ORIGINAL ARTICLE



Numerical modeling of spillway aerators in high-head dams

M. Cihan Aydin¹ · Ercan Isik¹ · A. Emre Ulu¹

Received: 4 July 2019 / Accepted: 20 December 2019 / Published online: 2 January 2020 @ The Author(s) 2020

Abstract

Due to high flow velocity, the spillway surfaces of high-head dams can expose to cavitational damage. The most effective and economical method of protection from this damage is aerated to flow using aerators. In this study, a spillway aerator of the roller-compacted concrete dam of 100 m height was analyzed using two-phase computational fluid dynamic model to overcome the cavitation damage on the spillway surface. The numerical analysis with prototype dimensions was performed for various flow conditions (5223, 3500, 1750 and 1000 m³/s of flow rate), and obtained results were compared with some experimental observation in the literature. Numerical and experimental results indicated that the cavitation occurs on the surface after a certain downstream point based on cavitation indices. The air entrainment rate and air concentrations supplied by means of the aerator were determined to avoid the cavitational damage. While the experimental results can contain considerable scale effect in terms of air entrainment rate owing to, e.g., viscous effects especially for small scales, the numerical models with prototype dimensions gave much more accurate results. In other words, it can be also mentioned that the actual aeration amount is much greater than that obtained from the model experiments. The results based on numerical analysis showed that the aerator device meet air demand to prevent the cavitation damage.

Keywords Dam · Cavitation · Computational fluid dynamics · Scale effect

Introduction

Spillways are crucial safety structures used for the discharge of flood discharges into dam reservoirs and for the operation of these dam reservoirs. It may be classified as controlled and uncontrolled in general, albeit in different types. In terms of body safety, it is preferred to place them in the right or the left shore in the earth-fill dams while and in the concrete dams it is placed on the body to be cost-effective. In arch dams, they can also be placed on the body due to the shortage of space. As a new type which has been preferred in recent years, roller-compacted concrete dams (RCC) can be constructed as safe and economical as dams. In these dams, thanks to its solid body, spillways can be placed on the body

 A. Emre Ulu aliemreulu@gmail.com
 M. Cihan Aydin mcaydin@gmail.com
 Ercan Isik eisik@beu.edu.tr

¹ Department of Civil Engineering, Bitlis Eren University, Bitlis, Turkey economically in these dams. Two important aspects should be taken into consideration in the design of the spillways. The first one is the scour failure in the dam's downstream due to the high-discharge and velocity flow. Various types of baffle blocks are used in the spillway outlets in order to prevent scour failure which may threaten the dam stability. The other significant issue is protecting the concrete surface from cavitation damage in the spillway channels exposed to high-velocity flows. Water vapor bubbles occur in the flow as a result of the pressure falling below the vapor pressure in small spaces due to irregularities (joint gap, construction defects etc.) on the surface of spillway under high-velocity flow. The vapor bubbles, which are subjected to repeated hydrostatic pressure in the flow discontinuities, explode with a large noise as they enter into the liquid phase, damaging concrete surface which is in physical contact with the water. This phenomenon is called 'cavitation damage'. Concrete surface exposed to the flow may cause more cavitation and leads to deepened damage and eventually make the spillway structure unusable. The cavitation damage in the spillways was first noticed in the period between 1970 and 1980 through extensive damages in the spillways of Keban Dam in Turkey and Karun Dam in Iran (Pfister and Hager 2010).



The most economical and effective method is to use aerating structures to protect the concrete surface from cavitation damage (as used in the Keban Dam spillway after damage). By means of these aerators providing a natural aeration mechanism, the risk of damage can be completely eliminated by mixing enough air into the cavitation zone. Usually, if the flow velocity in the spillway exceeds 20–30 m/s, it is recommended to use an aerator to protect the surface from cavitation damage (Chanson 1994). In some cases, it has been reported that precautions should be taken in case the flow velocities exceed 30–35 m/s even if the surface of the spillway is very smooth and well-constructed (Cassidy and Elder 1984; Chadwick and Morfett 1986; Novak et al. 1990).

Aydin et al. (2019) investigated bottom outlet of Ilisu Dam using CFD model. In the study, the performance of aeration galleries with different designs was investigated. At the end, two new aeration designs for that particular situation were proposed. Chanson (2009) stated that the extrapolation of laboratory results to large prototype hydraulic structures basically concerned despite recent advances in this area and discussed the dynamic similarity of the air entrainment processes. It is noted in the paper that physical model studies were performed generally using Froude similarity rule with smaller Reynolds number than corresponding prototype flow, and the concept of scale effect is closely related with the selection of relevant characteristic air-water flow properties. Kumcu (2017) used a CFD model with Flow-3D to investigate the flow over a full scaled (prototype) ogee spillway and compared to 1/50 scaled physical model results. The results of the numerical model well agree with the scaled physical model in terms of free-surface characteristics such as surface level, flow velocity and pressure, but any data about detail of air entrainment amount and its scale effects were not given besides some air concentration. Geun and Hyun (2005) investigated some flow characteristics of an ogee spillway by using CFD model (Flow-3D) to observe roughness and scale effects on them. They found that while the surface roughness and scale effects do not affect some results such as discharge, water surface and crest pressures too much, the roughness and scale effects are significant in maximum velocity location. Ferrari (2010) successfully performed a numerical study on the freesurface flow over a sharp-crested weir. The results were validated by comparing the free-surface profiles obtained from experimental measurements in the literature, and a good agreement was achieved. Heller (2011) presented a review on the Froude and Reynold model-prototype similarities to describe scale effects for typical hydraulic flows and discussed how scale effects were avoided or corrected. Felder and Chanson (2017) carried out some experiment of high-velocity mixing flow to investigate scale effects with respect to air-water flow. They presented a comprehensive



investigation on the air-water flow properties, e.g., the interfacial area, turbulence properties and particle sizes, which may be affected by scale effects. They also stated that the findings of the study are applicable to the other air-water flow type, but the prototype data were needed for final confirmation.

It is understanding from above researchers, for scaled model of hydraulic structures, while the scale effects are insignificant for some flow parameters such as discharge flow rate, water surface and pressures, it can be quite important for high-velocity air-water mixture flow, e.g., spillway flows with an aerator. In this study, the spillway aerator of the RRC type dam of 100 m height was selected as the model. The numerical model of the spillway was prepared in the prototype using a computational fluid model in different flow conditions (5223, 3500, 1750 and 1000 m³/s of flow rate), and the hydrodynamic behavior of the present design was analyzed with the help of computational fluid dynamics (CFD). The obtained numerical model results were compared with the results of the model test performed by DSI, and the obtained results were discussed presenting the hydrodynamic properties of the present design (Özcan 2011). The hydraulics of the flow on the spillway was first investigated with a single-phase flow model, and a doublephase (air-water) flow model was used for aerator performance on the spillway.

Materials and methods

The spillway design

The Kopru and HPP (hydroelectric power plant) installed on the Goksu Stream of Seyhan River in Turkey were constructed as RCC Dam with a power capacity of 154 MW. The energy dissipation of the uncontrolled spillway flow is provided by the flip bucket running as submerged at the downstream end. The design flow rates are considered as $Q = 3500 \text{ m}^3$ /s which is 1000-year period and the probable maximum flood discharge rate as Q = 5223 m³/s. The spillway's crest has a width of 125 m, and this width falls to 100 m for the flip bucket. The slope of the spillway chute channel is H:V=0.8/1, and its length is approximately 72 m. In the original project, the aeration troughs supplied by a 0.5×1.5 m rectangular aeration shaft on both sides were placed along the width of the spillway at 30 m downstream from the spillway crest to avoid cavitation are. The deflector ramp length used for aeration is designed to be 2.40 m with angle of 4°. The height of the aerator offset at the end of the ramp is given 0.5 m. After the project was revised, a second aeration gallery was added to aerate the middle regions and was given the two-chimney final form in Fig. 1.



Fig. 1 The details of spillway aerator geometry

 Table 1
 Discharge values selected for the model experiment and their tailwater levels (Özcan 2011)

Discharges, $Q (m^3/s)$	Model discharges $(\text{Scale} = 1/60) (1/s)$	Tailwater levels (m)	Periods
500	17.93	391.50	Minimum discharge
1000	35.86	321.80	10 years period
1750	62.76	324.20	100 years period
3500	125.51	328.30	2/3 of max. flood discharge
5223	187.30	331.00	Max. flood discharge

Physical model

In order to determine the hydraulic properties of the dam reservoir, the State Hydraulic Works (DSI) conducted physical model studies on a scale of 1/60 considering laboratory facilities (Özcan 2011). Some characteristic values used in physical model studies are given in Table 1. Some visuals of the physical model studies are given in Fig. 2. In the physical model studies, some flow characteristics (such as flow velocities, pressures, water levels) and performance efficiency of the aerator on the spillway chute were investigated. The studied discharge values were determined based on the discharge capacity and the cavitation risk of the spillway. A rectangular weir of 1.5 m wide with a sharp edge was used to adjust the discharge. In order to provide a uniform flow from the dam reservoir to the spillway, various baffle blocks and regulators were placed into the experimental setup after the measuring weir; then the water surface was smoothened. Physical models were first prepared according to the original project, and then the final design was obtained by improving the design in line with the hydraulic conditions.



Fig. 2 Visuals from the physical model setup (Özcan 2011)

Numerical model (CFD)

Numerical modeling of multiphase and free-surface flows is very difficult compared to other flow models i.e.



single-phase flows. Volume of fluid (VOF) method allows us to determine a clear free surface between the air and water. In high-velocity flows such as spillway flows, flow patterns should be used, which also take into account turbulence stresses and shear stresses between phases. In this study, numerical analysis was performed with the Flow-3D software. The governing equations of the fluid motion with the acknowledgment of the incompressible fluid (ρ = constant) are given below Flow Science (2014).

Mass continuity equation:

$$\frac{\partial}{\partial x}(uA_x) + \frac{\partial}{\partial y}(vA_y) + \frac{\partial}{\partial z}(wA_z) = 0$$
(1)

where ρ is the density of the fluid; A_x , A_y , A_z , respectively, are the partial areas opening to the flow in x, y and z directions; and u, v, w are velocity components in x, y and z directions, respectively.

Navier-Stokes Equations of fluid motion:

$$\frac{\partial u}{\partial t} + \frac{1}{V_{\rm F}} \left[uA_x \frac{\partial u}{\partial x} + vA_y \frac{\partial u}{\partial y} + wA_z \frac{\partial u}{\partial z} \right] = -\frac{1}{\rho} \frac{\partial p}{\partial x} + G_x + f_x$$

$$\frac{\partial v}{\partial t} + \frac{1}{V_{\rm F}} \left[uA_x \frac{\partial v}{\partial x} + vA_y \frac{\partial v}{\partial y} + wA_z \frac{\partial v}{\partial z} \right] = -\frac{1}{\rho} \frac{\partial p}{\partial y} + G_y + f_y$$

$$\frac{\partial w}{\partial t} + \frac{1}{V_{\rm F}} \left[uA_x \frac{\partial w}{\partial x} + vA_y \frac{\partial w}{\partial y} + wA_z \frac{\partial w}{\partial z} \right] = -\frac{1}{\rho} \frac{\partial p}{\partial z} + G_z + f_z$$
(2)

Here V_F is volume fraction open to flow; (G_x, G_y, G_z) : mass acceleration, (f_x, f_y, f_z) : viscous momenta. VOF scheme was performed for the solution of two-phase flows. In this method, if a cell is full by the primary phase (i.e. water), F=1; if the cell is empty F=0, in other words, the cell is full with the second phase (i.e. air); and if 0 < F < 1 then the cell contains an interface between two phases. The volume of the first fluid per unit volume is defined by the following VOF function (Hirt and Nichols 1981).

$$\frac{\partial F}{\partial t} + \frac{\partial 1}{V_{\rm F}} \left[\frac{\partial}{\partial x} (FA_x u) + R \frac{\partial}{\partial y} (FA_y v) + \frac{\partial}{\partial z} (FA_z w) + \varphi \frac{FA_x u}{x} \right] = F_{\rm D} + F_{\rm S}$$
(3)

Here F_D represents the term diffusion only involving twophase flow applications, and F_S represents the source term of density (Flow Science 2014)

In CFD analyses, while the single-phase flow models solve only one the constitutive equations, two-phase flow model solves the equations of motion for each fluid. Since the flows in the spillway have high velocity and high turbulence, the standard k- ϵ turbulence model, which is one of the most widely used turbulence models, is used in numerical analysis. The standard k- ϵ turbulence model which is one of the two-equation turbulence models are widely used in CDF



applications. In this model, the turbulent kinetic energy (*k*) and the dissipation rate with Reynolds stresses ($\varepsilon = k^{2/3}/L$, *L* is a scale of length) are solved together. This model is based on the assumption that the flow is completely turbulent besides viscous effects.

Geometry and boundary conditions

In numerical analyses, scale effects can be avoided by using prototype dimensions which are normally not possible to apply in physical laboratory models. In order to shorten the solution time, considering the symmetry of the spillway model with respect to the Y-axis, the solution domain was taken into consideration for half of the model as shown in Fig. 3a. In order to decrease the sensitivity of the results to mesh structure, 24,300,000 structural (square prismatic) elements were used by selecting the model's mesh structure sufficiently sensitive. These elements are excellent in terms of numerical computation, which do not have a distorted cell geometry in terms of numerical computation (since there will be no problem such as skewness). The boundary and initial conditions of the numerical model are presented in Fig. 3b. Here P is the specified pressure, S symmetry, O outflow boundary conditions. As initial condition, upstream and tail water levels according to the



Fig. 3 a Geometry and network. b Boundary and initial conditions of the numerical model

original project were defined in the model. Spillway discharge values are determined using the discharge-level curve of the spillway in the project.

Results and discussion

Scale effect

Physical hydraulic models of open channel flows are generally performed according to Froude's rules of similarity. Since the effects such as viscous and surface tension are neglected, significant scale effects occur especially in small-scale models when the Reynolds number (Re) is smaller than its prototype. Previous researchers have shown that significant scale effects occur especially in small-scale models of air mixture problems. In model experiments such as aerator design, it is reported that scales should be chosen to be larger than 1/10 in order to accurately estimate air entrainment and neglect scale effects (Kells and Smith 1991). In general, again, it is stated that the effects of scale can be neglected for $\text{Re} > 10^5$. Pinto (1988) conducted the 1/8 scale experiments of the Foz do Areia Dam spillway and discussed that it should be $\text{Re} > 3.3 \times 10^5$ to avoid scale effects. In their aerating experiments, Pinto and Neidert (1982) demonstrated that scale models larger than 1/15 had no observed effects on all flow rates, which was by means of scale models ranging from 1/8 to 1/50; they stated that models ranging between 1/30 and 1/50 do not have scale effects only for high flow rates. In some other studies carried out on the dam spillway aerators, Chanson (1989) chose the scale of 1/15 and Tan (1984) adjusted the model scale as 1/8 to avoid scale effects. In addition to this, Pinto (1984) identified scale effects of 33–67% in terms of air entrainment even in the 1/12 scale model tests of Terbela dam tunnel. Sakhuja et al. (1984), Vischer et al. (1982) stated that scale effects must be considered in terms of air entrainment in scale models smaller than 1/4. Sakhuja et al. (1984) proposed the below formula for Froude scale models:

$$\log_{10} \text{SEF} = 0.0048(\lambda - 1) \tag{4}$$

Here SEF is the scale effect factor and λ refers to the geometric scale ratio. After the scale effect factor is calculated, the prototype air entrainment rate can be calculated according to the model air entrainment rate with the formula $(\beta_p = \text{SEF} \times \beta_m)$. Based on the Froude scale, Bruschin (1984) gave the air entrainment scale with the following formula.

$$\frac{\beta_{\rm p}}{\beta_{\rm m}} = \sqrt{\lambda} \tag{5}$$

Kökpınar and Göğüş (2002) proposed the following formula to define the scale effects of the lower nappe air requirement.

$$\beta_{\rm p} = \xi \beta_{\rm m}^{\varepsilon} \tag{6}$$

Hereby β_p is the air entrainment rate according to the prototype calculation, β_m is the model air entrainment rate, ξ and ε are geometrically determined coefficients. Kökpınar and Göğüş (2002) gave these values ξ =5.194 and ε =1.150 for spillways with symmetric aerators and ξ =4.186 and ε =1.388 for spillways with asymmetric aerators. These coefficients for symmetric aerator were also used to eliminate the scale effects of the experimental results (Özcan 2011). Aydin and Ozturk (2009) obtained the coefficients of Eq. (6) as ξ =5.221 and ε =1.211 for bottom-inlet aerators. The scale effect factor for the air entrainment rate was taken as 1.4 in the 1/30 scale model test conducted (Demiröz 1982). This value is taken from the coefficient interval in Eq. (7) proposed by Eccher and Siegenthaler (1982).

$$(q_{\rm a})_{\rm p} = (1.11 - 1.43)(q_{\rm a})_{\rm m} \tag{7}$$

Here q_a refers to unit air discharge, *p* refers to prototype, and *m* represents model. Aydin et al. (2017) carried out the CFD analyses of 12-m-diameter sluice outlet of Ilisu Dam's tunnel using the two-phase flow model, compared it to the results of the 1/40 scale physical model and reported a scale effect of about 10% in terms of air entrainment rates based on the numerical results. In the single-phase flow model, a goodness of fit was obtained in terms of the hydraulics of the flow by means of experiments. In their study, they demonstrated that scale effects noticeably appeared in multiphase flow models such as air–water mixture.

Based on above explanations, it is obvious that the air entrainment rates obtained from the 1/60 physical model studies here will show significant scale effects. In studies of this nature, therefore, it is important to determine and eliminate the scale effects on the experimentally obtained air entrainment volumes.

Comparison with the experiment results

In order to determine the risk of cavitation in the probable maximum flood ($Q = 5223 \text{ m}^3/\text{s}$), cavitation indices were calculated using Eq. (8) at certain intervals along the axis of the spillway chute channel. According to the literature, if the cavitation index goes below 0.25, the risk of cavitation damage is reported to be very high (Falvey 1990). Accordingly, cavitation indices are calculated in Table 2 and compared with the results in experimental studies.

$$\sigma = \frac{P - P_{\nu}}{\frac{1}{2}\rho V^2} = \frac{g\left(\frac{P}{\gamma} - \frac{P_{\nu}}{\gamma}\right)}{\frac{1}{2}V^2} = \frac{g(h - h_{\nu})}{\frac{1}{2}V^2}$$
(8)



Applied Water Science (2020) 10:42

								(σ)	
								CFD	Experiment
7	0+001.57	30.32	7.20	-45	5.09	125.00	8.21	2.64	9.74
8	0 + 004.56	33.31	5.24	0	5.24	125.00	7.97	2.80	3.35
9	0+007.17	35.92	4.55	21.26	4.24	125.00	9.85	1.83	2.03
10	0+010.46	39.21	4.47	34.46	3.69	125.00	11.34	1.38	1.51
11	0+012.12	40.95	4.47	44.47	3.19	125.00	13.10	1.04	1.07
12	0+013.94	42.49	4.47	51.34	2.79	125.00	14.96	0.79	0.97
19	0+018.02	46.77	4.18	51.34	2.61	122.80	16.29	0.67	-
20	0+021.98	50.73	3.70	51.34	2.31	121.28	18.63	0.51	0.49
21	0+029.33	58.08	3.05	51.34	1.91	118.49	23.13	0.33	0.21
22	0+031.39	60.14	2.90	51.34	1.81	117.74	24.49	0.30	0.19
23	0+037.98	66.73	2.59	51.34	1.62	115.20	28.02	0.23	0.23
24	0+046.12	74.87	2.54	51.34	1.59	112.11	29.36	0.21	0.15
25	0+054.10	82.85	2.09	51.34	1.31	109.08	36.67	0.13	0.05
26	0+062.09	90.84	2.08	51.34	1.30	106.04	37.91	0.12	0.09
27	0+070.10	98.85	2.08	51.34	1.30	103.04	39.01	0.12	0.10
28	0+078.11	106.86	1.96	51.34	1.22	100.00	42.66	0.10	0.04

$$V = \frac{Q}{A} = \frac{Q}{B(y \times \cos \varphi)}$$

Point

ΚM

X(m)

v (m)

(9)

velocities (V > 20 m/s) in the aerator's upstream, it may still indicate that cavitation damage may occur in these areas.

Here σ represents cavitation index, P is absolute pressure, P_p is vapor pressure, ρ is water density, V is mean flow velocity, $g = 9.81 \text{ m}^2/\text{s}$ is gravitational acceleration, and h; h_p represents absolute pressure and water vapor pressure in terms of the water column. The average velocity in each section can also be calculated by Eq. (9) depending on the vertical water depth (y) and the slope of channel chute (φ) of the spillway. The absolute pressure head was taken as h = 9.30 m at 400 m altitude, where the dam is located and vapor pressure of water was taken as $h_p = 0.23$ m for a water temperature of 20 °C. According to the results obtained in Table 2, the cavitation index falls below over KM = 0 + 037.98. According to the model test results, there arises a risk of cavitation over KM = 0 + 029.33. According to the numerical model results, the cavitation risk is seen to be on the downstream. However, due to the high flow

Scale effect factors (SEF) and prototype values of air entrainment ratios determined based on the experimental results obtained according to three different flood discharges of the final design are given in Table 3. As can be seen in the table, the scale effect factors related to the air entrainment rates given in the literature vary widely. It is not possible to make arrive at a clear conclusion on the subject due to reasons such as deficiencies in prototype measurements, many of the experimental studies having been conducted according to Froude number and the fact that some of the formulas are given independently of the scale. Based on the information gathered so far, the volume of aeration from small-scale physical models has significant scale effects, and as the scale decreases, this effect can be said to grow and become more uncertain. Therefore, it is clear that a scaled model experiment for 1/60 will have significant scale effects. In the experimental studies, Eq. (6) proposed by Kökpınar and Göğüş (2002) was used to calibrate the results of the

 Table 3
 Scale effect factors and calculation of prototype air entrainment rates

$\overline{Q(m^3/s)}$	Model (1/60) β _m	This study		Equation (4) (Sakhuja et al. 1984)		Equation (5) (Bruschin 1984)		Equation (6) (Kökpınar and Göğüş 2002)		Aydin and Ozturk (2009)	
		SEF	$\beta_{\rm p}$	SEF	β _p	SEF	$\beta_{\rm p}$	SEF	$\beta_{\rm p}$	SEF	$\beta_{\rm p}$
1750	0.075	4.880	0.366	1.920	0.144	7.740	0.581	3.522	0.264	3.023	0.227
3500	0.050	4.520	0.226	1.920	0.096	7.740	0.387	3.314	0.166	2.775	0.139
5223	0.044	3.864	0.170	1.920	0.084	7.740	0.341	3.251	0.143	2.701	0.119
Mean		4.421		1.920		7.740		3.362		2.833	



model experiment given in Table 3. However, since this formula is independent of the scale, it may not be accurate. In Table 3, it is demonstrated in a study conducted by Aydin and Ozturk (2009) that the scale effect factors calculated by a larger scale such as 1/25 were smaller than the results obtained from this study and this supports the results of our study. This is because smaller scale leads to a greater scale effect. Equations of (4) and (5) given depending on the scale should not be taken into account since they yield very different results. Therefore, it can be said that the results obtained in this study are the most reasonable values shown in the table based on the results of the CFD analysis on the prototype scale.

Numerical model results

In the numerical analyses, discharges of $1000 \text{ m}^3/\text{s}$, $1750 \text{ m}^3/\text{s}$ and $3500 \text{ m}^3/\text{s}$ which are in different return period were considered in addition to the probable maximum flood discharge of $Q = 5223 \text{ m}^3/\text{s}$. In Fig. 4, two different numerical model outputs were given. Although a smoother water surface was obtained from the one-phase flow model (Fig. 4a), it is not possible to determine the air entrainment as in the two-phase flow model (Fig. 4b). To clearly obtain water surface profiles and the aeration performances, a higher mesh resolution including 24 million elements was needed for two-phase model. The solution domain of the numerical model was restricted as shown in Fig. 4b due to too much time and processor effort.

The aeration performance of the aerator together with some hydraulic parameters (e.g. flow depth, average flow velocities, upstream Froude number of aerator) obtained from CDF analysis by means of the two-phase flow model is given in Table 4. Two different Froude numbers were defined to examine the air entrainment rate and the change in air concentration with Froude number. The first one is the F_1 , calculated on the crest of the spillway, and the other is the F_2 , upstream Froude number calculated at just before the aerator. In the table, h_1 and h_2 are the flow depths at the crest and aerator upstream, respectively; V_2 is the upstream flow velocity, $Q_{\rm a}$ average air entrainment discharge provided from the aerator, $Q_{\rm w}$ water discharge, and lastly $C_{\rm a}$ represents average air concentration. In the literature, if the flow velocity in the spillway chute exceeds 20-30 m/s, it is recommended to use an aerator to protect the spillway surface from cavitation damage. It is seen in Table 4 that flow velocities are higher than 20 m/s. In Fig. 4a, it is also seen that the flow velocities increase up to 40 m/s toward the downstream of the chute. At these velocities, cavitation becomes inevitable and, as indicated, the flow must be aerated even if the surface of the spillway is well-constructed.

Based on the literature review, Aydin (2016) stated that the average air concentration in the chute flow should be



Fig. 4 Water surface velocity profiles ($Q = 5223 \text{ m}^3/\text{s}$): **a** one-phase flow model, **b** two-phase flow model

at least 6-8% in order to avoid cavitation damage. Also as seen in Table 4, the aerator provides the required minimum air entrainment rate and thus the air concentration. The average air concentration supplied to the flow is obtained by means of the formula of $C_a = \beta/(1+\beta)$. In calculations, usually, the upstream Froude number (F_2) is taken into account for determination of aerator performance. In the graphs of Fig. 5, the changes of air entrainment rates according to F_2 Froude numbers were plotted. The air entrainment rate is exponentially increased with F_2 . The fitted curve in the graph was obtained with a determination coefficient (R^2) of more than 99%, which means that the curves perfectly reflect its physical aspect. In addition, Eq. (10) has been determined to calculate the average air concentration from the aerator device with the regression coefficient of 99%.



 Table 4
 Aeration performances

 obtained from numerical
 simulations for different weir

 discharges
 discharges

Reservoir water level (m)	$Q_{\rm w}$ (m ³ /s)	<i>h</i> ₁ (m)	<i>h</i> ₂ (m)	<i>V</i> ₂ (m/s)	F ₂ (-)	$Q_{\rm a} ({\rm m^{3}\!/\!s})$	$\beta = Q_{\rm a}/Q_{\rm w}$ (-)	$C_{\mathrm{a}}(\%)$
417.15	5223	7.15	1.84	24.017	5.65	885.84	0.170	14.50
415.71	3500	5.71	1.10	26.921	8.20	790.28	0.226	18.42
413.71	1750	3.71	0.61	24.273	9.92	640.52	0.366	26.79
412.57	1000	2.57	0.41	20.636	10.29	432.58	0.433	30.20



Fig. 5 Variations of the air entrainment rate with Froude numbers

$$C_{\rm a} = 13.4 + 0.042 \exp\left(\frac{F_2}{1.72}\right) \tag{10}$$

Conclusions

This study attempts to determine the aerating performance of a spillway aerator by using the two-phase computational fluid dynamics. While the scale effects were not expected in the numerical model of the prototype scale, significant scale effects were determined in terms of air entrainment volume in the 1/60 scale physical model results. Therefore, it is seen that using only scaled hydraulic models may be inadequate in determining the air entrainment rates and that the results are needed with crude formulas in the literature so as to convert the results into the prototype values. According to the numerical model results, it can be said that the actual aeration amount is much greater than that obtained from the model experiments due to their scale effects. Therefore, it is noted that especially when prototype data are not available, numerical models validated can be preferred with or without experimental studies.

Acknowledgements This study was supported by Scientific Research Projects Department of Bitlis Eren University, which Project Number BEBAP-2018-02.

Compliance with ethical standards

Conflict of interest The authors declare that they have no conflict of interest.

Open Access This article is licensed under a Creative Commons Attribution 4.0 International License, which permits use, sharing, adaptation, distribution and reproduction in any medium or format, as long as you give appropriate credit to the original author(s) and the source, provide a link to the Creative Commons licence, and indicate if changes were made. The images or other third party material in this article are included in the article's Creative Commons licence, unless indicated otherwise in a credit line to the material. If material is not included in the article's Creative Commons licence and your intended use is not permitted by statutory regulation or exceeds the permitted use, you will need to obtain permission directly from the copyright holder. To view a copy of this licence, visit http://creativecommons.org/licenses/by/4.0/.

References

- Aydin MC (2016) Aeration in dam spillway (in Turkish). Türkiye Alim Kitapevi, Saarbrücken
- Aydin MC, Ozturk M (2009) Verification and validation of a computational fluid dynamics (CFD) model for air entrainment at spillway aerators. Can J Civ Eng 36(5):826–836
- Aydin MC, Ulu AE, Karaduman Ç (2017) 3D model-prototype comparison of air entrainment by two-phase CFD model. In: 8th International advanced technologies symposium, 19–21 October 2017, Elazığ, Turkey
- Aydin MC, Ulu AE, Karaduman Ç (2019) Investigation of aeration performance of Ilisu Dam outlet using two-phase flow model. Appl Water Sci 9(4):111
- Bruschin J (1984) Aeration off-sets for spillway chutes and bottom outlets. In: Symposium on scale effects in modelling hydraulic structures. International Association for Hydraulic Research, Esslingen, Germany, 3–6 September 1984, pp 4.3-1–4.3-4
- Cassidy J, Elder R (1984) Spillways of high dams. In: Novak P (ed) Developments in hydraulic engineering 2. Aplied Science Publishers Ltd., Essex, pp 153–173
- Chadwick AJ, Morfett JC (1986) Spillways in hydraulic in civil engineering. Harper Collins Academic, London, pp 390–403
- Chanson H (1989) Study of air entrainment and aeration devices. J Hydraul Res 27(3):301–319
- Chanson H (1994) Aeration and deaeration at bottom aeration devices on spillways. Can J Civ Eng 21(3):404–409



- Chanson H (2009) Turbulent air-water flows in hydraulic structures: dynamic similarity and scale effects. Environ Fluid Mech 9(2):125–142
- Demiröz E (1982) Ilısu Barajı dolusavak boşaltım kanalı havalandırması model çalışma raporu. Ankara
- Eccher L, Siegenthaler A (1982) Spillway aeration of aeration of the San Roque project. Water Power Dam Constr 34(9):37–41
- Falvey HT (1990) Cavitation in chutes and spillways. Engineering Monograph 42. Water Resources Technical Publication. US Printing Office. Bureau of Reclamation. Denver
- Felder S, Chanson H (2017) Scale effects in microscopic air-water flow properties in high-velocity free-surface flows. Exp Therm Fluid Sci 83:19–36
- Ferrari A (2010) SPH simulation of free surface flow over a sharpcrested weir. Adv Water Resour 33(3):270–276
- Geun KD, Hyun PJ (2005) Analysis of flow structure over ogee-spillway in consideration of scale and roughness effects by using CFD model. KSCE J Civ Eng 9(2):161–169
- Heller V (2011) Scale effects in physical hydraulic engineering models. J Hydraul Res 49(3):293–306
- Hirt CW, Nichols BD (1981) Volume of fluid (VOF) method for the dynamics of free boundaries. J Comput Phys 39(1):201–225
- Kells JA, Smith CD (1991) Reduction of cavitation on spillways by induced air entrainment. Can J Civ Eng 18:358–377
- Kökpınar MA, Göğüş M (2002) High-speed jet flows over spillway aerators. Can J Civ Eng 29(6):885–898
- Kumcu SY (2017) Investigation of flow over spillway modeling and comparison between experimental data and CFD analysis. KSCE J Civ Eng 21(3):994–1003
- Novak P, Moffat AIB, Nalluri C, Narayanan R (1990) Dam outlet works, Hydraulic structures. Unwin Hyman, London, UK, pp 153–176
- Özcan Ç (2011) Hydraulic model studies of Kopru Dam and HPP. Report no: Hi-1011, General Directorate of State Hydraulic Works-Hydraulic Model Laboratory, Ankara (**in Turkish**)

- Pfister M, Hager WH (2010) Chute aerators. I. Air transport characteristics. J Hydraul Eng 136:352–359
- Pinto NLDS (1984) Model evaluation of aerators in shooting flow. In: Symposium on scale effects in modelling hydraulic structures. International Association for Hydraulic Research, Esslingen, Germany, pp 4.2-1–4.2-6
- Pinto NLDS (1988) Cavitation and aeration. In: Jansen RB (ed) Advanced dam engineering for design, construction, and rehabilitation. Kluwer Academic Publishers, Dordrecht, pp 620–634
- Pinto NLDS, Neidert SH (1982) Model prototype conformity in spillway flow. In: International conference on hydraulic modelling of civil engineering structures, Coventry, England, pp 273–284
- Sakhuja VS, Paul TC, Singh S (1984) Air entrainment distortion in free surface flows. In: Symposium on scale effects in modelling hydraulic structures. International Association for Hydraulic Research, Esslingen, Germany, pp 4.8-1–4.8-4
- Sicence Flow (2014) Theory. Flow-3D user manual, v11.0.3. Flow Science Inc, Santa Fe
- Tan TP (1984) Model studies of aerators on spillways, Resarch Report 84–6. University of Canterbury, Christchuch, New Zealand
- Vischer D, Volkart P, Sigenthalear A (1982) Hydraulic modelling of air slots in open chute spillways. In: International conference on hydraulic modelling of civil engineering structures, Coventry, England, pp 239–252

Publisher's Note Springer Nature remains neutral with regard to jurisdictional claims in published maps and institutional affiliations.

