Computational Research Progress in Applied Science & Engineering

ISSN 2423-4591

CRPASE Vol. 03(04), 165-171, December 2017

Pressure Fluctuations Investigation on the Curve of Flip Buckets Using Analytical and Numerical Methods

Salar Khani[a*], Mehdi Azhdary Moghadam[b], Mohammad Nikookar[c]

[a*] *Department of Civil and Environmental Engineering, Tarbiat Modares University, Tehran, Iran*
[b] *Department of Civil Engineering, Faculty of Engineering, University of Sistan and Baluchestan, Zahedan, Iran*
[c] *Department of Civil Engineering, Faculty of Engineering, University of Guilan, Rasht, Iran*

Keywords

Flip bucket,
Spillway,
Pressure distribution,
Computational fluid dynamics,
Flow-3D.

Abstract

Flip buckets are one of the most important energy dissipating structures in the dams' downstream. The pressure on the surface of the flip bucket's curve is one of the most important design parameters. In these structures, the pressure fluctuations may cause a considerable lift force below the structure causing the rise and destruction of the floor slabs, fatigue of the used materials due to cyclic loading and unloading, floor destruction and the creation of cavitation, as a consequence. In this study, the computational fluid dynamics (CFD) is implemented for simulating the flow over the flip buckets. For this reason, the *Flow-3D®* code is used here. After validating the results of the software, the flip bucket of the Clyde dam spillway in New Zealand was investigated for different ratios of the spillway's design discharge (Q_d) and the numerical results associated with the pressure distribution over the flip bucket's structure were presented for $0.25Q_d$, $0.5Q_d$, $0.75Q_d$, Q_d, $1.25Q_d$, and $1.5Q_d$. Afterward, the pressure values were theoretically estimated along the flip bucket's curve. The results obtained showed that for different flow rates, the maximum pressure occurs near the midpoint of the horizontal length of the bucket's curve. For different flow rates, the theoretical and numerical results showed a difference of 23 to 41 percent. In addition, it was seen that the theoretical pressure distribution curve is less variable and changes with a slight slope, while the numerical results provide a nonuniform pressure distribution that is more consistent with the experimental results given by the previous studies.

1. Introduction

Energy dissipating structures are constructed at the end of spillways discharge channels to dissipate the extra energy. In general, hydraulic jump stilling basins, roller buckets, ski jump buckets or other energy dissipating structures are implemented for dissipating the extra energy. If the selected choice for the energy dissipation is a ski jump or a flip bucket, a curved circular surface called bucket will be founded at the end of spillway chute to cause the mild transition of the flow from the spillway surface to the downstream in the outlet channel. Flip buckets are used for energy dissipating and avoiding the river bed scoure in the downstream [1].

The flip bucket is a kind of ski jump spillway usually used as an end to a chut's or tunnel spillway in the case of appropriate geological and topological conditions [19]. The flip bucket itself is not considered as an energy dissipator; however, it is an integral part of the energy dissipation system. The aim of the flip bucket is to guide the high-velocity flow (the jet) to the farther location rather than the dam, powerhouse, spillway and other parts of the dam structure. A small amount of water's energy is dissipated due to the friction through the bucket. In the hydraulic design of the flip buckets, those parameters which are of high importance for the designers include the bucket's geometry, pressure exerted on its boundaries, and the jet trajectory characteristics [2].

Ski jumps or flip buckets have been proposed as a successful hydraulic design in Dordogne hydraulic projects which have been performed in France in the middle of 1930s and also upon the experimentations carried out by Maitre and Obolenski on the jet flows in 1954 [3]. Rohne and Peterka [4] conducted a study on an improved design of flip buckets

* Corresponding Author:
E-mail address: salar.khani@modares.ac.ir – Tel, (+98) 9126799470

Received: 12 August 2017; Accepted: 28 November 2017

performed by US bureau of reclamation. The simplest flip bucket is in the shape of a cylindrical shell's sector tangent to the floor of the flood conduit (chute or tunnel). This kind of flip bucket was founded as the energy dissipating system after the studies conducted in 1993 [5] on Grand Coulee dam located in Washington. Then, in 1945, the modified version of this structure named the slotted bucket was founded in Angostura dam located in South Dakota. This structure has been also constructed for Brantley dam of New Mexico.

Joun and Hager [6] evaluated the flip buckets both in the type of a prismatic rectangular channel and a bucket having a lateral flow deflector. In their research, the scale effects in the hydraulic models, pressure distribution through the bucket, flow projectile, shock-waves' creation conditions and governing choking relations have been investigated. Jorabloo et al. [7] compared the results of the numerical simulation of flow over the flip bucket with those obtained via the experimental model. They implemented *Fluent*® software for their numerical analysis. They concluded that the pressure distribution and jet trajectory obtained via the numerical analysis were in close agreement with the experimental results. Yamini and Kvianpour [8] investigated the hydrostatic and dynamic pressures and their distributions on the flip bucket curve. In their study they examined the effects of geometry and flow characteristics on pressure fields. Yamini et al. [9] experimentally investigated the pressure fluctuation and also the effect of entrance flow conditions on the pressure fluctuation on the bed of compound flip buckets of the Gotvand dam in Iran.

Flip buckets are specifically designed for a specific project and the designs are improved using the scale models [10]. Till now, the design of the flip bucket structures has been frequently based on the hydraulic models' studies and due to their site specifications. Therefore, limited design patterns are available in this regard.

2. Pressures Exerted on the Flip Bucket's Surface

In flip buckets, the pressure fluctuations may cause a considerable lift force below the structure causing the rise and destruction of the floor slabs, fatigue of the used materials due to cyclic loading and unloading, floor destruction and the creation of cavitation, as a consequence. As a sample, in April 27, 1993, Karun dam (Sh. Abbaspour) spillway due to the cavitation phenomenon in the bucket's place, led to remarkable damages which was unexampled during the 16 years of exploitation. This event further highlights the importance of paying more attention to the investigation of pressures on the structure of the bucket.

In order to determine the pressure distribution in the curvy flows, one has to consider the forces applying vertically on a column of the fluid with the cross section ΔA (see Figure 1).

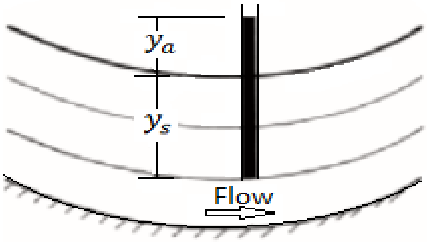

Figure 1. Pressure in the curvy flow [11]

If r and V define the streamline's curvature radius and flow speed at the assumed point, and $\rho y_s \Delta A$ be the mass of the fluid column, respectively; then, $\frac{V^2}{r}$ and $\rho y_s \Delta A \frac{V^2}{r}$ will stand for the centrifugal acceleration and centrifugal force, respectively.

Upon dividing the centrifugal force by the column cross section and converting the pressure to pressure head, one reaches to the Eq. (1)

$$y_a = \frac{1}{g} y_s \frac{V^2}{r} \tag{1}$$

in which, y_a is the pressure exerted on the end of fluid column caused by the centrifugal acceleration. As can be seen from Figure 1, this pressure is in the same direction as the fluid column's weight. Therefore, the total head on the end of column is the summation of the pressure values due to the centrifugal effects and the fluid column's weight. The total pressure head is obtained as Eq. (2)

$$h = y_s \left(1 + \frac{1}{g}\frac{V^2}{r}\right) \tag{2}$$

It is worthwhile mentioning that the first term in the above relation stands for the pressure head due to the static conditions, while, the second term describes the pressure head due to the centrifugal effects. Therefore, the fluid inside the inserted piezometer to the flow goes up [11].

The key parameters for the flip bucket design are including the approach flow velocity and depth, the radius r of the bucket, and the lip angle β. For a two-dimensional circular bucket, the pressure head can be computed for irrotational flow. The experimental achievements verify these values for the maximum pressure head, while, the theoretical results show a non-uniform pressure distribution (Figure 2) [10].

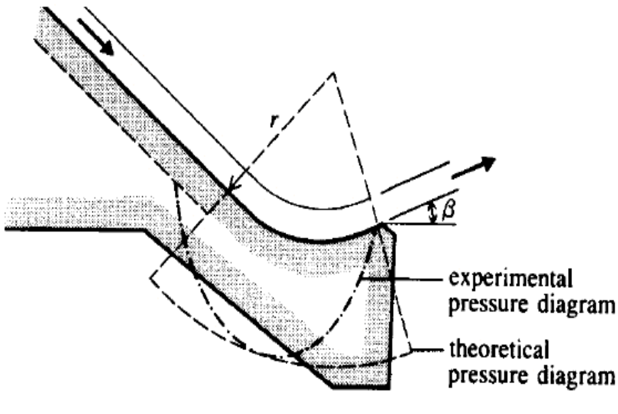

Figure 2 Comparison of the theoretical and experimental pressure distribution on the flip bucket [19]

The hydraulic forces acting on the flip buckets are considered in design of these structures. Theoretical studies and model and prototype data illustrate that the pressures resulting from the change in direction of the flow are changing continuously throughout the curve and they are affected by the curve radius, flow velocity, and discharge [2].

3. Numerical Analysis of the Flow Over the Flip Bucket

Although, flip buckets have been recently used in many hydraulic designs, the number of studies conducted on their basic hydraulic specifications is scarce. A great number of observations are position-oriented and specially designed for a specific hydraulic project. For this reason, the design guides of the flip buckets are not complete at this time. Therefore, many of these hydraulic structures will be examined by physical models before approving the final design [12].

Physical models are constructed in order to study the spillways flow specifications in hydraulic laboratories. However, they are expensive, cumbersome, and time-consuming [13]. Computational fluid dynamics (CFD) has been increasingly implemented by engineers for modelling and analyzing complicated problems related to the hydraulic design [14]. CFD models are capable of predicting many flow specifications on a spillway. Most of the investigations dealing with the CFD implementation for simulating the spillways have been using *Flow-3D*® code and this software has been successful at representing either the physical model's results or design curves of U.S. army corps of engineers (USACE) and U.S. bureau of reclamation (USBR) [15].

Savage and Johnson compared the *Flow-3D*® simulation results with those obtained via the physical model and USBR and USACE data for a standard ogee-crested spillway [16]. Their estimated values for pressure heads on the spillway surface and flow rates were in close agreement with the experimental achievements. The two-dimensional flow analysis at the toe of a dam were performed by Heidarinejad and Najibi [17]. Ho et al. [18] examined the behavior of a spillway with a standard WES profile under increased maximum flood. Eklund [19] investigated the jet trajectory of a ski-jump spillway in Stornnforsen using physical and CFD models.

The numerical model implemented here is constructed using the *Flow-3D*® code. The available obstacle in the calculation area is identified using fractional area/volume obstacle representation (FAVOR) technique. This approach which has been presented by Hirt et al. [20] uses values between 0 and 1 in order to identify the obstacle existence in each cell. The value of 1 indicates the case when the whole cell is filled with the obstacle. However, 0 points out to a completely empty cell [21, 22]. This program uses a finite volume calculation approach and the volume of fluid (VOF) method which has been previously improved by Hirt and Nichols [23] for computing free surface motion. Implementing this method lead to a significant decrease in the simulation time [24]. In the VOF method, it is assumed that the two fluids (water and air) do not interfere [25]. For each phase (fluid), a variable is defined called the volume fraction which represents the occupied amount of computational cells by that phase [26].

In this research, to investigate the pressure fluctuations on the flip buckets, the spillway of Clyde dam in New Zealand has been studied. After plotting the spillway's geometry in Autocad software, the model is imported into the *Flow-3D*® code. The turbulent model using the modified Reynolds equations (RNG) have been applied due to its good capability as well as precision in analyzing the free surface flows. The three-block meshing pattern with mesh sizes of 0.25m in the spillway area and 1m outside this area, has been used for the flow simulation.

4. Governing Equations

The governing equations of continuity and momentum for the incompressible flow including VOF and FAVOR variables are written as Eqs. (3) and (4).

$$\frac{\partial}{\partial x}\left(uA_x\right)+\frac{\partial}{\partial y}\left(vA_y\right)+\frac{\partial}{\partial z}\left(wA_z\right)=0 \tag{3}$$

$$\frac{\partial U_i}{\partial t}+\frac{1}{V_F}\left(U_j A_j \frac{\partial U_i}{dX_j}\right)=\frac{1}{\rho}\frac{\partial P'}{dX_i}+g_i+f_i \tag{4}$$

In the Eq. (3) and Eq. (4), u, v and w stand for the velocity components in *x*, *y* and *z* directions, respectively, V_F describes the volume fraction of each cell, A_x, A_y and A_z are the fractional surfaces in the above-mentioned directions, respectively. Also, ρ is the density, P' the pressure, g_i the gravitational force in the ith direction and f_i denotes Reynolds stresses of the turbulent model. In the cells completely filled by the fluid, V_F and A_j (cell surface) are equal to 1. Therefore, Eqs. (3) and (4) will be converted to the basic equation of incompressible Reynolds averaged Navier-Stokes (RANS).

5. Software Validation

In order to accredit the results obtained from *Flow-3D*® code, the achievements will be compared with those via real data or the experimental model. In this study, the Wuskwatim dam's spillway which located on the Burntwood River in North Manitoba of Canada has been simulated for validating the results of the software. This dam's spillway is composed of three openings with vertical sliding gates. Each opening has 9 m width and 13 m height and the spillway is designed to handle the probable maximum flood (PMF) level of 2650 m3/s at a design head of 15.8 m [27]. This ogee-crested spillway is modelled as a single opening with fully opened gates. Figure 3 depicts the velocity field on the spillway for the design discharge (Q_d).

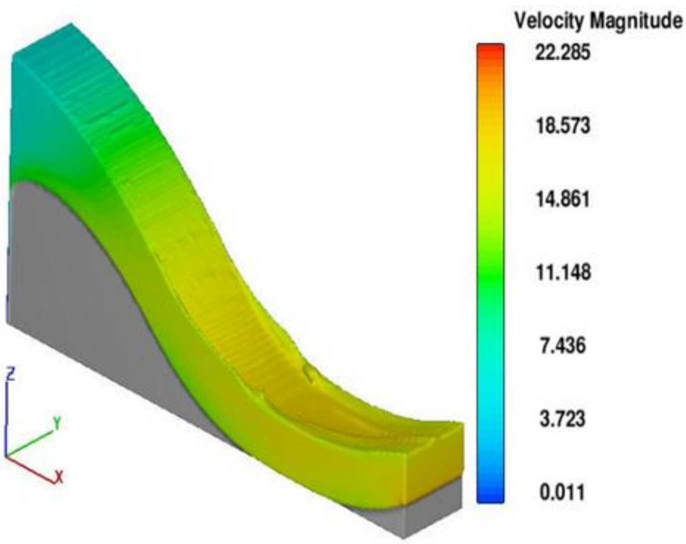

Figure 3. The geometry and velocity field on the spillway of Wuskwatim dam

A comparison is given in Figure 4 for the water surface profile obtained via CFD simulation with mesh size of 0.25 m in the central line for the water level of 244.7 m and those achieved via the physical model. It shoud be mentioned that in Figure 4, the results of CFD flow profile in each point are deduced from the nearest possible point on the spillway surface. Although this case is the major cause of the error occurrence, it would be observed that *Flow-3D*® results are in close agreement with the values estimated by the physical model.

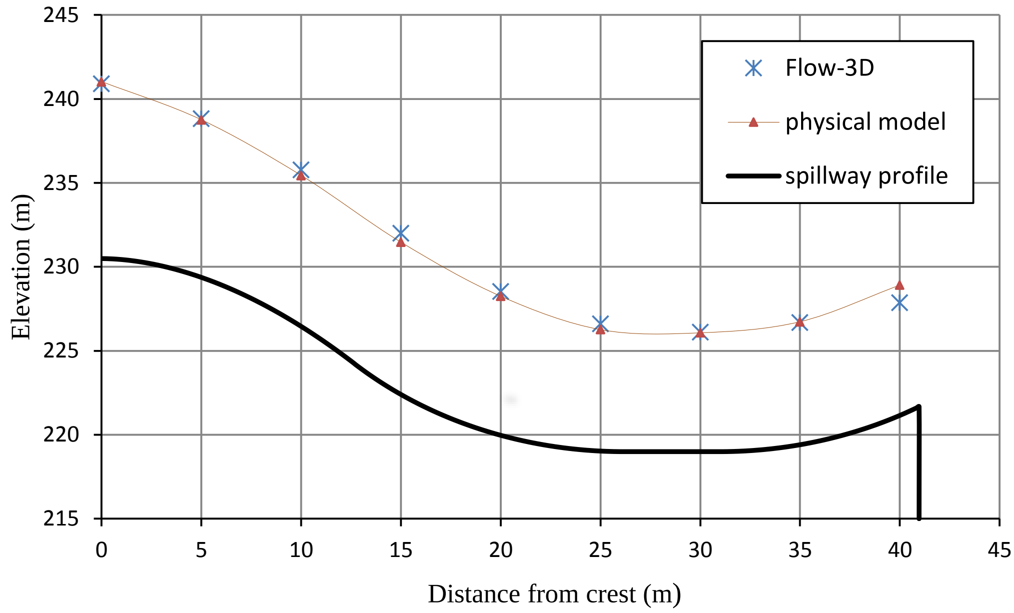

Figure 4. Comparison of *Flow-3D*® results and the physical model achievements for the water surface profile in the central line of the Wuskwatim dam's spillway

6. Model Description

Clyde dam with a height of 64.5 m is founded on Clutha River in New Zealand. The spillway of this dam is designed to pass the flow of 4100 m3/s in the PMF level of 195.1 m. This spillway includes four radial gates each 14.3 m high by 10 m wide. The spillway's length in about 70 m and leads to the stilling basin with a reverse slope of 1:8 [28].

To investigate the flip buckets of Clyde dam spillway, the geometrical model of the spillway has been constructed. After plotting the two-dimensional geometry of the spillway together with the stilling basin and flip bucket, the three-dimensional model is constructed and after importing the corresponding file into *Flow-3D*® code, the numerical model will be prepared for the simulation. The turbulent model together with the RNG equations model is implemented for the problem analysis due to its high precision as well as capability. The fluid specifications are given for water with applying the boundary conditions in terms of the flow velocity at upstream inlet and the output flow at downstream. The whole simulation is performed by mesh sizes of 0.25m in the spillway area and 1m outside this area. In this study, the flip bucket of the Clyde dam spillway is investigated using *Flow-3D*® code for various inlet flows ($0.25Q_d$, $0.5Q_d$, $0.75Q_d$, Q_d, $1.25Q_d$, and $1.5Q_d$).

7. Results and Discussion

After performing the flow analysis for various flow rates in *Flow-3D*® code, the numerical results have been obtained. It is seen that for the flow of $0.25Q_d$, the stream is undergone the hydraulic jump in the stilling basin before entering the flip bucket and loses a lot of its energy. As illustrated by Figure 5, the severe turbulence in the flow during the hydraulic jump and the mixture of water and air are clearly observed.

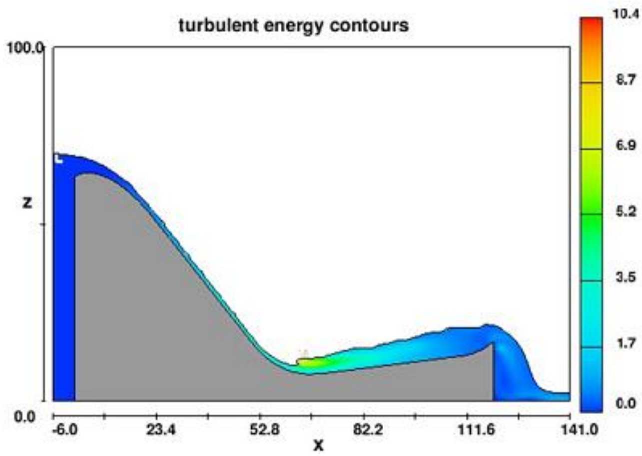

Figure 5. The turbulent energy contours from the crest to the end of spillway bucket for flow of $0.25Q_d$

The pressure contours on the flip bucket are depicted in Figure 6 for flows of $0.25Q_d$, $0.5Q_d$, $0.75Q_d$, Q_d, $1.25Q_d$, and $1.5Q_d$. The logical trend of the pressure variation on the bucket is notable for different values of the flow. As can be seen for various flow rates, the maximum pressure is happened in the proximity of the midpoint of the bucket's length. Further to this, the pressure exerted on the bucket's surface increases with an increment in the flow value. Also, the decreasing trend of the pressure near the bucket's end lip is observable for all values of the flow.

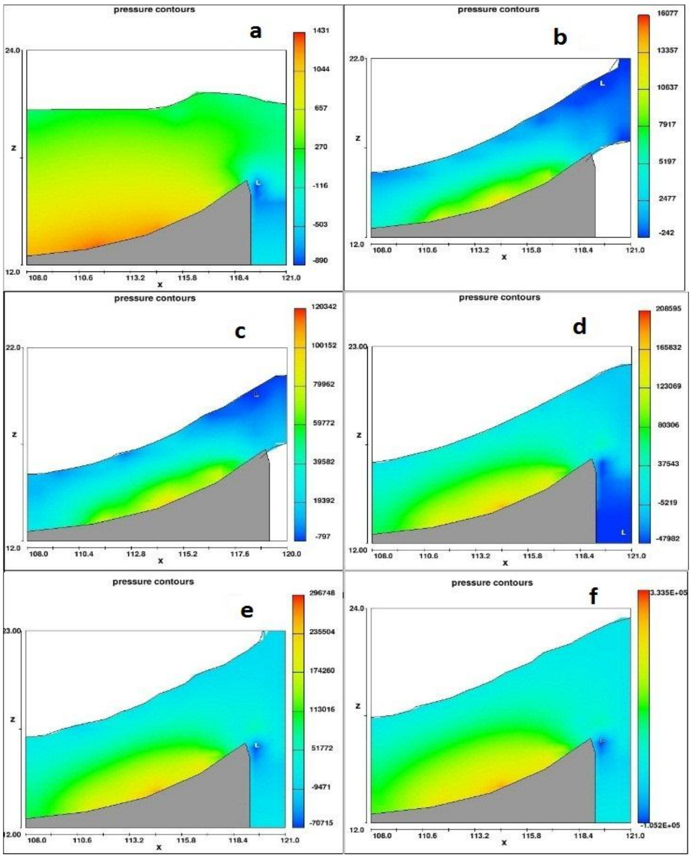

Figure 6. The pressure contours on the flip bucket for various flow rates, (a) $0.25Q_d$, (b) $0.5Q_d$, (c) $0.75Q_d$, (d) Q_d, (e) $1.25Q_d$, and (f) $1.5Q_d$

Figure 7 plots the pressure variation on the flip bucket for different values of the flow. It should be mentioned that the bucket's starting point and lip are located in the distance of 110.98 m and 119.2 m from the spillway's crest, respectively. It is observed that the value of pressure is negligible for flow rates of $0.25Q_d$ and $0.5Q_d$, however, it significantly increases as the flow comes up to $0.75Q_d$. With further increase in the flow value from $0.75Q_d$ to $1.5Q_d$, the pressure further increases and the pressure variation trends are the same along the bucket's length. For various flow rates, the maximum pressure occurs near the midpoint of the bucket's horizontal length which located in a horizontal distance of 114 m from the spillway's crest. Then, the pressure on the bucket gradually decreases and reaches to zero near the bucket's lip and meets the negative pressure values in some flow rates. Therefore, attention must be paid to the cavitation occurrence possibility in the bucket's lip.

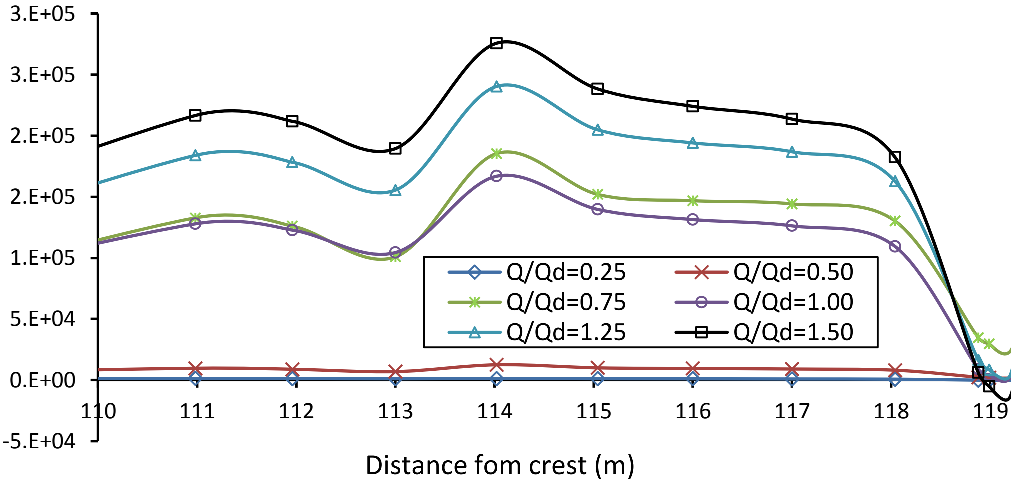

Figure 7. Pressure distribution on the flip bucket for various flow rates (*Flow-3D*® results)

In the following, the pressure distribution along the bucket's curve length is estimated theoretically by using Eq. (2). The results of these manipulations are given in Figure 8 for $0.75Q_d$, Q_d, $1.25Q_d$, and $1.5Q_d$. For flow value of $0.25Q_d$, the theoretical model yield the unreasonable results due to the hydraulic jump occurrence and high increase of the stream's height over the bucket. As would be observed again, the pressure variation trends are the same along the bucket's length for various flow rates.

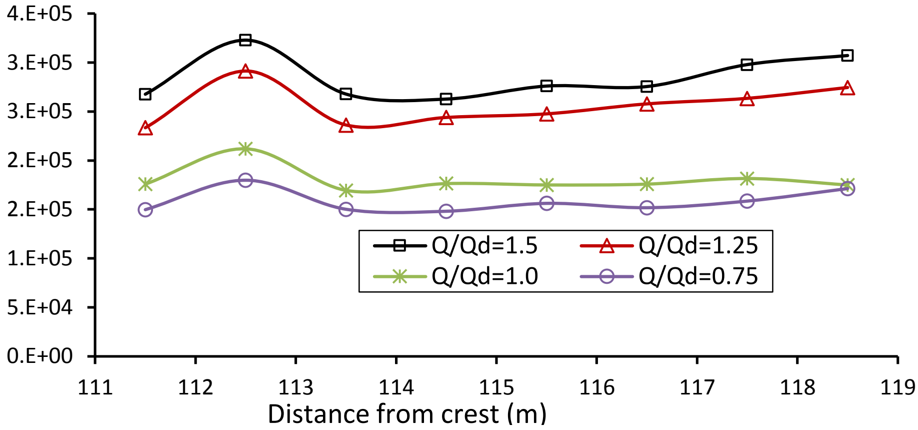

Figure 8. Pressure distribution on the flip bucket for various flow rates (theoretical results)

The theoretical results illustrate the maximum pressure's occurrence location in the distance of 1.5 m before the midpoint of the bucket's curve. Also, comparing with the numerical results, the gradient slope of the pressure oscillations is slower and, especially after the maximum point, shows slight pressure variations. This is in agreement with the theoretical pressure diagram presented in Figure 2. In Figure 2, the pressure diagram of the experimental results has been also given. It is observed that the experimental achievements indicate the more severe pressure oscillations rather than the theoretical results and the maximum pressure occurs in the vicinity of the midpoint of the bucket's horizontal length. Furthermore, the experimental results indicate a pressure value of about zero in the bucket's lip. Since the numerical models solve the governing flow equations with high precision, the pressure variations predicted by the numerical simulation using *Flow-3D*® code admit the remarkable accuracy in comparison with those of experimental pressure diagrams in Figure 2.

Table 1 gives a comparison between the theoretical results and those via *Flow-3D*® code for the maximum pressure on the flip bucket of Clyde dam spillway for flow rates of $0.75Q_d$, Q_d, $1.25Q_d$, and $1.5Q_d$. The theoretical results are calculated according to the depth and velocity of the flow and using Eq. (2). The difference between the theoretical and numerical results varies from 23.39% to 41.32% for various flow rates. It can be seen that the theoretical and numerical results are closer to each other for the design flow.

Table 1. Comparison between the theoretical and *Flow-3D*® results for the maximum pressure on the bucket

Q/Q_d	y_s (m)	V (m/s)	P_{Theory} (Pa)	$P_{Flow\text{-}3D}$ (Pa)	Differences (%)
0.75	3.2	19.59	108000	185000	41.32
1.00	3.9	19.13	128000	167000	23.39
1.25	4.9	19.56	166000	240000	30.89
1.50	5.8	19.79	199000	276000	27.67

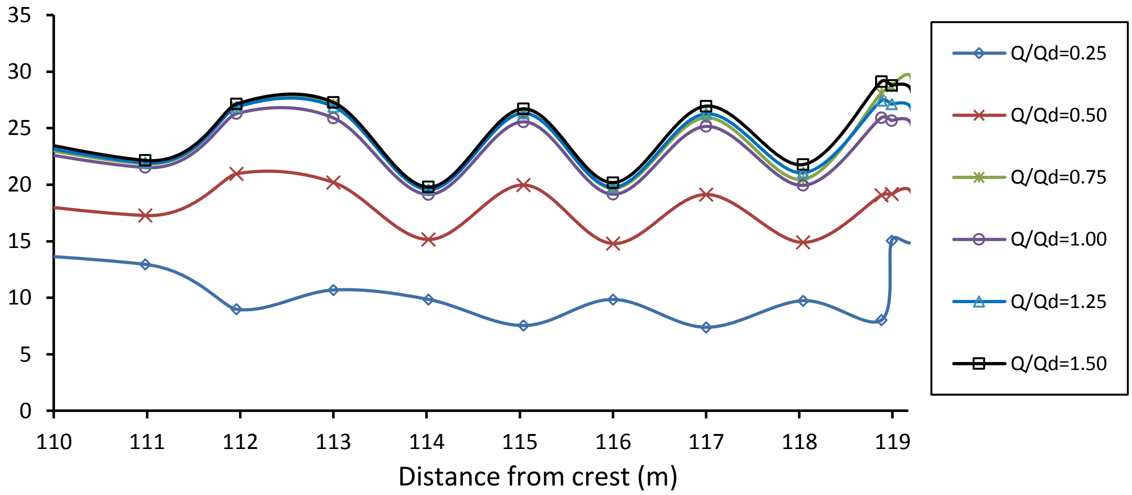

Figure 9. Distribution of the velocity magnitude on the bucket for different flow rates

Figure 9 depicts the variations of flow velocity from the beginning to the end of the bucket and for different flow values. It is observed that by initially entering the flow into the bucket, the flow velocity decreases, but the velocity variation trend is oscillatory in the bucket's length with the similar behavior for various flows. The velocity values on the bucket are very close to each other for flow values varying from $0.75Q_d$ to $1.5Q_d$, but the difference becomes more prominent in the bucket's lip. As expected before, the velocity is minimum in the vicinity of the bucket length's midpoint. This point in the in the bucket's length is the location of the maximum pressure which has been mentioned before.

8. Conclusions

Although a great number of flip buckets have been constructed in various dam projects around the globe, but the design of these structures are accomplished upon conducting specific studies and there are still no comprehensive and exact design guidelines in this subject area. The experimental studies are performed for the flow analysis over the flip buckets and the results will be used for the design of these structures. These studies are often time consuming and expensive. Using the flow simulating software is an appropriate option of increasing speed and reducing the cost of studies. In addition, in the case of performing laboratory studies, the results of the software would be used for verifying the experimental findings. In the present research, the simulation of Clyde dam spillways's flip bucket using *Flow-3D®* code led to satisfacory results. Outputs have been presented in terms of the pressure and velocity distributions on the flip bucket's curvy surface. These values are of high importance as they will be implemented for predicting the exerted pressures on the bucket in different flow values and the cavitation occurrence possibility on these structures. The numerical results have been compared with those achieved via the theoretical model. This comparison indicated that the results of these two approaches are closer for the design flow. Also, the numerical results obtained via *Flow-3D®* code for the pressure variation diagram in the bucket's length are in close agreement with the available experimental achievements in the literature.

References

[1] Design of Hydraulic Structure, Ministry of Science and Technology, Department of Technical and Vocational Education Design of Hydraulic Structures

[2] High Overflow Dams, Hydraulic Design Criteria, U.S. Army Corps of Engineers, Waterways Experiment Station, 1970.

[3] R. Maitre, S. Obolensky, Etude de Quelques Caractéristiques de l'Ecoulement dans la Partie Aval des Evacuateurs de Surface, La Houille Blanche 4 (1954) 481–511.

[4] T.J. Rhone, A. J. Peterka, Improved tunnel spillway flip buckets, Journal of Hydraulic Engineering, ASCE 126 (1959) 1270–1291.

[5] A.C. WATERS, Terraces and coulees along the Columbia River near lake Chelan, Washington. Geological Society of America Bulletin 44 (1933) 783–820.

[6] R. Joun, W.H. hager, Flip bucket without and with deflectors, Journal of Hydraulic Engineering, ASCE 126 (2000) 837–845.

[7] M. Jorabloo, R. Maghsoodi, H. Sarkardeh, 3D Simulation of flow over flip buckets at dams, Journal of American Science 7 (2011) 931–936.

[8] O. A. Yamini, M. R. Kavianpour, Experimental study of static and dynamic pressures over simple flip bucket, 5th symposium on advances in science and technology, Khavaran Higher-education Istitute, Mashhad, Iran. May 12-14 (2011).

[9] O.A. Yamini, M.R. Kavianpour, S.H. Mousavi, A. Movahedi, M. Bavandpour, Experimental investigation of pressure fluctuation on the bed of compound flip buckets, ISH Journal of Hydraulic Engineering 1 (2017) 1–8.

[10] P. Novak, C. Nalluri, R. Narayanan, Hydraulic Structures, Forth Edition, Taylor & Francis, New York (2007) 246–265.

[11] M.H. Chaudhry, Open-Channel Flow, Second Edition, Springer, 2008.

[12] L. Schmocker, M. Pfister, W.H. Hager, H. E. Minor, Aeration characteristics of ski jump jets, Journal of Hydraulic Engineering, ASCE 134 (2008) 90–97.

[13] M. R. Bhajantri, T.I. Eldho, P.B. Deolalikar, Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model, Sadhana 31 (2006) 743–754.

[14] F.A. Bombardelli, I. Meireles, J. Matos, Laboratory measurements and multi-block numerical simulation of the mean flow and turbulence in the non-aerated skimming flow region of steep stepped spillways, Environ Fluid Mech, Springer 11 (2011) 263–288.

[15] P.G. Chanel, J.C. Doering, An evaluation of computational fluid dynamics for spillway modelling, 16th Australian Fluid Mechanics Conference, Crown Plaza, Gold Coast, Australia, (2007).

[16] B. M. Savage, and M. C. Johanson, Flow over ogee spillway: physical and numerical model case study, J. of Hydraulic Engineering, ASCE 127 (2001) 640–649.

[17] G. Heidarinejad, and A. Najibi, Two-dimentional analysis of flow at the toe of a dam, Proceedings of Intl. Conf. on Hydraulic Structures, Shahid Bahonar Kerman University, Kerman, (2001).

[18] D.K. Ho, H.K.M. Boyes, S.M. Donohoo, Investigation of spillway behaviour under increased maximum flood by computational fluid dynamics technique, 14th Australian Fluid Mechanics Conference, Adelaide University, Adelaide, Australia, (2001).

[19] S. Eklund, CFD modelling of ski-jump spillway in Stornnforsen, Master's Thesis, Royal Institue of Technology, Sweden, (2017).

[20] C.W. Hirt, B.D. Nichols, N.C. Romero, SOLA: A numerical solution algorithm for transient fluid flows (No. LA--5852). Los Alamos Scientific Lab., N. Mex.(USA), (1975).

[21] Flow-3D® Manual, Version 8.1

[22] C.W. Hirt, J.N.A. Silcilian, Porosity technique for the definiotion of obstaacles in rectangular cell meshes, Proc., 4th Int. Conf. Ship Hydro., National Academy of Science Washington, D.C., (1985) 1–19.

[23] C.W. Hirt, B.D. Nichols, Volume of fluid (VOF) method for the dynamics of free boundaries, Journal of computational physics 39 (1981) 201–225.

[24] W. Choi W. and M. H. Kim, Free-surface fluid flows over Spillway, 4th International Conference Hydro-Science and Engineering, Seoul, 2000.

[25] B. Margeirsson, Computational modeling of flow over a spillway in vatnsllsstifla dam in Iceland, Master's Thesis, Department of Applied Mechanics, Chalmers University of Technology, Sweden, (2007).

[26] M. Sartaj, K. Beirami, A.M. Fooladgar, Analysis of two-dimentional flow over standard ogee spillway using RNG turbulence model, 7th International Congress on Civil Engineering, Tarbiat Modares University, Tehran, Iran. (2006).

[27] E. Teklemariam, B.W. Korbaylo, J.L. Groeneveld, D.M. Fuchs, Computational fluid dynamics: diverse applications in hydropower project's design and analysis, Water Management in a Changing Climate, CWRA 55th Annual Conference – Winnipeg, MB, (2002).

[28] H. Chanson, Study of air entrainment and aeration devices on spillway model, PHD Thesis, University of Canterbury, Christchurch New Zealand, (1988).