Physical Modeling of Flow Behavior on Stepped Spillway Case Study : Mae Suai Dam in Chiang Rai*

Received:	December	31, 2018
Revised:	June	17, 2019
Accepted:	June	18, 2019

Treerat Kuljitjuawong**
Duangrudee K. Kongkitkul***
Chaiwat Ekkawatpanit****
Rachapol Sukjan*****

Abstract

Dam construction for decreasing flood have to consider the size of reservoir and location of the dam. The size of the reservoir should have enough capacity to store a large amount of water. Limitation of a store the amount of water in the reservoir is controlled for the safety of the structure. If there are too much of amount of water, it is drained by using spillway that the high energy dissipation occurs along with the drainage large amount of water and then, the erosion on the spillway. The aim of this study is to observe the differences between the results from the numerical model simulation and the physical model test. The less difference of results will assure that the numerical model simulation results are reliable and can be used to design the prototype. The velocity, energy dissipation, and flow behavior on stepped spillways from the physical model will be compared with the numerical models. The percent of energy dissipation of up to 99 % was found. The percentage of difference in velocity from physical and numerical models are around 2 -4 %.

Keywords: Flow Behavior, Numerical model, Physical model, Stepped Spillway, Energy Dissipation

---

* This manuscript is a part of the research work done during the study towards the degree of Master of Civil Engineering Program.

** Student., Department of Civil Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, Thailand treerat.kul@mail.kmutt.ac.th, (66)952528761

*** Assistant Professor Dr., Department of Civil Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi, Thailand. duangrudee.kos@kmutt.ac.th

**** Assistant Professor Dr., Department of Civil Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi., Chaiwat.ekk@gmail.com

***** Technician, Department of Civil Engineering, Faculty of Engineering, King Mongkut's University of Technology Thonburi.

Veridian E-Journal, Science and Technology Silpakorn University
Volume 6 Number 1 January – February 2019 ISSN 2408 - 1248

Introduction

A structure of the dam for decreasing flood or hydrological problems should have an enough capacity to store a large amount of water. High discharge or velocity of the flow, occurs in a huge structure, can cause the erosion of the dam. A stepped spillway can be an interesting alternative in solving the erosion problem that shows up in the spillway. The reason for choosing the stepped spillway in this study are; it can deliver a high amount of discharge and relieve damage in terms of erosion problem and collapse surface that cause the disaster.

This study focuses on the difference between the results from the simulation and the physical model test. The processes in this study are separated into 2 parts; the first part is using a physical model of stepped spillways to create the flow over the steps and the second part is using Computation Fluid Dynamic (CFD) to simulate the flow behavior.

The flow behavior, the discharge of the flow on the stepped spillway, and the energy dissipation on the spillways from the physical and numerical models are compared.

1. Literature Review

1.1 *Stepped Spillway*

Chatila and Jurdi (2004) proposed that the stepped spillway can decrease the energy dissipation from the high discharge which caused erosion on the spillway. Moreover, the cost of construction is not too expensive. Lesleighter et al. (2014) defined that this kind of spillway is widely used in some countries such as Malaysia.

1.2 *Energy dissipation*

The energy dissipation in stepped spillways has been a research focus currently. Barani et al (2005) and Mero and Mitchell (2016) studied the energy dissipation over various forms of stepped spillways. Felder and Chanson (2011) studied the energy dissipation in stepped spillway with non-uniform step heights. Chafi et al (2010) has noted that the energy dissipation is inversely proportional to the discharge value. Terms of energy equation (1) consist of velocity head, pressure head, and potential head as shown in Fig.1.

$$\frac{V_1^2}{2g} + y_1 + Z_0 = \frac{V_2^2}{2g} + y_2 + \Delta E \qquad (1)$$

Where	$\frac{V_1^2}{2g}, \frac{V_2^2}{2g}$	=	Velocity head
	y_1, y_2	=	Pressure head
	Z_0	=	Potential head
	ΔE	=	Head loss

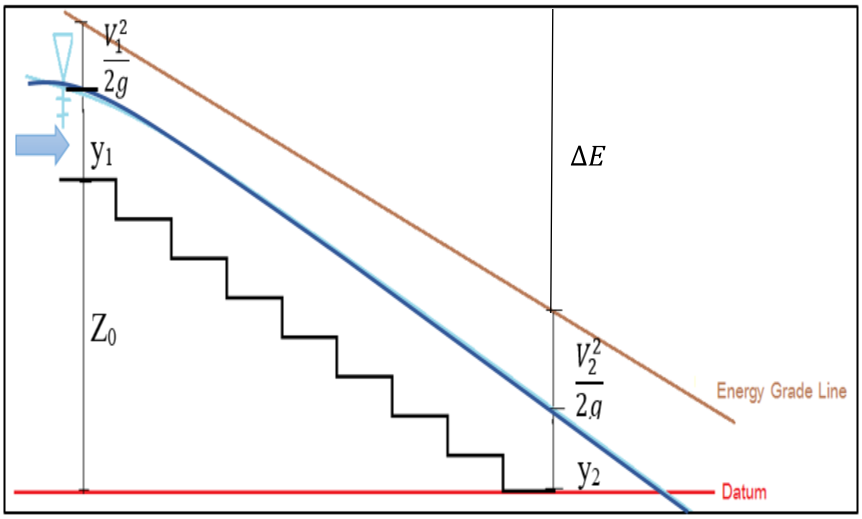

Fig. 1. Cross-section of the typical stepped spillway dam with quantities in the empirical equation**.**

1.3 *Flow regimes*

Flow regimes on the spillway are classified into three types: nappe, transition and skimming flows. Napped flow has occurred at low discharge and it affects from step to step with a full nappe cavity (1996). Skimming flow has occurred while the water falls down the stepped face at high discharge (2002). The transition flow occurs when the discharge is higher than the maximum of nappe flow but lower than the skimming flow.

1.4 *Numerical model*

The numerical method uses numerical analysis to solve and analyze the fluid problem that involves the flow behavior of water passed through the stepped spillway and the flow velocity on the stepped spillway. The numerical method can be used to solve problems easily because it changes from a partial differential equation (Lottes, 2016).

Computational fluid dynamics (CFD) is one of numerical modeling technique capable of solving a wide range of fluid flow problems. Moreover, CFD is the simulation of fluids engineering systems using modeling (mathematical physical problem formulation) and numerical methods (discretization methods, solvers, numerical parameters, and grid generations, etc.) (Zuo, 2016).

2. Materials and Methodology

2.1 *Physical model*

The chute is approximately 2.10 m long, 0.20 m wide, 0.25 m high, and 0.60 m deep of each step. The model consists of 10 steps, the total height is 2.50 m. Acrylics with the thickness of 5 mm were installed at both sides of a chute to provide observation of flow in the chute as shown in Fig. 2.

Veridian E-Journal, Science and Technology Silpakorn University
Volume 6 Number 1 January – February 2019 ISSN 2408 - 1248

Fig. 2. Physical model.

Numerical model

Visavale (2016) defined the process of computational fluid dynamics (CFD) used in the numerical model as shown in Fig. 3.

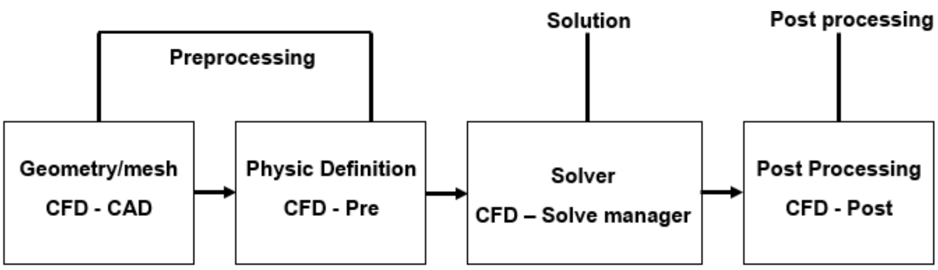

Fig. 3. Process of CFD.

Pre-processing are the step for geometric design, mesh defining and boundary condition defining. The importance of this part is to create things that are needed for this test and to know how it is functioning (Patel, 2013). The process of making the geometry of a model can be described in three parts. The first part is to design the x-y plane. Afterward, the edge on each point is being made which can be easily explained that is the connection line between the vertexes. The final part of making geometry is making the face of the model. After the process of making geometry is created, mesh defining and boundary condition are important in this process. The precision and accuracy of the result depend on how the mesh is defined and from edging and facing process which is a boundary condition, the role set up for geometry (Werayoot and Wirachai, 2008).

The solution step is the setting of initialization and calculation activities. First, the initialization step is to create the condition of value that is needed to show in the model. Second, calculation activities are the specification of the database where the result can be kept while the program is operated (Versteeg and Malalasekera, 1995).

Post – Processing is the step after getting the results to analyze the result by different methods such as means of color plots, contour plots (Lahamornchaikul, 2014). Setting up contour is similar to some graphics capabilities that will make it much easier to illustrate the flow behavior of fluid. Moreover, presentation in various types such as graphics, plots, animations or report can be chosen in a post-processing step.

3. Results and discussion

3.1 *The results from the physical model*

The velocity, energy dissipation, and behavior of flow affected by changing discharge are the results from the physical model. The increased discharge affect the velocities and behavior of flow on both upstream and downstream.

Figs. 4 – 8. show the behavior of flow on each step from the different discharges. The water depth on the 1^{st} step at upstream is lower than the water depth on the 4^{th} step at downstream. Due to the maximum velocity occurs at the upstream and decrease, respectively, so on the 1^{st} step (upstream), the water depth is the lowest but the highest water depth occurs on the 4^{th} step.

Veridian E-Journal, Science and Technology Silpakorn University
Volume 6 Number 1 January – February 2019 ISSN 2408 - 1248

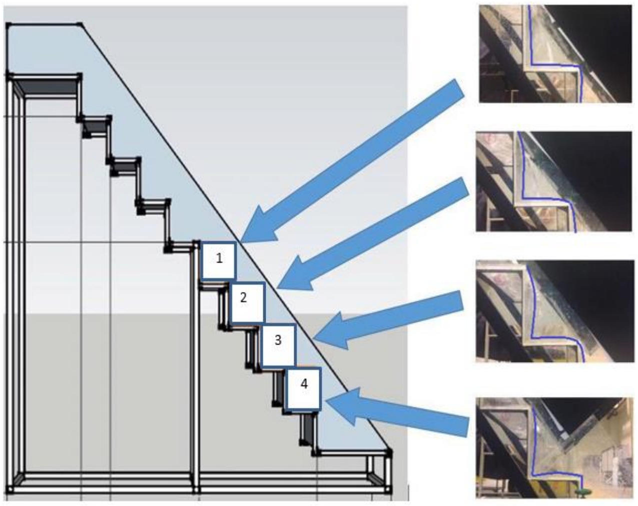

Fig. 4. Flow behavior on the physical model at $Q_1 = 4.21 \times 10^{-4}\ m^3/s$.

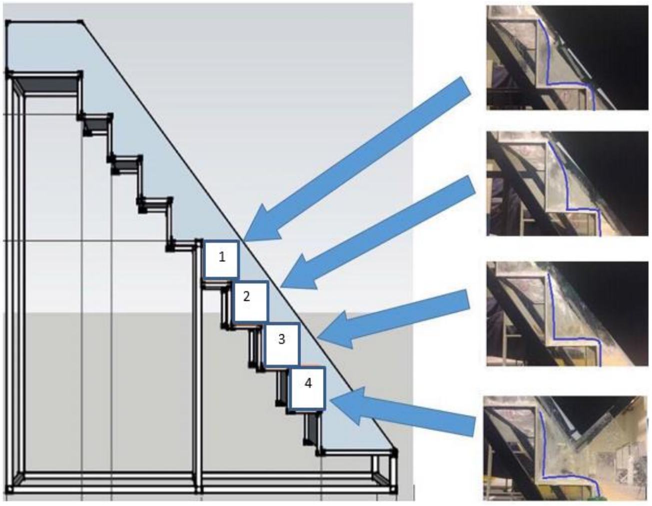

Fig. 5. Flow behavior on the physical model at $Q_2 = 6.53 \times 10^{-4}\ m^3/s$.

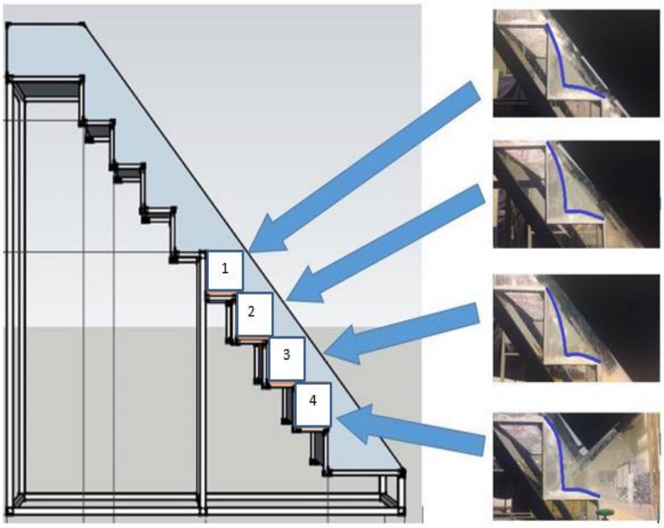

Fig. 6. Flow behavior on the physical model at $Q_3 = 9.10 \times 10^{-3}\ m^3/s$.

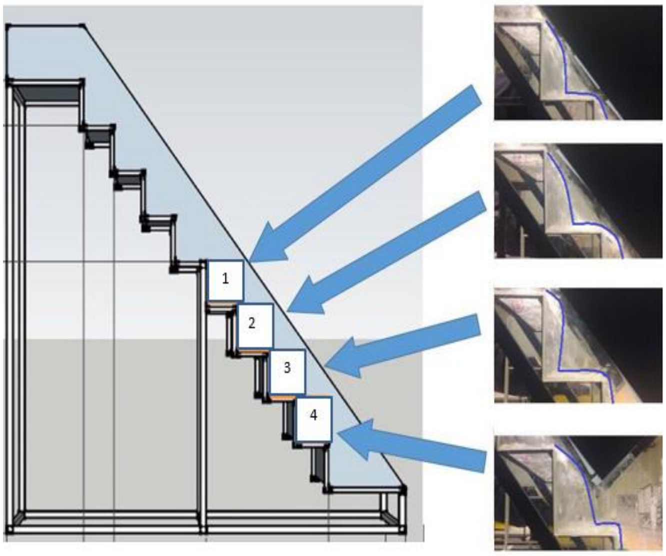

Fig. 7. Flow behavior on the physical model at $Q_4 = 1.216 \times 10^{-3}\ m^3/s$.

There are 5 cases in the present study by using different discharge. The results from the physical model show that stepped spillway can be used to dissipate up to 90 % of energy as shown in Table 1.

Table 1. Results of different energy and percent of energy decreasing.

Case No.	Discharge (m^3/s)	Energy on the 1^{st} step (m)	Energy on the 4^{th} step (m)	% dissipate of energy
Avg.1	0.000421	0.7589	0.0061	99.19
Avg.2	0.000653	0.7671	0.0074	99.03
Avg.3	0.000910	0.7699	0.0093	98.79
Avg.4	0.001216	0.7836	0.0114	98.54
Avg.5	0.001462	0.8050	0.0141	98.25

The R–square is the percent of the response variable variation that is explained by a linear model. It is a statistical measure of how close the data are to the fitted regression line so it means the higher the R-squared, the better the model fits the data [20]. Fig. 9 shows the relationship between the percentage of energy dissipation and discharge in the form of a linear relationship that the trend of energy dissipation varies inversely with discharge. The graph shows that the highest percentage of energy dissipation occurs at the lowest discharge. On the other hand, the lowest percentage of energy dissipation occurs at the highest discharge. It means increasing of discharge affect the decreasing of the percent of energy dissipation. The coefficient of determination or R-square which gets from the graph is 0.9741. The percent of energy dissipation from using the stepped spillway is calculated by the linear equation (2) that is:

$$E = -902.15Q + 99.601 \tag{2}$$

where E = percent of energy dissipation

Q = discharge in m^3/s

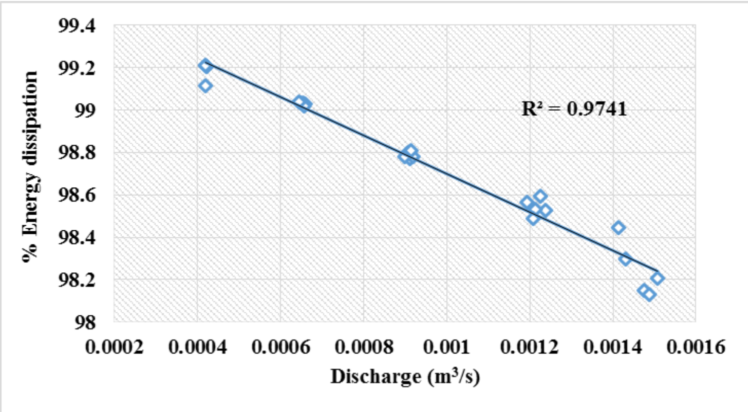

Fig. 9. The relationship between the percentage of energy dissipation and discharge.

3.2 *The results from the numerical model*

The results from the numerical model were focuses on the velocity and flow behavior on the stepped spillway. From Figs. 10 – 14, the flow behavior in each case which gets from the numerical model are quite similar in terms of water depth. The water depth on the 1st step at upstream is lower than the water depth on the 4th step at downstream.

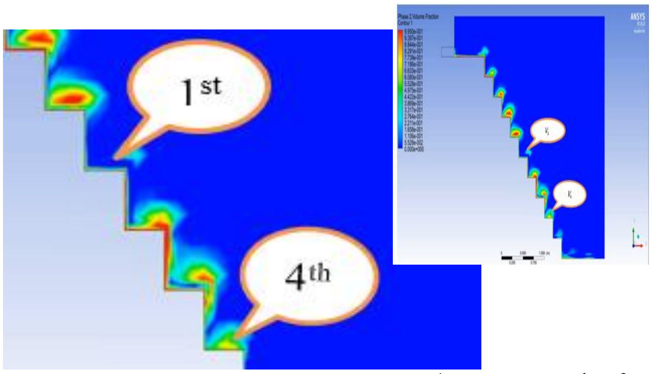

Fig. 10. Flow behavior on the numerical model at Q^1= 4.21 x 10^{-4} m^3/s.

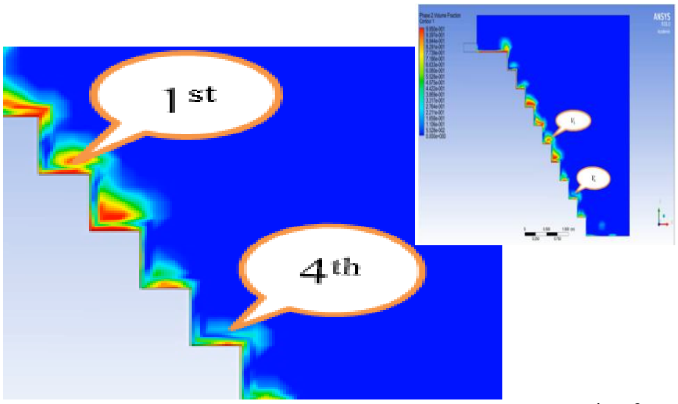

Fig. 11. Flow behavior on the numerical model at Q_2 = 6.53 x 10^{-4} m^3/s.

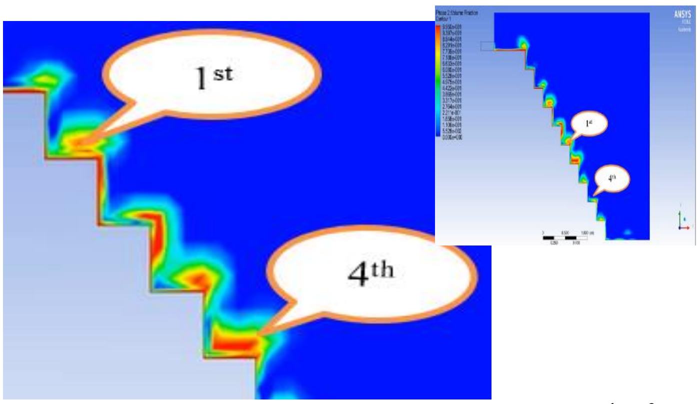

Fig. 12. Flow behavior on the numerical model at Q_3 = 9.10 x 10^{-4} m^3/s.

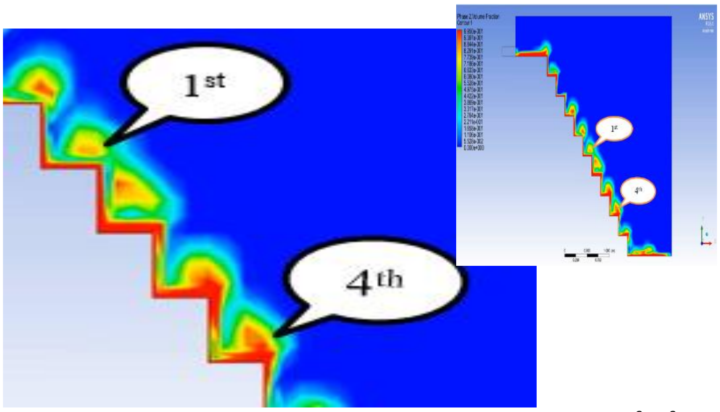

Fig. 13. Flow behavior on the numerical model at $Q_4 = 1.216 \times 10^{-3} m^3/s$.

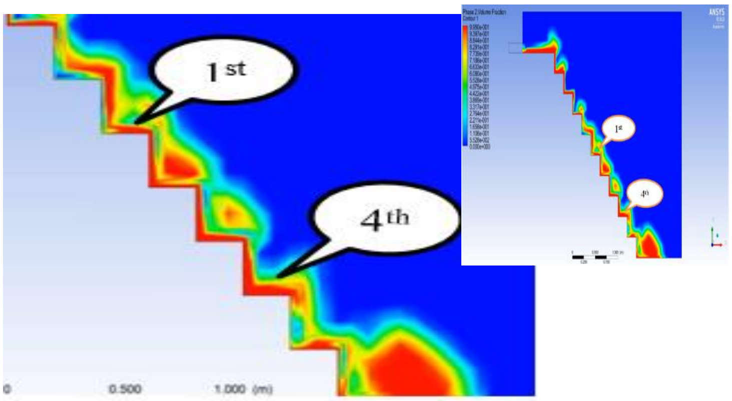

Fig. 14. Flow behavior on the numerical model at $Q_5 = 1.462 \times 10^{-3} m^3/s$.

The velocity profiles, as shown in Figs. 15 – 19, are the results from the numerical model. The results demonstrate that the velocity at the upstream on the 1st step is faster than the velocity at the downstream on the 4th step. From calculation by using continuity equation, the water depth at the upstream is lower than at downstream and the results show that the velocity at the upstream on the 1st step is higher than the velocity at the downstream on the 4th step.

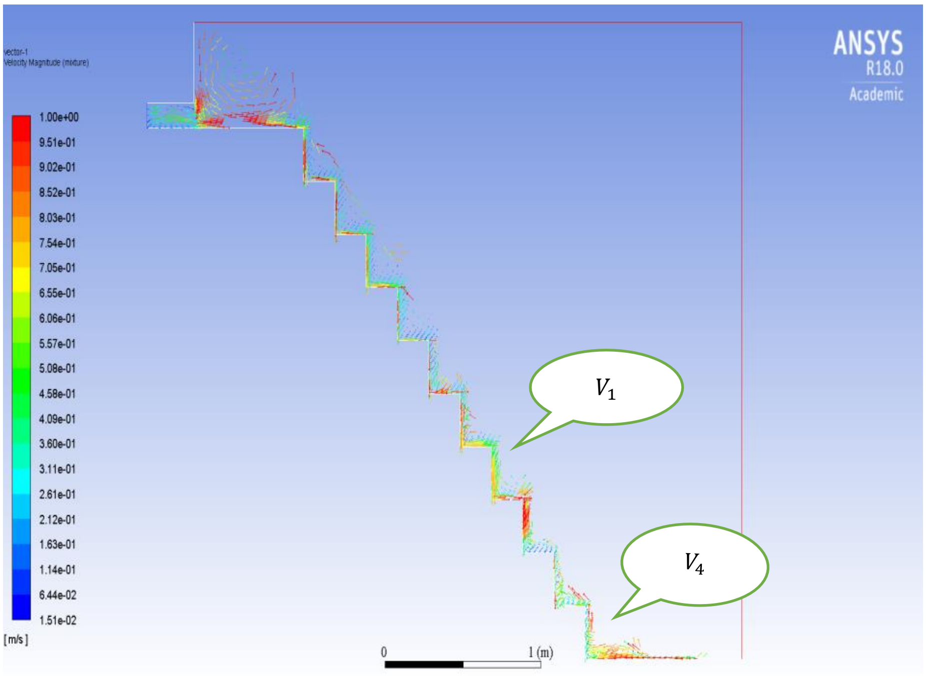

Fig. 15. Velocity on the numerical model at Q_1 = 4.21 x 10^{-4} m^3/s.

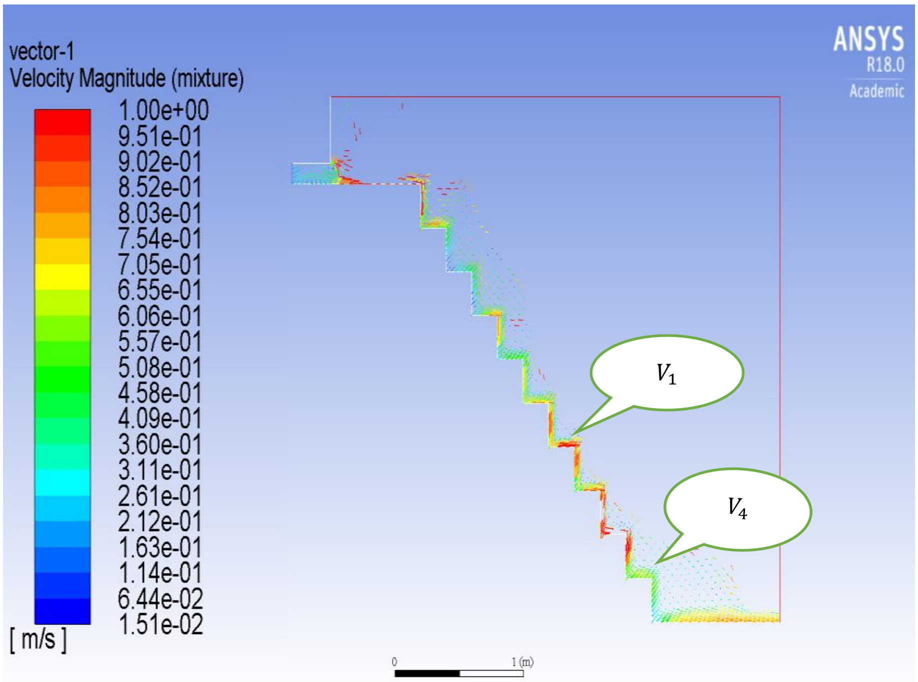

Fig. 16. Velocity on the numerical model at Q_2 = 6.53 x 10^{-4} m^3/s.

Veridian E-Journal, Science and Technology Silpakorn University
Volume 6 Number 1 January – February 2019 ISSN 2408 - 1248

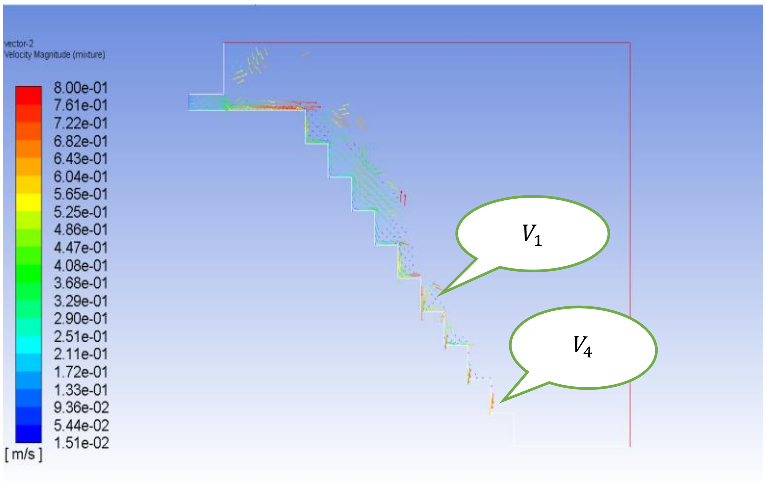

Fig. 17. Velocity on the numerical model at Q_3 = 9.10 x 10^{-4} m^3/s.

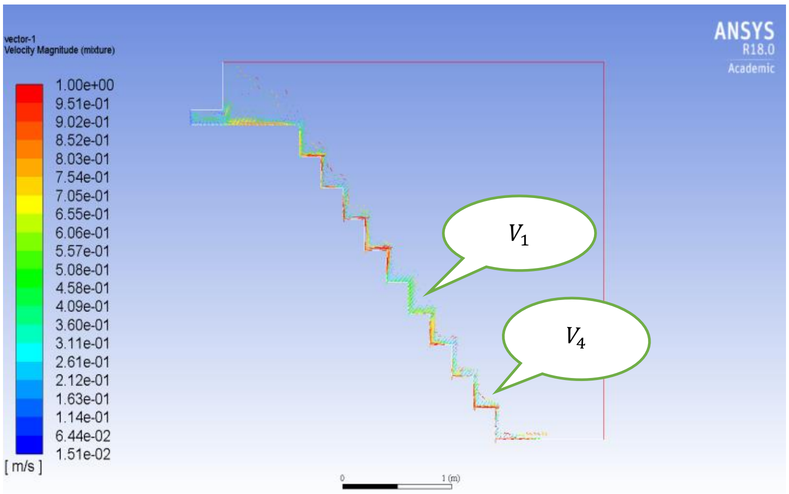

Fig. 18. Velocity on the numerical model at Q_4 = 1.216 x 10^{-3} m^3/s.

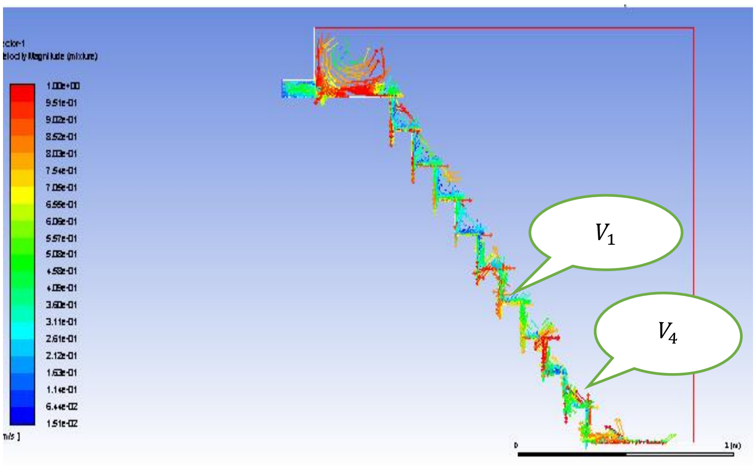

Fig. 19. Velocity on the numerical model at Q_5 = 1.462 x 10^{-3} m^3/s.

The increased discharge affects to the velocity and behavior of flow of both upstream and downstream because of the continuity equation, the velocity and the flow behavior (in term of water depth at the upstream and the downstream) are directly varied with the discharge.

3.3 *Comparison between physical and numerical models.*

The velocity and flow behavior of the physical model can be compared with the results from the numerical model. The flow behavior of both models is likely similar to each other than the water depth on the 1st step at the upstream is lower than the 4th step at the downstream. The water depth of both physical and numerical models on the 4th step is significantly the highest of all steps. The graph is shown in Fig. 20. is the comparison between the velocities from physical and numerical models that in all cases. The velocity from the physical model is lower than the numerical model both at the upstream and the downstream. Moreover, Table 2 and Table 3 show that the percentage difference in velocity on the 1st step is ranged between 3.262 – 3.887 percent and on the 4th step is ranged between 2.258 – 2.954 percent.

It is obvious that the numerical model can be used in a work without any physical model and it can be used in a work such as the analysis of velocity on the stepped spillway. The advantages of applying the numerical model are; it helps to predict the velocity of flow that occurs on the stepped spillway, decreases the duration of the process of analysis for foresees the possibility.

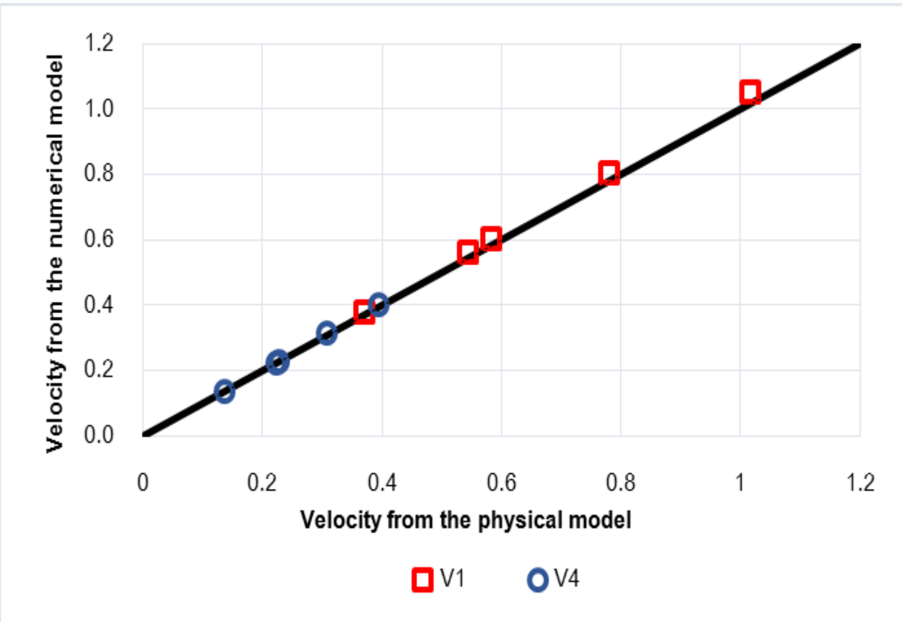

Fig. 20. Comparison of velocity between physical and numerical models.

Table 2. Comparison of velocity on the 1st step between physical and numerical models.

Run no.	Discharge (m^3/s)	V_1 (m/s)		% diff. of velocity
		Physical model	Numerical model	
Avg.1	0.000421	0.369	0.383	3.642
Avg.2	0.000653	0.544	0.566	3.887
Avg.3	0.000910	0.583	0.604	3.421
Avg.4	0.001216	0.779	0.805	3.262
Avg.5	0.001462	1.016	1.056	3.791

4. Conclusion

From the flow behavior on the stepped spillway, the results of both the physical and the numerical models show that the water depth on the 1st step at upstream is always lower than the water depth on the 4th step at downstream.

The advantage of using stepped spillway is the reduction of the energy which occurs on the step. It is causing damage to the dam structure. The less energy also increases the lifetime of this dam, which will help to decrease cost and time consuming about maintenance.

In the study of energy dissipation, the results represent that stepped spillway can be used to reduce the energy up to 99 %.

The results of both the physical and the numerical models proved that the numerical model can be used to analyze the flow behavior. The percentage difference between velocity from physical and numerical models are around 2.2 - 3.9 %. Moreover, both the velocity and flow behaviors on each step of the physical model is in the same trend with the numerical model.

Acknowledgement

The authors gratefully acknowledges the financial support from Department of civil engineering, KMUTT, under the project of research capacity strengthening, fiscal year 2018.

References

Barani G.A., Rahnama M.B., and Sohrabipoor N. (2005). Investigation of flow energy dissipation over different stepped spillways. J.phys.chem.c, 117(14), 7123-7137.

Chatila, J.G. and Jurdi, B.R. (2004). Stepped Spillway as an Energy Dissipater. Canadian Water Resources Journal / Revue canadienne des ressources hydriques 29, 148-156.

Chafi, C., Hazzab, A. and Seddini, A. (2010). Study of flow and Energy Dissipation in Stepped Spillway. Jordan Journal of Civil Engineering, 4(1), 1-11.

Chanson, H. (1996). Prediction of the Transition Nappe/Skimming Flow on a Stepped Channel. Journal of Hydraulic Research, 34(3), 421-429.

Chanson, H. (2002). The Hydraulics of Stepped Chutes and Spillways. Netherlands: Balkema.

Felder S., and Chanson H. (2011). Energy dissipation down a stepped spillway with non-uniform step heights. Journal of Hydraulic Engineering, 137(11), 1543-1548.

Lesleighter, E.J., Tullis, B.P. and Andrews, D. (2014). Stepped Spillway Model Pressures Characteristics, Susu Dam Malaysia. Proc. Of 5th International Symposium on Hydraulic Structures Hydraulic Structures and Society: Engineering Challenges and Extremes. Brisbane: Australia.

Mero S., and Mitchell S. (2016). Investigation of energy dissipation and flow regime over various forms of stepped spillways. Water & Environment Journal, 31(2017), 127-137.

Lottes, S. (2016). Computation Fluid Dynamics. November 15, 2016, Website: https://www.tracc.anl.gov/index.php/transportation-research/computational-fluid-dynamics,

Lahamornchaikul, W. (2014). Computational Fluid Dynamics. Journal of Ladkrabang Engineering, 31(4), 1-6.

Patel, M. (2013). Typical Steps of CFD Simulation Process. November 15, 2016, Website: http://www.hitechcfd.com/cfd-knowledge base/seven-stages-of-a-typical-cfd-simulation.html.

Visavale, G. (2016). Understanding CFD Simulation Process with Examples. November 15, 2016, Website: https://www.lear ncax.com/knowledge-base/blog/bycategory/cfd/understandding-cfd-simulation-process-with-examples.

Versteeg, H.K. and Malalasekera, W. (1995). In An Introduction to Computational Fluid Dynamics:The finite Volume Method 2nd Edition. London: Pearson Education Limited.

Veridian E-Journal, Science and Technology Silpakorn University
Volume 6 Number 1 January – February 2019 ISSN 2408 - 1248

Werayoot, L. and Wirachai, R. (2008). The Performance Analysis of the Micro Turbine Generate Using Computational Fluid Dynamics (CFD). Proceeding of Thailand Renewable Energy for Community Conference (pp.112-128). Thailand

Zuo, W. (2015). Introduction of Computational Fluid Dynamics. November 15, 2016, Website: http://www.mayr.informatik.tumuenchen.de/konferenzen/Jass05/courses/2/Zuo/Zuo_paper.pdf

.