Smart Structures and Systems, *Vol. 35, No. 2 (2025) 89-99*
https://doi.org/10.12989/sss.2025.35.2.089

FE modeling of flow over stepped spillways: A case study of spillway of Siah-Bishe dam

Amirhosein Parvizi [1a], Vahidreza Amiresmaili [2b], Seyyed Behnam Beheshti [3c],
Seyed Mohammad Ali Razavizadeh [4b], Mojtaba Gorji Azandariani*[5] and Mohammad Reza Halvaeyfar [6d]

[1] *Department of Civil Engineering, Shahr-e-Qods Branch, Islamic Azad University, Tehran, Iran*
[2] *Department of Civil Engineering, University of Saravan, Saravan, Iran*
[3] *Department of Civil Engineering, Khomein Branch, Islamic Azad University, Khomein, Iran*
[4] *Department of Civil Engineering, National University of Skills (NUS), Tehran, Iran*
[5] *Structural Engineering Division, Faculty of Civil Engineering, Semnan University, Semnan, Iran*
[6] *Department of Civil Engineering, Dezful Branch, Islamic Azad University, Dezful, Iran*

(*Received October 20, 2020, Revised February 11, 2025, Accepted February 12, 2025*)

Abstract. Stepped spillway is one of the common types of spillways for water projects, and nowadays, it is considered due to its many benefits and they are used in many dam construction projects. The correct design and ensuring from proper these spillways require accurate analysis of the flow and has always been among the concerns of the dam's experts. Nowadays, using computers and numerical methods are a powerful tool for analyzing the flow and designing of water structures, and it is very useful in designing the primary spillways and optimizing them along with the construction of the hydraulic model. In this study, numerical modelling of flow over stepped spillways with different steps are presented, as well as optimization of the heights of steps. Also, a case study has been carried out about the Siah Bishe Dam spillway, in which the advantages and disadvantages of the numerical methods have been provided for optimizing the design. The computational fluid dynamics module of the finite element ANSYS FLUENT software was applied to assess the velocity vectors and pressure of the flow. According to the real flow is turbulent, the k–ε flow model was used. The predicted velocity vectors and pressure by numerical model was compared to the values attained hydraulic model. The use of the results of this study will be useful for experts and stepped spillways designers.

Keywords: CFD simulation; flow analysis; numerical method; optimization; stepped spillway; steps dimensions

1. Introduction

Spillways as an integral part of a dam regulate water flow and reservoir level to prevent overflow and damage to the dam. Spillways have different types, depending on the type of dam and site conditions (Chanel and Doering 2008, Chatila and Tabbara 2004, Imanian and Mohammadian 2019). Stepped spillways are one of the most common types of spillways used for embankment dams, rock-fill dams, Roller-compacted concrete dams, etc. These Spillways perform well in flood discharge and are also the most cost-effective solutions to this problem (Fig. 1). Studies about dam construction projects in the world as well as the results of the hydraulic model of various projects indicate that stepped spillways have a high kinetic energy dissipation. The energy dissipation in this type of spillway is about 70% more than the smooth and non-stepped spillway (Castro-Orgaz *et al.* 2008).

In addition to having considerable energy dissipation, it also reduces the size of the tailwater structure. The advantages of stepping on the spillway can be summarized as follows:

- Low operating costs compared to other types of spillways,
- Reduced construction and implementation time,
- Smooth operation and maintenance,
- Reducing downstream bed erosion,
- High energy dissipation rate.

Proper design and optimum performance of these spillways require a detailed understanding of flow behavior. In the meantime, using numerical and computational methods has been considered as a powerful tool along with the results of a hydraulic model. In this research, the study of the spillway flow using the finite element method (FEM) is attempted and the numerical methods for optimizing the geometry of the spillway are described (Chamoli 2015, Simonetti and Gentile 2019, Tabbara *et al.* 2005).

The flow over a stepped spillway can be divided into three groups: nappe, transition, and skimming flow. At low flow rates, stepping forms a series of cascades represented by a sequence of free-falling nappes. In the nappe flow regime, the height of the entire spillway is divided into

---

*Corresponding author, Research Associate Professor,
E-mail: MGorji@semnan.ac.ir; gorji1365@yahoo.com
[a] M.Sc.
[b] Instructor
[c] Ph.D. Candidate
[d] Assistant Professor

http://www.techno-press.org/?journal=sss&subpage=7
ISSN: 1738-1584 (Print), 1738-1991 (Online)

Fig. 1 Stepped spillway of Kosar dam (2006)

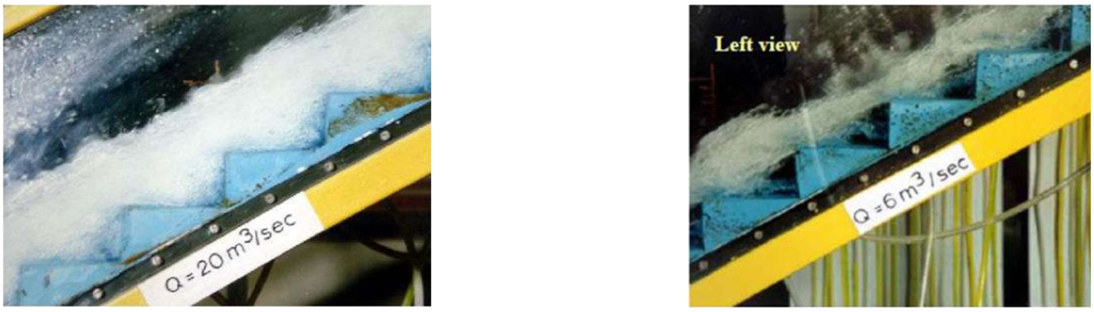
several vertical cascades, and the water flow in contact with successive steps can result in a complete or incomplete hydraulic jump (depending on the horizontal length of the steps). The nappe flow regime takes place at low flow rates and with increasing step height. The energy dissipation is due to the air/water mix flow contact with the steps and the formation of complete or incomplete hydraulic jumps. Water flow at the surface of each step is accompanied by three phenomena: free fall, air-cavity formation, and combination with water and the generation of rotational flow.

Passing from nappe flow to skimming flow is a gradual and continuous process. In the transition flow, the nappe flow appears on some steps and the skimming flow (exciting vortex) appears on others (Chinnarasri and Wongwises 2006). The hydraulic oscillations and water spraying near the free surface are some characteristics of the transition flow. In addition to throwing droplets of water at a height several times higher than the height of the step of the spillway, the transition flow has a turbulent shape (Jacobsen and Olsen 2010). In the skimmed flow, steps act as a large roughness against the flow in the. In this type of regime, a pseudo-bottom that connects the ends of the successive steps is formed. In this type of flow, a large proportion of energy dissipation is handled by rotational flows beneath the pseudobottom. Stable whirl vortices are also formed vertically under the pseudo-bottom on the surface of the steps. As discharge increases, the velocity of these vortices increases as well. In the skimming flow, two distinguishable areas are (Fig. 2):

- The upper area on the edges of steps, which acts as a pseudo-bottom,
- Triangular pseudo-bottom space and the whirling vortex inside it.

2. Literature review

Multiple studies examining the flow over stepped spillways have been performed employing experimental with scaled models and analytical and computational methods. Chamani and Rajaratnam (1999) conclude that due to lack of a precise solution to specify the energy dissiption in the skimming flow regime reason several problems for comparing the energy loss in the two types of skimming and nappe flow regimes. Chen *et al*. (2002) and Tabbara *et al*. (2005) applying the computational method and using k−ε model to examine the turbulence flow, can be analyzed the transmission flow of spillway. Among the effective parameters in the elasticity coefficient of step shapes, the Froude number turned out to be the dominant parameter. Chinnarasri and Wongwises (2006) proposed equations for inclined stepped spillways. The minimum critical depth for having skimming flow and the maximum critical depth for nappe flow in simple and inclined spillway are obtained from these equations. Varjavand (2008) investigated energy dissipation in the stepped spillway using numerical model of Fluent and concluded that with regard to the upstream stable energy, increasing the number of steps leads to a noticeable increase in energy dissipation. Carosi and Chanson (2008) provided the investigation of the air–water flow properties in skimming flows with a focus on the turbulent characteristics. Measurements showed some relatively good correlation between turbulence intensities and turbulent length and time scales. These measurements also illustrated large turbulence levels and large turbulent time and length scales in the intermediate region between the spray and bubbly flow regions. Carvalho and Martins (2009) investigated the hydraulic jumps on the steps of a stepped spillway analytically, physically, and numerically. They proposed preliminary design criteria for stepped spillways with hydraulic jump formation of simple shapes and adequate relations of critical depth/step height and the application of computational fluid dynamics to such problems is studied. Hanbay *et al*. (2009) presented two intelligent models to predict flow conditions and aeration performance in stepped cascades applying critical flow depth, step height, and



(a) Nappe flow (b) Skimming flow

Fig. 2 Types of flow in the stepper spillway of the hydraulic model of the Siah Bishe project (2006) channel slope. Bombardelli *et al*. (2011) provided and evaluated the results of a comprehensive investigation addressing the non-aerated area of the skimming flow in steep stepped spillways. They used a relatively large physical model of the spillway to acquire data on flow velocities and water levels. Numerical simulations using a commercial code were then performed to reproduce those experimental conditions. Dastgheib *et al*. (2012) developed a numerical modeling of the flow over different forms of stepped spillway. The computational fluid dynamics model which solves the RANS equations coupled to a surface-capturing algorithm to predict the main features of flow water. The model was developed using the volume of fluid (VOF) method. In this paper, the authors compared the flow velocity at the end of spillway and energy dissipation rate in different height of steps. In addition, they used both the RNG model of turbulence with wall functions to compute the eddy viscosity and structured hybrid grids to accommodate the geometry of the stepped spillway included upstream, steps, and downstream, respectively. Agreement between numerical results and experimental data showed that RNG turbulent model and VOF method for predicting the water surface in the stepped spillway were suitable. Felder and Chanson (2013) investigated the air–water on some flat and pooled stepped spillways with slopes of 8.9° and 26.6° in transition and skimming flows. This study comprised the observations of the flow patterns, characteristic air–water flow properties, and energy dissipation performances. The air–water flow properties showed some differences in terms of interfacial velocity, bubble count rate, and turbulence intensity between the stepped chutes for the two channel slopes. These differences were also reflected in the residual energy data, highlighting a better energy dissipation rate for the pooled stepped spillway with slope of 8.9°. However, the aerated flows on the pooled stepped spillways exhibited some hydrodynamic instability, and a safe operation must be tested in physical models. The flat stepped spillway appeared to be the preferable design in terms of energy dissipation and flow stability. Estrella *et al*. (2015) in their paper reported the results of the physical modeling of a generic stepped spillway without sidewalls. The resulting data demonstrate that the distribution of the different variables studied depend on the specific discharge at the entrance and the spillway height.

However, many studies have been done on these types of spillways, but recently a limited number of numerical studies have been done in this regard. Using numerical models to determine the geometry of construct proper to the discharge present in nature is helpful in reducing costs and saving time.

It is obvious that there has been furthed concern in the apply of flow over stepped spillways in hydraulic engineering. However, there was low progress in the numerical modelling of flow over stepped spillways. Encouraged to identify the requirement for extensive numerical modelling, the finite element method is applied to modelling of flow over stepped spillways. The aim of this investigate is assessment the numerical model in estimating and evaluation of rate of pressure and velocity in flow over stepped spillway and measurement of numerical and experimental results. Numerical modelling is according to ANSYS FLUENT software using k–ε turbulence model.

3. Case study

Siah Bishe Dam and Pumped Storage Power Plant project started in 1979 intending to supply electricity during periods of high energy demand as well as decreasing thermal power depreciation (Fig. 3). To use the existing potential of Alborz mountains, the project site was selected at the distance of 125 kilometers north of Tehran, at Mazandaran Province. This project has been constructed on the Chalus River, near the village of Siah Bishe and now is in its final stage.

Besides the generation of electricity, the project has also other goals, including the creation of a recreational and tourism environment, downstream water supply and creating jobs in the region during its construction and

Fig. 3 Different Components of the Siah Bishe Project (2006)

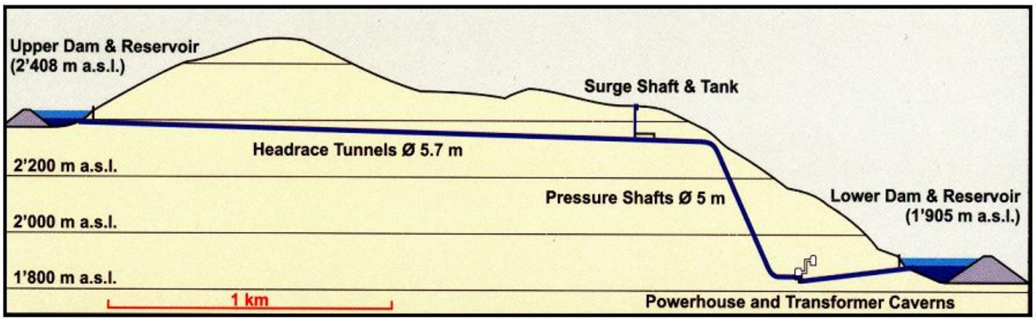


Fig. 4 A view of the Siah Bishe dam (2006)

operation. The project has four generator units with a capacity of 1060 megawatts. Its main components are the upper reservoir of the dam, the headrace tunnel, the pressure shafts of the reservoirs, the powerhouse and the lower dam with its reservoir. The dams in this project are concrete face rock-fill dams. Each of these dams has a stepped spillway that is responsible for flood drainage. The maximum discharge of the upper dam spillway is 203 cubic meters per second and the maximum discharge of the lower dam spillway is 830 cubic meters per second. These components are shown in Fig. 4. To achieve the optimal design of spillway, its hydraulic model with a reduced scale of 1:15 was built and studied at the Water Research Center. This paper describes how to use numerical methods for flow simulation in the upper dam spillway and how to get the optimal height of the steps.

4. Numerical analysis

4.1 Theoretical concept

The ANSYS FLUENT software has been used for numerical flow simulation. The capabilities of this software include two-dimensional, three-dimensional, rotational, stable, unstable, compressible, incompressible, non-viscous, calm, turbulent, Newtonian, non-Newtonian and multiphase flow analysis (Cao and Chen 2017, Phillips *et al.* 2002). Due to its many capabilities, this software is widely used in various industries. Along with the construction of hydraulic models and practical experiments, this software can be used to simulate flow at a very low cost. The flood drainage system of the upper dam is located on the left side of the dam structure. It includes the approach channel, stepped spillway and stilling basin. ANSYS Fluent is a computational fluid dynamics (CFD) software widely used for simulating fluid flow, turbulence, and heat transfer in complex geometries. It employs the finite volume method (FVM) to discretize and solve the governing equations of fluid motion. In this study, the numerical analysis of flow over the stepped spillway was conducted using ANSYS Fluent, with the Volume of Fluid (VOF) method applied to capture the air-water interface. The simulations were carried out using a pressure-based solver, suitable for incompressible flows, and the SIMPLE algorithm was used for pressure-velocity coupling.

The governing equations used in this analysis include the continuity equation (mass conservation) and the Navier-Stokes equations (momentum conservation), which are expressed as follows

$$\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho V) = 0 \tag{1}$$

$$\frac{\partial (\rho V)}{\partial t} + \nabla \cdot (\rho V V) = -\nabla P + \nabla \cdot (\mu \nabla V) + F \tag{2}$$

where ρ is the fluid density and V is the vector velocity, P is the pressure, μ is the dynamic viscosity, and F represents external forces such as gravity.

To model turbulence, the standard k–ϵ turbulence model was used, which is based on transport equations for turbulent kinetic energy (kkk) and its dissipation rate (ϵ\epsilonϵ). The Turbulent Kinetic Energy (k) and Dissipation Rate (ϵ) equations are given by

$$\frac{\partial (\rho k)}{\partial t} + \nabla \cdot (\rho k V) = -\nabla \cdot \left[\left(\frac{\mu}{\sigma_k} \right) \nabla k \right] + Gk - \rho \epsilon \tag{3}$$

$$\frac{\partial (\rho \epsilon)}{\partial t} + \nabla \cdot (\rho \epsilon V) = \nabla \cdot \left[\left(\frac{\mu}{\sigma_\epsilon} \right) \nabla \epsilon \right] + C_1 \frac{\epsilon}{k} G_k - C_2 \rho \frac{\epsilon^2}{k} \tag{4}$$

where G_k represents the generation of turbulent kinetic energy due to velocity gradients, and C_1, C_2, σ_k, and σ_ϵ are empirical constants.

4.2 Numerical modeling details

In this study, the ANSYS Fluent software was used for Computational Fluid Dynamics (CFD) simulations. The numerical model was developed to simulate the turbulent multiphase flow over a stepped spillway, considering the complex interactions between water and air phases. The computational domain was constructed based on the geometric parameters of the Siah-Bishe spillway, and meshing was performed using the GAMBIT software. An unstructured mesh with local refinements near the steps was employed to improve accuracy in capturing flow separation, turbulence, and energy dissipation. The boundary conditions were carefully defined: the inlet was set as a mass flow boundary condition corresponding to different discharge rates (ranging from 25 to 203 m^3/s), the outlet was modeled as a pressure boundary with atmospheric conditions, and the spillway surfaces were treated as no-slip walls to account for wall shear effects.

The turbulence model selection was critical for accurately predicting flow behavior over the stepped spillway. Given the highly turbulent nature of the flow, the standard $k{-}\epsilon$ model was used due to its robustness in handling complex free-surface flows. The Volume of Fluid (VOF) method was implemented to track the air-water interface, allowing for an accurate representation of skimming, nappe, and transition flow regimes. The simulations were conducted using a pressure-based solver with the SIMPLE algorithm for pressure-velocity coupling. Momentum and turbulence equations were discretized using a second-order upwind scheme to enhance numerical stability and accuracy. The solution was considered converged when the residuals for continuity, momentum, and turbulence equations dropped below 10^{-4}, and mass balance was verified across the computational domain. These refinements ensure that the numerical model provides a reliable prediction of velocity distribution, pressure variations, and energy dissipation over the stepped spillway.

Based on the plans hydraulic design this spillway must be capable to discharge the Probable Maximum Flood (PMF) with a flow rate of 203 cubic meters per second. The steps part of the spillway started from the level of 2366 and continues to the level of 2314. The horizontal length of this part is also about 130 meters (Fig. 5). For the preliminary design of the steps and computational flow analysis on the steps, several options were used (Table 1). Also, the necessary discharges for flow analysis on the spillway are

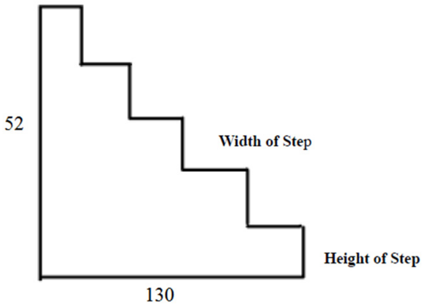


Fig. 5 Spillway sizes and schematic shape of Siah Bishe dam

Table 1 Different options for the initial plan of the Siah Bishe spillway

Option	Height of step (m)	Relative width of step (m)	Relative number of steps
1	0.6	1.5	87
2	0.8	2	65
3	1	2.5	52
4	1.2	3	43
5	1.4	3.5	37

25, 40, 60, 92, 150, 180 and 203 m^3/s.

The flow analysis settings and border conditions are as follows:

- Two-dimensional analysis and selection of the standard K-ε model (Carvalho and Martins 2009, Kim *et al*. 2014, Park and Cho 2012).
- Steady solution and multiphase flow-air and water as two existing phases (Carosi and Chanson 2008, Jun *et al*. 2015).
- The selected mass flow inlet boundary condition in which the input discharge is different and the input discharge of air has been also considered to be zero (open airflow) (Dastgheib *et al*. 2012, Esmaeili *et al*. 2018).
- The outlet boundary condition has been equal to the atmospheric pressure (Iqbal *et al*. 2020, Jun *et al*. 2015, Nair and Bhattacharyya 2018).
- The boundary condition of the upper part of the spillway is also considered to be equal to theatmospheric pressure (Fig. 7) (Jun *et al*. 2015, Nair and Bhattacharyya 2018, Ozdogan *et al*. 2017).
- The steps of the spillway are considered with the wall boundary condition and non-slip.

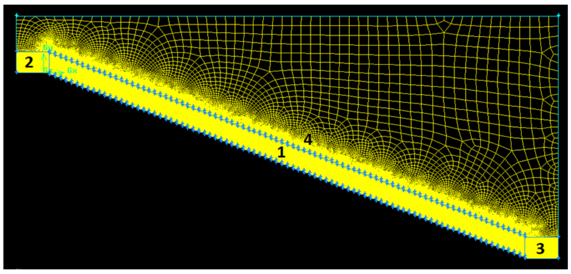


Fig. 6 A plot mesh of the spillway

In this study, the pressure distribution along the stepped spillway was analyzed using both numerical and analytical approaches. The governing equation for pressure distribution is derived from the fundamental principles of fluid dynamics and hydrostatic pressure theory. The total pressure at any point on the spillway can be determined using the Bernoulli equation along a streamline

$$P+\frac{1}{2}\rho V^2+\rho g h = constan \tag{5}$$

where g is the gravitational acceleration (9.81 m/s²), h is the height above a reference datum (m).

For the stepped spillway, the pressure at a given step can be estimated using the hydrostatic pressure equation

$$P=\rho g h_{step} \tag{6}$$

where h_{step} represents the local water depth at each step. The total pressure accounts for both dynamic and hydrostatic contributions, which were validated using numerical simulations in ANSYS Fluent. The numerical results were compared to analytical values derived using these equations, confirming the accuracy of the computational model.

Meshing is performed using the finite element software, Gambit. Mesh sizes are very small in the regions where water flows and at the steps of the spillway and they are

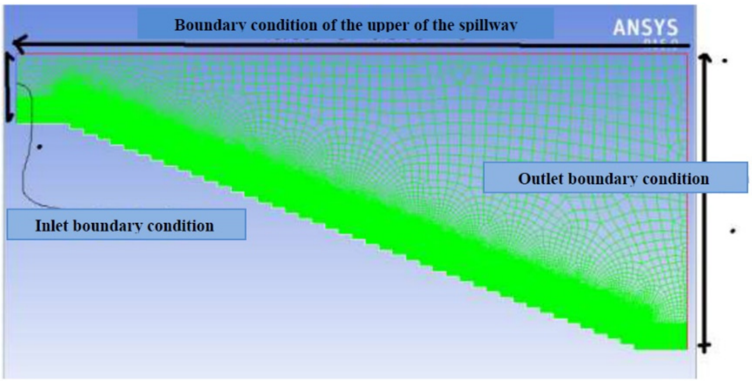


Fig. 7 Boundary conditions in different areas of the model

larger in upper regions where there is free air (Fig. 6).

The selection of flow rates for this study was based on hydraulic design principles and flood frequency analysis, ensuring that the investigated discharge values represent realistic operating conditions for the Siah-Bishe stepped spillway. The flow rates were determined considering the following factors:

(i) The maximum flow rate (203 m³/s) corresponds to the Probable Maximum Flood (PMF) for the spillway, which represents the extreme flood event that the structure is designed to withstand. Intermediate flow rates (25, 40, 60, 92, 150, and 180 m³/s) were chosen to analyze spillway performance under various operating conditions.

(ii) Historical hydrological data of the Chalus River and statistical flood return period analysis were used to estimate expected flood discharges. The selected flow rates correspond to different return periods, ensuring the model captures both frequent and extreme flow conditions.

(iii) The physical hydraulic model used for experimental validation was built at a reduced scale (1:15), and the flow rates were scaled accordingly to maintain Froude number similarity. This ensures that the results obtained from the model accurately represent real-world spillway behavior.

5. Results and discussion

After modeling and analyzing the flow, some of the results are mentioned here. The phase plot means that the simulation has two phases (air and water phases). The water phase is shown by blue and the air phase with red and the phases in-between which are the contact surface, bubble and eddy areas are shown by other colors (yellow and green). These contours do not have a unit; the number 1, which is in the highest state (red), represents the pure phase of air and the zero number (blue) represents the pure liquid phase (Fig. 8). The phase diagram, namely two-phase (VOF) is a simulation (weather phase). The water phase is shown by the blue and the air phase with the red and the phases between these two colors, which actually are the contact surface and the bubble and eddy areas are shown by the colors between these (yellow and green). These contours do not have a unit; in fact, the number 1, which is in the highest state (red), represents the pure phase of the air and the zero number (blue) represents the pure liquid phase.

The velocity vectors of flow over stepped spillway are exhibited in Fig. 9. In Fig. 9 is clearly exhibited, the energy dissipation in the flow over stepped spillways is rotational flow that happen after each step. This is rotating clockwise and organized near the stairs. The amount of velocity vector in superficial flow is large than rotational flow on the steps. According to the interaction between surface and rotational flow above it, energy dissipation is great.

The velocity vector is shown in Fig. 10, that to some extent it shows the formation of the air vortex in the water. Such vectors will separate the phases, while simultaneously showing the flow movements (for example, vortex formation). In this figure, the existence of vortex in the steps is quite visible and the flow is completely skimming. According to Fig. 10, it can be derived that numerical model with an error of about 7.5% was predicted. Regarding the comparison of velocity in the hydraulic and numerical models (Fig. 10), it can be said that there is a good correlation between the results and the velocities predicted by the numerical model are correct. The channel side walls, forces caused by surface tension and viscosity at low discharges and the accuracy of hydraulic model are the factors that cause the difference between the numerical and hydraulic model.

Considering the pressure, due to the constant oscillations of the pressure on the surface of the steps, the existence of skimming flow and air vortices in the steps, as

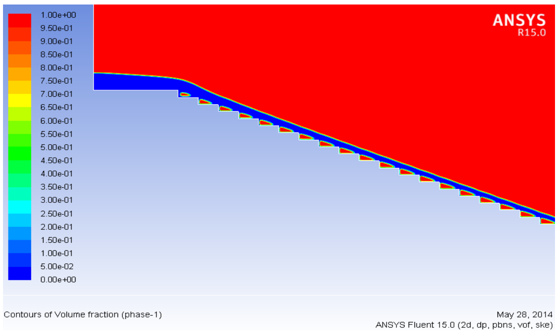


Fig. 8 An example of the flow phase graph in a 52-step spillway with a flow rate of 203 cubic meters per second

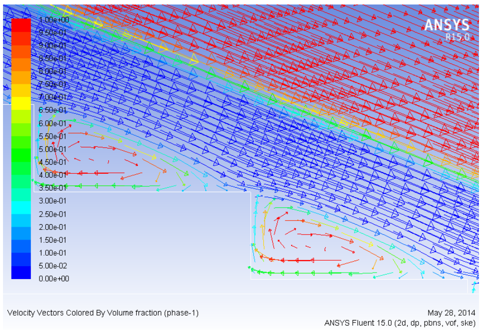


Fig. 9 An example of velocity vectors in an 87-step spillway with a flow rate of 203 cubic meters per second

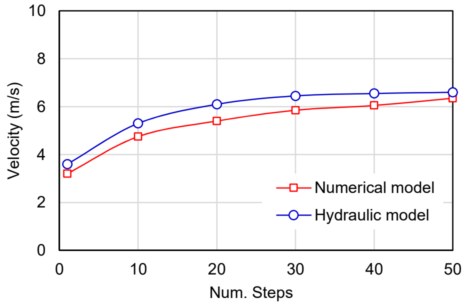
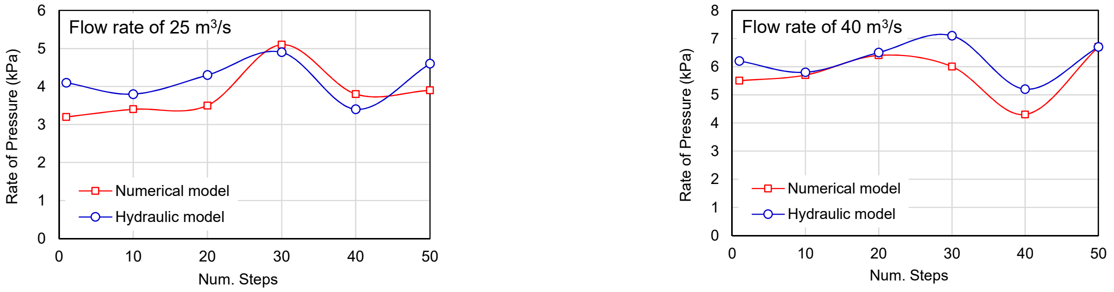


Fig. 10 Comparison of velocity in the hydraulic and numerical models

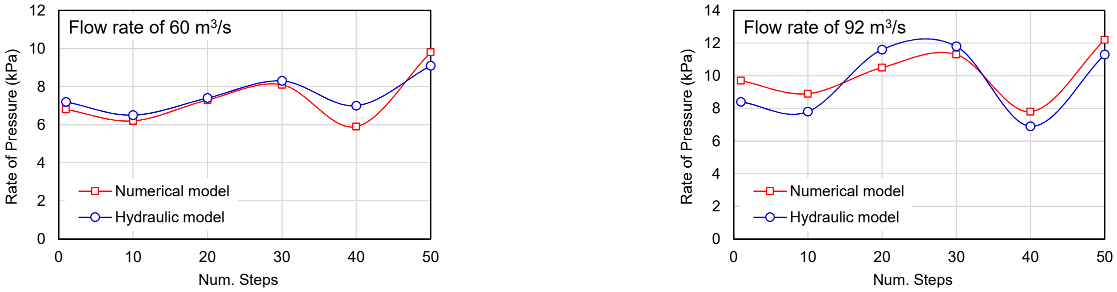
Table 2 Comparison of pressure results in hydraulic and numerical models

Step num.	Rate of Pressure (kPa)													
	25 m³/s		40 m³/s		60 m³/s		92 m³/s		150 m³/s		180 m³/s		203 m³/s	
	Num.[a]	Hyd.[b]	Num.	Hyd.	Num.	Hyd.	Num.	Hyd.	Num.	Hyd.	Num.	Hyd.	Num.	Hyd.
1	3.2	4.1	5.5	6.2	6.8	7.2	9.7	8.4	9.9	14.9	13.9	18.0	13.8	17.5
10	3.4	3.8	5.7	5.8	6.2	6.5	8.9	7.8	8.1	13.3	14.4	17.8	14.8	11.8
20	3.5	4.3	6.4	6.5	7.3	7.4	10.5	11.6	16.6	18.0	14.4	15.0	14.7	12.7
30	5.1	4.9	6	7.1	8.1	8.3	11.3	11.8	16.9	20.2	19.4	23.3	13.7	18.0
40	3.8	3.4	4.3	5.2	5.9	7	7.8	6.9	11.1	10.0	6.1	4.0	3.8	4.0
50	3.9	4.6	6.7	6.7	9.8	9.1	12.2	11.3	16.7	22.1	26.5	22.9	24.3	23.6

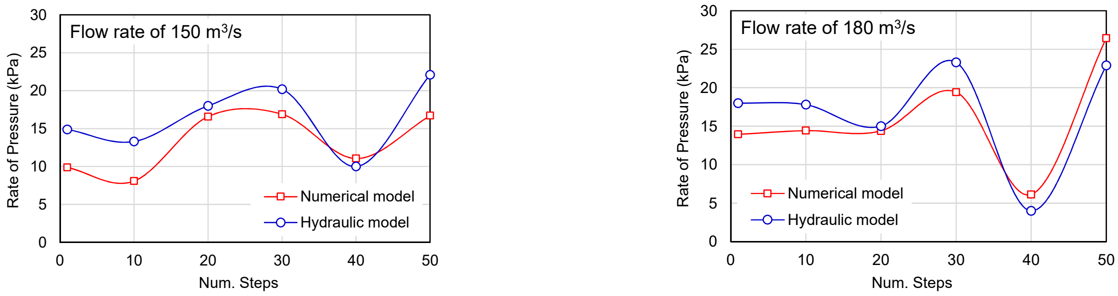
[a] Numerical model; [b] Hydraulic model



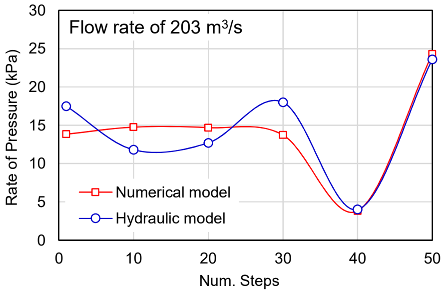
(a) Flow rate of 25 m³/s (b) Flow rate of 40 m³/s



(c) Flow rate of 60 m³/s (d) Flow rate of 92 m³/s



(e) Flow rate of 150 m³/s (f) Flow rate of 180 m³/s



(g) Flow rate of 203 m³/s

Fig. 11 Comparison of rate of pressure in the hydraulic and numerical models

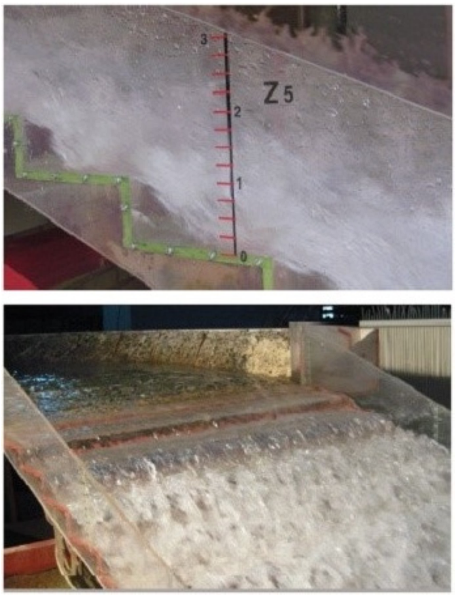


Fig. 12 Results of the flow type in the hydraulic model at a flow rate of 25 cubic meters per second (2006)

well as the multiphase flow, there is no systematic and predictable trend of the pressure oscillations. The results of both the hydraulic and the numerical models have great dispersions and it is not predictable. In Table 2, some of the recorded averaged static pressure results on the step surface are presented. Regarding great dispersions in the hydraulic and numerical models (Fig. 11), it can be said that there is a good correlation between the results and the rate of pressure are well predicted by the numerical model.

5.1 Discharge with 25 cubic meters per second

The results of the hydraulic model indicate that the first two steps of the spillway are overflowed and airless (Fig. 12). In the following, the flow of the surfaces of the spillway steps take form as overflowing, and in some steps, there are small rotational flows and air vortexes. The flow cannot get out from the overflowing mode, because these rotational and eddy flows are weak. The results of the numerical model in this mode have a relatively good agreement with the results of the hydraulic model.

5.2 Discharge with 40 cubic meters per second

Hydraulic model results indicate that in this case, the first two steps of the spillway are deep in water without air. After the third step, the air enters the flow and forms the air vortexes. These vortices are gradually enlarged and this process continues to the sixth step (Fig. 13). From the sixth step, the flow process is uniform and the size of the vortices is constant. In this case, the flow is also overflowing and the results of the numerical model are slightly different with the hydraulic model. In the numerical model, the air vortices grow from the first step gradually. This process continues until the middle of the spillway, and then the vortices remain almost constant. In this case, the power of the air vortices is higher than the flow rate of 25 cubic meters per second, but the flow is still as overflowing mode.

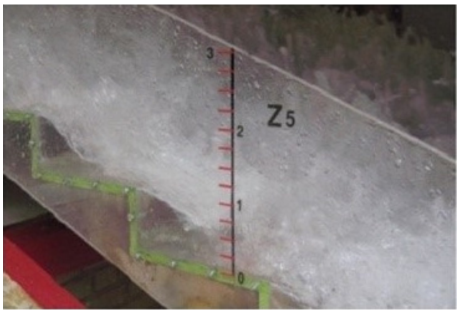


Fig. 13 Results of the flow type in the hydraulic model at a flow rate of 40 cubic meters per second (2006)

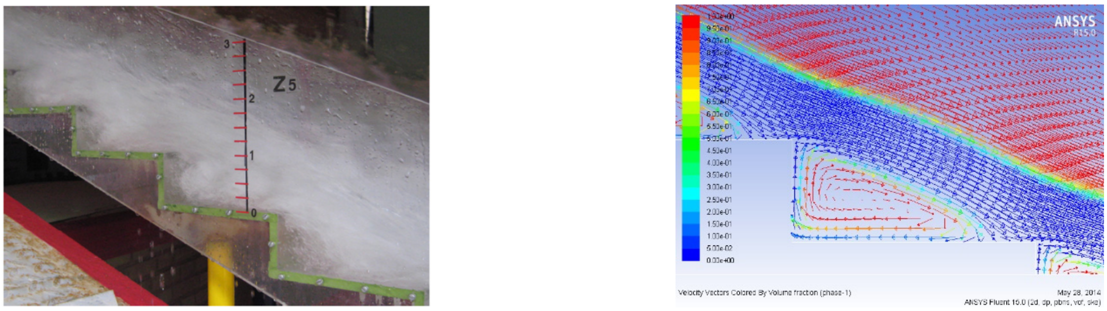
Fig. 14 The results of the flow type in the hydraulic model with a discharge of 92 cubic meters per second (2006)

5.3 Discharge with 92 cubic meters per second

The results of the hydraulic model indicate that in the first six steps, the flow occurs without air entering and creating the vortex (Fig. 14). After the sixth step, a skimming flow is formed and the rotation of the flow in the vortices is formed in the clockwise direction. A flow rate of 92 cubic meters equals flood discharge with a return period of 10,000 years. In this case, the results of the numerical model differ with the hydraulic model results, and in the numerical model, the vortex flow is formed from the second step. In this case, as in the hydraulic model, the flow is in the form of a skimming flow and the velocity vectors show the flow movement in the clockwise direction.

5.4 Discharge with 92 cubic meters per second

The results of the model in this case are also two-phase mode completely and the mixture of water and air and



(a) Hydraulic model (2006) (b) Numerical model

Fig. 15 Comparison of the flow type at a flow rate of 203 cubic meters per second

Table 3 Comparison of velocity values at different flow rates in hydraulic and numerical models

Step num.	Velocity (m/s)													
	25 m^3/s		40 m^3/s		60 m^3/s		92 m^3/s		150 m^3/s		180 m^3/s		203 m^3/s	
	Num.[a]	Hyd.[b]	Num.	Hyd.	Num.	Hyd.	Num.	Hyd.	Num.	Hyd.	Num.	Hyd.	Num.	Hyd.
1	**1.2**	**1.5**	**1.8**	**2.2**	**2.6**	**3.0**	**3.8**	**4.2**	**4.5**	**5.1**	**5.3**	**6.0**	**5.8**	**6.5**
10	**1.8**	**2.1**	**2.6**	**3.0**	**3.8**	**4.0**	**4.9**	**5.4**	**6.1**	**6.8**	**7.2**	**8.0**	**7.8**	**8.5**
20	**2.3**	**2.7**	**3.2**	**3.8**	**4.5**	**5.0**	**6.1**	**6.9**	**7.8**	**8.5**	**8.9**	**9.8**	**9.5**	**10.2**
30	**2.7**	**3.1**	**3.8**	**4.3**	**5.2**	**5.9**	**7.3**	**8.0**	**9.2**	**10.1**	**10.4**	**11.2**	**11.1**	**11.9**
40	3.0	3.5	4.1	4.7	5.8	6.5	8.0	8.9	10.5	11.2	11.8	12.6	12.3	13.0
50	3.4	3.9	4.8	5.4	6.4	7.1	8.8	9.6	11.3	12.0	12.6	13.3	13.1	14.0

[a] Numerical model; [b] Hydraulic model

scattering water from the fifteenth step is completely visible (Fig. 15). In the numerical model, this state is also quite evident and the two-phase flow density increases in this case. Also, the results of the hydraulic model show that the flow rate was about 40 m^3/s as the overflowing state and it is a transitional flow between 40-90 m^3/s and at discharges greater than 90 m^3/s, is completely as skimming flow. This state is also visible in the result of the numerical model, and in general there is a fairly good fit between the two models.

The velocity distribution over the stepped spillway plays a crucial role in understanding flow behavior, energy dissipation, and pressure variations. In this study, velocity values were extracted from both numerical simulations and hydraulic model experiments to evaluate flow characteristics under different discharge conditions. The velocity at different discharges was analyzed, focusing on surface flow velocity and rotational flow along the steps. The results indicate that as the discharge increases, the velocity also increases, leading to more pronounced skimming flow. Table 3 presents the velocity values obtained from both numerical and experimental analyses at selected locations along the spillway. The velocity results confirm that as discharge increases, the flow transitions from nappe to skimming flow, with higher velocities leading to increased turbulence and energy dissipation. This analysis aligns with the study's objectives, providing a comprehensive understanding of velocity distribution over the spillway steps.

A direct comparison between the hydraulic (experimental) and numerical models is crucial to validate the accuracy of the numerical simulations. To provide a quantitative assessment, numerical values for velocity and pressure at different steps and discharge rates have been tabulated along with their corresponding experimental values. The percentage error (E%E \%E%) is calculated using the following formula

$$\text{Error}(\%) = \left(\frac{|V_{Num.} - V_{Hyd.}|}{V_{Hyd.}}\right) \times 100 \qquad (6)$$

where $V_{Num.}$ represents the numerical value and $V_{Hyd.}$ represents the experimental value.

The results indicate that the numerical model provides a reasonable prediction of velocity and pressure values, with percentage errors generally within an acceptable range (typically below 12% for velocity and 17% for pressure). The largest deviations occur at higher flow rates due to the complexities associated with turbulence and air entrainment in the skimming flow regime. However, overall, the comparison confirms that the numerical model effectively captures the flow characteristics over the stepped spillway.

6. Conclusions

The main conclusions of this study can be summarized as follows:

- The use of a hydraulic model to control the numerical results are necessary and eliminates the deficiency of the numerical models.

- In the stepped spillway of this research, the results of the flow velocity are approximately the same for both the numerical and hydraulic models and there is an appropriate coordination between them.
- Due to the nature of the stepped spillways and the two-phase flow in them and mixing the water and air with each other, the results of the average pressure are not accurately predictable and don't have a regular process in both hydraulic and numerical models.
- Due to the complexity of the nature of the issue about the scattering water around, which is clearly visible in the hydraulic model, this issue is not visible in the numerical model and only the increase of the thickness can be seen in the two-phase flow, therefore, it can be considered as one of the weaknesses of the numerical method.
- On the issue of the flow type, which is as a skimming flow, there is a relatively good correlation between the results of hydraulic and numerical models.
- Both models indicate the formation of air vortices and the rotation of the flow, but there are differences about the location of the formation of the first vortexes between the two models. As the discharge in the hydraulic model increases, the vortex formation begins on the lower steps, but in the numerical model, it is observed that the vortices form after the second step in almost all the discharges above 60 m^3/s.

According to the above issues that have been discussed in this research, and in order to continue the study in this field, it can be stated the following suggestions:

(i) In the next step of this study due to the large number of numerical models, the effects of the asymmetric steps can be evaluated with different ratios of height to width. It is also possible to examine the effects of the slope of the step on the flow.

(ii) The effects of obstacles such as concrete blocks in the width of the steps can be examined in future studies.

(iii) It is also possible to examine the effects of water depth in the stilling basin in the flow on the steps.

(iv) It is hoped that this research can be a useful guide for developing and applying the numerical methods for modeling flow in stepped spillways among spillway designers in dam construction projects.

References

Bombardelli, F.A., Meireles, I. and Matos, J. (2011), “Laboratory measurements and multi-block numerical simulations of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways”, *Environ. Fluid Mech.*, **11**(3), 263-288. https://doi.org/10.1007/s10652-010-9188-6

Cao, Y. and Chen, H.-C. (2017), “CFD simulation of vortex-induced vibration of free-standing hybrid riser”, *Ocean Syst. Eng., Int. J.*, **7**(3), 195-223.

Carosi, G. and Chanson, H. (2008), “Turbulence characteristics in skimming flows on stepped spillways”, *Can. J. Civil Eng.*, **35**(9), 865-880. https://doi.org/10.1139/L08-030

Carvalho, R.F. and Martins, R. (2009), “Stepped spillway with hydraulic jumps: application of a numerical model to a scale model of a conceptual prototype”, *J. Hydraul. Eng.*, **135**(7), 615-619. https://doi.org/10.1061/(ASCE)HY.1943-7900.0000042

Castro-Orgaz, O., Giráldez, J.V. and Ayuso, J.L. (2008), “Critical flow over spillway profiles”, In: *Proceedings of the Institution of Civil Engineers-Water Management*, **161**(2), 89-95. https://doi.org/10.1680/wama.2008.161.2.89

Chamani, M.R. and Rajaratnam, N. (1999), “Characteristics of skimming flow over stepped spillways”, *J. Hydraul. Eng.*, **125**(4), 361-367. https://doi.org/10.1061/(asce)0733-9429(1999)125:4(361)

Chamoli, S. (2015), “A Taguchi approach for optimization of flow and geometrical parameters in a rectangular channel roughened with V down perforated baffles”, *Case Stud. Therm. Eng.*, **5**, 59-69. https://doi.org/10.1016/j.csite.2015.01.001

Chanel, P.G. and Doering, J.C. (2008), “Assessment of spillway modeling using computational fluid dynamics”, *Can. J. Civil Eng.*, **35**(12), 1481-1485. https://doi.org/10.1139/L08-094

Chatila, J. and Tabbara, M. (2004), “Computational modeling of flow over an ogee spillway”, *Comput. Struct.*, **82**(22), 1805-1812. https://doi.org/10.1016/j.compstruc.2004.04.007

Chen, Q., Dai, G. and Liu, H. (2002), “Volume of fluid model for turbulence numerical simulation of stepped spillway overflow”, *J. Hydraul. Eng.*, **128**(7), 683-688. https://doi.org/10.1061/(ASCE)0733-9429(2002)128:7(683)

Chinnarasri, C. and Wongwises, S. (2006), “Flow Patterns and Energy Dissipation over Various Stepped Chutes.” J. Irrig. Drain. Eng., 132(1), 70–76.

Dastgheib, A., Niksokhan, M.H. and Nowroozpour, A.R. (2012), “Comparing of flow pattern and energy dissipation over different forms of stepped spillway”, In: *World Environmental and Water Resources Congress 2012*: *Crossing Boundaries*, Reston, VA, USA, pp. 1246-1259. https://doi.org/10.1061/9780784412312.126

Esmaeili, H.A., Khaki, M. and Abbasi, M. (2018), “Dynamic stability of nanocomposite Mindlin pipes conveying pulsating fluid flow subjected to magnetic field”, *Struct. Eng. Mech., Int. J.*, **67**(1), 21-31. https://doi.org/10.12989/sem.2018.67.1.021

Estrella, S., Sánchez-Juny, M., Bladé, E. and Dolz, J. (2015), “Physical modeling of a stepped spillway without sidewalls”, *Can. J. Civil Eng.*, **42**(5), 311-318. https://doi.org/10.1139/cjce-2014-0427

Felder, S. and Chanson, H. (2013), “Aeration, flow instabilities, and residual energy on pooled stepped spillways of embankment dams”, *J. Irrig. Drain. Eng.*, **139**(10), 880-887. https://doi.org/10.1061/(ASCE)IR.1943-4774.0000627

Hanbay, D., Baylar, A. and Batan, M. (2009), “Prediction of aeration efficiency on stepped cascades by using least square support vector machines”, *Expert Syst. Applicat.*, **36**(3), 4248-4252. https://doi.org/10.1016/j.eswa.2008.03.003

Imanian, H. and Mohammadian, A. (2019), “Numerical simulation of flow over ogee crested spillways under high hydraulic head ratio”, *Eng. Appl. Comput. Fluid Mech.*, **13**(1), 983-1000. https://doi.org/10.1080/19942060.2019.1661014

Iqbal, W., Jalil, M., Khadimallah, M.A., Ayed, H., Naeem, M.N., Hussain, M., Bouzgarrou, S.M., Mahmoud, S.R., Ghandourah, E., Taj, M. and Tounsi, A. (2020), “Runge-Kutta method for flow of dusty fluid along exponentially stretching cylinder”, *Steel Compos. Struct., Int. J.*, **36**(5), 603-615. https://doi.org/10.12989/SCS.2020.36.5.603

Jacobsen, J. and Olsen, N.R.B. (2010), “Numerical modelling of the capacity for a complex spillway”, *Proceedings of the*

Institution of Civil Engineers-Water Management, **163**(6), 283-288. https://doi.org/10.1680/wama.2010.163.6.283

Jun, C.L., Xiang, J.Y. and Dong, H.Y. (2015), "CFD simulations of the fluid flow behavior in a spacer-filled membrane module." *Membr. Water Treat*, *Int. J.*, **6**(6), 513-524. https://doi.org/10.12989/mwt.2015.6.6.513

Kim, D.H., Chang, Y.S. and Jhung, M.J. (2014), "Numerical study on fluid flow by hydrodynamic loads in reactor internals." *Struct. Eng. Mech.*, *Int. J.*, **51**(6), 1005-1016. https://doi.org/10.12989/sem.2014.51.6.1005

Nair, V.V. and Bhattacharyya, S.K. (2018), "Water impact of three dimensional wedges using CFD", *Ocean Syst. Eng.*, *Int. J.*, **8**(2), 223-246. https://doi.org/10.12989/ose.2018.8.2.223

Ozdogan, M., Sungur, B., Namli, L. and Durmus, A. (2017), "Comparative study of turbulent flow around a bluff body by using two- and three-dimensional CFD", *Wind Struct.*, *Int. J.*, **25**(6), 537-549. https://doi.org/10.12989/was.2017.25.6.537

Park, S.W. and Cho, J.R. (2012), "Adaptive fluid-structure interaction simulation of large-scale complex liquid containment with two-phase flow", *Struct. Eng. Mech.*, *Int. J.*, **41**(4), 559-573. https://doi.org/10.12989/sem.2012.41.4.559

Phillips, D.G., Richards, P.J. and Flay, R.G.J. (2002), "CFD modelling and the development of the diffuser augmented wind turbine", *Wind Struct.*, *Int. J.*, **5**(2), 267. https://doi.org/10.12989/was.2002.5.2_4.267

Siah-Bishe Dam (2006), Siah-Bishe Pumped Storage Project Hydraulic Model of Flood Outlet System Final Report; Siah-Bishe, Iran.

Simonetti, M. and Gentile, V.M. (2019), "CFD optimization of large water storages for efficient cooling of high power intermittent thermal loads", *Case Stud. Therm. Eng.*, **14**, 100466. https://doi.org/10.1016/j.csite.2019.100466

Tabbara, M., Chatila, J. and Awwad, R. (2005), "Computational simulation of flow over stepped spillways", *Comput. Struct.*, **83**(27), 2215-2224. https://doi.org/10.1016/j.compstruc.2005.04.005

Varjavand, P. (2008), "Investigation the energy dissipation in the stepped spillway using the fluent numerical model", M.Sc., Tabriz University, Iran.

CC