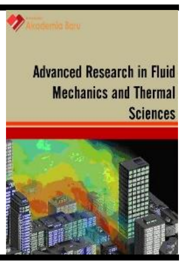
Penerbit Akademia Baru

Journal of Advanced Research in Fluid Mechanics and Thermal Sciences

Journal homepage: www.akademiabaru.com/arfmts.html
ISSN: 2289-7879



Numerical Investigation of Hydraulic Characteristics on 3D Unsymmetrical Smooth Spillway

Open Access

Mohd Safarul Izmi Saidin[1], Mohd Sharizal Abdul Aziz[1,*], Mohd Remy Rozainy Mohd Arif Zainol[2]

[1] School of Mechanical Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Pulau Pinang, Malaysia
[2] School of Civil Engineering, Universiti Sains Malaysia, Engineering Campus, 14300 Nibong Tebal, Pulau Pinang, Malaysia

ARTICLE INFO

Article history:
Received 7 October 2019
Received in revised form 13 November 2019
Accepted 15 November 2019
Available online 15 December 2019

Keywords:
Smooth spillway; volume of fluid; numerical simulation; hydraulic characteristics

ABSTRACT

In recent decades, several studies of hydraulic characteristics of spillway have been conducted. At the initial stage of design, hydraulic characteristics of spillway shall be investigated thoroughly either experimentally or numerically to provide better insight of flow behavior and predicting any critical area which can endanger the structure and the environment. The present paper focused on the numerical study of free surface flow over the three dimensional (3D) unsymmetrical smooth spillway. This paper aims to investigate hydraulic characteristics by using 3D numerical simulation in terms of water surface profiles, surface velocity profile and velocity profile at the downstream. The governing equation of Reynolds-averaged Navier-Stokes (RANS) equation is numerically solved using volume of fluid (VOF) method and Realizable κ-ε (RKE) turbulence model. The numerical simulation successfully estimated hydraulic characteristics and show a good agreement in terms of profile pattern compared to establish papers.

1. Introduction

The smooth spillway is a conventional hydraulic structure that been used to release a surplus of water over the crest down to the toe properly and prevent any damages to the surrounding by dissipating excess kinetic energy [1]. By combining with the stilling basin at the toe of the spillway, the efficiency of energy dissipation will increase. At the upstream of the spillway, crest or labyrinth been used to spill the water efficiently while the chute will function to carry down the upstream water to the stilling basin which functions as energy dissipator [2].

The experimental study shall be considered as the most reliable study, but the model setup can be a time-consuming process and costly compared to numerical study [3-5]. Alternatively, the numerical simulation had been used in many years to estimates the hydraulic performance of smooth spillway geometry. Several authors have focused on ogee-type and circular-type of smooth spillway

* *Corresponding author.*
E-mail address: msharizal@usm.my (Mohd Sharizal Abdul Aziz)

[1,3-7]. All these authors were focused on discharge coefficient, discharge flow, flow pattern, pressure characteristics, water surface profiles, and velocity profiles on certain critical areas. Most of them were used two-dimensional (2D) geometry except [5-7] where three-dimensional (3D) geometry was used in the numerical simulation of the spillway. Most of them were applying standard VOF method and κ-ε turbulence model such as standard κ-ε (SKE) [3, 7-8], re-normalization group κ-ε (RNG) [6-7, 9-11] and realizable κ-ε (RKE) [7,12] which provide good agreement with the experimental result in terms of hydraulic characteristics. In previous literature, many researchers used and proposed RKE turbulence model for numerical simulation of spillway due to stability and performances [13-15].

In general, flow characteristics along the spillway can be categorized base on dimensionless Froude Number, Fr value. The Froude number can be defined as $Fr = u/(gh)^{1/2}$ where u is the approach velocity (m/s), g is gravitational acceleration (m/s^2) and h is the water depth (m). There are three types of flow conditions which are subcritical flow (Fr > 1), critical flow (Fr =1) and supercritical flow (Fr >1). The subcritical flow is a gentle flow while the supercritical condition is a rapid flow and the critical flow is the transition flow between subcritical and supercritical [2,8].

Water surface and velocity profiles of spillway are essential hydraulic characteristics for the researcher to investigate. In the open channel flow study, the water surface profile or water depth is unknown in advance compared to water flow in the pipe where the water depth will depend on the diameter of the pipe [2]. Both characteristics were depending on the water discharge, Q and the Froude number. Base on previous publish paper mentioned, commercial numerical simulation software such as ANSYS had been used to estimate and captured various hydraulic characteristics such as velocity profiles, pressure distribution, water surface profiles, turbulence kinetic energy and so forth. Thus, in this present paper aim to investigate of capability numerical model to estimate water surface profile and hydraulic characteristics along the 3D unsymmetrical smooth spillway by applying VOF method and RKE turbulence model.

2. Numerical Simulation Methodology

By using the geometry of the unsymmetrical smooth spillway model constructed at the hydraulic laboratory of Universiti Sains Malaysia (USM), 3D water flow simulation was performed by using ANSYS software to solve numerical simulation involving Reynolds Average Navier Stokes (RANS) equation. Spillway geometry with 10.9 m (Length) x 1.8 m (Height) x 3.7 m (maximum width) with the chute slope of 8.6 degrees had been drawn by using SpaceClaim tools which built- in the ANSYS as shown in Figure 1 below.

This model consists of four main components which are a labyrinth, chute piers, smooth chute, and stilling basin. The water will flow from the inlet and flow over the labyrinth down through the chute piers and chute towards the stilling basin as depicted in Figure 2.

Base on this solid geometry, volume extraction been generated to get a volume domain. In order to make the meshing process smoother, some modification on geometry had been done to replicate as per laboratory model, and the volume domain had been cut into five bodies for better mesh pattern.

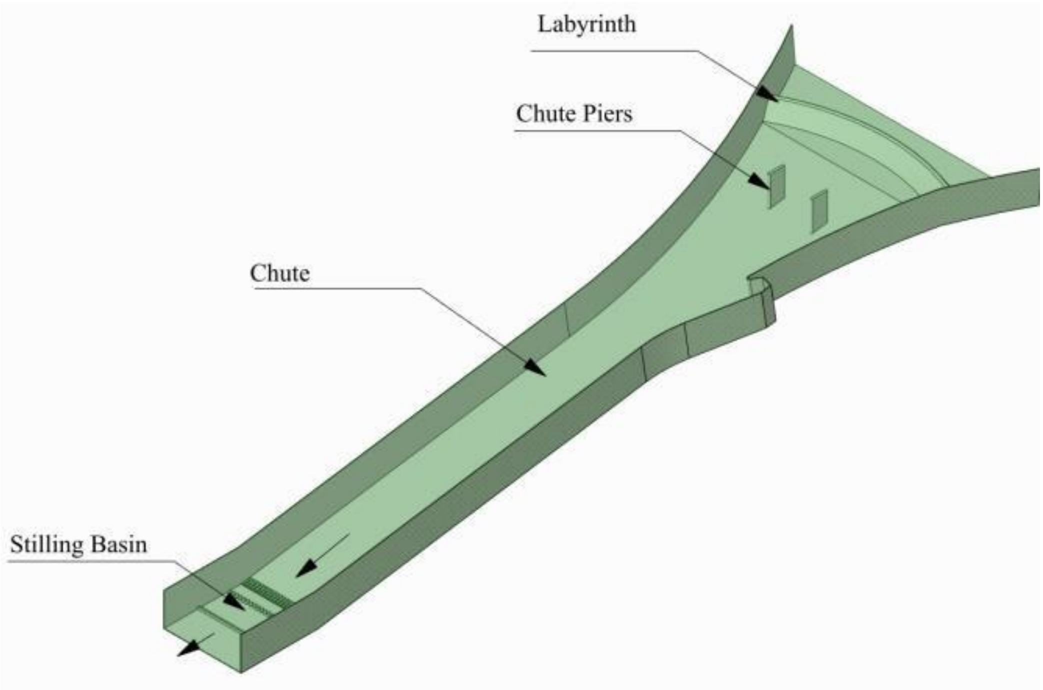


Fig. 1. Detail of spillways geometry

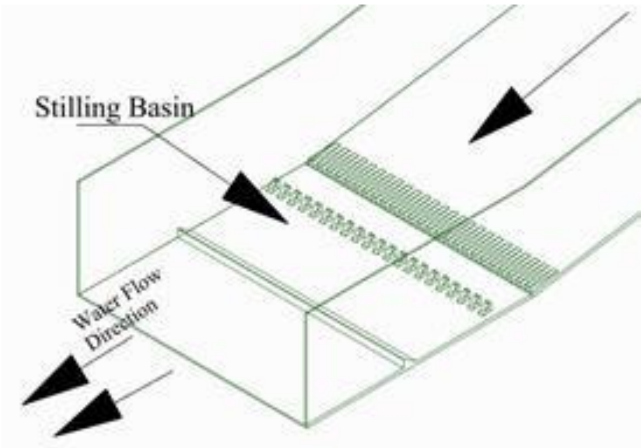


Fig. 2. Stilling basin region

Due to the large and complex model, unstructured tetrahedrons mesh was applied to fit the unsymmetrical geometry. To provide a better boundary layer of flow in simulation, the inflation layer has also been applied at the bottom of the spillway geometry. The mesh rendering process successfully produces 2,335,718 elements with the minimum size of mesh was 1.5mm as shown in Figure 3. The minimum size of the mesh must be considered in this rendering process to prevent defeaturing occurred at the stilling basin region due to small geometry of baffles arrangement.

The generated mesh had been assigned with the named selection for boundary conditions (Figure 4). This name selections will provide initial information to the software about the boundary condition type that will be used during CFD-Post process in Fluent ANSYS. Due to VOF application, water and air domain had assigned during the meshing process while the fluid properties and phases will be determined during numerical setup in the Fluent module.

Fig. 3. The unstructured mesh of volume geometry

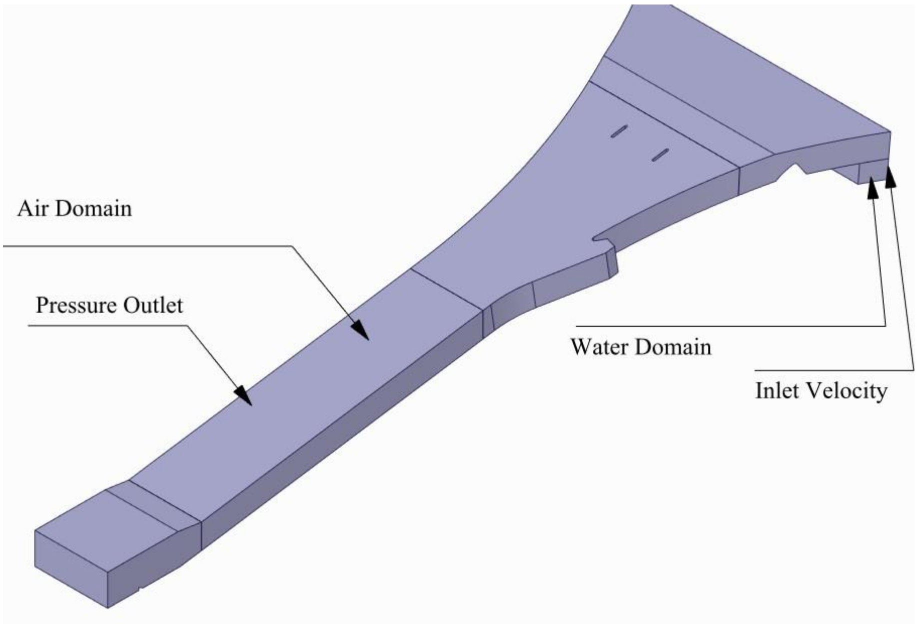


Fig. 4. Volume domain and boundary condition

After finishing the meshing process, the spillway simulation setup has been done in the Fluent module. In this module, the RANS equation was solved by using the VOF method outlined by Hirt and Nichols [17] and the RKE turbulence model introduced by Shih *et al.,* [18]. The VOF method was used to track interaction between volume fraction of air, α_a and volume fraction of water, α_w by using transport equation [19]. Fluid properties of air and water had been chosen and the initial volume of each fluid also been determined in this module. The water flow had been setting in term of inlet velocity, and the free flow will be accumulated at the upstream region before it spills down to the downstream over the labyrinth. In this transient simulation, the total simulation time for the water to flow down to the end of the outlet boundaries was 18 seconds and the total calculation time had been recorded was 3.6 days by using provided workstation computer. The summary of the numerical setup was shown in Table 1.

Table 1
Summary of numerical setup

Numerical set up	Details
Total volume bodies	5 bodies
Size function	Proximity and curvature
Minimum mesh size	1.5 mm
Maximum mesh size	192.0 mm
Inflation layer	5 layers
Growth rate	1.2
Nodes	617,705
Elements	2,335,718
Multiphase model	VOF
Turbulence model	Realizable κ-ε (RKE) turbulence model
Boundary conditions	Water and air domain; Velocity inlet, pressure outlet and wall
Inlet water flow, Q	0.060 m^3/s
Inlet velocity, u	0.051 m/s
Total time step	1800
Simulation time	18 s
Calculation time	3.6 days

3. Results and Discussion

3.1 Results

Base on a 3D simulation been conducted, hydraulic characteristics of water flow over the spillway are presented below in terms of water surface profile, surface velocity, and downstream velocity profiles (tailwater region). To analyze the result, the volume fraction of water which had a range of volume fraction $0 \leq \alpha_W \leq 1$ was rendered, and the contour was depicted in Figure 5. The volume fraction contour provides the maximum water surface profiles simulated at the time value of 18s. The highest water depth was a sudden rise in the stilling basin went the high-velocity water flow collided with the baffle at the stilling basin region. This transition is called hydraulic jump phenomena.

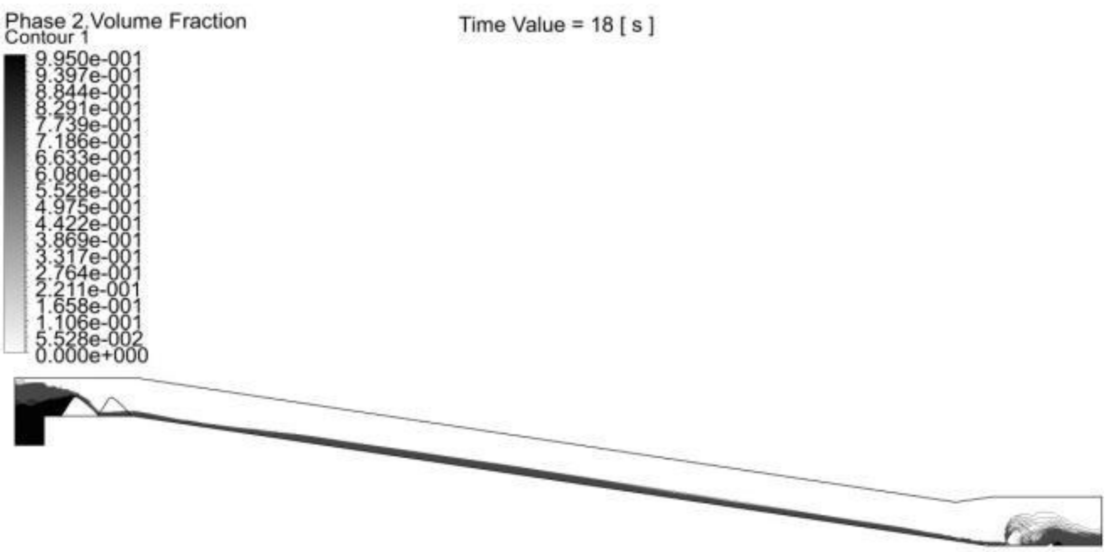


Fig. 5. Midplane volume fraction contour of the spillway

All the results shown below were extracted and plotted base on $\alpha_W = 0.5$ and this condition which was considered as a standard setting to extract the average water surface profile from the results [10]. The water discharge flow over the crest down through the chute spillway was very shallow with estimated depth 0.02m on the chute as shown in Figure 6 before the water depth arose back when the hydraulic jump occurred at the stilling basin region. The maximum water depth at the top of the labyrinth was estimated 0.03m, and maximum water depth estimated 0.09m occurred at the stilling basin area.

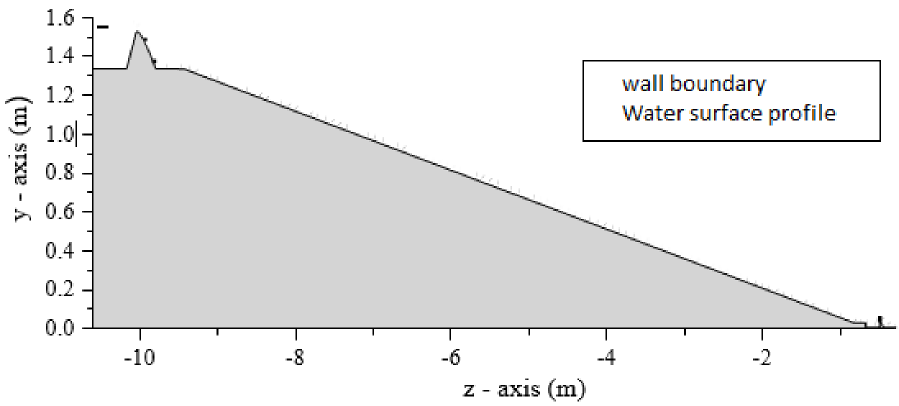


Fig. 6. Water surface profile along the spillway

Base on Figure 7 and Figure 8, the first peak of velocity occurred at the end of the labyrinth, and water flow starts to decelerate to the end of the flat basin before it accelerates back and reaches the maximum velocity at the stilling basin. The stilling basin was performed well to slow down the water flow from upstream before it accelerates back to the end of the boundary. The stilling basin successfully decelerate water flow by 40.16 % from the maximum velocity.

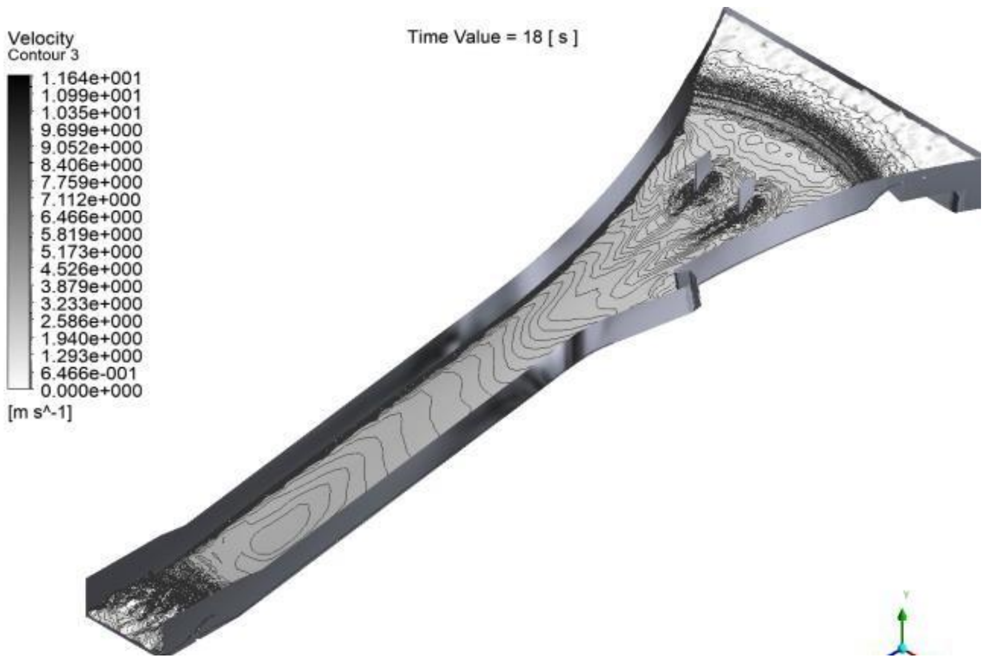


Fig. 7. Velocity contour along the spillway

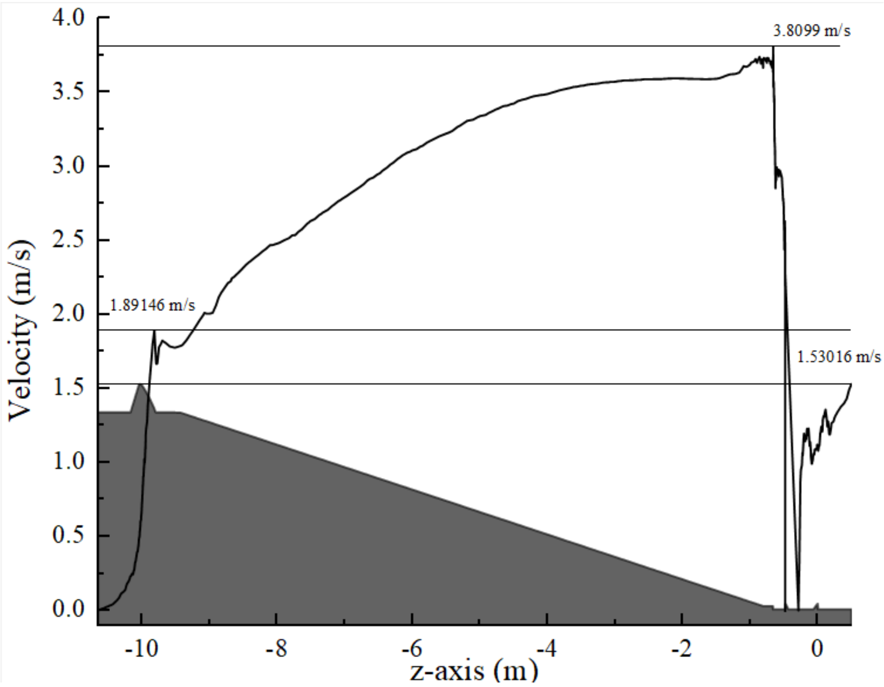


Fig. 8. Surface velocity profiles along the spillway

The velocity distributions were plotted at the tailwater region after the stilling basin region and were depicted in Figure 9. At 15 location by using 50 samples point for each location, the graph provides a smooth pattern of velocity distributions. Base on this result, the transition from turbulent profile to the laminar profile was captured and been rearrange in the graph as shown in Figure 10. At z = 0.030 m, the local maximum velocity = 1.353 m/s and keep increase to end of the boundary at z = 0.492 m with the local maximum velocity = 1.53 m/s. The water depth at this tailwater area slightly uneven where it keeps decreasing from 0.10450 m to 0.03231 m.

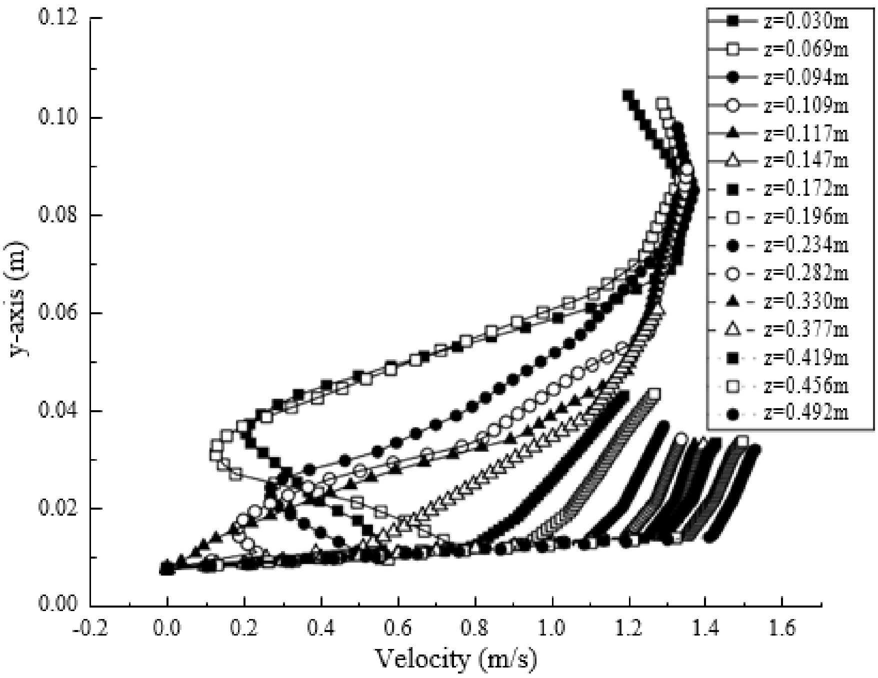


Fig. 9. Velocity distributions from z= 0.030 ~ 0.492m

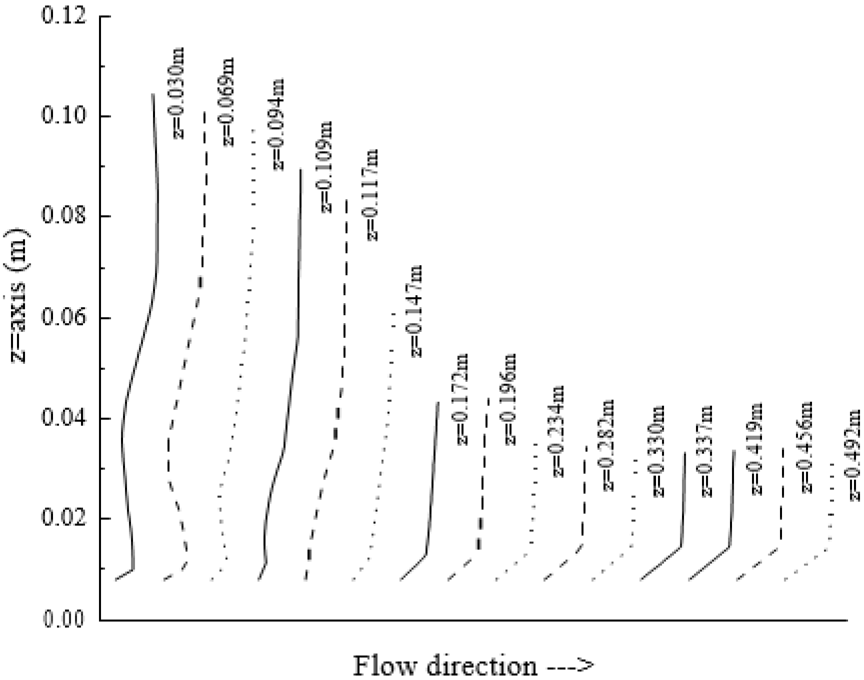


Fig. 10. Velocity profiles from z= 0.030 ~ 0.492m

3.2 Discussion

At the upstream region, water flow was in the subcritical condition where Froude No. (Fr) is less than 1 which can be observed from Figure 8 where the velocity is less than 1 m/s and water depth were high. The supercritical flow (Fr >1) occurred at the chute area, where the water flow accelerated by gravity with high velocity and the water depth was shallow. When the water reaches the stilling basin, hydraulic jump occurred, and the flow transformed to the subcritical flow again. The maximum velocity occurred when the water flow reached the stilling basin area. The hydraulic jump occurred in this region were assisted by baffle blocks to dissipate the excess kinetic energy of water flow [2,20].

By comparing Figure 9 and Figure 10 with other papers such as [3,8,19,21], the results showed similarity in terms of the pattern at a certain point as shown in Figure 11 and Figure 12. Figure 9 indicates that the velocity gradually increased towards the channel and the acceleration will keep increasing until it reaches the equilibrium condition [2]. To get this result, the boundary of the downstream channel shall be extending to a certain range, and the equilibrium velocity profiles can be plotted.

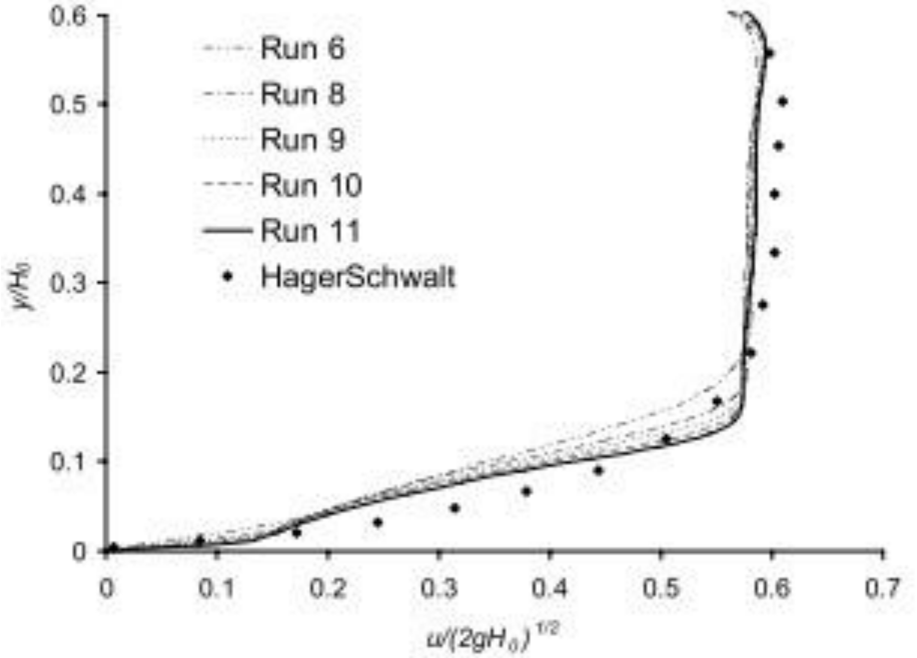


Fig. 11. Velocity profiles on the flat rectangular flume after the weir [19]

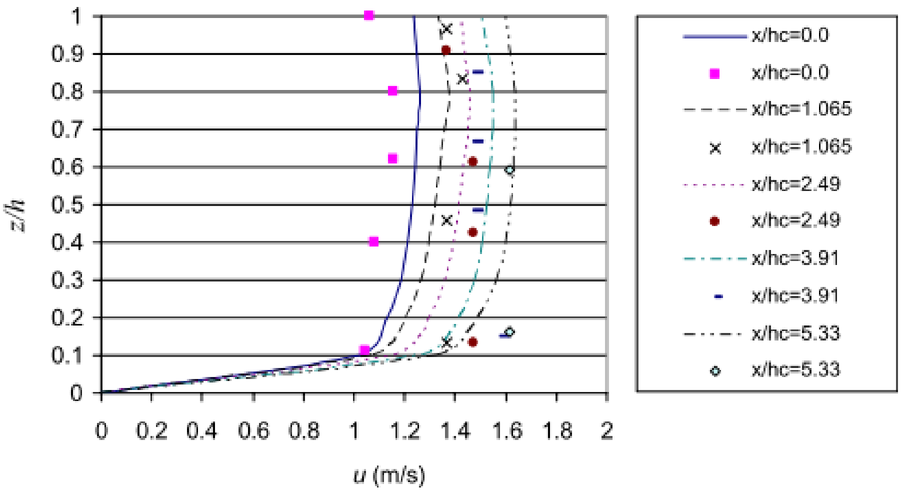


Fig. 12. Velocity profiles at multiple locations along the chute spillway [8]

4. Conclusions

The numerical investigation of smooth spillway with the unsymmetrical geometry by using VOF method and realizable κ-ε turbulence model was carried out. The numerical method applied successfully estimated water surface profiles, surface velocity profiles along the spillway and velocity profiles at the end of the boundary. The results indicate three peaks of maximum velocity before water flow reached the downstream channel and the laminar flow development after the stilling basin towards the channel. The numerical model successfully provides reasonably flow characteristics results and additional referral paper for researcher especially in the computational fluid dynamic field of research. With the motivation and confidence gained in this study, it is planned to use a similar method and software to further study on this geometry with multiple flow configuration and geometry modification by using establish validation method by comparing the numerical result with the experimental result.

Acknowledgement
The authors would like to extend their appreciation to Universiti Sains Malaysia (Research University Grant (RUI) 1001/PMEKANIK/ 8014072) for providing the financial and technical support for this research. The authors also acknowledge the School of Mechanical Engineering and School of Civil Engineering, Universiti Sains Malaysia for the lab facilities. The author also acknowledges the Department of Irrigation and Drainage Malaysia (JPS) and Public Service Department of Malaysia (JPA) for the scholarship. Special thanks to those who contributed to this project directly or indirectly.

References

[1] Chatila, Jean, and Mazen Tabbara. "Computational modeling of flow over an ogee spillway." *Computers & structures* 82, no. 22 (2004): 1805-1812.
[2] Chanson, Hubert. *Hydraulics of open channel flow*. Elsevier, 2004.
[3] Kirkgoz, Mehmet Salih, Mevlut Sami Akoz, and Ahmet Alper Oner. "Numerical modeling of flow over a chute spillway." *Journal of Hydraulic Research* 47, no. 6 (2009): 790-797.
[4] Savage, Bruce M., and Michael C. Johnson. "Flow over ogee spillway: Physical and numerical model case study." *Journal of hydraulic engineering* 127, no. 8 (2001): 640-649.
[5] Dehdar-Behbahani, Sadegh, and Abbas Parsaie. "Numerical modeling of flow pattern in dam spillway's guide wall. Case study: Balaroud dam, Iran." *Alexandria Engineering Journal* 55, no. 1 (2016): 467-473.
[6] Dargahi, Bijan. "Experimental study and 3D numerical simulations for a free-overflow spillway." *Journal of Hydraulic Engineering* 132, no. 9 (2006): 899-907.
[7] Rahimzadeh, H., R. Maghsoodi, H. Sarkardeh, and S. Tavakkol. "Simulating flow over circular spillways by using different turbulence models." *Engineering Applications of Computational Fluid Mechanics* 6, no. 1 (2012): 100-109.
[8] Mohamed, Hassan Ibrahim. "An investigation of turbulent accelerated flow along smooth prismatic chutes." *Water and Environment Journal* 27, no. 1 (2013): 71-78.
[9] Kim, Dae Geun, and Jae Hyun Park. "Analysis of flow structure over ogee-spillway in consideration of scale and roughness effects by using CFD model." *KSCE Journal of Civil Engineering* 9, no. 2 (2005): 161-169.
[10] Johnson, Michael C., and Bruce M. Savage. "Physical and numerical comparison of flow over ogee spillway in the presence of tailwater." *Journal of hydraulic engineering* 132, no. 12 (2006): 1353-1357.
[11] Gadhe, Vaishali, R. G. Patil, and V. V. Bhosekar. "Performance assessment of upgraded spillway–case study." *ISH Journal of Hydraulic Engineering* (2018): 1-9.
[12] Teng, Penghua, and James Yang. "CFD modeling of two-phase flow of a spillway chute aerator of large width." *Journal of Applied Water Engineering and Research* 4, no. 2 (2016): 163-177.
[13] Cheng, Xiangju, Yongcan Chen, and Lin Luo. "Numerical simulation of air-water two-phase flow over stepped spillways." *Science in China Series E: Technological Sciences* 49, no. 6 (2006): 674-684.
[14] Jahani, Matin, Hamed Sarkardeh, and Ebrahim Jabbari. "Effect of using guide walls and piers with different geometries on the flow at entrance of a spillway." *The European Physical Journal Plus* 133, no. 3 (2018): 91.
[15] Valero, Daniel, Daniel B. Bung, and Brian M. Crookston. "Energy dissipation of a Type III basin under design and adverse conditions for stepped and smooth spillways." *Journal of Hydraulic Engineering* 144, no. 7 (2018): 04018036.
[16] Chow, Ven T. "Open-channel hydraulics." *McGraw-Hill civil engineering series* (1959).
[17] Hirt, Cyril W., and Billy D. Nichols. "Volume of fluid (VOF) method for the dynamics of free boundaries." *Journal of computational physics* 39, no. 1 (1981): 201-225.
[18] Shih, Tsan-Hsing, William W. Liou, Aamir Shabbir, Zhigang Yang, and Jiang Zhu. "A new k-ϵ eddy viscosity model for high reynolds number turbulent flows." *Computers & Fluids* 24, no. 3 (1995): 227-238.
[19] Hargreaves, D. M., H. P. Morvan, and N. G. Wright. "Validation of the volume of fluid method for free surface calculation: the broad-crested weir." *Engineering Applications of Computational Fluid Mechanics* 1, no. 2 (2007): 136-146.
[20] Li, Lian-xia, Hua-sheng Liao, Da Liu, and Sheng-yin Jiang. "Experimental investigation of the optimization of stilling basin with shallow-water cushion used for low Froude number energy dissipation." *Journal of Hydrodynamics* 27, no. 4 (2015): 522-529.
[21] Qingchao, Liu, and Uwe Drewes. "Turbulence characteristics in free and forced hydraulic jumps." *Journal of hydraulic research* 32, no. 6 (1994): 877-898.