Sigma J Eng & Nat Sci 34 (2), 2016, 279-290

Sigma Journal Engineering and Natural Sciences
Sigma Mühendislik ve Fen Bilimleri Dergisi

Research Article / Araştırma Makalesi

NUMERICAL SIMULATION OF FLOW ON A SIPHON SPILLWAY AND INVESTIGATION OF THE EFFECT OF A BOTTOM/OUTLET ANGLE ON HYDRAULIC PARAMETERS

Farshid PAKGAR[1], Rasoul DANESHFARAZ*[2], Ali Rezazadeh JOUDI[3]

[1]*M. Sc Student of Hydraulic Structures, Islamic Azad University, Maragheh Branch, Maragheh-IRAN*
[2]*Department of Civil Engineering, University of Maragheh, Maragheh-IRAN*
[3]*Young Researchers and Elite Club, Maragheh Branch, Islamic Azad University, Maragheh-IRAN*

Received/Geliş: 29.12.2015 Revised/Düzeltme: 25.03.2016 Accepted/Kabul: 03.06.2016

ABSTRACT

Spillways are the main hydraulic structures used to drain excess flow volume of dam reservoirs. It is essential to study siphon overflows due to simple geometric structure, automatic function and acceptable capacity. Numerical values of overflow siphon hydraulic parameters were calculated in this study. For this purpose, the geometrical domain of the solution was designed using GAMBIT Software, and the numerical simulation of flow was carried out using Fluent Software. Then, the experimental results were compared with actual results. The data used in the validation section was collected from calculation of absolute pressure in the lower part of the spillway body. After verification of the results, the study mainly aimed to evaluate the effect of a cup-shaped damper angle of the spillway outlet on hydraulic parameters of flow. For this purpose, four groups of hydraulic parameters were used: absolute pressure in the lower part of the spillway body, distribution of velocity at the spillway downstream, energy dissipation and the siphon spillway discharge coefficient. The cup-shaped damper radial angles were 30, 45 and 60 degrees. The results showed that mean velocity at a spillway downstream at a 60° outlet angle was higher than other outlet angles in all discharges. The greatest energy dissipation was observed in the spillway with a 30° outlet angle. In addition, the discharge coefficient increased by increasing the cup-shaped damper angle. However, the discharge coefficient decreased by increasing flow discharge. Absolute pressure on the lower part of spillway body was the only parameter that was not affected by changes in the spillway outlet angle.
Keywords: Siphon spillway, numerical simulation, absolute pressure, energy dissipation, discharge coefficient.

SİFON SAVAK AKIMININ SAYISAL SİMÜLASYONU VE KUPA ŞEKİLLİ SÖNÜMLEYİCİ AÇININ HİDROLİK PARAMETRELER ÜZERİNDEKİ ETKİSİ

ÖZ

Dolusavaklar baraj haznelerinden aşırı akımın drene edilmesi için kullanılan temel hidrolik yapılardır. Bu arada, basit geometrik yapı, otomatik akim fonksiyonu ve kabul edilebilir bir kapasiteye nedeniyle sifon savak akımı çalışmaları özel öneme sahiptir. Bu çalışmada sifon akımı hidrolik parametrelerinin sayısal değerleri hesaplanmıştır. Bu amaçla, çözüme ait geometrik ağ Gambit yazılımı kullanılarak tasarlanmış ve akım simülasyonu FLUENT yazılımı kullanılarak gerçekleştirilmiştir. Ardından laboratuvar sonuçları Simüle Edilmiş sonuçlarla karşılaştırılmıştır. Validasyon kısmında kullanılan veri, savak gövdesinin alt kısmına ait mutlak basınç hesabından alınmıştır. Sonuçların doğrulanmasından sonra çalışmada, savak çıkışına ait kupa şekilli sönümleyici açının akımın hidrolik parametreleri üzerindeki etkisinin değerlendirilmesi esas olarak amaçlanmıştır. Bu çerçevede, savak gövdesinin alt kısmına ait mutlak basınç, savak mansabındaki hızın dağılımı, enerji kaybı ve sifon savak katsayısı olmak üzere dört adet hidrolik parametre kullanılmıştır. Kupa şekilli sönümleyici radyal açıları 30°, 45° ve 60° olarak seçilmiştir. Sonuçlar tüm debiler için 60 derece çıkış açılı savak mansabındaki ortalama hızın, diğer çıkış açılılara göre daha yüksek olduğunu göstermiştir. En yüksek enerji kaybı 30 derece çıkış açılı savakta gözlenmiştir. Ayrıca, Kupa şekilli sönümleyici açısı arttıkça debi katsayısı artmıştır. Bununla birlikte, debi katsayısı akım debisi arttıkça azalmıştır. Savak gövdesinin alt kısmına ait mutlak basınç değişik savak çıkış açılarından etkilenen tek parametre değildir.
Anahtar Sözcükler: Sifon savak, sayısal simülasyon, mutlak basınç, enerji kaybı, debi katsayısı.

* Corresponding Author/Sorumlu Yazar: e-mail/e-ileti: daneshfaraz@yahoo.com, tel: 00989143202126

1. INTRODUCTION

A spillway is a hydraulic structure that releases floodwater that cannot safely be stored in reservoirs in order to prevent damage to the dam downstream. Spillways with improper design or insufficient capacity can lead to dam failure. Siphon spillways are used in many dams for various reasons, such as passage of full discharge through the spillway with a minimum increase in upstream head. Other advantages of this type of spillways are as follows (Houichi *et al.*, 2009):

1. Relatively automatic effect on flow
2. Adequate capacity for at least spillway width
3. Reliable, simple function and construction, regardless of maintenance, repair and operating costs

Siphon spillways generally consist of closed ducts with rectangular sections (Vischer and Hager, 1997). Limitations in large discharges is a disadvantage for this type of spillways (Roberson *et al.*, 1998). In general, a siphon spillway is used where there is not enough space to construct an overflow spillway and where outburst discharge is not considerably important. This spillway functions rapidly and increases water level in the lake behind the dam to the water level of the spillway as quickly as possible. Flow passes through the siphon spillway either at a free surface or under pressure. The siphon acts as an ogee spillway at a free surface, which was not considered in this study. The spillway becomes an orifice under pressure. Pressure should not drop to negative levels in a siphon spillway. Large negative pressure leads to cavitation at crest and consequent dam failure. Oliver was the first scholar who constructed a physical model of a siphon spillway in 1980. He conducted discharge rate increases as siphon is empty of air. Holder and Eschimpff (1999) determined discharge rate of siphon spillway flow in Conklingville Dam (New York) in laboratory. The discharge rate was determined earlier based on original design theory data. Dornack *et al.* (1999) developed a physical model of a siphon spillway of Oker Dam (Germany) in laboratory and studied the behavior of air valve and control valve. They determined the least area of air inlet opening and local head drop. Babaeyan and Koopaei (2002) evaluated hydraulic performance of a siphon spillway in Brent Reservoir in the mid-1830s with physical modeling. Houichi *et al.* (2006) studied the siphon spillway in four alternative models with different configurations based on changes in cross sections. They formulated the discharge coefficient as a function of head ratio in vertical dimension of siphon crest and Froude number. Finally, the linear and nonlinear relationships between flow on the spillway (which acts as a catchment edge) and the siphon spillway were investigated. Jourablou *et al.* (2010) studied different rectangular, square and circular sections and showed that a siphon spillway with a rectangular cross section has higher efficiency than other sections in immersion status. Lucke and Beecham (2010) evaluated air inlet through the siphon spillway in irrigation systems. They injected a known quantity of air and studied the effect of trapped air on flow rate and system capacity. Mousavi-Jahromi (2011) performed experimental and numerical investigation of flow on the siphon spillway and studied piezometric pressure in the spillway body. In laboratory, 10 piezometers were installed at the lower body of the siphon spillway, and pressure values were measured. Numerical simulation was performed using FLUENT software. Mousavi-Jahromi (2011) also compared the results of a physical model with numerical results of the simulated model and showed an acceptable consistency between the results. Ghafourian *et al.* (2011) investigated the hydraulic of the siphon spillway using physical and numerical modeling. Comparison of the discharge coefficient in a pressurized spillway with both free and submerged outlets shows that submerging an outlet is effective in promoting spillway efficiency.

In this study, the flow of the siphon spillway was simulated as numerical values using FLUENT software, and the effect of a cup-shaped damper angle on the spillway outlet on hydraulic parameters was studied. After numerical modeling of the flow on siphon spillways, the results were compared with the laboratory results of Mousavi-Jahromi (2011).

2. MATERIAL AND METHODS

2.1. Governing Equations

Governing equations on a viscous incompressible fluid in a turbulent state were expressed using Reynolds Averaged Navier Stokes equations. These included continuity and motion equations.

$$\frac{\partial u_i}{\partial x_i} = 0 \tag{1}$$

$$\frac{\partial u_i}{\partial_t} + \frac{\partial u_i u_j}{\partial x_i} = -\frac{1}{\rho}\frac{\partial P}{\partial x_i} + g_i + \frac{\partial}{\partial x_j} S_{ij} \tag{2}$$

In the above equations, u_i shows velocity component at x_i direction, P represents total pressure, ρ denotes fluid density, g_i represents gravitational acceleration at x_i direction and S_{ij} denotes stress tensor. The following equation was expressed for turbulent flow:

$$S_{ij} = \left[\rho(v + v_t)\left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i}\right)\right] - \left[\frac{2}{3}\rho(k + v_t)\frac{\partial u_i}{\partial x_i}\delta_{ij}\right] \tag{3}$$

Shear stress in turbulent flows consisted of two terms. In addition to shear stress caused by components of mean flow, another shear stress caused by components of velocity fluctuations occurs known as Reynolds stresses, which is expressed in the following equation:

$$S_{ij} = -\rho\overline{u_i^{'} u_j^{'}} = \rho v_t \left(\frac{\partial u_i}{\partial x} + \frac{\partial u_j}{\partial x}\right) - \frac{2}{3}\rho k \delta_{ij} \tag{4}$$

In the equation 4, v_t shows kronecker, which is a function of flow characteristics as well as flow and turmoil characteristics. δ_{ij} is used for kronecker. Turbulent kinetic energy per unit mass is expressed as equation 5.

$$k = \frac{1}{2}\left(\overline{u_i^{'}}^2 + \overline{u_j^{'}}^2 + \overline{u_k^{'}}^2\right) \tag{5}$$

$$\delta_{ij} = \begin{cases} 1 & i = j \\ 0 & i \neq j \end{cases} \tag{6}$$

In order to solve turbulent flow field based on equations of continuity and Reynolds Averaged Navier Stokes, it is essential to model Reynolds stress in equations in a specific way. In this case, four unknowns (velocities in three perpendicular directions and pressure) were obtained for a three-dimensional flow with four equations (continuity and motion in three dimensions). Turbulence models were used for governing the following equations. In this study, k-ε RNG turbulence models were used.

2.2. Statement of the Problem

In this study, the flow on a siphon spillway was numerically simulated. The siphon spillway was simulated based on the physical model developed by Mousavi-Jahromi (2011) at Shahid

Chamran University of Ahwaz. The spillway was installed within 5.2 meters of a flume with 5.5 meter length. The flume width was 31 cm, the height was 45 cm and the flume was 1.5 meters above the ground. The studied siphon spillway consisted of two upper and lower parts, which made possible rapid changes in flow cross section. The profile of the lower part of the spillway corresponded with the ogee part of the spillway. A ness was located in the direction of flow. Pressure on the lower part of the spillway was measured using ten pressure gauges placed at equal intervals ~~at~~ along the length of the spillway. A cup-shaped damper was also located at the end of the spillway. Figure 1 shows a schematic of the simulated siphon spillway. In figure 1, P_1 to P_{10} show piezometers installed in the lower part of the spillway. The radius of R_1, R_2, R_3 and R_4 curvature were respectively ~~as~~ 0.15, 0.1, 0.0085 and 0.11 meters. Other geometrical dimensions of the spillway are also shown in Figure 1. In this study, after modeling flow on the spillway, the effect of a cup-shaped damper on hydraulic parameters was discussed. For this purpose, 30°, 45° and 60° angles (angle θ in Figure 2) were considered for the cup-shaped damper angle of a siphon spillway. It should be noted that θ angle was 60° in the study conducted by Mousavi-Jahromi (2011).

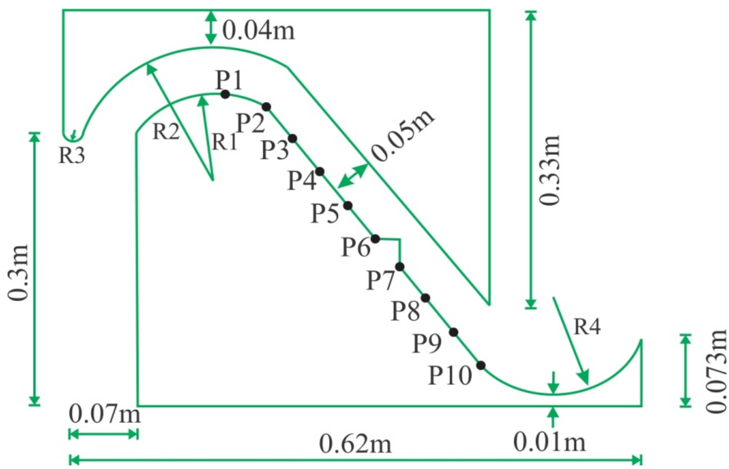

Figure 1. Dimensions and geometry of the studied siphon spillway.

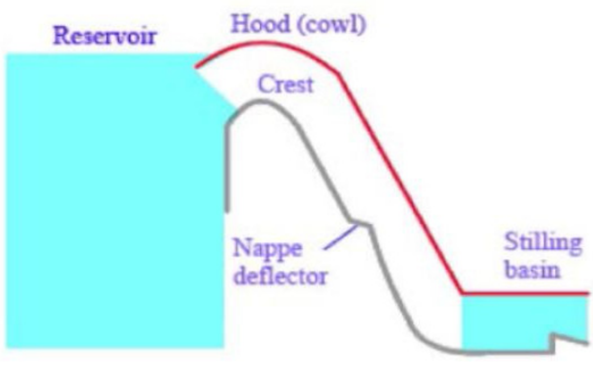

Figure 2. Longitudinal profile of a physical model of the siphon spillway (Mousavi-Jahromi 2011)

2.3. Boundary Conditions

In Figure 3, the geometry of a siphon spillway by Mousavi-Jahromi (2011) is shown in GAMBIT. The boundary conditions are also shown in this figure. The boundary of a water inlet due to given flow rate and head height of the water inlet and inlet velocity were as boundary of inlet pressure where inlet flow velocity and water head were determined at inlet boundary. The outlet boundary was as outlet pressure with atmospheric pressure. The floor, upper and lower boundaries of the spillway at the siphon section were as wall boundaries on which no-slip condition was applied.

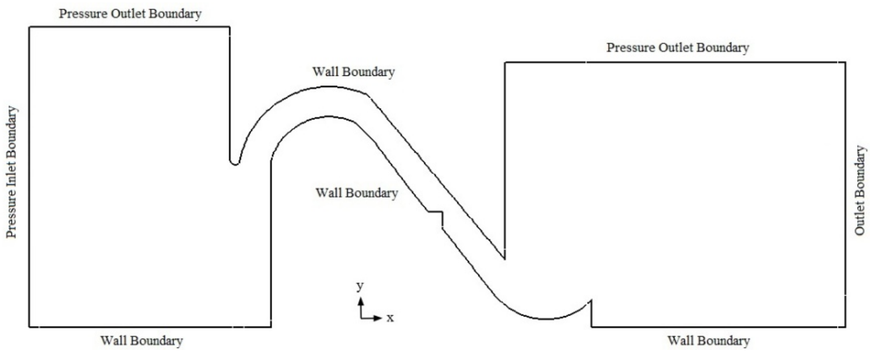

Figure 3. Geometry of the spillway and boundary conditions in GAMBIT software.

Upstream head, inlet velocity and water load for simulated models of siphon spillways are provided for three different flow rates in Table 1. It should be noted that the values of the water load and absolute inlet discharge were intended as those values obtained in the experiments performed by Mousavi-Jahromi (2011). Values of inlet velocity were calculated based on the principle of continuity.

Table 1. Boundary conditions of the spillway for different dischargers

Upstream head (m)	Inlet velocity (m/s)	Water load (m)
0.4299	2.24	0.2371
0.4	1.91	0.2097
0.3801	1.48	0.1672

2.4. Meshing and Discretization the Equations

Tetrahedral meshes in GAMBIT software were used to mesh the domain of solutions. Mesh details for siphon spillways with different outlet angles are presented in Table 2. Structured meshes were used in most domains of solutions in order to increase the speed of analysis in FLUENT software. Unstructured meshes were used due to the complexity of the geometry of the siphon spillway channel. The mesh of the siphon spillway with a 45^o damper angle in GAMBIT medium is shown as an example in Figure 4.

Table 2. Number of used mesh in siphon spillways with different outlet angles.

Outlet angle (°)	Number of mesh
30	3918
45	4008
60	4248

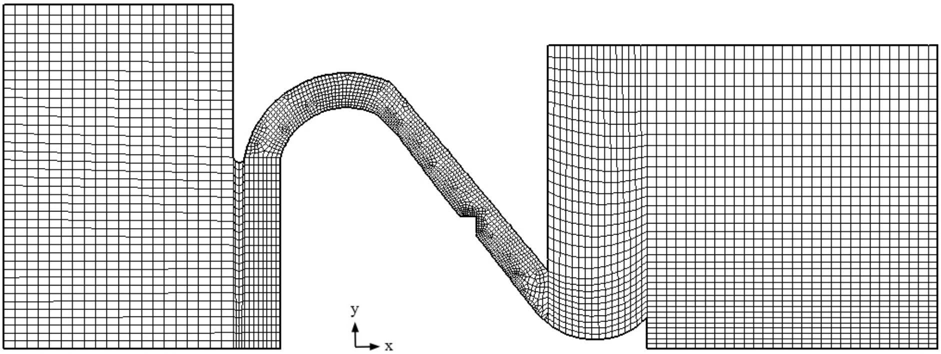

Figure 4. Meshing of a siphon spillway with a 45-degree cup-shaped damper.

As mentioned earlier, numerical simulation of fluid flow governing equations are solved numerically with a finite volume method using FLUENT software. This method aims to discretize governing differential equations and convert them into algebraic equations. FLUENT software is equipped with a feature to select discretization type of various terms in governing equations. Accordingly, a standard design was used for discretization of pressure, momentum equations were used for separation of movement terms, first order upwind design was used for movement terms in turbulence equations and simple algorithm was used to solve synchronization of velocity and pressure.

3. RESULTS

The results of absolute pressure at 10 piezometers obtained from numerical simulation analysis and experimental results of Mousavi-Jahromi (2011) are shown in Figures 5 and 6 for θ=60°. In these figures, the horizontal axis shows the piezometer number, and the vertical axis indicates the absolute pressure (kPA). Figures 5 and 6 are respectively provided for the validation of 0.0260 and 0.0316 m^3/s discharges. After the piezometer number 6, because of the nose, the pressure drop can be seen. The pressure loss can be caused by increased velocity due to the constriction of the section. Failure to observe this phenomenon in the experimental results may be due to the lack of accurate pressure gauges in the laboratory. Precise investigation of these figures shows an acceptable agreement between numerical and experimental results. Relative error levels in Figures 5 and 6 are 0.60% and 0.83%, respectively. It should be noted that relative error level is calculated by the following equation:

$$\text{Relative Error} = 100 \times \sum \frac{R_F - R_M}{R_M} \tag{7}$$

In this regard, R_M and R_F respectively represent experimental and simulated absolute pressures.

In numerical simulation results, a jump can be observed among 6 and 7 piezometers, which is normal due to the presence of ness. According to absolute pressure in the upstream area of the ness, absolute pressure suddenly increases and decreases in the downstream area. This type of behavior may be due to lack of precision of pressure gauges in the laboratory.

The results of absolute pressure on the body of a simulated siphon spillway are presented in figures 7-9. In these figures, the horizontal axis represents the piezometer number, and the vertical axis shows absolute pressure in kilopascal.

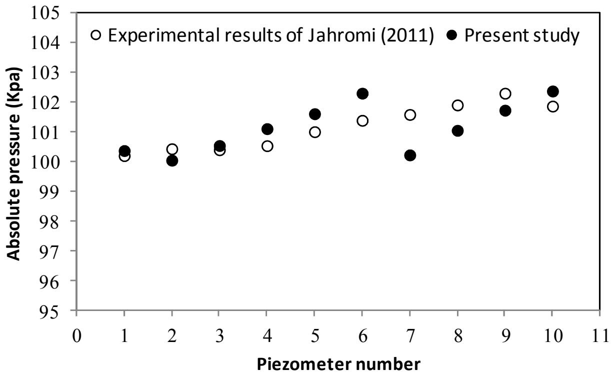

Figure 5. Comparison of numerical and experimental results of absolute pressure on the wall of the siphon spillway in discharge of 0.026 (m3/s)**.**

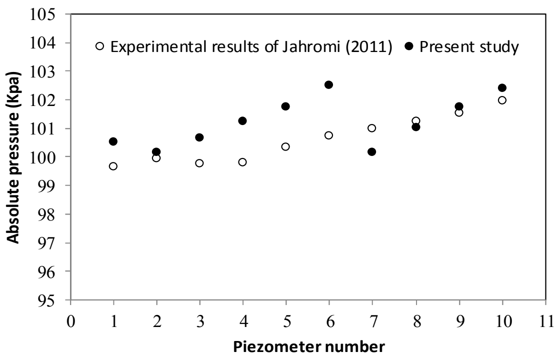

Figure 6. Comparison of numerical and experimental results of absolute pressure on the wall of the siphon spillway in discharge of 0.0316 (m^3/s).

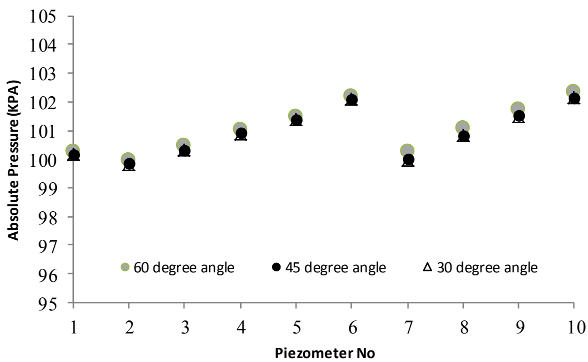

Figure 7. Absolute pressure on the body of a simulated siphon spillway in discharge of 0.0197 (m^3/s).

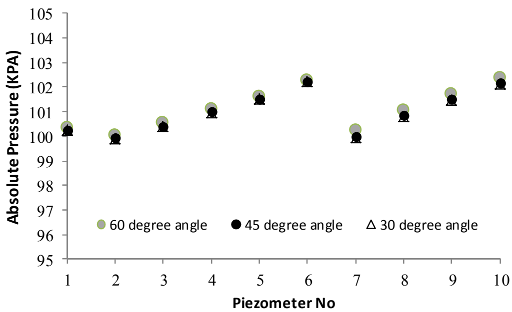

Figure 8. Absolute pressure on the body of a simulated siphon spillway in discharge of 0.0260 (m^3/s).

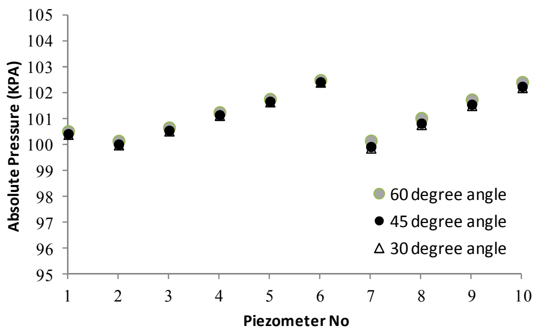

Figure 9. Absolute pressure on the body of a simulated siphon spillway in discharge of 0.0316 (m^3/s).

The results of velocity profiles downstream from the simulated siphon spillways are presented in figures 10 to 12. In these figures, the horizontal axis represents longitudinal velocity in meters per second, and the vertical axis shows flow depth, or the distance of water level from the bottom of the channel, in meters.

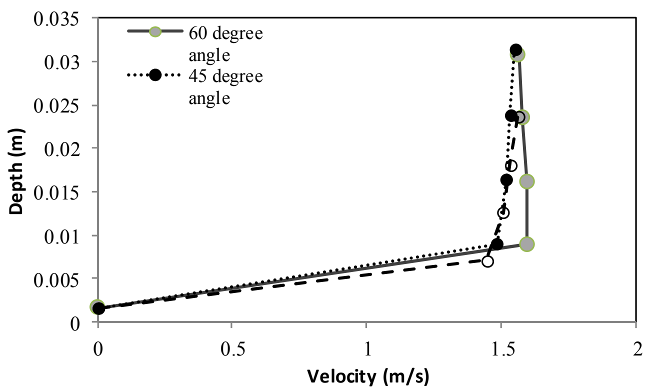

Figure 10. Velocity distribution downstream from the siphon with a discharge of 0.0197 (m^3/s).

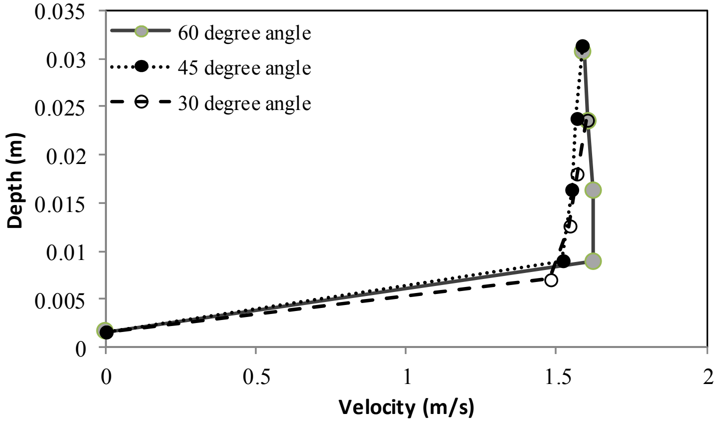

Figure 11. Velocity distribution downstream from the siphon with a discharge of 0.0260 (m3/s).

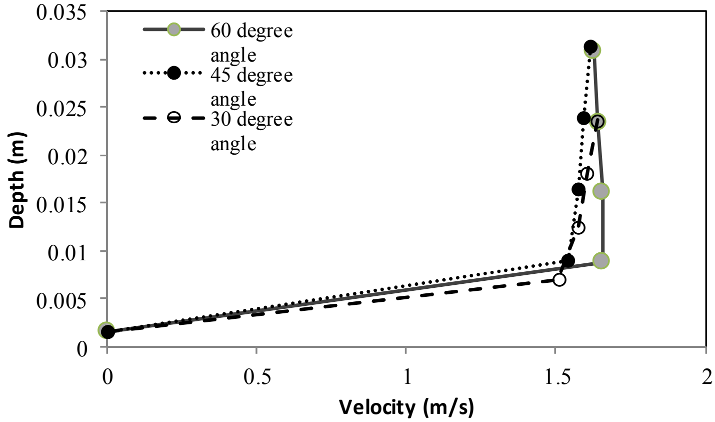

Figure 12. Velocity distribution-downstream from siphon in discharge of 0.0316 (m3/s).

Figures 10-12 show that the highest mean velocity was observed downstream from the siphon spillway with a cup-shaped damper angle (60 degrees). This is due to the formation of a vortex flow~~at~~ downstream of the cup-shaped damper, which acts as a shoot and increases velocity at the spillway outlet. In the following, energy dissipation parameters are discussed. Energy dissipation was calculated through relative energy loss between input of solution domain (spillway upstream) and output of solution domain (spillway downstream) using the following equation:

$$\frac{(E_A - E_B)}{E_A} \times 100 = \frac{(y_A + \frac{V_A^2}{2g}) - (y_B + \frac{V_B^2}{2g})}{y_A + \frac{V_A^2}{2g}} \times 100 \quad (8)$$

In this regard, *E*, *y* and *V* denote specific energy, water depth and mean velocity. Subtitles of A and B respectively represent the inlet section upstream from the spillway reservoir and the outlet section downstream from the spillway reservoir. Relative energy dissipation parameters were measured using Equation 8, and flow simulation results are shown in Table 3.

Table 3. Comparison of relative energy drop (%) in siphon spillways

		Angle of cup-shaped damper		
		30	45	60
Discharge (m^3/s)	0.0197	63.71%	61.29%	58.83%
	0.0260	64.15%	61.92%	59.81%
	0.0316	65.48%	63.77%	61.45%

According to Table 3, the highest relative energy loss was obtained in the spillway with a 30-degree outlet angle. In other words, relative energy loss reduced by increasing outlet angle. Relative energy loss increased in all cases with increasing flow rate. Finally, the results of the discharge coefficient in outlet angles are presented in Figure 13. In figure 13, the horizontal axis represents the cup-shaped damper angle, and the vertical axis shows the discharge coefficient. The results were obtained through numerical simulation. The discharge coefficient was calculated from the ratio of outlet discharge on the spillway to nominal discharge on the spillway. It should be noted that the discharge coefficient depends on the geometry of the spillway and passing discharge. The figure shows that the discharge coefficient increased when increasing the cup-shaped damper angle. The discharge rate also decreased with increasing flow discharge.

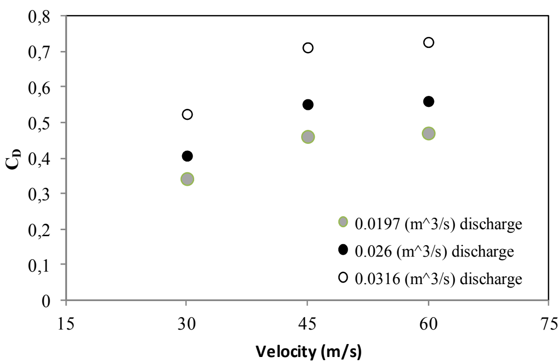

Figure 13. Comparison of discharge coefficients in siphon spillways.

4. CONCLUSION

Siphon spillways are used in many dams for various reasons, such as passage of full discharge through the spillway with a minimum increase in upstream head. Understanding energy dissipation downstream from any spillway is of paramount importance, as the energy of outlet water from the spillway can threaten the stability of upstream and downstream structures. One method of energy dissipation in the downstream of spillways is to use a bucket. In this study, the effect of three different output angles of the bucket for energy dissipation in a siphon spillway is evaluated. Furthermore, the effect of the bottom/outlet angle on the discharge coefficient is also examined. In this regard, the values of the outlet angles were considered as 30, 45 and 60 degrees (sectional angles). Three different flow rates 0.0197, 0.0216 and 0.0316 m^3/s were applied to the angles. The effect of the outlet angle on four groups of flow parameters absolute pressure, velocity distribution downstream of the spillway, energy dissipation and discharge rate was evaluated in given discharges. It was shown that changes in the outlet angle had no effect on absolute pressure in the lower side of the siphon spillway. Longitudinal velocity distribution upstream and downstream of the spillway was also determined. The mean velocity in the spillway with a 60-degree angle was higher than other outlet angles. This was due to the formation of a vortex flow downstream from the cup-shaped damper, which acts as a shoot and increases velocity at the spillway outlet. In the following, the effect of the outlet angle on energy dissipation was evaluated. The amount of energy at the inlet and outlet of solution domain was measured. The highest relative energy loss was obtained at a 30-degree outlet angle of the spillway. In other words, relative energy loss decreases as the outlet angle increases. Also, in all cases, relative energy loss increases with increasing flow discharge. The discharge coefficient was the last studied parameter. The results showed an increase in the cup-shaped damper angle as the discharge coefficient was increased. The results showed that the discharge coefficient decreased with increasing flow discharge.

REFERENCES / KAYNAKLAR

[1] Babaeyan-Koopaei, K., Valentine, E.M., Alan Ervine, D. (2002) Case study on hydraulic performance of Brent Reservoir siphon spillway. Journal of Hydraulic Engineering. ASCE, 128(6): 562-67.

[2] Chamani, M R., Dehghani, A A., Beirami, M K., Gholipour, R. (2009) Fluid Mechanics, Isfahan Industrial University's Publication, Iran. (In Persian).

[3] Daneshfaraz, R. (2010) Fluid Mechanics. Maragheh University's Publication, Iran. (In Persian).

[4] Daneshfaraz, R., Nikpour, V., Sadeghi, H. (2013) Simulation of hydraulic problems. Amidi Publications, Tabriz, Iran. (In Persian).

[5] Dornack, S., Horlacher, H. B. (1999) Hydraulic model experiments for the siphon spillway of the Oker Dam. In *28th IAHR Congress*.

[6] Ghafourian, A., Mousavi jahromi, H., Shafaee bajestan, M. (2011) Hydraulic of Siphon Spillway by Physical and Computational Fluid Dynamics. World applied sciences journal, 14(8): 1240-1245.

[7] Hirt, C.W., Nichols, B.D. (1981) Volume of fluid (VOF) method for the dynamics of free boundaries. J. Computational. Physics, 39: 201-5.

[8] Holder, G.K., Schimpff, M.C.E.J. (1999) West Project: analysis of discharge structures. Proc. of Waterpower Conference. Peggy A. Brookshire editor, Las Vegas, Nevada, USA.

[9] Houichi, L., Ibrahim, G., Achour, B. (2006) Experiments for the discharge capacity of the siphon spillway having the Creager-Ofitserov profile. Int. J. Fluid Mechanics Res., 5: 395-06.

[10] IUT Computational Fluid Dynamics Group Website

[11] Jourabloo, M., (2010) Investigation on siphon spillway's hydraulic. M Sc thesis, Islamic Azad University, Tehran's Science and Research branch. (In Persian).

[12] Lucke, T., Beecham, S. (2010) Capacity loss in siphonic roof drainage systems due to aeration. *Building Research & Information*, *38*(2), 206-217.

[13] Mousavi-Jahromi, S.H., (2011) Simulation of pizometric pressure in dam siphon spillways, World Applied Sciences Journal, 12 (7): 1074-1083.

[14] Oliver, G. (1980) The full-scale behavior of air-regulated siphon spillways. In ICE Proceedings (Vol. 69, No. 3, pp. 687-706). Thomas Telford.

[15] Patankar S.V. (1980) Numerical heat transfer and fluid flow. Hemisphere Publishing Corporation, Taylor & Francis Group, New York.
[16] Roberson, J.A., Cassidy, J.J., Chaudhry, M.H. (1998) Hydraulic engineering, Second edition. John Wiley & Sons, New York, USA.
[17] Roberson, J.A., Cassidy, J.J., Chaudhry, M.H. (1998) Hydraulic engineering, Second edition. John Wiley & Sons, New York, USA.
[18] Shojaei fard, M H., and Nourpour, A. (2010) An Introduction on Computational Fluid Dynamics. Science and Industry University's Publication. (In Persian).
[19] Tasi, W., Yue, D. (1996) Computation of nonlinear free surface flows. Annu. Rev. Fluid Mech., 28: 249.
[20] Vischer, D.L., Hager, W.H. (1997) Dam hydraulics, John Wiley and Sons Ltd, England.