World Applied Sciences Journal 14 (8): 1240-1245, 2011
ISSN 1818-4952

Hydraulic of Siphon Spillway by Physical and Computational Fluid Dynamics

[1]A. Ghafourian, [2]H. Mousavi jahromi and [3]M. Shafaee Bajestan,

[1]Hydraulic Structures, Azad university, Science and research of Tehran branch
[2]Department of Water Engineering, University of Chamran, Ahvaz, Iran
[3]Department of Water Engineering University of Chamran, Ahvaz

Abstract: Siphon spillway is a hydraulic structure which has been used in many dams and irrigation networks. Estimation of pressure distribution within this structure and head-discharge, relationship are the most important design's parameters of siphon. At present the hydraulic of siphon spillway usually is done by physical modeling. Physical modeling requires many times. On the other hand, numerical modeling has the advantage of doing very fast. Therefore it is the main purpose of this study to show that, the numerical model also can be performed to have accurate results. To reach the purpose of this study first a physical model of siphon spillway was constructed and placed in a test flume in hydraulic laboratory of Azad university- Garmsar branch. At the same time an available CFD program, which solves the Navier-Stokes equations, was run by using data obtained from physical model. The result of investigations shows that: Comparison of discharge coefficient in pressurized spillway with both free and submerged outlet shows that submerging in outlet is effective in promotion of spillway efficiency. It is shown that there is reasonably good agreement between the physical and numerical models for both pressures and discharges and CFD modeling can be used for this purpose.

Key words: Siphon spillway · Modeling · CFD · Navier-Stokes equations

INTRODUCTION

A siphon spillway is a closed waterway system formed in the shape of an overturned U, positioned so that the inside of the curve of the upper entrance is at normal lake storage level. The primary discharges of the spillway, as the lake level increases above normal, are alike to flow over an ogee spillway [1]. Siphonic action takes place after the air in the upper curve has been drained. Continuously flow is supported by the suction result due to the gravity pull of the water in the lower leg of structure. Detailed explanations of siphons may be found e.g. by Govinda Rao, Press, Samarine, *et al.* and Preissler and Bollrich [9]. Consistent with Head a differentiation is made between the Blackwater siphon and the air-regulated siphon [7]. Blackwater siphon can only operate on an on-off basis (i.e. full-bore, or no flow rate at all) but air-regulated structure or whitewater siphon automatically adjusts its flow rate over the full range of flow to maintain a virtually constant headwater level. The rehabilitation of a siphon with a comparatively large head of 16 m is defined by Bollrich [9]. The characteristic of the whitewater structure is that it sucks in both water and air from where the white water creates.

Up until now, the use of physical scaled model was the only investigation methods. The routine usage of numerical tools such as finite element, finite difference and computational fluid dynamics (CFD) analysis software in engineering design has expanded in current years [3]. Advances in software and hardware technology mean more nonlinear, complex three-dimensional analyses are being performed. Now the use of numerical methods such as CFD analyses is attractive in term of lower cost and substantially reduced preparation time and results can be obtained throughout the flow domain rather than at selected monitoring locations. Early difficulties involving moving mesh or grid to track the free water surface and to obtain a converged solution were reported. Nowadays, more efficient CFD codes can solve the Navier-Stokes equations in three-dimensions and free surface computation in a significantly improved manner [10]. As this type of siphon spillway analysis technique was used for the first time and the need to carry out validation was essential. The fundamentals of verification

Corresponding Author: A. Ghafourian, Hydraulic Structures, Azad university,
Science and research of Tehran branch No.54, east 74 Ave, Dardasht St., Narmak, Tehran, Iran.
E-mail: ab_kooshan@yahoo.com, ghafourian.ali@gmail.com.

and validation of CFD simulations have been done by the physical model. The interrelationship between the physical model and computer model has been thoroughly researched and established. The results in two models were compared in order to provide a level of confidence in applying CFD modeling in future studies. This paper begins by describing the general background of siphon spillway and in particular, the tracking of free surface and under pressure flows for spillway. Then physical model designing has been reported. The process of conducting an extensive validation exercise by analysing a siphon spillway profile under various flood conditions in both two- and three-dimensions will be described.

MATERIALS AND METHODS

A siphon is a ducted over flow structure with either free surface or pressurized flow [8]. Siphons are used as spillway either in alongside or addition to other flood opening or intake of small power plants, such as described by Xian-Huan [10]. The hydraulic design of a siphon mentions especially to the maximum discharge, i.e. pressurized flow. In accordance with the energy equation the flow rate Q is:

$$Q = A_d V_d = ab\eta(2gH_0)^{1/2}$$

Where A_d and v_d are cross-section and average velocity of the siphon outlet, a and b are height and width at the siphon crest and η is a siphon efficiency coefficient. In practice typical value of η are between 0.7 and 0.9. By analogy to the standard spillway, the minimum pressure occurs at the siphon crest. Assuming a flow with concentric streamlines in the crest region yields an expression for the discharge Q_c with incipient cavitation as [9]:

$$Q_c = br_i \ln(\frac{r_a}{r_i})\left[\frac{2g(h_A - h_c + t)}{1+\sum \zeta_i}\right]^{1/2}$$

Here, r_i and r_a are inner and outer crest radii, respectively, h_a is the atmospheric pressure head at elevation A above sea level, h_c is the pressure head at incipient cavitation, t is the sum of all head loss coefficient [5].

Design of Physical Model: The required model of the spillway have a size of 620*mm**300*mm**400*mm* containing two lower and upper parts. The reason behind this is for the easy installation and the possibility of calculation of water cross section. The lower part has a longitudinal profile which is length conforms to the ogee spillway. Of course a nose has been installed in the flow direction. The supports and frame of the spillway all are made of MDF to make it impermeable to the water. The flow section has been made of galvanized steel.

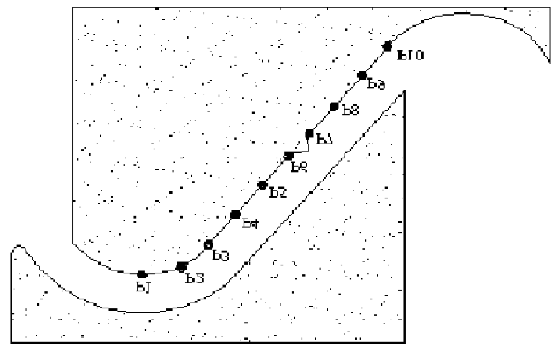

Fig. 1: Siphon spillway, Location of piezometers

At the outlet of the spillway, there is a place for installation of some barriers to submerge the flow. In this research, a barrier with 15*m* height has been installed to raise the tailwater level to 20*cm*. A number of 10 piezometers have been installed in the central axis of the spillway on its lower part with equal intervals to measure the water pressure in the spillway. The reason behind the installation of piezometers on central axis is for elimination of side compaction effect (Figure 1).

The siphon spillway is installed in a flume with 5500*mm**1300*mm**1900*mm* (L*W*H) size. Two control valves have been installed at inlet and outlet of pump to control and regulated the acceptable range of discharge in the flume. The bottom of the flume was flat and the spillway was installed in a part of flume which had a glass wall and 2.5*m* length.

Computation Fluid Dynamics (CFD): Prior of discussion on the numerical model of siphon spillway, we have to outline the principles and fundamentals of this branch of science. Firstly, we try to address this question: What is fluid dynamics? The physical specifications of a fluid flow are controlled by three main factors: 1- Conservation of mass, 2- Newton's 2nd Law 3- Conservation of Energy [6]. These three laws of physics could be expressed by use of a mathematical model which is generally an integral equation or a partial differential equation. The Computational Fluid Dynamics is the art of replacement of integrals or partial differential equation (as case may be) with simple algebraic expressions. Historically, this method was developed in 1960s and 1970s due to the need in air and space industry [3]. But, today, the use of fluid dynamics methods have been became as a norm, including:

Diverse application in Hydropower project's design and analysis [13]; Numerical flow analysis for ogee spillways [12]; Physical and numerical model of flow in labyrant spillway [3]; Volume of fluid model for turbulence numerical simulation of stepped spillway overflow [4]; Modeling entrainment of air at turbulent free surfaces [2].

The commercially available CFD package FLOW-3D uses the finite-volume method to solve the RANS equations. The computational domain is subdivided using cartesian coordinates into a grid of variable-sized hexahedral cells. For each cell, average values of the flow parameters (pressure and velocity) are computed at discrete times using a staggered grid technique. The genaral governing RANS and continuity equations for incompressible flow, including the VOF and FAVOR variables, are outlined in continuity and momentum equation such as:

Continuity:

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

Momentum:

$$\frac{\partial U_i}{\partial t}+\frac{1}{V_F}\left(U_J A_J \frac{\partial U_i}{dx_i}\right)=\frac{1}{\rho}\frac{\partial p}{dx_i}+g_i+f_i$$

The variables u, v and w represent the velocities in the x, y and z directions; V_F= volume fraction of fluid in each cell; A_x, A_y and A_z= fractional areas open to flow in the subscript directions; ρ= density; P is defined as pressure; g_i= gravitational force in the subscript direction; and f_i represents the Reynolds stresses for which a turbulence model is required for closure. The FAVOR method, outlined by Hirt and Sicilian and Hirt, is a porosity technique used to define obstacles [2]. The grid porosity value is zero within obstacles and 1 for cells without the obstacle. Cells only partially filled with an obstacle have a value between zero and 1, based on the percent volume that is solid. Therefore, the siphon spillway surface is defined by cells within the grid that have a porosity value between 1 and zero. The location of the interface in each cell is defined as a first-order approximation - a straight line in two dimensions and a plane in three dimensions, determined by the points where the obstacle intersects the cell faces. In essence, the siphon structure is constructed of a series of short chords that define the spillway curves. Given this fact, it is obvious that smaller size cells produce a much smoother numerical obstacle boundary.

Two networks are used for simulation of numerical model. In first situation for the free surface flow, the spillway was defined as an obstacle which has been located at 1.8m*0.44m network, in flow-3D software. In second condition, we have under pressure flow. Therefore the siphon was defined as the closed conduit for passing the water through it. The left and right boundaries of the control volume are in inlet and outlet of structure respectively. For the purpose of convergence of the calculations and rapidly achieving to the steady condition, the multi grid method was used. In the beginning of computations was chosen relatively large cells in networks ($\Delta x=\Delta y$). In order to the high accuracy in the calculations, the scales of cells were gotten smaller gradually, until the optimum amounts of Δx and Δy were achieved. In the numerical model with free surface, the computations media were divided to 15*28 cells and in under pressure condition, 15*30 cells were used in the network. Control volume in free surface flow consists of four boundaries: atmospheric pressure in above; impermiable layer in downward; hydrostatic head in up-stream or left and continuity equation in down-stream or right.

RESULTS AND DISCUSSION

In this research, the model spillway was tested by 8 different flow ranges. In free surface flow, the discharges were selected on the basis of tests done by savage (2001) to enable us to compare the results. The discharges in pressurized flow have been selected close to each other as far as possible to enable us to investigate the discharge changes more accurate.

The discharge coefficient calculation was also taken place in two free surface and under pressure states. Since the calculated discharge coefficient in physical model was used as basis for CFD, the test was repeated 5 times to make sure of the appropriate accuracy. The calculated discharge coefficient in free state is higher than expected for a ogee spillway and the reason behind it is due to flow a discharge less than design discharge for ogee overflow. The main target of this research is formation of a physical and numerical siphon spillway model and comparison of the results under each environmental condition. Therefore, for the purpose of comparison, the discharge and pressure were calculated on central line of physical and numerical model.

10 nipples were placed on the central axis of the spillway and all connected the read board. The same boundary conditions have been used for construction of numerical model. The CFD models have been designed for 8 flow rate in two dimensional states.

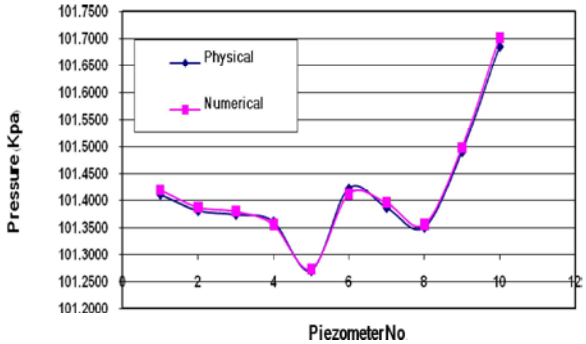

Fig. 2: Comparison of presures in both with free-surface flow (Q = 3.78 L/s)

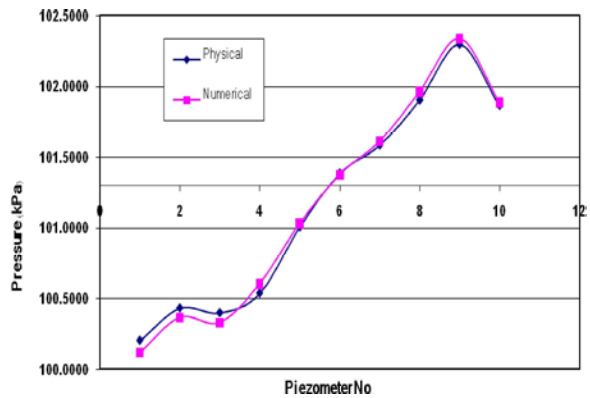

Fig. 3: Comparison of pressures in both models with under-pressure flow (Q=26 L/s)

The pressure reading took place in Three different conditions and the comparison was made for these three sections: A) The pressure in spillway with free surface flow: The calculated pressure of 10 piezometers on the center line of spillway for both model, have been indicated in figure 2. The pressures are decreased from piezometer 1 to 5 (which is due to increase in flow rate) and the lowest measuring is in point 5. In piezometer No. 6, due to contact of flow with deflector, we see pressure increases and by approximation to the fall point and this is due to decrease in flow rate and accumulation of water which results in increase of static pressure. B) Pressure in pressurized spillway with free outlet: Figure 3, shows the changes in pressure along the spillway for 26 L/s discharge. In this condition, the differences between the physical and numerical data of piezometers No. 1 to 4 are higher than other ones and the reason behind it is error in simplifications in CFD. C) Pressure in pressurized spillway with submerged outlet: In this case, the siphon operates with lower discharge rather than the previous state. Figure 4, shows the pressure changes along the spillway for 21 L/s flow rate. The pressure in physical and numerical models along the structure have almost linear

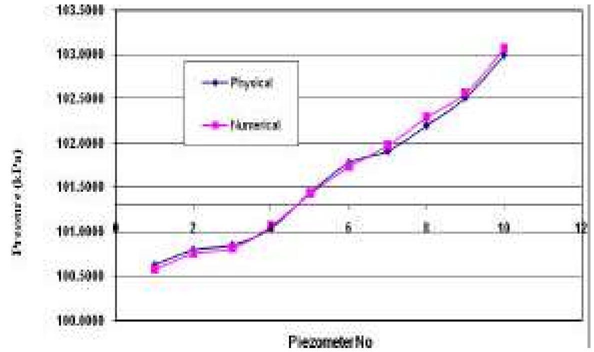

Fig. 4: Comparison of pressures in both models with submerged outlet and under-pressure flow (Q=21 L/s)

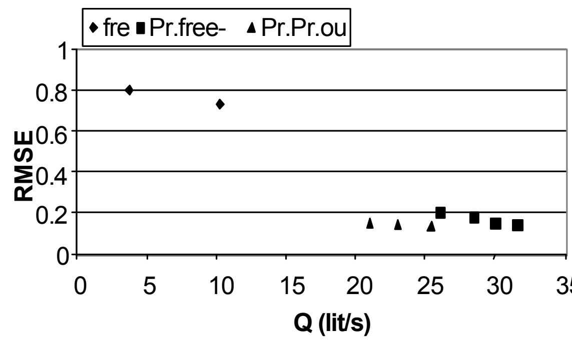

Fig. 5: RMSE of pressures in both models

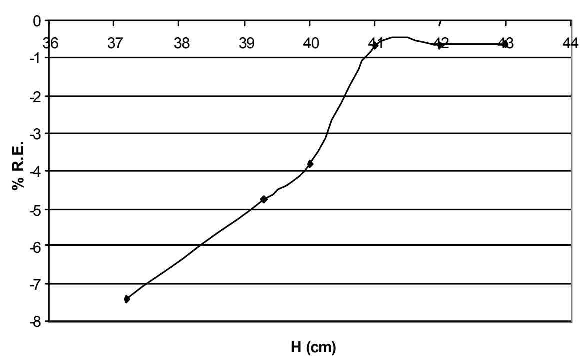

Fig. 6: Comparison of discharge of spillway in both model with free outlet

and ascending trend. The pressures are negative in 4 first piezometers and positive in all other ones. For the purpose of investigation and determination of the CFD model's accuracy, the Root Mean Square Error (RMSE) is used (Figure 5).

For the purpose of comparison, the flow on spillway was classes as two free outlet and submerged outlet forms; for example, for free condition, figure 6, shows the relationship between discharge and resource's head in both models. For the purpose of more accurate criterion for comparison, the relative error percent was used in this section. This parameter is defined as [(Qc-Qm)/Qm]*100,

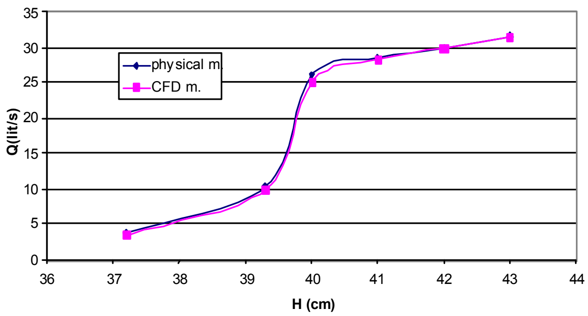

Fig. 7: Relative Error percent of spillway with free outlet

where Qm is the discharge in physical model and Qc denotes the discharge in numerical model. The relative error percentage for spillway with free outlet has been shown in figure 7. The relative error percentage is about 7% for 11 L/s discharge and reduces to 1% by discharge increase i.e. finds a decreasing trend. Therefore, by increasing the discharge, the results of physical and numerical models become closer to each other. This is true for both free outlet and submerged outlet spillway conditions.

The Results of Both Physical and Numerical Models of Siphon Spillway Could Be Summarized as Follows:

- The discharge coefficient in free outlet condition is 3.5. The coefficient increases by increasing flow rate. Therefore, the siphon spillway in free surface flow has less efficiency than ogee spillway.
- The discharge coefficient in under pressure spillway with free outlet is about 0.73 to 0.83. The study conducted by Gramathy (1970) with same condition the coefficient range is between 0.7 up to 0.85 [11]. Therefore, the coefficient trend is same in both tests and the coefficient increases by increase in flow rate.
- The discharge coefficient in pressurized structure with submerged outlet is from 0.71 up to 0.81. For the same condition, Gramathy (1970) has determined a range of 0.69 up to 0.79 which in turn evidences the correctness of the results of tests.
- Comparison of discharge coefficient in pressurized spillway with both free and submerged outlet shows that submerging in outlet is effective in promotion of spillway efficiency and, in other words, the siphonic action is commenced in lower discharge.
- The investigation and assessment of piezometerics pressures show that the 3D effect of the flow is negligible. In free surface flow, we can see water level rise in areas close to side walls, which is due to viscosity effect. The pressure changes along the siphon is sensible, which the results obtained from both physical and numerical models evidences this trend.
- The high effect in numerical model design is related to the blocks in network and in the subsequent steps the solving system and pressure - velocity algorithm, with more than 5 % error, have relatively high effect. The lowest effect is relating to the solving system tolerance and repeat P-V algorithm. We can also point out the error evaluation blocks as well which their changes result in less than 0.1 % error.
- The comparison of discharges coefficient in physical model and 2D and 3D numerical models show relatively similar results - 3.5 for physical model, 3.51 for 2D and 3.52 for 3D.
- The siphon spillway is more advantages of Ogee spillway because of its operation with more discharge with less head in spite of comparison with Ogee.
- This research shows that numerical model can simulate the stream through the siphon spillway. Therefore, flow conditions such as velocity, pressure, etc in this structure can be computed with suitable accuracy.

REFERENCES

1. Davies, B., M. Larson and G. Wright, 2004. Fool Hollow Spillway Rehabilitation Project, J. ASCE.
2. Souders, D.T. and C.W. Hirt, 2004. Modeling Entrainment of Air at Turbulent Free Surfaces, J. ASCE.
3. Savage, Frizell. and Crowder, 2003, Brains Versus Brawn: The Changing World of Hydraulic Model Studies, Mc Neese State University
4. Chen, Q., G. Dai and H. Liu, 2002. Volume of Fluid Model for Turbulence Numerical Simulation of Stepped Spillway Overflow, J. ASCE,
5. Lee, W. and J.A. Hoopes, 1999, Prediction of Cavitation Damage for Spillways, J. Hydr. Engrg., pp: 122.
6. Pagram, G.S. and A.K. Officer, 1999. Hydraulics of Skimming Flow on Modeled Stepped Spillways, Journal of Hydr. Engrg., pp: 125.
7. Head, 1975. Siphon Spillway, Journal of ASCE,
8. Streeter, Wylie and Bedford, 1998. Fluid Mechanics, McGraw-Hill.
9. Vischer, D.L. and W.H. Hager, 1997. DAM HYDRAULICS, John Wiley and Sons. pp: 81-95.
10. Featherstone, R.E. and C. Nalluri, 1995, Civil Engineering Hydraulics, Blackwell Sci.,

11., Gramathy, G. and K. Robinson, 1970. Siphon Spillway, Thesis, California Institute of Technol.,
12. Ho, Boyes, Donohoo and Cooper, 2003. Numerical Flow Analysis for Spillways, 43th ANCOLD Conference, Tasmania, Oct.,
13. Teklemariam, E. and W. Korbaylo, 2002. Computational Fluid Dynamics: Diverse Application in Hydropower project's Design and Analysis, CWRA 55 th Annual Conference- Winnipeg, M.B.,