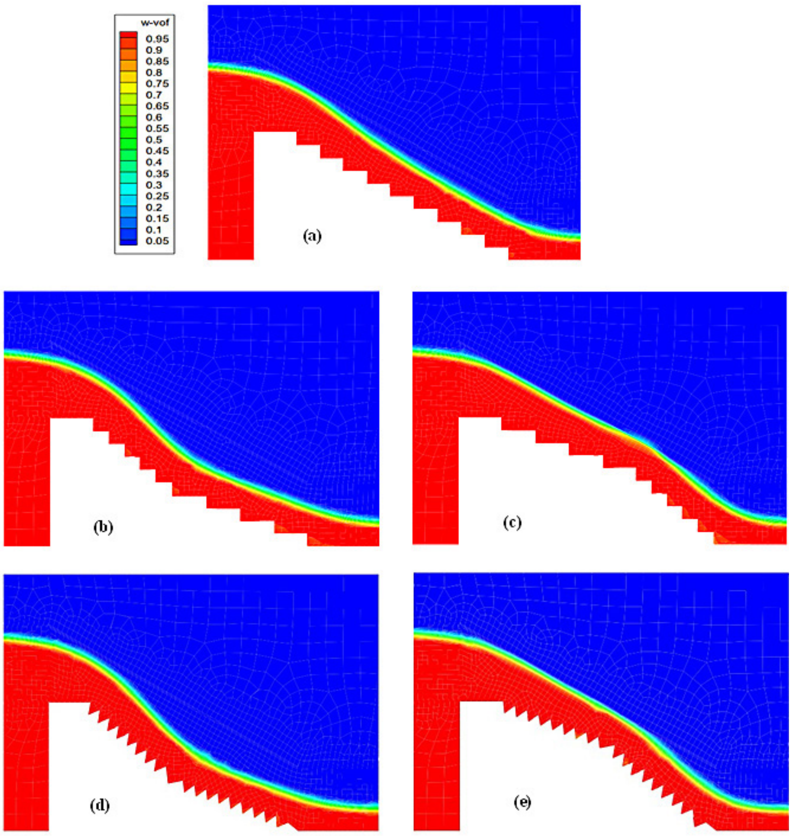

Figure 4. Skimming flow displayed by using w-vof on; (a) MS1, (b) MS2, (c) MS3, (d) MS4, and (e) MS5

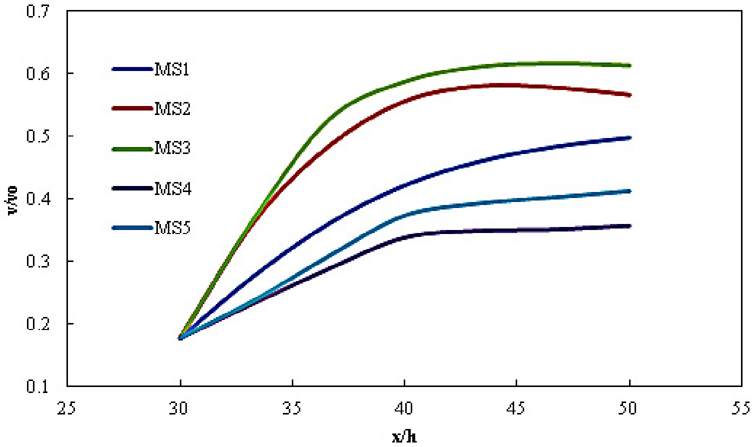

Figure 5. The relative velocity along the spillway body at q = 0.050 (m^3/s)/m

is a critical indicator of the energy loss between two sections of a fluid flow, mainly due to its ability to characterize the mean velocity gradients that precede its formation. Measuring the root-mean-square of velocity fluctuations is one way of quantifying TKE whereby it is calculated by finding the root mean square of consecutive velocity values taken along the flow direction (v_{x1}, v_{x2}, v_{x3}, …, v_{xn}), where v is the velocity in x direction. In addition to measuring the energy exchange between two points of a system, TKE can also be used to quantify the magnitude of turbulence associated with the flow. Higher values of TKE are usually associated with high flow turbulence, whereas lower TKE values are often seen with laminar flows. Therefore, it is essential to accurately measure the TKE of a system in order to make informed decisions about the flow regime. This v_{rx} value can be determined mathematically as:

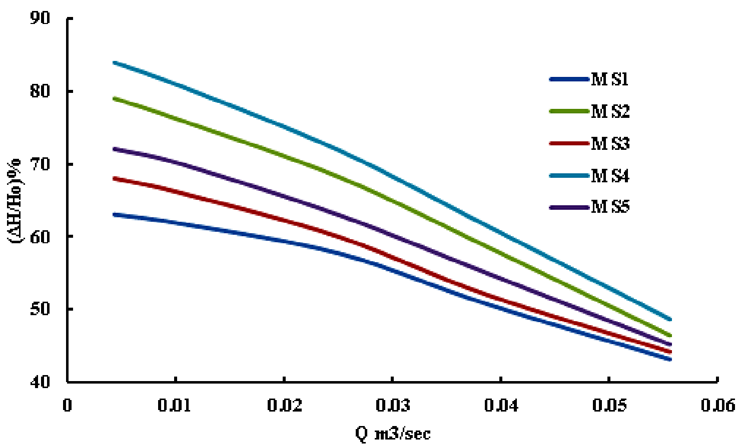

Figure 6. Energy dissipation when the flow varied from Nappe to Skimming flow

$$v_{rx}=\sqrt{(v_{x1}^2+v_{x2}^2+v_{x3}^2+\cdots+v_{xn}^2)/n} \quad (14)$$

$$\mathrm{TKE}=0.5(v_{rx}^2+v_{ry}^2+v_{rz}^2) \quad (15)$$

The results of the TKE change process with different discharges q = 0.009 $(m^3/s)/m$ and q = 0.050 $(m^3/s)/m$ suggest that the change in step direction with the downstream slope arrangement has a stronger intensity of turbulence than other arrangements. This was highlighted by Figure 7, in which a change in step direction with

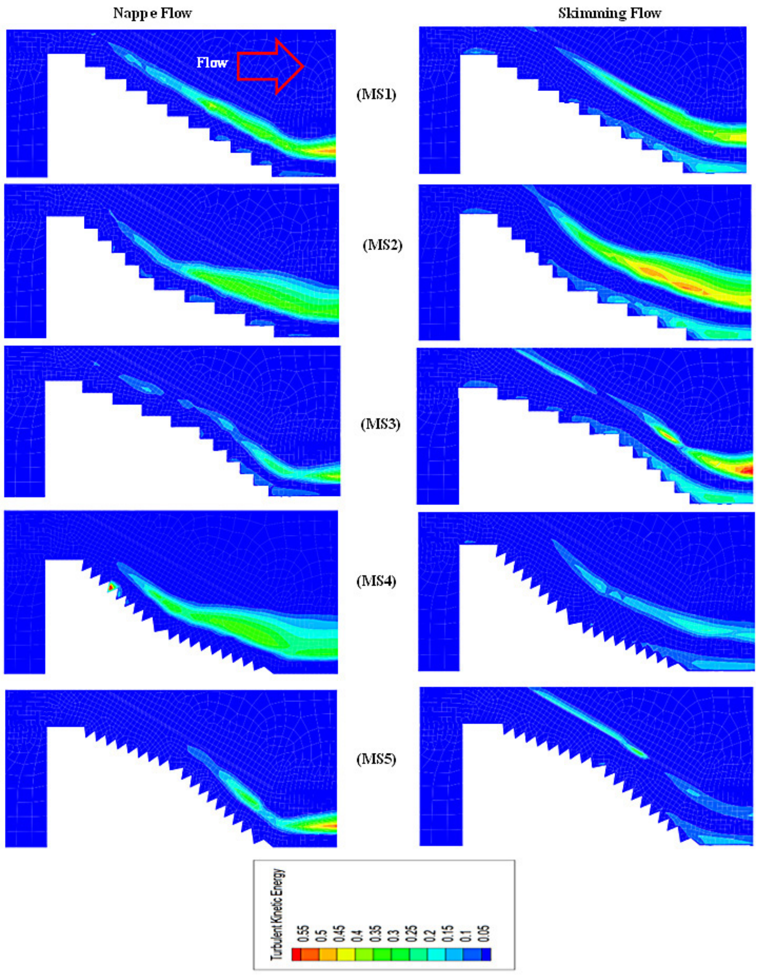

Figure 7. TKE values for different arrangement slopes for discharges q = 0.009 $(m^3/s)/m$ and q = 0.050 $(m^3/s)/m$

the downstream slope increased the deflected jets on the steps and gave the highest TKE at the compound slope with 18 steps for the model MS4. In addition, the findings by (Ghaderi and Abbasi, 2021), were validated as the TKE gradually increased as one moved along the spillway, confirming that a change in the step direction combined with the downstream compound slope arrangement increases the turbulence along the spillway more effectively than other methods for this specific discharge. Further studies should be conducted to see if such an arrangement is equally effective in other applications with varying levels of discharge.

CONCLUSIONS

The success of spillway design is contingent on the downstream slope being appropriate for the anticipated flow rate conditions. To further investigate this, a numerical examination of a stepped spillway featuring a compound slope was conducted using the ANSYS Fluent Model and the Volume of Fluid (VOF) Method, as well as the turbulence RNG-k-ε model. To validate the model, an experimental study of a flat-stepped spillway was also conducted, and the results were compared with those from the numerical simulation. In the end, these findings indicated that the parameters of the spillway operated effectively when the compound slope was properly designed. This study further reveals the potential of using this design to accommodate various flow conditions, enabling efficient and reliable operations of the spillway. The conclusions obtained from this study are as follows:

- Comparisons between numerical predictions and laboratory experiments demonstrated that the CFD model accurately captured the fundamental flow characteristics. The water profiles predicted by the CFD model corresponded well to the experimental results. Additionally, the various flow patterns (nappe, transition, and skimming) that arose due to changes in slope and geometry were thoroughly modelled by the numerical simulation. Investigation of the predicted VOF further indicated the effect of various slopes on the flow patterns and the energy dissipation rates. Compound slope models produced a strong recirculation with a single vortex of regular shape, whereas the model (MS1) generated a single shear-induced vortex similar to a flow over a backward-facing step.
- The flow patterns of various models with different geometries were investigated, and the observations were consistent with previous research. The results showed that the flow patterns varied in regard to the slope, geometry, and flow rate. Additionally, the onset of transition and skimming flow patterns were observed to be contingent upon the slope, with a slow shift in flow patterns. Model simulations MS2 and MS4 clearly displayed the change from nappe, to transition, and then to skimming flow.
- The numerical model indicates that energy dissipation rates of compound slopes, such as MS2 and MS4, are greater than that of the MS1 model under the same flow conditions. This is attributed to the slope changes that result in increased scale and intensity of vortex structures, thereby promoting energy dissipation. The trend of energy dissipation was found to be linear and increased with downstream distance.
- It has been noted that the use of compound slope as opposed to normal slope results in increased TKE values. Furthermore, the intensity of the turbulence generated on a compound slope has been found to be greater than that produced by other types of sloping arrangements.

With this knowledge about the compound slope's hydraulic characteristics in hand, the downstream slope can be modified to achieve the desired flow rate. There has to be more research done to form a complete vision. This provides a viable option for thinking about the problem of spillway design and operation.

REFERENCES

1. Castillo, L.G., Carrillo, J.M., García, J.T., & Vigueras-Rodríguez, A. 2014. Numerical simulations and laboratory measurements in hydraulic jumps. 11th International Conference on Hydroinformatics, HIC 2014.
2. Chamani, M.R., 2020. Air inception in skimming flow regime over stepped spillways. In Hydraulics of stepped spillways, 61–67, CRC Press.
3. Chanson, H., 2022. Energy dissipation on stepped spillways and hydraulic challenges-Prototype and laboratory experiences. Journal of Hydrodynamics,

34(1), 52–62.

4. Dong, Z., Wang, J., Vetsch, D.F., Boes, R.M. and Tan, G., 2019. Numerical simulation of air–water two-phase flow on stepped spillways behind x-shaped flaring gate piers under very high unit discharge. Water, 11(10),1956.
5. Felder, S. and Chanson, H., 2015. Aeration and air–water mass transfer on stepped chutes with embankment dam slopes. Environmental Fluid Mechanics, 15, 695–710.
6. Felder, S., 2013. Air-water flow properties on stepped spillways for embankment dams: Aeration, energy dissipation and turbulence on uniform, non-uniform and pooled stepped chutes. PhD thesis, Queensland University 2013.
7. Ghaderi, A. and Abbasi, S., 2021. Experimental and numerical study of the effects of geometric appendance elements on energy dissipation over stepped spillway. Water, 13(7), 957.
8. Ghaderi, A., Abbasi, S. and Di Francesco, S., 2021. Numerical study on the hydraulic properties of flow over different pooled stepped spillways. Water, 13(5),710.
9. Hamedi, A. and Ketabdar, M., 2016. Energy loss estimation and flow simulation in the skimming flow regime of stepped spillways with inclined steps and end sill: A numerical model. International Journal of Science and Engineering Applications, 5(7), 399–407.
10. Hamedi, A., Mansoori, A., Shamsai, A. and Amirahmadian, S., 2014. Effects of end sill and step slope on stepped spillway energy dissipation. Journal of Water Sciences Research, 6(1), 1–15.
11. Jahad, U.A., Al-Ameri, R. and Das, S., 2022. Investigations of velocity and pressure fluctuations over a stepped spillway with new step configuration. Water Supply, 22(7), 6321–6337.
12. Lebdiri, F., Seghir, A. and Berreksi, A., 2022. Multi-objective optimization of stepped spillway and stilling basin dimensions. Water Supply, 22(1), 766–778.
13. Ma, X., Zhang, J. and Hu, Y., 2022. Analysis of Energy Dissipation of Interval-Pooled Stepped Spillways. Entropy, 24(1), 85.
14. Mero, S. and Mitchell, S., 2017. Investigation of energy dissipation and flow regime over various forms of stepped spillways. Water and Environment Journal, 31(1), 127–137.
15. Raza, A., Wan, W. and Mehmood, K., 2021. Stepped spillway slope effect on air entrainment and inception point location. Water, 13(10), 1428.
16. Roushangar, K., Akhgar, S., Salmasi, F. and Shiri, J., 2014. Modeling energy dissipation over stepped spillways using machine learning approaches. Journal of Hydrology, 508, 254–265.
17. Salmasi, F. and Abraham, J., 2022. Effect of slope on energy dissipation for flow over a stepped spillway. Water Supply, 22(5), 5056–5069.
18. Shih, T.H., Liou, W.W., Shabbir, A., Yang, Z. and Zhu, J., 1994. A new k-epsilon eddy viscosity model for high Reynolds number turbulent flows: Model development and validation (No. CMOTT-946).
19. Torabi, H., Parsaie, A., Yonesi, H. and Mozafari, E., 2018. Energy dissipation on rough stepped spillways. Iranian Journal of Science and Technology, Transactions of Civil Engineering, 42, 325–330.
20. Valero, D. and Bung, D.B., 2018. Reformulating self-aeration in hydraulic structures: Turbulent growth of free surface perturbations leading to air entrainment. International Journal of Multiphase Flow, 100, 127–142.
21. Wan, W., Raza, A. and Chen, X., 2019. Effect of height and geometry of stepped spillway on inception point location. Applied Sciences, 9(10), 2091.
22. Yalcin, E.E., Ikinciogullari, E. and Kaya, N., 2023. Comparison of Turbulence Methods for a Stepped Spillway Using Computational Fluid Dynamics. Iranian Journal of Science and Technology, Transactions of Civil Engineering, 1–17.
23. Zhou, Y., Wu, J., Ma, F. and Hu, J., 2020. Uniform flow and energy dissipation of hydraulic-jump-stepped spillways. Water Supply, 20(4), 1546–1553.