WATER SPILLING FLUID DYNAMIC ANALYSIS ON SECTOR GATE OPENING IN DAM SPILLWAY

N H Hassan[1*], M H Zawawi[1,*], M A Abas[2], M R M Radzi[3], A Hassani[3], W N Yusairah[1], F Nurhikmah[1], Nurhanani A Aziz[1], Aisyahira Melan[1]

[1]*Department of Civil Engineering, Universiti Tenaga Nasional, 43000 Kajang, Selangor, Malaysia*
[2]*School of Civil Engineering, Universiti Sains Malaysia, Engineering Campus, Penang, Malaysia*
[3]*Generation Division, Tenaga Nasional Berhad*
[*]For correspondence; Tel. + (60) 179509799, E-mail: nhh.husnahassan@gmail.com
[*]For correspondence; Tel. + (60) 164886106, E-mail: MHafiz@uniten.edu.my

ABSTRACT: *This study analyses the fluid dynamic on the sector gate in the dam spillway with a 16 ft gap opening. The parameters determined in this study are velocity, pressure, and streamline. A 3D Computational Fluid Dynamics (CFD) model of dam spillway structure and fluid boundary conditions were developed using Ansys FLUENT. The result of the CFD model shows that the highest velocity value which is 14.8 m/s and the largest hydraulic jump occurred in the middle of the dam spillway between the two energy dissipators. The hydraulic jump type that occurred which is oscillating jump is determined by the Froude value of 2.988. High velocity may contribute to the large hydraulic jump that then gives a more damaging effect to the stilling basin surface. Future mitigation measures have to take to ensure the sustainability of structural integrity.*

Keywords: Dam spillway, velocity, pressure, Computational Fluid Dynamics (CFD), hydraulic jump

1. INTRODUCTION

Numerical modeling techniques have been rapidly developing and have increased to the point where numerical solutions are now possible for many applications for many years. It led to the widespread use of numerical modeling as a design tool in many engineering disciplines such as hydraulic engineering [1]. Fundamental principles upon which all numerical models are comparable for any models regardless of the broad range of numerical modeling executions [2]. There are a few types of numerical methods such as Finite Element Method (FEM) or Finite Volume Method (FVM) is used to devise a set of algebraic equations that represent Partial Differential Equations (PDE) [3]. By means of some forms of either iterative or matrix solution, an approximate solution to those algebraic equations is obtained. It usually very computationally intensive which makes the use of modern computational power extremely essential to numerical models uses [4].

Computational Fluid Dynamics (CFD) is one of the algorithms and numerical modeling brunch that has been established for analyzing and solving problems that involve fluid flow [5]. Using CFD for modeling the hydraulic structures can reduce the cost and time of run experiments [6]. Thus, the interest on the part of the pertinency of CFD to model the flow of fluid for hydraulic engineers is considerable. Based on the fact that CFD is being utilized for modeling flow in all areas of a generating station, this study is focusing on the use of CFD to model the water flow through a spillway of a hydroelectric dam.

The hydraulic design of a spillway has been one of deliberated subjects in hydraulic engineering virtually [7]. The spillway is among the most major dam project structures [7]. It is a passage throughout in a dam structure in which the design flood could be spilled-off to downstream safely and dependable [8]. Spillway design in all types of dams is very essential because the incapability of spillway to remove probable maximum flood (PMF) discharge may give rise to overtopping or water overflow which eventually leads to the disfigurement of the dam or any other harms [9]. It is essential to design spillway facilities with adequate capacity to avoid overtopping of a dam [10]. Furthermore, determining flow pressure and velocity parameters along the apron of the spillway is very important [11-12].

Research on spillway generally done with making physical models [13-14]. But recently, hydraulic characteristics study on this structure has been done with advancing in the field of CFD [15]. It is particularly in the initial stages of design and analysis where the development of physical models would be excluded on the basis of cost and time [16]. Users are allowed to produce an instantaneous evaluation of the existing conditions with recent CFD techniques [16]. FLUENT in Ansys software packages is provided to use the CFD techniques [15] in this study. Numerical method has been used in this paper to analyze flow dynamics at the spillway of a hydroelectric dam.

2. MATERIALS AND METHODS

In this section, the materials and methods of the model and numerical modeling will be explained. The methodology involved pre-processing, numerical simulation, and post-processing.

Pre-Processing

For the geometry, a 3D model was developed based on an as-built drawing that was prepared for spillway design. In this study, the fluid dynamic is discussed on the sector gate with a 16ft gap opening. 16ft gap opening of the sector gate is one of the operating conditions of the spillway of a hydroelectric dam which is the highest gap gate opening. All processes of 3D model development were conducted in SolidWorks. Then, the file of 3D model sketching was uploaded in FLUENT software and defining structure in the software. Figure (1) shows a 3D sketch of the spillway of a hydroelectric dam and Figure (2) shows the uploading file of the sketch in FLUENT in Ansys software. In Figure (2), the considered boundary condition which is water and air boundary is shown. Fluid elevation and outflow were considered as a boundary condition at the entrance and end of the spillway,

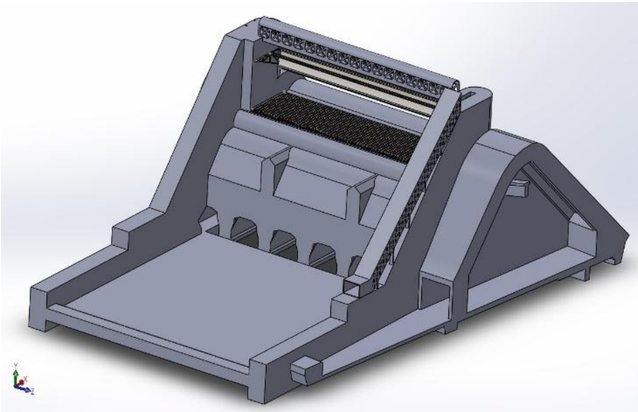


Fig (1) 3D sketch of the spillway

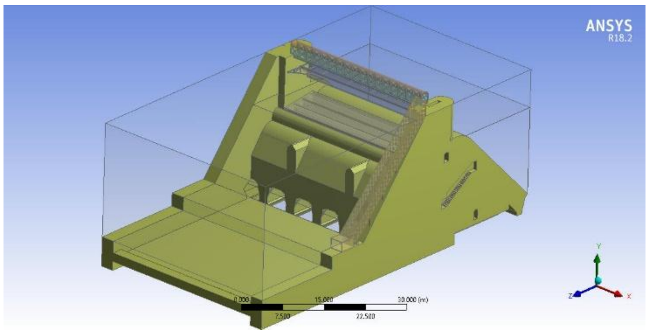


Fig (2) 3D model of the spillway in Ansys software

An assessment of the mesh sensitivity is conducted. It is analyzing the mesh size by a trial-and-error process where the mesh size is evaluated form the largest to the smallest as shown in Figure (3). Computation cost is increased with increase model accuracy by fining the mesh size [6]. Numerically, Ansys solved the Navier–Stokes equation by using FVM. A brief reference on the equations used in the software is shown below. Given Eq (1) as continuity equation at three-dimensional (3D) Cartesian coordinates.

$$v_f \frac{\partial \rho}{\partial t} + \frac{\partial}{\partial x}(uA_f) + \frac{\partial}{\partial x}(vA_f) + \frac{\partial}{\partial x}(wA_f) = \frac{PSOR}{\rho} \qquad (1)$$

where u, v, z are velocities component in the x, y, z direction; ρ fluid density; Ax, Ay, Az cross-sectional area of the flow; PSOR the source term and vf is the volume fraction of the fluid in 3D momentum equations given as shown in Eq (2).

$$\frac{\partial u}{\partial t} + \frac{1}{v_f}\left(uA_x\frac{\partial u}{\partial x} + vA_y\frac{\partial u}{\partial y} + wA_z\frac{\partial u}{\partial z}\right) = -\frac{1}{\rho}\frac{\partial P}{\partial x} + G_x + f_x$$

$$\frac{\partial v}{\partial t} + \frac{1}{v_f}\left(uA_x\frac{\partial u}{\partial x} + vA_y\frac{\partial u}{\partial y} + wA_z\frac{\partial u}{\partial z}\right) = -\frac{1}{\rho}\frac{\partial P}{\partial y} + G_y + f_y \qquad (2)$$

$$\frac{\partial w}{\partial t} + \frac{1}{v_f}\left(uA_x\frac{\partial u}{\partial x} + vA_y\frac{\partial u}{\partial y} + wA_z\frac{\partial u}{\partial z}\right) = -\frac{1}{\rho}\frac{\partial P}{\partial z} + G_z + f_z$$

where Gx, Gy, Gz the acceleration created by body fluids; P is the fluid pressure; vf is related to the volume of fluid and fx, fy, fz acceleration of viscosity in three dimensions defined in Eq. (3). For free surface profile modelling, Volume of Fluid (VOF) technique on regards volume fraction of the computational cells had been used. It takes value between 0 and 1 since volume fraction, F depicts fluid amount in each cell [6].

$$\frac{\partial F}{\partial t} + \frac{1}{v_f}\left[\frac{\partial}{\partial x}(FA_x u) + \frac{\partial}{\partial y}(F_y v) + \frac{\partial}{\partial z}(FA_z w)\right] = 0 \qquad (3)$$

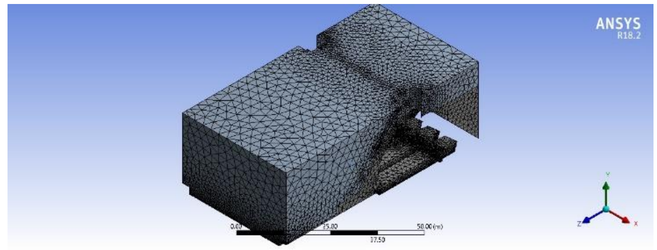


Fig (3) Fluid meshing

A spillway model requires the presence of a free surface that represents the air-water interface. VOF model is one of the approaches for certainly tracks interface between immiscible liquids [17]. Figure (2) shows the boundary conditions of water and air were set while meshing processing for the geometry of fluid which is water and air is shown in Figure (3). Reynolds-Averaged Navier-Stokes (RANS) equations solve the CFD applied to spillway modeling using Ansys. The alterations include algorithms to traverse free surface and model the flow past. Navier-Stokes equation resolved by powerful numerical methods which are FEM and FVM in the CFD field [6],[18]. In order to get a closed-form Navier-Stokes equations turbulence model, realizable *k-ε* (RKE) has been presented [19]. This approach requires statistical methods to take out an averaged equation that related to turbulence quantities. The turbulence model is requisite to directly capture every motion scale [19]. CFD user typically wants a steady-state solution rather than a detailed time-accurate one that captures every minuscule vortex [20]. As a result, there are turbulent which are unsteady motions affecting the flow that cannot be resolved directly thus it has to be modeled [21].

*k-ε is r*easonably accurate for a wide variety of flows [19]. Generally, *the k-ε* turbulence model is used to simulate mean flow characteristics for turbulent flow conditions in CFD [22-23]. It satisfies reliable mathematical limitations on the Reynolds stresses that consistent with turbulent flows physics. It also probably provides high performance for flows that involve boundary layers under strongly unfavorable pressure recirculation, gradients, separation, and rotation [22]. Based on an exact equation for the transport of the mean-square vorticity fluctuation, a new transport equation for the dissipation rate, ε, has been derived [19]. The first transported variable is the turbulence kinetic energy (k) while the second transported variable is the dissipation rate of turbulence energy (ε) [21]. RKE indicates a high capability to determine the mean flow of complex structures in every measure of comparison virtually [21].

Numerical Simulation

The simulation was initiated with a water elevation of 147 ft which is equal to spillway crest elevation with. At the end of the upstream model, a uniform inflow velocity was applied to get a target flow rate. All velocities value is set 1 m/s. Computational grid contains 207,926 cells. General boundary conditions used in the simulation are as summarized as shown in Figure (2). Velocity inlet boundary conditions are considered to assess the scalar and velocity properties of the flow for a mixture phase of the inlet.

Water velocity flow into the inlet is set at 1 m/s while pressure at the inlet and outlet equals the atmospheric pressure which is 0 Pa. The volume fraction under the multiphase section is set at 1 for the water phase of the inlet. In the simulation, the time step size used is set at 0.1 s and the end time is 5 s.

Post-Processing

Post-processed is to extract quantities of interest of velocity, pressure, and streamline. The visualizations of the simulation are shown in the figures below.

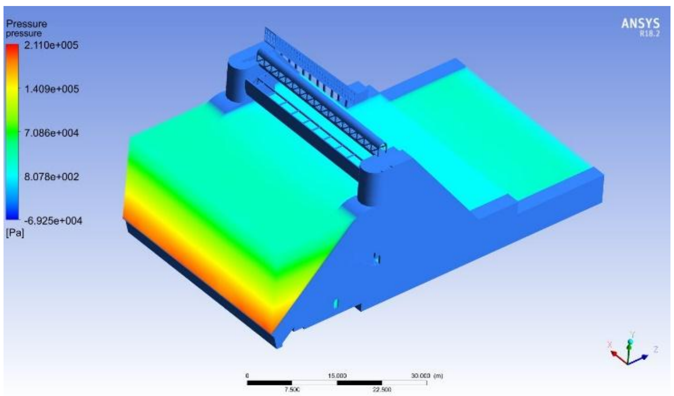


Fig (4) Pressure of water

The water pressure of the spillway is shown in Figure (4). The maximum value for the pressure is 0.08078 Pa. Upstream boundary conditions can be determined with one of two pressure boundary conditions which is static. Approach velocity may not be notable and therefore it is omitted in a few applications [11]. However, the approach velocity is significant at the higher flows. Thus, the maximum velocity value is taken in this study. Figure (5) shows the velocity of water at the spillway. The maximum value for the mean velocity is 0.1266 m/s.

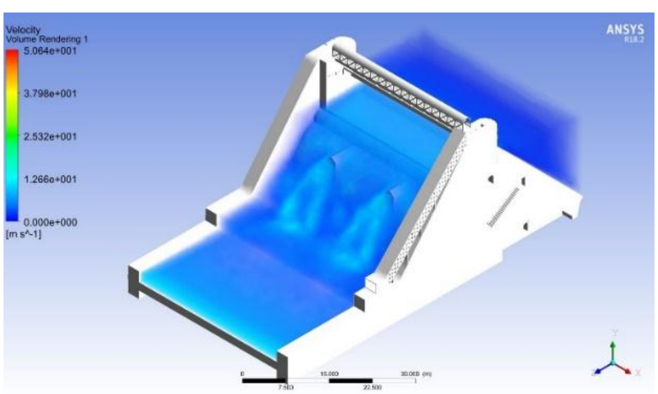


Fig (5) Velocity of water

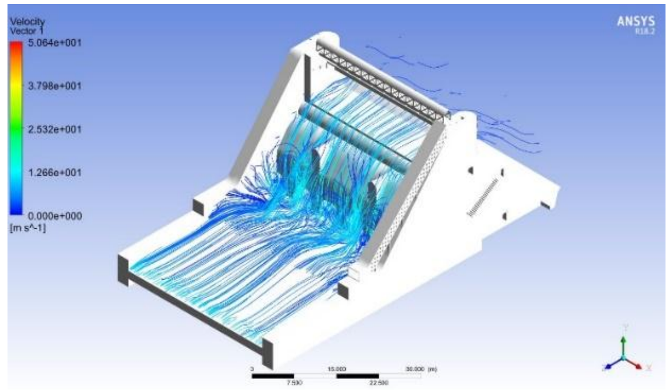


Fig (6) Flow pattern

Streamline in Figure (6) shows the flow pattern through the spillway from the upstream to the downstream. Despite this study is not including the experiment part, all of the methods and fundamentals are similar to the Sultan Abu Bakar Dam study that has been done by Abas (2018) [24].

3. RESULTS AND DISCUSSION

The principles of fluid dynamics can be used to understand an almost unimaginable variety of phenomena [25]. This study is focusing to model the flow of water through a spillway on the use of CFD. Water behavior in the spillway is highly affected by the flow pattern at the entrance of the spillway. Flow pattern formation at the entrance is affected by the operating condition effect on increasing the capability of the spillway for easy passing the PMF. Therefore, any nonuniformity in the flow of approach channel may reduce spillway capacity and reduction in the spillway discharge coefficient [11]. By optimizing the sector gate operating may lead to turbulence loss and flow disturbances at the spillway [11,26].

Based on Figure (4), the pressure is measured from upstream and it instantly increases to reach close to zero at downstream. The measurements of velocity in Figure (5) considered spurious and most of these vectors were located near the spillway crest and energy dissipater parts. The dam flow rate is dependent on the water level. A hydraulic jump occurs when water in the spillway is flowing supercritical and is slowed by a deepening of the channel and obstruction in the spillway as shown in Figure (6). The slowing causes the water to suddenly jump to the other specific energy state. The rise in the Froude number of the supercritical flow by the amount of energy dissipated in a jump is increasing. Prediction of flow profile due to downstream flow will allow the prediction of such occurrences. Estimation of the super- and subcritical flow will also allow specific control to be applied at the spillway side by using an energy dissipator [27]. Diversion in streamline and vortex flow create a cross wave and cause nonuniformity through the spillway at the downstream area.

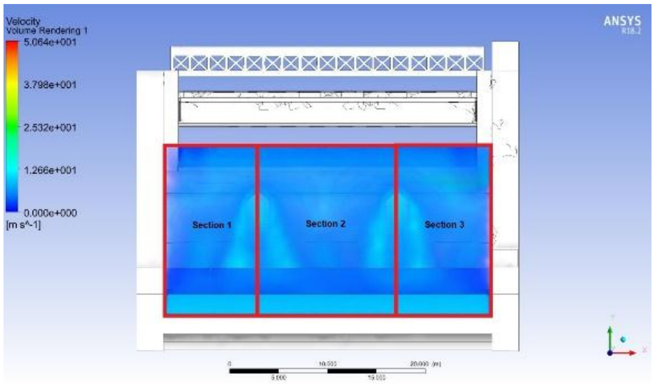


Fig (7) Sections divided by the energy dissipaters

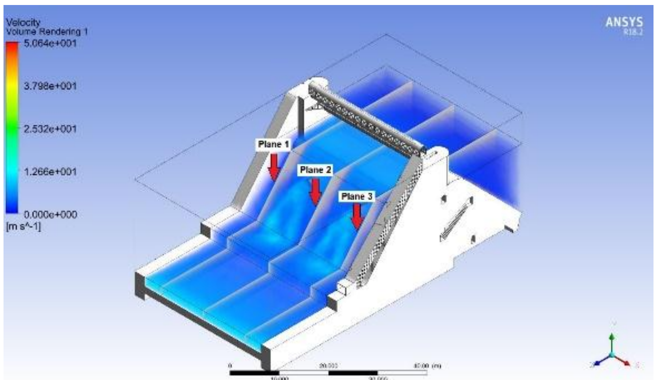
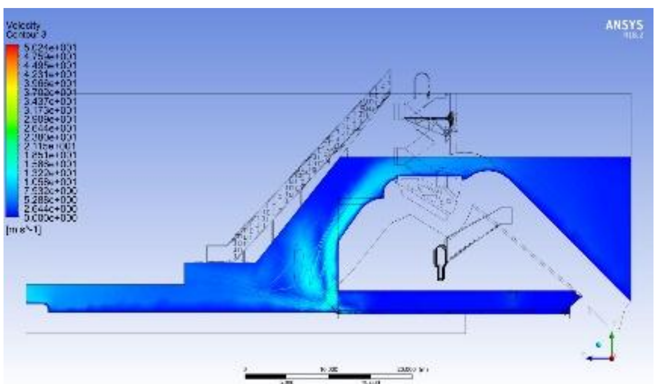
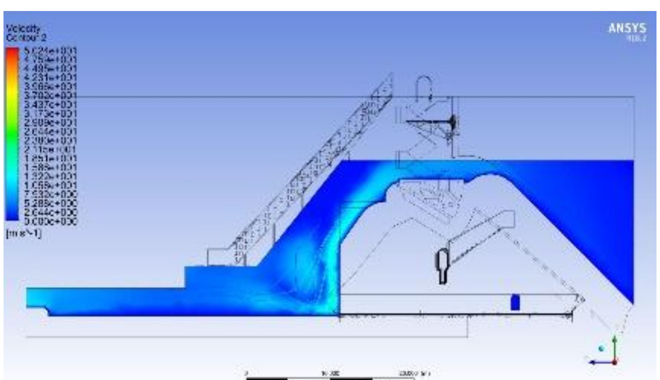


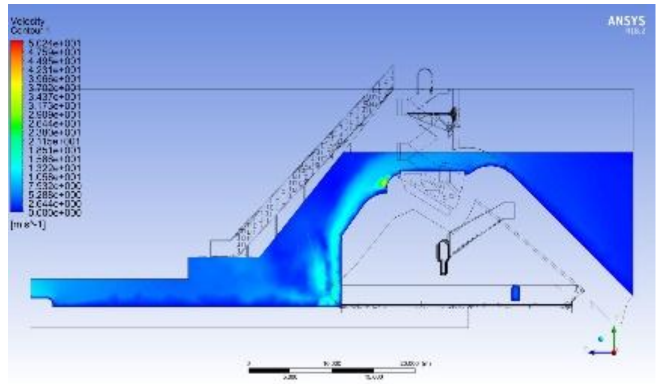
Fig (8) Planes for the cross-sections



(a) Plane 1



(b) Plane 2



(c) Plane 3

Fig (9) Cross-sections of the water flow velocity

Three sections were divided by the energy dissipaters as shown in Figure (7). Figure (8) shows that three planes were made to capture the cross-sections of the water velocity and vortex and Figure (9) shows each of the cross-sections of the water velocity. Types of the jump as shown in Table 1 is determined by the value of Froud before the jump, Fr_1.

Table (1) Type of The Jump

Froude Value	Jump Types
$Fr_1 = 1.0 – 1.7$	Undular Jump
$Fr_1 = 1.7 – 2.5$	Weak Jump
$Fr_1 = 2.5 – 4.5$	Oscillating Jump
$Fr_1 = 4.5 – 9.0$	Steady Jump
$Fr_1 > 9.0$	Strong Jump

Table (2) Results for hydraulic jump parameters

Cross Section	Plane 1	Plane 2	Plane 3
Froude Value, Fr_1	2.523	2.988	2.843
Energy Loss, E_L (m)	1.873	3.484	2.923
Height of Jump (m)	5.257	6.888	6.379
Length of Jump, L_j (m)	36.27	47.53	44.02
Jump Type	Oscillating Jump	Oscillating Jump	Oscillating Jump

Based on the numerical simulation, the Froude value then was calculated based on the hydraulic jump equation derived from the momentum equation. All of the three hydraulic jumps were determined based on the Froude value obtained and there are classified as oscillating jump which Froude value is between 2.5 to 4.5. Energy loss, height, and length of the jump were also obtained for all the planes.

Generally, physical models were used to determine flow conditions and characteristics for given dam design in the past meanwhile nowadays it is being used for simulation validation purposes. The validation purpose for this study uses the Sultan Abu Bakar Dam study by Abas (2018) [24]. The method and fundamentals being used for fluid dynamic analysis for this study compared to Sultan Abu Bakar fluid dynamic study are similar. Therefore, 90% precision of physical and numerical modeling results in the Sultan Abu

Bakar Dam study can be used to validate this dam numerical simulation result. It compares physical scaled-down experimental and numerical simulation. They presented both numerical simulation and experimental approaches to analyze fluid dynamics through the gap height of the radial gate [24]. It was shown that there is a reasonably good understanding between the numerical and physical models in the characteristic of flow [24]. Therefore, the simulation analysis in this study referred to the study for validation purposes.

4. CONCLUSIONS

From this study, it can be concluded that there are three different velocities in the receiving ends at the spillway stilling basin. The largest hydraulic jump occurred is at Plane 2 which dissipates 3.484 m energy compared to Plane 1 and Plane 3 which are 1.873 m and 2.923 m energy respectively. The high amount of energy dissipate contributes to high-stress distribution through the collision of water with the surface of the dam stilling basin. Therefore, it can be good information for dam operators to take future mitigation measures to ensure the sustainability of structural integrity.

5. REFERENCES

[1] P. G. Chanel, "An Evaluation of Computational Fluid Dynamics for Spillway Modeling," p. 84, 2008.
[2] H. H. Hu, *Computational Fluid Dynamics*. 2012.
[3] F. D. Background, "1 . Fundamentals of Fluid Dynamics," pp. 1–39.
[4] A. Itou, "Technical Paper High performance parallel computing for Computational Fluid Dynamics (CFD) Parallel Computer in this Study," *Fluid Dyn.*, vol. 51, no. 156, pp. 1–7, 2005.
[5] C. Rumsey, "Introduction: Computational Fluid Dynamics Validation for Synthetic Jets," *AIAA J.*, vol. 44, no. 2, pp. 193–193, 2006.
[6] A. Parsaie, A. H. Haghiabi, and A. Moradinejad, "CFD modeling of flow pattern in spillway's approach channel," *Sustain. Water Resour. Manag.*, vol. 1, no. 3, pp. 245–251, 2015.
[7] H. W. Coleman, C. Y. Wei, and J. E. Lindell, "Chapter 17. Hydraulic Design Of Spillways," *Hydraul. Des. Handb.*, pp. 1–54, 2004.
[8] W. A. Glaeser and R. W. Bruce, "33.1 Introduction," pp. 1–6, 1968.
[9] V. D. M. Oliviera and L. Thompson, "Spillway Design," pp. 1–8.
[10] G. Mohamad *et al.*, "Prediction of the Flow-Induced Vibration Response of the Chenderoh Dam Left Bank Section," vol. 01001, pp. 1–6, 2018.
[11] M. H. Zawawi, A. Saleha, A. Salwa, N. H. Hassan and Z. C. Muda, "Computational Fluid Dynamic Analysis at Dam Spillway due to Different Gate Openings," vol. 020245, 2018.
[12] D. K. H. Ho, et al., "Investigation of Spillway Behaviour under Increased Maximum Flood by Computational Fluid Dynamics Technique," *14th Australas. Fluid Mech. Conf.*, no. December, pp. 577–580, 2001.
[13] H. Chanson, "Physical modelling of hydraulics," *Hydraulic Open Channel Flow*, pp. 261–283, 1999.
[14] A. Moradinejad, A. Parssai, and M. Noriemamzade, "Numerical Modeling Of Flow Pattern In Kamal Saleh Dam Spillway Approach Channel," *Appl. Sci. Reports*, vol. 10, no. 2, 2015.
[15] V. Kiricci and A. O. Celik, "Modeling hydraulics structures with Computational Fluid Dynamics (CFD)," vol. 2014, no. October 2014, pp. 585–591, 2014.
[16] T. Ewing et al. "Efficient and cost-effective modelling and analysis of hydraulic structures using CFD," 2015.
[17] A. Serafeim, L. Avgeris, V. Hrissanthou, and K. Bellos, "Experimental and numerical simulation of the flow over a spillway," no. 2015, pp. 253–260, 2017.
[18] M. H. Zawawi et al., "Condition Assessment on Hydropower Dam based on Simulation Approach : A Review," vol. 020254, 2018.0.
[19] I. FLUENT, "Modeling Turbulent Flows," *ANSYS.Inc.*, pp. 6–2, 6–49, 2006.
[20] E. S. Researcher, "Application of CFD modelling in Water Resources Engineering Partners and," 2017.
[21] Kharagpur, "Lesson 8 Spillways and Energy Dissipators," p. 75.
[22] Gianluca Iaccarino, "Simulation of Turbulent Flows," *Stanford Lect. Notes, Course ME469B*, 2004.
[23] E. Stenmark, "On Multiphase Flow Models in ANSYS CFD Software," p. 59, 2013.
[24] F. C. Ng *et al.*, "Fluid/structure interaction study on the variation of radial gate's gap height in dam," *IOP Conf. Ser. Mater. Sci. Eng.*, vol. 370, no. 1, 2018.
[25] N. R. B. Olsen, *CFD modelling for hydraulic structures*, no. May. 2001.
[26] Y. Peltier et al., "Pressure and velocity on an ogee spillway crest operating at high head ratio: Experimental measurements and validation," *J. Hydro-Environment Res.*, vol. 19, no. May, pp. 128–136, 2018.
[27] B. W. H. Gray, M. Aeronautical, and L. Field, "File Copy," no. 3125, 1943.

*For correspondence; Tel. + (60) 179509799, E-mail:nhh.husnahassan@gmail.com

*For correspondence; Tel. + (60) 164886106, E-mail:MHafiz@uniten.edu.my