Research Article

Comparison of the Flow Behaviours of Physical and Numerical Models on a Stepped Spillway

Treerat Kuljitjuawong, Duangrudee Kositgittiwong*, Chaiwat Ekkawatpanit and Rachapol Sukjan
Department of Civil Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, Bangkok, Thailand

* Corresponding author. E-mail: duangrudee.kos@kmutt.ac.th DOI: 10.14416/j.asep.2020.05.004
Received: 5 August 2019; Revised: 21 February 2020; Accepted: 23 March 2020; Published online: 22 May 2020

Abstract
A massive water release can create high discharge and high velocity, which in turn can lead to erosion at the surface of a spillway. The difficulties of data collection and the high cost of experiments impose barriers to the study of large spillways. A numerical model that can simulate the flow behaviour is a valid choice to solve this problem. The objective of this research is to study the flow behaviour and flow velocity triggered by physical model tests and numerical model simulations. The small differences between the results of the two models illustrate that the numerical model is reliable and can be used to design a stepped spillway. Computational fluid dynamics (CFD) is a numerical modelling technique used in this study. For the numerical modelling of turbulence, the k–epsilon turbulence model is used. This model comprises 3 submodels: The standard k–epsilon, renormalized group (RNG) k–epsilon and realizable k–epsilon models. The results show that the RNG k–epsilon model is the most suitable model. The results reveal that the stepped spillway results in a 98% reduction of energy dissipation based on the present case study. The quantified differences between the results of the physical model and those of the numerical model are approximately 0.1–11.23%.

Keywords: Energy dissipation, Flow behaviour, Numerical model, Physical model, Stepped spillway

1 Introduction

Currently, flood problems frequently occur due to human behaviour and nature. Dam construction is one method used to solve this problem, as dam spillways are designed to release floodwater that is in excess of a reservoir capacity and can reserve substantial amounts of water for use in a dry season. However, dam construction must consider the area, environment, and population density to evaluate the value of construction.

The water released from upstream to downstream of a large structure, such as a dam, affects the discharge and velocity due to the substantial movement of water. The design of hydraulic structures must certify that the water is discharged in a safe manner to prevent any damage to the structure and enclosed locations. High levels of discharge can cause erosion at the surface of a spillway. The spillway is an important component because it is built to control the water depth in the reservoir at a safe level by releasing water sufficiently downstream to prevent the overflow of water in the reservoir. Dissipation of energy of flow along the spillway is a common work in the hydraulic projects. To this purpose, depending on the amount of energy that must be dissipated, suitable approach and structure are proposed [1]. In this case, a stepped spillway is preferred because there are many advantages, such as increasing the energy dissipation rate in the chute and reducing the size of the spilling basin downstream;

Please cite this article as: T. Kuljitjuawong, D. Kositgittiwong, C. Ekkawatpanit, and R. Sukjan, "Comparison of the flow behaviours of physical and numerical models on a stepped spillway," *Applied Science and Engineering Progress*, vol. 14, no. 2, pp. 165–174, Apr.–Jun. 2021, doi: 10.14416/j.asep.2020.05.004.

thus, a stepped spillway can be used to solve the problem of high discharge from released water while reducing the chance of erosion at the surface. Economic analysis of stepped spillway showed that it is expensive compared with standard stilling basin [2]. For both economic reason and hydraulic characteristics of stepped spillways, they have been more attention especially in the last few decades. Studies on flow characteristics initially were conducted using physical hydraulic modelling [3], [4].

There are some results demonstrated that in the napped flow regime, roughing the steps increased the performance of stepped spillways regarding energy dissipation about 15–20% as compared with smooth stepped spillways [5]. For the hydraulic properties of circular crested stepped spillways, results showed that an exponential function could mathematically model the stage-discharge relation. The energy dissipation of 40% compare to a smooth type was found [6]. In both nappe flow and skimming flow regimes, the design of a stepped spillway is a very efficient method to dissipate a large part of the flow energy along the spillway, and up to 99% of the total head available can be dissipated [7].

Various techniques of numerical model have been used for the comparison of numerical and physical models for example; multivariate adaptive regression, artificial neural network, support vector machine, Flow 3D software as computational fluid dynamic (CFD) tool, genetic programming, group method of data handling [8]–[13]. The data from those models show a good agreement on the comparison.

This study focuses on the flow behaviour and flow velocity triggered by a physical model test and a numerical model simulation. The suitable turbulent model and energy reduction will be proposed. The energy dissipation is also discussed.

2 Literature Review

2.1 *Stepped spillway*

The stepped spillway is one of many types of spillway and has been in use for 300 years. A stepped spillway is a spillway with steps on the spillway chute that assist in the dissipation of the kinetic energy of the descending water. Currently, stepped spillways are often paired with dam construction using roller-compacted concrete (RCC) and enhance the rate of energy dissipation compared to a smooth chute design [14]. This spillway design increases energy dissipation and thus greatly reduces the need for a large energy dissipater at the toe of the spillway or chute.

2.2 *Dimensional analysis*

Dimensional analysis can reduce complex physical problems to the simplest (that is, most economical) form prior to obtaining a quantitative answer. The concept of similarity is the key to dimensional analysis. In physical terms, similarity refers to some equivalence between two things or phenomena that are actually different. Most applications of dimensional analysis are not in question, no doubt because they are well supported by experimental facts. Moreover, dimensional analysis is rooted in the common artifices that are constructed to describe the physical world and explain its functioning in quantitative terms [15].

Experimental work in the laboratory can be expensive and time consuming. Thus, dimensional analysis is an important tool that helps in relating analytical results with experimental data for such unknown flow problems. The important terms used in dimensional analysis may be defined as dimensional analysis, dimensional homogeneity, dimensional variables, dimensional constants and pure constants [16].

A representation of a physical system that is used to predict the behaviour of the system in some desired respect is called the “model”. A “prototype” is a physical system for which the predictions are to be made. Normally, a model is smaller than the prototype and can enable laboratory experiments [16].

2.3 *Energy dissipation*

A dissipative process is a process in which energy (internal, flow kinetic, or potential) is transformed from some initial form to a final form that is normally lower than that of the initial form. Dissipation is the result of an irreversible process that occurs in thermodynamic systems. Energy dissipation can be calculated by using the energy equation, which is applied from the conservation of energy [17].

The energy dissipation can be observed and calculated from the energy or head between the inlet

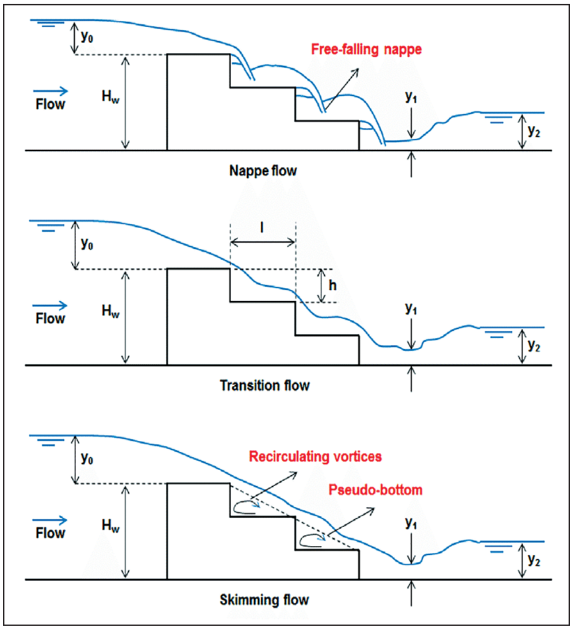

Figure 1: Schematic classification of 3 flow regimes.

section at the approach channel of a spillway, E_0, and any step section of interest, E_i.

The calculation of the energy between the inlet section, E_0, and any step section of interest, E_i, can be calculated by using the elevation head, H, pressure head, y, and velocity head, $\frac{v^2}{2g}$. [Equations (1) and (2)]

$$E_0 = \frac{v^2}{2g} + y_c + H_0 \tag{1}$$

$$E_i = \frac{v_i^2}{2g} + y_i \tag{2}$$

2.4 *Flow regimes*

The flow over a stepped spillway can be divided into three types as shown in Figure 1 (i.e., nappe, transition or skimming flow) depending on a function of the discharge and step geometry.

2.4.1 Nappe flow

The nappe flows can be developed in the low discharge and large depth of the step and are characterized by a succession of free-falling nappes at each step edge, followed by the nappe impact on the downstream step [18]. Nappe flow can be classified into two types: a nappe flow with a fully developed hydraulic jump for low discharge and small flow depth and a nappe flow with partially developed hydraulic jump [17]. The energy dissipation in the steps is due to the kind of hydraulic jump and the mixing of air with the flow water.

2.4.2 Skimming flow

The skimming flow regime occurs at high discharges. In this regime, the flow creates a coherent stream parallel to and above the pseudo-bottom. Recirculating vortices are created in the step cavities below the pseudo-bottom [19]. At each step, whether air entrainment occurs or not, a stable vortex develops, and the overlaying flow moves down the spillway supported by these vortices. There is a continuous exchange of flow between the top layer and the vortices formed on the steps [20].

2.4.3 Transition flow

The transition flow regime occurs between the two previous regimes with significant longitudinal variations. This flow may be observed by a strong decrease in the x-axis step with downstream splashing [17].

2.5 *Numerical model*

In engineering, modelling is categorized into two major approaches: physical/empirical modelling and theoretical/analytical modelling. Physical modelling can be tested in the laboratory, where engineers and scientists can obtain helpful information to develop empirical or semiempirical algorithms for exact applications. In most engineering cases, theoretical modelling is used to solve complex problems that are difficult to describe. A numerical model is necessary when calibrating and validating pre-existing data and analytical results [21].

There are many advantages of Computational fluid dynamics, CFD, over experiment-based approaches to fluid system design:

- Factual reduction of time consumption and construction costs.
- Ability to study systems where controlled

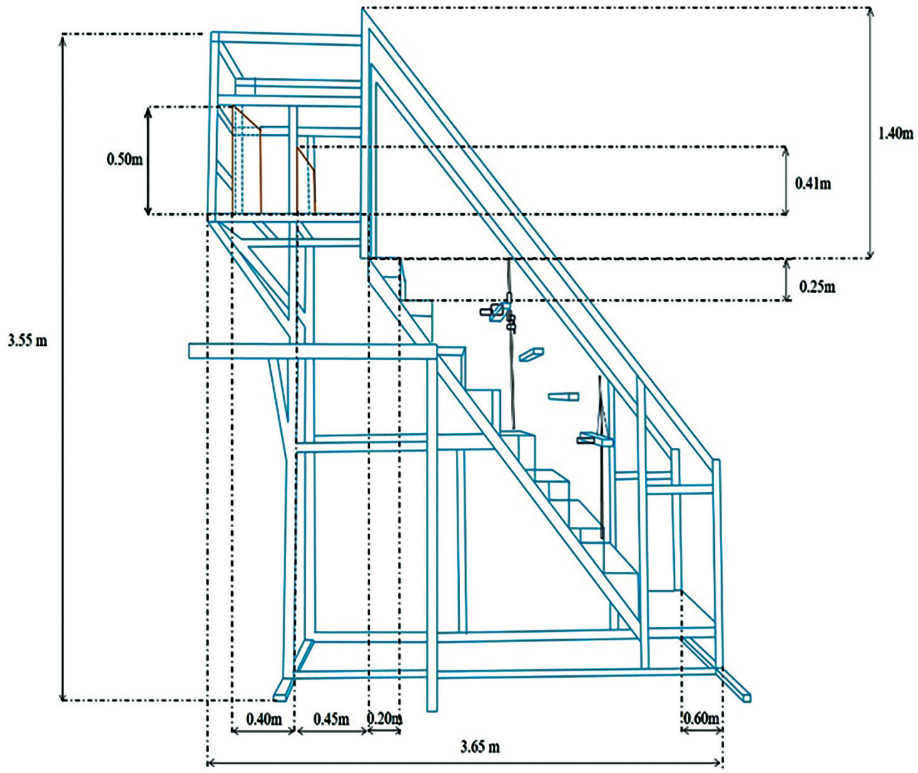

Figure 2: Schematic diagram of the physical model of the stepped spillway.

experiments are difficult or impossible to perform.

- Ability to study systems under hazardous conditions at and beyond their normal performance limits.

CFD approaches are often more flexible than a physical model in changing the physical geometry and/or hydraulic conditions, allowing many configurations to be tested for comparison. With a CFD model, pressures and velocities can be taken at any location in the model, and pressure or velocity contour plots can be created on any plane, allowing the modeler to visualize flow patterns or pressures at any location in the model [22].

3 Materials and Methodology

3.1 *Physical model*

The main concept of the model is it is similar to the prototype in every dimension at the same geometrics in order to obtain the most accurate data from the physical model test. The model, which was designed in metres, is shown in Figure 2. There are 10 spillway face steps with a constant width l = 0.2 m and height h = 0.25 m (l/h = 0.8 m); the total height is 3.55 m, the length is 3.65 m and the width of the model is 0.6 m. The spillway is made of colourless acrylic with a thickness of 0.5 mm, and the structure of the model is made of steel with a thickness of 0.05 m. A depth gauge is installed for measuring water depth at the centre above the 6th step and the 9th step.

The model was tested in a laboratory to obtain the required results, including the discharge, flow behaviour and water depth, after the physical model was constructed following the plan and the design. The discharge was measured in various cases by calculations from the continuity equation. Then, the water depths on the 6th step and the 9th step above the stepped spillway were collected by using the depth gauge. Finally, the data were completely recorded, and all the results were calculated to determine the velocity and energy of the flow.

3.2 *Dimensional analysis for stepped spillways*

The parameters can be divided into 3 types as follows [23]:

- The fluid properties: energy upstream (E_0), energy downstream (E_i), and acceleration of gravity (g).
- Flow characteristics by critical depth (y_c) and discharge (Q).
- The shape properties for the spillway: the height of the spillway (H), area (A), height of the step (h_s), and width of the step (T).

Creating a dimensional analysis of stepped spillways is necessary to use the Buckingham theory for dimensional analysis. Using the pi-theorem of dimensional analysis, the following functional relationship can be ascertained [Equation (3)]:

$$f\left(\frac{E_i}{E_0}, Fr, \frac{y_c}{h_s}\right) = 0 \qquad (3)$$

3.3 *Numerical model*

Computational fluid dynamics (CFD) is a numerical modelling technique capable of solving a wide range of fluid flow problems with or without solid interactions. A CFD analysis conducts an examination of a fluid flow in concordance with its physical properties, such as velocity, pressure, temperature, density, and viscosity. Moreover, both a mathematical model of the physical case and a numerical model are used in a software tool to analyse the fluid flow.

Navier-Stokes equations are often specified as the mathematical model of a physical case. Heat transfer, mass transfer, phase change, and chemical reaction are among the problems that the mathematical model has to vary in accordance with the equations. Furthermore, the verification of the mathematical model is exceedingly important to create an accurate case for solving the problem. The overall process of CFD consists of 3 primary steps: A preprocessing step, a solution step, and a postprocessing step [23].

3.3.1 Preprocessing step

This step consists of defining the geometry to define our domain of interest. The setting accuracy of the data is displayed from this step because the accuracy depends on mesh resolution and geometry.

3.3.2 Solution step

This step starts from mathematical fundamentals to choose a flow model that fits with the simulations, such as Spalart-Allmaras, k-epsilon, k-omega, etc. If possible, the equations are simplified by checking symmetry and dominant flow direction (1D/2D) [24].

3.3.3 Post processing step

This process consists of postprocessing the integral parameters (drag, lift, etc.) and visualizing them in different dimensions, which can be divided into 4 types:

- 1-D: function values connected by straight lines
- 2-D: Contour levels, colour diagrams, and streamlines
- 3-D: Isosurfaces, isovolumes, cutlines, and cutplanes
- Animation of flow, arrow plots, particle tracing, and statistical analysis.

Furthermore, postprocessing includes verification and validation [24].

3.4 *Comparing the data between the physical and numerical models*

After the discharges were obtained by using time recording and calculation, the results from the physical and numerical models will be compared to prove that the numerical model can be used in actual work.

4 Results and Discussion

4.1 *Results from the physical model*

4.1.1 Water depth

The physical model consists of 10 steps in which the 6th step and the 9th step on the spillway are upstream and downstream, respectively, as shown in Figure 3.

The results show that the water depth on the 6th step is lower than the water depth on the 9th step. Moreover, the water depth can indicate the trend of the velocity that occurs on the step. When the discharge is compared with the water depth of both steps, it can

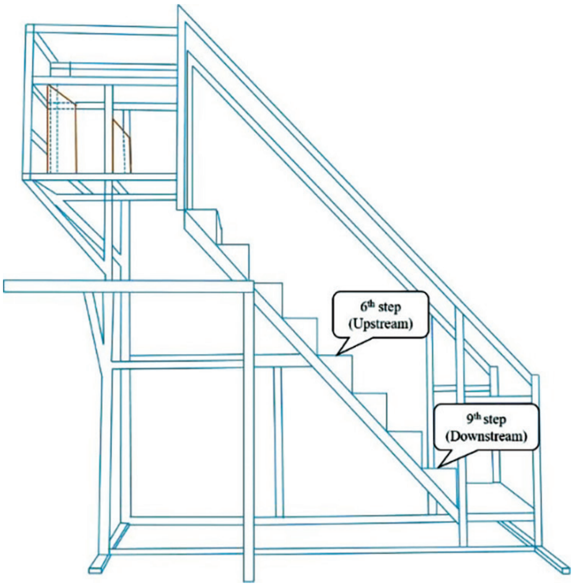

Figure 3: Positions upstream and downstream on the spillway of the physical model.

be seen that when the discharge increases, the water depth increases.

The graph shows that the water depth on the 6th step was lower than the water depth on the 9th step, and the trends of both steps were quite parallel, as shown in Figure 4. The water depth on the 6th step was lower because the amount of water on the 6th step was rapidly drained because the location of the 6th step was near the point of water release, so the amount of water that remained on the 6th step was less than the amount of water on the 9th step. Moreover, the high discharge also affects the water depth because in the lower level of discharge, the water flow passed through the whole surface and then cascaded down the spillway as a succession of free-falling nappes from one step to another, as shown in Figure 5. The trend of the water depth in both steps increases with increasing discharge. Normally, nappe flow was observed in this research.

4.1.2 The velocity on the steps

The upstream and downstream velocities are calculated from the continuity Equation (4), where the discharge from the previous level and the water depth on the 6th step and the 9th step are calculated by the continuity equation:

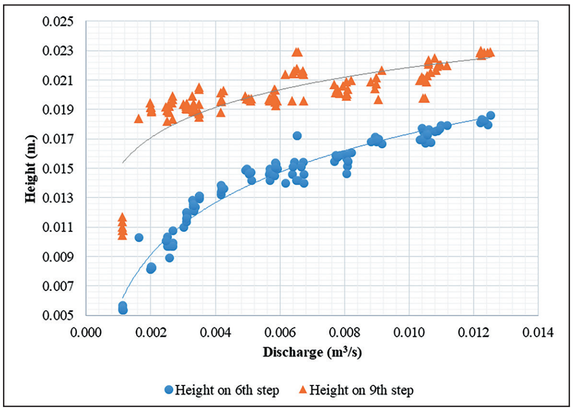

Figure 4: Relationship between discharge and water depth on the 6th step and the 9th step.

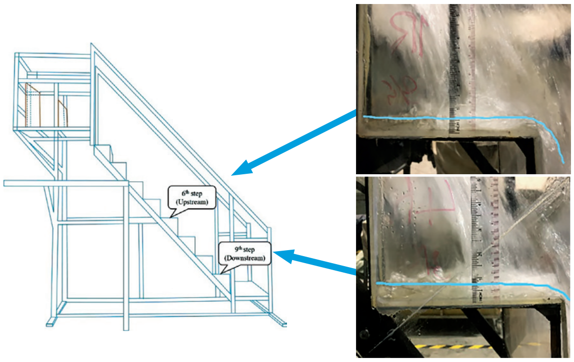

Figure 5: Example of the water depth.

$$Q = AV \quad (4)$$

where Q = Discharge in m^3/s,
A = Area in m^2,
V = Velocity in m/s.

4.1.3 The velocity on the steps

The upstream and downstream velocities are calculated from the continuity Equation (4), where the discharge from the previous level and the water depth on the 6th step and the 9th step are calculated by the continuity Equation (4).

The flow velocity increases continuously when the discharge increases. It can be seen that the flow velocity can be affected by the changing discharge. Furthermore, the R-square values shown in Figure 6 on the 6th step and the 9th step are approximately 0.99. Thus, the velocity and discharge have the same trend.

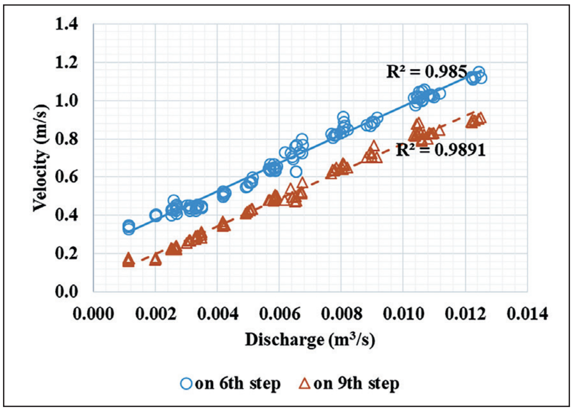

Figure 6: Relationship between discharge and flow velocity.

4.1.4 The energy dissipated on the step

The energy of the flow is calculated by using the energy Equation (5). The results from the experiment illustrate that the stepped spillway can reduce the energy of flow; the percentage of energy dissipation varies from 98.36 to 92.32%. The lower effectiveness of the high discharge may be because the flow of water does not pass through the whole surface. Normally, high energy occurs on the 6th step (upstream), whereas low energy occurs on the 9th step (downstream), as shown in Figure 7. The energy dissipation decreases with increases in discharge for a constant step height. The Froude number, Fr, is a dimensionless value that describes different flow regimes of open channel flow. The Froude number is a ratio of inertial and gravitational forces that is calculated by Equation (6). The Froude number increases as the discharge increases. The percentage of energy dissipation can be predicted by a graph, as shown in Figure 8. The percentage of energy dissipation can be calculated by linear Equation (7). However, the limitation of this equation concerns the discharge; it should be used with discharge ranges between 0.0011 m^3/s and 0.0123 m^3/s.

$$E = \frac{V^2}{2g} + y + H \tag{5}$$

$$Fr = \frac{V}{\sqrt{gD}} \tag{6}$$

where V = Flow velocity in m/s,

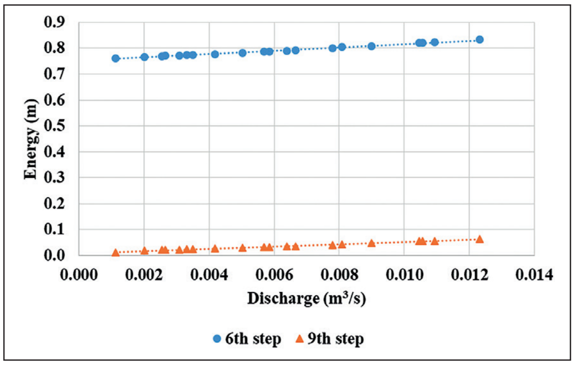

Figure 7: Relationship between discharge and energy.

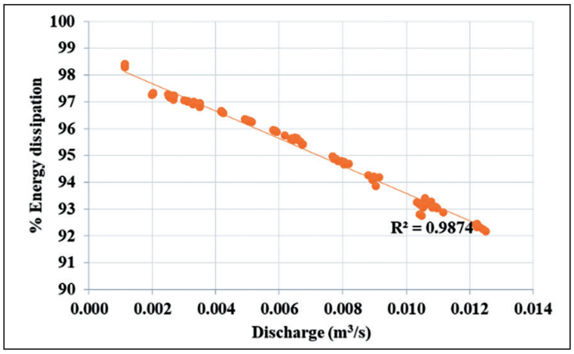

Figure 8: Percent of energy dissipation.

D = Hydraulic depth in m,
g = Gravity in m/s^2

$$E = -510.45Q + 98.707 \tag{7}$$

where E = Percent of energy dissipation,
Q = Discharge in m^3/s.

4.2 *Results from the numerical model*

To create a reliable CFD model to simulate the fluid dynamics behaviour, several mathematical models are applied. In this thesis, the k–epsilon turbulence model is chosen as a mathematical model. In the k–epsilon turbulence model, there are three types of models: the standard k–epsilon turbulence model, the RNG k–epsilon turbulence model, and the realizable k–epsilon turbulence model.

The velocities on the 6th step and 9th step, which were collected by using three models, are shown in Tables 1 and 2, respectively. When the velocities on the

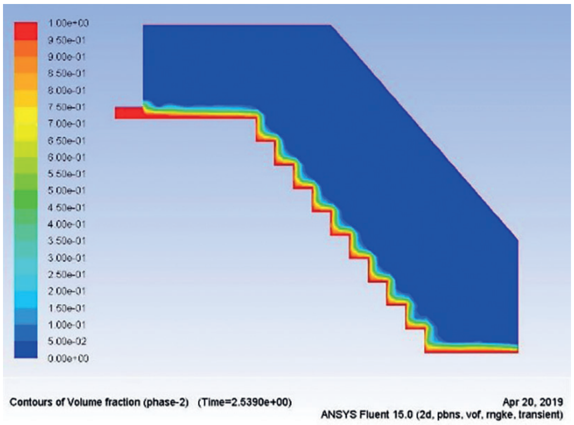

Figure 9: The flow behaviour at Q = 0.0031 m^3/s.

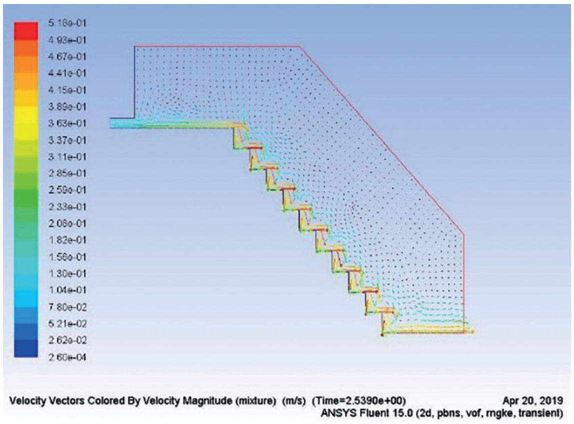

Figure 10: Nappe flow at Q = 0.0031 m^3/s.

6th step and 9th step were compared, the velocities on the 6th step were faster than the velocities on the 9th step because the water depth that remained on the 9th step was greater than the water depth on the 6th step, as shown in Figure 9. Moreover, most of the low discharge flows were nappe flows, as shown in Figures 10 and 11.

Table 1: Results of the velocity on the 6th step from using three submodels

Case	Q (m^3/s)	Velocity on 6th Step (m/s)		
		RNG Model	Standard Model	Realizable Model
1	0.0011	0.319	0.321	0.333
2	0.0020	0.418	0.383	0.406
3	0.0025	0.421	0.425	0.431
4	0.0027	0.429	0.496	0.433
5	0.0031	0.438	0.455	0.450

Table 2 Results of the velocity on the 9th step from using three submodels

Case	Q (m^3/s)	Velocity on 9th Step (m/s)		
		RNG Model	Standard Model	Realizable Model
1	0.0011	0.192	0.193	0.195
2	0.0020	0.168	0.192	0.205
3	0.0025	0.241	0.233	0.215
4	0.0027	0.246	0.248	0.232
5	0.0031	0.292	0.280	0.271

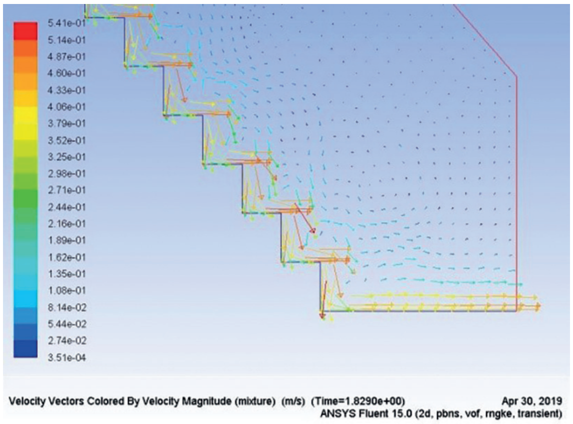

Figure 11: Nappe flow at Q = 0.0033 m^3/s.

4.3 *Comparison of the flow velocity between the physical and numerical models*

The percentage difference in the velocity is a factor that can indicate the accuracy of the results. The percentage differences of velocities from the RNG model are the lowest when the percentage differences of velocities of both models were compared with the physical model, as shown in Figures 12 and 13. Thus, the RNG model is appropriate to simulate the flow behaviour in this research.

The results from the RNG model simulation are chosen for comparison with the results from the physical model. The percentage differences of velocities between the physical model and RNG model are approximately 0.085–11.228%, and the percentage

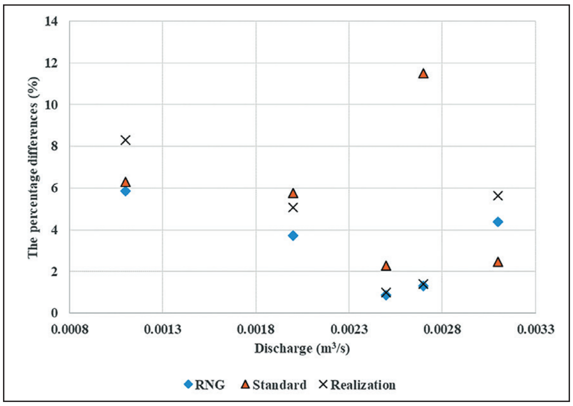

Figure 12: Percent differences in velocity on the 6th step between the physical and three models.

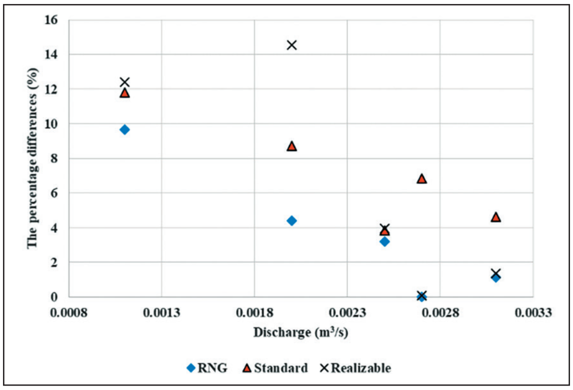

Figure 13: Percent differences in velocity on the 9th step between the physical and three models.

differences on the 6th step and 9th step are not the same. Thus, at the same discharge, the percentage difference on the 6th step may be the highest, but the lowest percentage difference may occur on the 9th step.

5 Conclusions

5.1 *Flow behaviours*

The water depth directly varies with the value of the discharge. Furthermore, the water depth that remains upstream is lower than the water depth downstream. Meanwhile, the flow velocity upstream is faster than the flow velocity downstream. Based on experimental observations, the flow regime that occurs on stepped spillways is nappe flow. The flow regimes in this research can be identified by observing the flow behaviour and proportion between the critical depth and the height of the step. Nappe flow, which occurs in this research, is observed by using a small discharge, and the water flow passes through each step by a free-falling succession at each step edge. A comparison of flow velocity and flow behaviour between the physical and numerical models shows the same trend.

5.2 *Energy dissipation*

The energy dissipation from using a stepped spillway can reduce up to 98% of the energy. Moreover, it can be seen that the discharge affects the reduction of energy because the energy is more dissipated with a low discharge; conversely, it is less dissipated when the discharge increases. The energy that is reduced from each step helps to extend the lifetime of the dam and reduces repair expenses. Furthermore, stepped spillways can be built with small areas.

5.3 *Numerical model*

The percentage difference of velocity between the physical model and numerical model using RNG ranges from 0.09–11.23%. The flow behaviour from both the physical and numerical models is in the same pattern. The results of the percentage difference of velocity and the flow behaviour can confirm the reliability of using the numerical model simulation.

The numerical model simulation can be used instead of a physical model. This approach can reduce the construction costs and time consumption based on experiments. Moreover, the model can predict the flow behaviour that will occur in the future and protect against damage.

Acknowledgements

The authors gratefully acknowledge financial support from the CE-KMUTT Research and strategy fund, provided by Civil Engineering Department, King Mongkut's University of Technology Thonburi.

References

[1] A. J. Aisenbrey, "Design of small canal structures," U.S. Government Printing Office, 1978.
[2] R. M. Sorensen, "Stepped spillway hydraulic model

investigation,"*Journal of Hydraulic Engineering*, vol. 111, no. 12, pp. 1461–1472, 1985.

[3] G. C. Christodoulou, "Energy dissipation on stepped spillways," *Journal of Hydraulic Engineering*, vol. 119, no. 5, pp. 644–650, 1993.

[4] F. Salmasi and M. Ozger, "Neuro-fuzzy approach for estimating energy dissipation in skimming flow over stepped spillways," *Arabian Journal for Science and Engineering*, vol. 39, no. 8, pp. 6099–6108, 2014.

[5] H. Torabi, A. Parsaie, H. Yonesi, and E. Mozafari, "Energy dissipation on rough stepped spillways," *Iranian Journal of Science and Technology*, vol. 42, pp. 325–330, 2018.

[6] A. Parsaie and A. H. Haghiabi, "The hydraulic investigation of circular crested stepped spillway," *Flow Measurement and Instrumentation*, vol. 70, p. 101624, 2019.

[7] H. Chanson, "Stepped spillway flows and air entrainment," *Canadian Journal of Civil Engineering*, vol. 20, no. 3, pp. 422–435, 1993.

[8] A. Parsaie, A. H. Haghiabi, M. Saneie, and H. Torabi, "Prediction of energy dissipation on the stepped spillway using the multivariate adaptive regression splines," *ISH Journal of Hydraulic Engineering*, vol. 22, no.3, pp. 281–292, 2016.

[9] A. H. Haghiabi, H. M. Azamathulla, and A. Parsaie, "Prediction of head loss on cascade weir using ANN and SVM," *ISH Journal of Hydraulic Engineering*, vol. 23, no.1, pp. 102–110, 2017.

[10] A. Parsaie, A. H. Haghiabi, M. Saneie, and H. Torabi, "Applications of soft computing techniques for prediction of energy dissipation on stepped spillway," *Neural Computing and Applications*, vol. 29, no.12, pp. 1393–1409, 2018.

[11] S. Dehdar-behbahani and A. Parsaie, "Numerical modeling of flow pattern in dam spillway's guide wall. Case study: Balaroud dam, Iran," *Alexandria Engineering Journal*, vol. 55, no. 1, pp. 467–473, 2016.

[12] A. Parsaie, S. Dehdar-Behbahani, and A. H. Haghiabi, "Numerical modeling of cavitation on spillway's flip bucket," *Frontiers of Structural and Civil Engineering*, vol. 10, no. 4, pp. 438–444, 2016.

[13] A. Parsaie, A. H. Haghiabi, M. Saneie, and H. Torabi, "Prediction of energy dissipation of flow over stepped spillways using data-driven models," *Iranian Journal of Science and Technology, Transactions of Civil Engineering*, vol. 42, no. 1, pp. 39–53, 2018.

[14] K. W. Frizell and H. K. Frizell, "Guidelines for hydraulic design of stepped spillways," in *Hydraulic Laboratory Report Hl-2015-06*. Colorado: U.S. Department of Interior, 2015, pp. 1–38.

[15] A. A. Sonin, *The Physical Basis of Dimensional Analysis*. 2nd ed. Massachusetts: Massachusetts Institute of Technology, 2001, pp. 9–53.

[16] V. Kulkarni and N. Sahoo, "Module 6 : Lecture 1 dimensional analysis (Part - I.)," 2019. [Online]. Available: https://nptel.ac.in/ courses/101103004/pdf/mod6.pdf

[17] Y. Srihiran, P. Sumphun, and T. Kuljitjuawong, "Study of flow behavior on stepped spillway case study: Mae Suai Dam in Chiang Rai," B.S. thesis, Department of Civil Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, 2016.

[18] M. M. R. Tabari and S. Tavakoli, "Effects of stepped spillway geometry on flow pattern and energy dissipation," *Arabian Journal for Science and Engineering*, vol. 41, pp. 1215–1224, 2015.

[19] S. Terrier, A. Schleiss, and M. Fister, *Hydraulic Performance of Stepped Spillway Aerators and Related Downstream Flow Features*. Lausanne, Switzerland: EPFL, 2016, p. 360.

[20] A. D. Rock, R. Zhang, and D. Wilkinson, *Velocity Variations in Cross-Hole Sonic Logging Surveys Causes and Impacts in Drilled Shafts*. Colorado: U.S. Department of Interior, 2008, pp. 27–43.

[21] S. Barfuss, M. Johnson, Z. Sharp, and B. Tullis, "Hydraulic model studies," 2019. [Online]. Available: https://uwrl.usu.edu/hydrau lics/hydraulic-modeling.

[22] R. A. Jahad, A. Udai, and S. Das, "Energy dissipation and geometry effects over stepped spillways," *International Journal of Civil Engineering and Technology (IJCIET)*, vol. 7, pp. 188–198, 2016.

[23] W. Lahamonchaiyakul and W. Roynarin, "The performance analysis of the micro turbine generate using computational fluid dynamic (CFD)," in *Proceeding of Thailand Renewable Energy for Community Conference*, 2008, pp. 112–128.

[24] Jousfm, "CFD analysis processs," 2019. [Online]. Available: https://www.sim scale.com/forum/t/cfd-analysis-process/66560