International Journal of Research in Engineering, Science and Management
Volume 6, Issue 11, November 2023
https://www.ijresm.com | ISSN (Online): 2581-5792

Numerical Study of Energy Dissipation in Baffled Stepped Spillway Using Flow-3D

Ahmed Ashour[1*], Emam Salah[2]
[1]*Department of Civil Engineering, University of Cairo, Cairo, Egypt*
[2]*Department of Civil Engineering, University of Tanta, Tanta, Egypt*

***Abstract*: One of the most obvious characteristics of spillways is their energy consumption, which has been extensively researched to understand how to estimate and increase it. Among these spillways, stepped spillways exhibit significant energy consumption potential compared to other types due to the artificial roughness of the bed. In this study, hydraulic modeling of the baffled stepped spillway in three dimensions and the two-phase flow system was conducted using FLOW-3D software. The results obtained were then compared with relevant laboratory experiments. Upon examining the modeling results, it was observed that the unique shape of the blocks in the aforementioned structure led to an increase in flow irregularity, resulting in a 77% reduction in flow energy. The proximity of the numerical model's results to the experimental model (RMSE=0.02) indicates that numerical methods can effectively simulate hydraulic behavior. Overall, it can be concluded that numerical methods are well-suited for accurately simulating hydraulic behavior.**

***Keywords*: Baffled Stepped Spillway, Energy Dissipation, Flow-3D.**

1. Introduction

The construction of the dam has always aimed to achieve efficient reservoir creation, storage, and optimal utilization, given the substantial expenses associated with building both the dam and its accompanying structures. From an economic perspective, this approach is more rational and suitable. The hydraulic design and construction of dams have consistently constituted a crucial aspect of the planning and building phases, with a focus on identifying optimal dimensions. Among the auxiliary structures of a dam, spillways serve the primary purpose of safely directing surplus water downstream. This process converts the potential energy of water into kinetic energy, subsequently leading to an increase in water velocity and erosion at the base of the dam. As erosion poses a significant threat to dams, measures must be taken to dissipate kinetic energy effectively [1]-[3].

Kinetic energy dissipation methods are generally employed in three ways: dissipation through spillway, dissipation at the base of the spillway (utilizing calm pools), and a combination of both (in the spillway and downstream). One suggested type of spillway, known as a stepped case spillway, facilitates the transfer of excess water while concurrently dissipating energy. By shaping the channel bed and enhancing the inflow of air into the water stream, the flow velocity of the water is reduced, leading to the dissipation of kinetic energy at the base of the spillway structure [4], [5].

There are various types of stepped spillways categorized based on the shape and design of their steps and their construction method. These include stepped spillway with simple steps, edged steps, sloping steps, bowl steps, and more. The type of flow generated on the spillway depends on the quantity of discharge passing through and the configuration of the steps. These flow types encompass Nappe Flow, Skimming Flow, and Transition Flow.

Stepped spillways offer numerous advantages, including high energy dissipation, reduced implementation time, straightforward operation and maintenance, decreased erosion downstream in the riverbed, increased energy dissipation, cost savings in implementation, the creation of calming pools, and prevention of cavitation [6], [7]. Extensive research has been conducted in various fields related to these spillways over the years. Among these research endeavors, one of the most significant focuses on identifying the parameters that have the greatest impact on increasing energy dissipation during these spillway operations [8]-[10].

To enhance energy efficiency in spillways, particularly stepped spillways, numerous efforts have been undertaken, some of which are briefly summarized as follows:

Tabbara et al., used numerical simulations of water flow over stepped spillways with different step configurations in their work, the finite element computational fluid dynamics module of the ADINA software was used to predict the main characteristics of the flow [11]. Chanson and Toombes did an experimental study focused on the flow down a stepped case channel characterized by very strong flow aeration and turbulence, showed the increase in turbulence levels, compared to single-phase flow situations, is proportional to the number of entrained particles [12]. Elnikhely conducted experimental runs to investigate the effect of installing cylinder blocks on the back slope of the spillway on the dimensions of the scour hole downstream of the spillway under various flow conditions [13]. Zhenwei et al. simulated the flow characteristics of an entire spillway using the VOF method, multidimensional two-phase flow model, and the standard k-ε method through FLUNT software. Through model tests and numerical simulations, they

*Corresponding author: ahmedashour2530@gmail.com

discovered that the flow pattern and surface elevation in the case of two holes were complex due to boundary conditions. However, these complexities were resolved after modifying it to a single hole [14].

Laboratory methods for investigating the behavior of hydraulic structures are highly accurate but require significant financial resources and time. However, with the advancement of computer technology, numerical methods have gained more capability to solve complex problems efficiently. These numerical methods offer a quicker and cost-effective way to provide comprehensive details and information. Given the energy efficiency potential of stepped spillways and the substantial expenses associated with laboratory models, numerical methods can offer a viable solution for addressing issues in modeling stepped spillways.

The laboratory-based relationships for energy consumption in spillways are based on ideal conditions specific to the laboratory setting. In contrast, numerical methods allow for the creation of various model variables and the examination of new conditions. Many researchers have conducted studies on stepped spillways over the years in the field of spillway research. However, the combination of stepped spillways with numerical methods remains an unexplored area, representing a research gap that this study aims to address.

The present research focuses on the numerical modeling of flow and energy consumption in a unique stepped spillway model with a composite shape that primarily affects floor roughness, as well as the influence of various hydraulic and geometric parameters. It aims to assess the existing conditions and quantify the anticipated energy losses.

2. Laboratory Model

The laboratory model setup comprises a staggered stepped spillway and a sand basin at the bottom. This setup allows for testing flow passage under various discharges and spillway slopes, considering different dimensions of notched blocks. These blocks, denoted as A, B, C, D, and E, are constructed from rough materials and their geometric shapes are illustrated in Figure 1-A. The blocks come in three different dimensions, and in all the experiments (as depicted in Figure 5), they are positioned on the channel bed alongside each other, with each block rotated 90 degrees relative to the adjacent one.

Inspired by the experimental methodology employed by Kamyab Moghaddam et al., who conducted a study on a stepped spillway featuring inclined steps and two discharge levels, encompassing both skimming and nappe flow regimes. Their findings demonstrated that the energy levels derived from the model and Bernoulli equation were trustworthy. Building upon their approach, the recent study followed a similar methodology, examining both skimming and nappe flow regimes, and compared the numerical model's results with those obtained by Kamyab Moghaddam and colleagues to gauge the precision of the model [15]. In Table 1, hydraulic data obtained from the laboratory model for Tooth No. C, featuring an 18.8-degree angle, is presented. This data covers six different discharge types and includes hydraulic parameters such as hs (the depth of the downstream irrigation pit), dc (the critical height), H dam (the spillway height), and q (the flow rate per unit width). Additionally, the table includes the percentage of energy dissipation in the spillway.

Table 1
Hydraulic data of the laboratory model on tooth No. C and with an angle of 18.8 degrees

Hs(m)	Dc(m)	Hdam(m)	ΔE/EO (%)	Q(m^2/s)
1.5	0.045	1.289	93.090	0.03
3	0.063	1.289	91.234	0.05
5	0.083	1.289	89.004	0.075
8.3	0.101	1.289	85.744	0.1
11	0.117	1.289	83.992	0.125
13	0.132	1.289	82.103	0.15

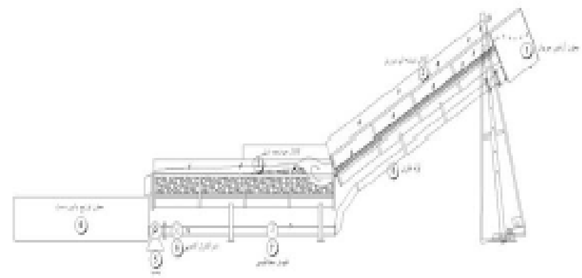

Fig. 1. An overview of the laboratory model

Fig. 2. Details of the laboratory model

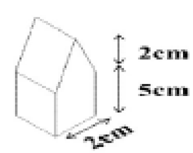

Fig. 3. One sample block

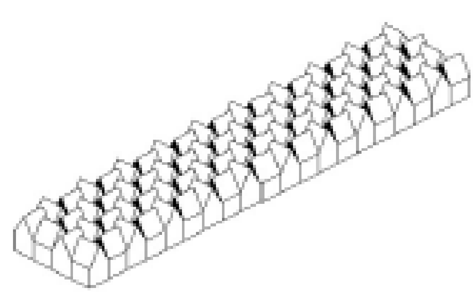

Fig. 4. Blocks arrangement and location

3. Numerical Modeling

The FLOW-3D software stands out as one of the most powerful models in the field of fluid dynamics. This model offers a three-dimensional analytical capability for the examination of flow fields, making it highly versatile in addressing fluid-related issues. The governing equations in this mathematical model encompass mass and momentum conservation equations, and the solution grid consists of rectangular cubic cells. Although this type of grid may seem limiting at first, it offers distinct advantages due to its ease of mesh generation and efficient memory usage. Furthermore, this software serves as a valuable tool, incorporating various methods such as the Volume of Fluid (VOF) technique for modeling free-flow surfaces, the Area Fraction Method (FAVOR) for simulating surfaces and solid volumes, including geometric boundaries, and the ability to utilize various turbulence models, including Prandtl Mixing Length, One Equation, Two Equations K-ξ, RNG K-ξ, and the 7 LES method. [16], [17].

A. Governing Equations

The laws governing the flow of an incompressible fluid are described by the constant equations of mass and momentum in the directions of the three coordinate axes, as follows:

$$\frac{\partial(u_i)}{\partial x_i} = 0$$

$$\frac{\partial u_i}{\partial t} + u_j \frac{\partial u_i}{\partial x_j} = \frac{1}{\rho}\frac{\partial p}{\partial x_i} + g x_i + v\nabla^2 u_i$$

1) Turbulence model equations

Most of the flows are turbulent in nature, so it is necessary to consider the turbulence and turbidity of the flow in order to solve the field. Turbulent flow based on continuity and Reynolds equation requires that Reynolds stresses be modeled in a special way. One of the most used two-equation turbulence models is the K-ξ-RNG model. This model is used in strong rotating flows and flows that depend on time [18], [19].

2) Fluid Volume Model (VOF)

The limited volume model was proposed by Hert and Niklas in 1981 and has been proposed to determine the common surface of two fluid phases in many hydrodynamic problems, and in hydrodynamic phenomena, the free flow surface of a special state. It is edible. In the VOF method, a differential equation is solved for each cell volume component, which ultimately determines the amount of each fluid volume component in each cell [20].

4. Meshing and Boundary Conditions of the Model

The meshing of each numerical model must be in such a way that the stability of the solution of the problem exists, the boundary conditions and meshing based on the laboratory data are validated and in this order the size of the meshes in the calculations according to the size of the teeth 2 A millimeter was taken into account. The boundary conditions of the model are also determined based on the residential model as follows.

In the inlet section of the spillway (X min) from the boundary condition (Volume flow rate) according to Table 3, in the direction of the spillway at the end of the downstream channel (X max) from the outlet condition (Out flow), on the side of the walls (Y max, Y min) according to the condition Symmetry has been applied to the wall (Z min) and to the upper border (Z max).

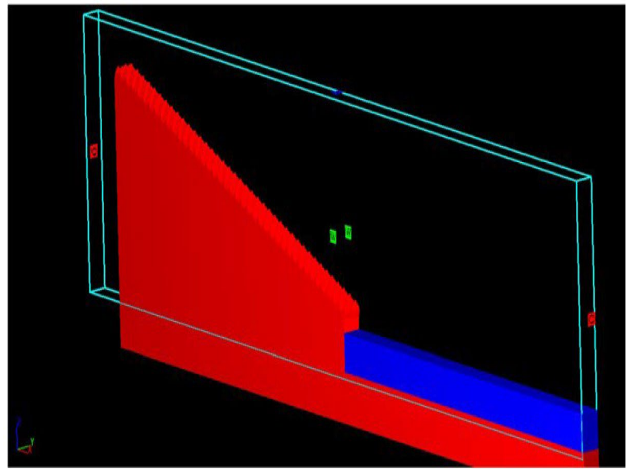

Fig. 5. Boundary conditions of the numerical model

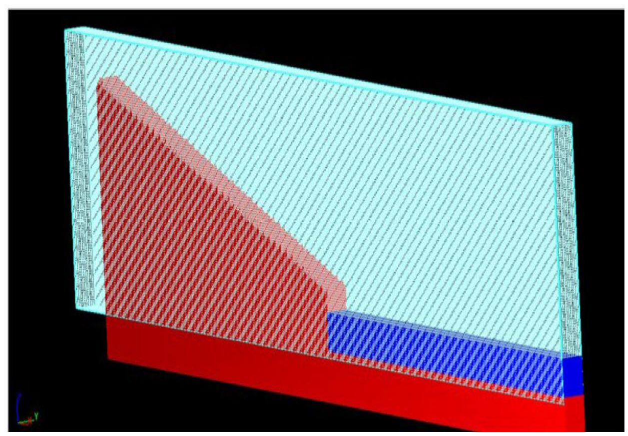

Fig. 6. Numerical model computing network

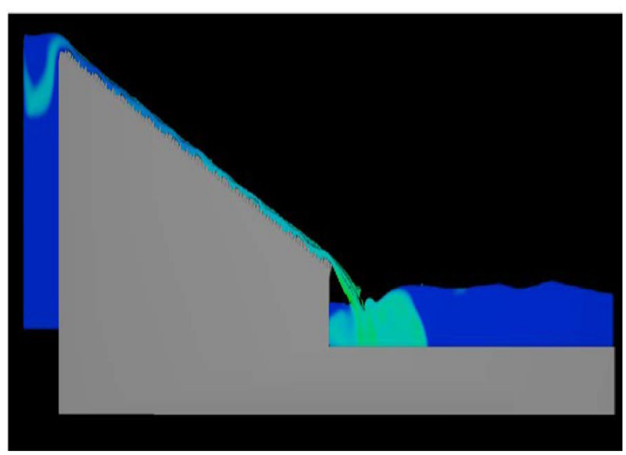

Fig. 7. Modeling of serrated stepped spillway

Table 2
Modeling results of relative energy dissipation and error value on block No. C with an angle of 18.8 degrees, at 6Q flow

$Q(m^2/s)$	Hdam(m)	dc(m)	ΔE/EO (%) Numerical Method	ΔE/EO (%) Experimental Method	MSE
0.03	1.2891	0.045108	0.9064132	0.930901	0.03
0.05	1.2891	0.063409	0.8903003	0.912342	0.02
0.075	1.2891	0.083089	0.8801885	0.890039	0.01
0.1	1.2891	0.100655	0.8475657	0.857442	0.01
0.125	1.2891	0.1168	0.826699	0.839915	0.02
0.15	1.2891	0.131896	0.8125506	0.821028	0.01

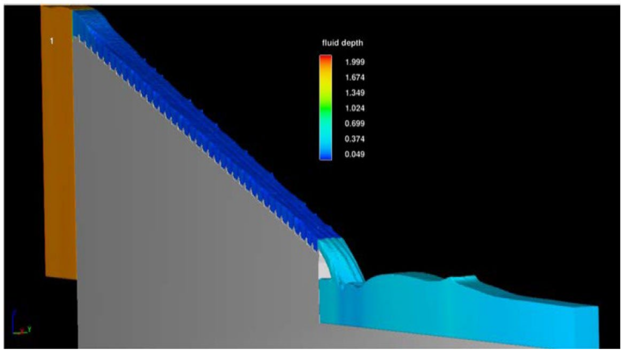

Fig. 8. Three-dimensional modeling of a serrated stepped spillway

5. Modeling Results

Stepped Spillway modeling was conducted based on the laboratory model, following the specifications outlined in Table 2. Six different flow rates were simulated on the teeth of model C, which had an angle of 18.8 degrees. The obtained results were then compared with those from the laboratory model (see Table 2). The results, calculated using the Mean Squared Error (MSE) and Root Mean Squared Error (RMSE) error rates, were found to be 0.02, indicating a small difference between the laboratory model and the numerical model results. This confirms the accuracy of the numerical model simulation. Additionally, the block consumed 77% of the total energy.

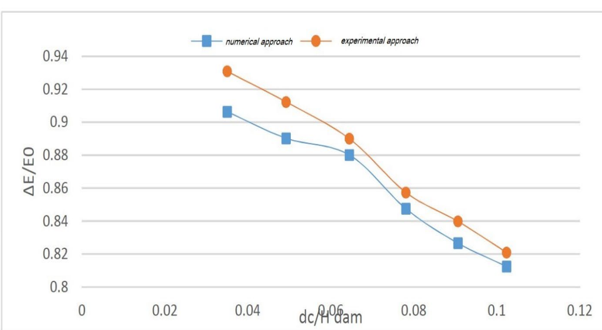

Fig. 9. The graph of the relative energy dissipation (ΔE/EO) via the relative critical depth (dc/H dam)

6. Conclusion and Suggestions

In this research, numerical modeling of the flow over a spillway with a new geometric configuration and rough arrangement designed for energy dissipation was conducted, as illustrated in Figure 6. The distinctive shape of the toothed blocks and the spacing between them led to the generation of flow turbulence. Based on the data presented in Table 2 and Figure 7, it has been determined that the innovative stepped spillway with block structures induces significant energy reduction due to the turbulence it imparts to the flow. Upon reviewing the results, it can be concluded that numerical evaluation of the model's hydraulic behavior is valuable, suggesting the use of a numerical model in conjunction with a laboratory model for simulations.

Given the heightened energy consumption and the need to mitigate cavitation issues in this spillway type, it is recommended to establish design guidelines specific to this type of spillway for its optimal implementation.

References

[1] H. W. Shen and B. C. Yen, "Advances in open-channel hydraulics after V.T. Chow's book," *Journal of Hydrology*, vol. 68, no. 1, pp. 333–348, Feb. 1984.
[2] M. Jovanović, "The Hydraulics of Open Channel Flow – An Introduction, Hubert Chanson; Wiley, New York, 1999, 495 pages (index included), pbk, ISBN 0-470-36103-4 (£35.00), http://www.arnoldpublishers.com," *Urban Water*, vol. 1, no. 3, p. 270, Sep. 1999.
[3] C. T. Haan, B. J. Barfield, and J. C. Hayes, "4 - Open Channel Hydraulics," in *Design Hydrology and Sedimentology for Small Catchments*, C. T. Haan, B. J. Barfield, and J. C. Hayes, Eds., San Diego: Academic Press, 1994, pp. 104–143.
[4] R. D. Reitz and Y. Sun, "18 - Advanced computational fluid dynamics modeling of direct injection engines," in *Advanced Direct Injection Combustion Engine Technologies and Development*, vol. 2, H. Zhao, Ed., Woodhead Publishing, 2010, pp. 676–707.
[5] P. Wesseling and C. W. Oosterlee, "Geometric multigrid with applications to computational fluid dynamics," *Journal of Computational and Applied Mathematics*, vol. 128, no. 1, pp. 311–334, Mar. 2001.
[6] M. Pfister and W. H. Hager, "Self-entrainment of air on stepped spillways," *International Journal of Multiphase Flow*, vol. 37, no. 2, pp. 99–107, Mar. 2011.
[7] J. Wu and C. Luo, "Effects of entrained air manner on cavitation damage," *Journal of Hydrodynamics, Ser. B*, vol. 23, no. 3, pp. 333–338, Jun. 2011.
[8] J. Chatila and M. Tabbara, "Computational modeling of flow over an ogee spillway," *Computers & Structures*, vol. 82, no. 22, pp. 1805–1812, Sep. 2004.
[9] H. Chanson, "20 - Design of drop structures and stepped cascades," in *Hydraulics of Open Channel Flow (Second Edition)*, H. Chanson, Ed., Oxford: Butterworth-Heinemann, 2004, pp. 431–439.
[10] A. Bagis and D. Karaboga, "Evolutionary algorithm-based fuzzy PD control of spillway gates of dams," *Journal of the Franklin Institute*, vol. 344, no. 8, pp. 1039–1055, Nov. 2007.
[11] M. Tabbara, J. Chatila, and R. Awwad, "Computational simulation of flow over stepped spillways," *Computers & Structures*, vol. 83, no. 27, pp. 2215–2224, Oct. 2005.
[12] H. Chanson and L. Toombes, "Air–water flows down stepped chutes: turbulence and flow structure observations," *International Journal of Multiphase Flow*, vol. 28, no. 11, pp. 1737–1761, Nov. 2002.
[13] E. A. Elnikhely, "Investigation and analysis of scour downstream of a spillway," *Ain Shams Engineering Journal*, vol. 9, no. 4, pp. 2275–2282, Dec. 2018.
[14] M. Zhenwei, Z. Zhiyan, and Z. Tao, "Numerical Simulation of 3-D Flow Field of Spillway based on VOF Method," *Procedia Engineering*, vol. 28, pp. 808–812, Jan. 2012.
[15] A. Kamyab Moghaddam, A. Hamedi, and S. Amirahmadian, "Experimental Study of Energy Loss in a Stepped Spillway Equipped with Inclined Steps in the Nappe and Skimming Flow Regimes," *International Journal of Science and Engineering Applications*, vol. 11, no. 12, pp. 346–350, 2022.
[16] O. Herrera-Granados, "Chapter 19 - Theoretical background and application of numerical modeling to surface water resources," in *Current Directions in Water Scarcity Research*, vol. 7, M. Zakwan, A. Wahid, M.

Niazkar, and U. Chatterjee, Eds., in Water Resource Modeling and Computational Technologies, vol. 7. Elsevier, 2022, pp. 319–340.

[17] W. de Q. Lamas, F. F. Bargos, G. E. O. Giacaglia, F. J. Grandinetti, and L. de Moura, "Numerical modelling and simulation of multi-phase flow through an industrial discharge chute," *Applied Thermal Engineering*, vol. 125, pp. 937–950, Oct. 2017.

[18] R. Gentle, P. Edwards, and B. Bolton, "3 - Fluid mechanics," in *Mechanical Engineering Systems*, R. Gentle, P. Edwards, and B. Bolton, Eds., in IIE Core Textbooks Series, Oxford: Butterworth-Heinemann, 2001, pp. 112–168.

[19] W. C. Chin, "10 - Advanced Modeling Methods," in *Computational Rheology for Pipeline and Annular Flow*, W. C. Chin, Ed., Woburn: Gulf Professional Publishing, 2001, pp. 241–253.

[20] G. H. Yeoh and J. Tu, "Chapter 3 - Solution Methods for Multi-Phase Flows," in *Computational Techniques for Multiphase Flows*, G. H. Yeoh and J. Tu, Eds., Oxford: Butterworth-Heinemann, 2010, pp. 95–242.