Applied Science Reports
www.pscipub.com/ASR
E-ISSN: 2310-9440 / P-ISSN: 2311-0139
DOI: 10.15192/PSCP.ASR.2014.10.2.8289

App. Sci. Report.
10 (2), 2014: 82-89
© PSCI Publications

Numerical Modeling Of Flow Pattern In Kamal Saleh Dam Spillway Approach Channel

Amir Moradinejad[1], Abas Parssai[2], Mohamad Noriemamzade[3]

1. PHD student Water Engineering Department, Faculty of Agriculture,Lorestan University, Khorramabad, Lorestan Province,Iran And Master of Institute of Applied Science Department of Agriculture and jehad Ministry
2. PHD student Water Engineering Department, Faculty of Agriculture,Lorestan University,Khorramabad,Lorestan Province,Iran
3. Associate Professor, Water Engineering Department,Faculty of Agriculture, Sharekord University, Sharekord, Charmahal Province,Iran.

Corresponding author email: Amir Moradinejad

Paper Information

Received: 10 March, 2015

Accepted: 7 May, 2015

Published: 20 May, 2015

Citation

Moradinejad A, Parssai A, Noriemamzade M. 2015. Numerical Modeling Of Flow Pattern In Kamal Saleh Dam Spillway Approach Channel. Applied Science Reports, 10 (2), 82-90. Retrieved from www.pscipub.com (DOI:10.15192/PSCP.ASR.2015.10.2.8290)

A B S T R A C T

Analysis of behavior and hydraulic properties of flow over the spillway dam is a complicated phenomenon that takes lots of money and time. To modeling of Hydraulic characteristics several ways as physical modeling and numerical methods has proposed. Today, using the software of Computational Fluid Dynamics (CFD) and modern computers, have developed the usage of numerical methods in the analysis of flow. The software related to CFD is Flow 3D that has the capability to analyze two and three dimensional flow field. In this paper the flow pattern at guide wall of the Kamal-Saleh has been modeled by Flow 3D software. The result shows that the current shape of the left guide wall cased to unsteady flow and making the secondary flow at beginning the weir. This shape of guide wall maybe reduced the performance of weir to remove the pick flood discharge.

© 2015 PSCI Publisher All rights reserved.

Key words: *Approach channel, Kamal-Saleh dam, Guide wall, Spillway, numerical modeling.*

Introduction

Spillways are one of the main structures that used in the Dam projects. Design of spillway in all type of dams, Specifically Earthen dams is more important because inability to remove Probable Maximum Flood (PMF) discharge by spillway may cause to overflow of water on the earthen dam which ultimately leads to destruction the dam(DAS & DAS SAIKIA, 2009; E, 2013; Novak et al., 2007). So study on the hydraulic characteristics of this structure is important. Hydraulic properties of spillway including flow pattern at the entrance of the guide walls and along the chute, Moreover, estimating the values of velocity and pressure parameters of flow during along the chute is very important(Chanson, 2004; Chatila & Tabbara, 2004). The purpose of the study on the flow pattern is the effect of wall geometry to creation the transverse waves, Flow instability, rotating and reciprocating flow through the inlet of spillway and its chute(Wang & Jiang, 2010). The purpose of study on the values of velocity and pressure are calculating the potential of the structure to occurrence the phenomena such as cavitation (Fattor & Bacchiega, 2009; Ma et al., 2010). Sometimes, it can be seen that the spillway design parameters of pressure and velocity is very suitable But geometry is considered is not suitable for conducting walls Causing unstable flow pattern over the spillway, Rotating flows at the beginning the spillway and Its designed cased to reduce the flood discharge capacity(Fattor & Bacchiega, 2009). Study on spillway, usually done with making physical models (Su et al., 2009; Suprapto, 2013; Wang & Chen, 2009; Wang & Jiang, 2010). But recently, with advancing in the field of CFD, study on Hydraulic characteristics of this structure has been done with these techniques (Chatila & Tabbara, 2004; Zhenwei et al., 2012). Using the CFD as a powerful technique for modeling the Hydraulic structures can reduce the time and cost of experiments(Tabbara et al., 2005). In CFD field, the Navier-Stokes equation has solved by powerful numerical methods such as finite element method and finite volumes (Kim & Park, 2005; Zhenwei et al., 2012). In order to obtain closed form Navier-Stokes equations turbulence models, such $k-\varepsilon$ and RNG is presented. To use the technique of computational fluid dynamics software packages such as Fluent and Flow 3D, etc. are provided, Recently, These two software packages widely used in hydraulic engineering because the performance and these accuracy are very suitable(Gessler; Kim, 2007; Kim et al., 2012; Milési & Causse, 2014; Montagna et al., 2011). In this paper, to assessing the flow patter at Kamal-Saleh guide wall the numerical method has been used. All the stage of numerical modeling was conducted in the Flow 3D software. Kamal -Saleh Dam basin has an area of 655 square kilometers. Is located in South West Markazi Province a west of Iran. Figure 1 shows the picture of this structure.

Figure 1. Kamal Saleh guide walls and Spillway

Materials and Methods

Firstly a three dimensional model has constructed according to two dimensional map that prepared for designing of spillway Then a small model was prepared with scale of 1:80 and entered the Flow 3D software, all stage of model construction was conducted in AutoCAD 3D. Flow 3D software solved numerically The Navier-Stokes equation in both compressible and incompressible modes by finite volume method. Below is a brief references on the equations that used in the software.

Review of the governing equations in software Flow3D

Continuity equation at three dimensional Cartesian coordinates given as Eq(1).

$$v_f \frac{\partial \rho}{\partial t}+\frac{\partial}{\partial x}(uA_x)+\frac{\partial}{\partial x}(vA_y)+\frac{\partial}{\partial x}(wA_z)=\frac{PSOR}{\rho} \quad (1)$$

Where u, v, z are Velocity component in the x, y, z direction. A_x, A_y, A_z Cross Sectional area of the flow, ρ Fluid density, $PSOR$ the source term, v_f is the volume fraction of the fluid and Three-dimensional momentum equations given in the Eq (2).

$$\begin{aligned}
&\frac{\partial u}{\partial t}+\frac{1}{v_f}\left(uA_x\frac{\partial u}{\partial x}+vA_y\frac{\partial u}{\partial y}+wA_z\frac{\partial u}{\partial z}\right)=-\frac{1}{\rho}\frac{\partial P}{\partial x}+G_x+f_x\\
&\frac{\partial v}{\partial t}+\frac{1}{v_f}\left(uA_x\frac{\partial v}{\partial x}+vA_y\frac{\partial v}{\partial y}+wA_z\frac{\partial v}{\partial z}\right)=-\frac{1}{\rho}\frac{\partial P}{\partial y}+G_y+f_y \quad (2)\\
&\frac{\partial w}{\partial t}+\frac{1}{v_f}\left(uA_x\frac{\partial w}{\partial x}+vA_y\frac{\partial w}{\partial y}+wA_z\frac{\partial w}{\partial z}\right)=-\frac{1}{\rho}\frac{\partial P}{\partial y}+G_z+f_z
\end{aligned}$$

Where, P is the fluid pressure, G_x, G_y, G_z The Acceleration created by body fluids, f_x, f_y, f_z Viscosity acceleration in three dimensions and v_f is related to the volume of fluid, defined that by the equation (3). For modeling of free surface profile the VOF Technique based on the volume fraction of the computational cells has been used. Since the volume fraction F represents the amount of fluid in each cell, takes Value between 0 and 1.

$$\frac{\partial F}{\partial t}+\frac{1}{v_f}\left[\frac{\partial}{\partial x}\left(FA_x u\right)+\frac{\partial}{\partial y}\left(FA_y v\right)+\frac{\partial}{\partial y}\left(FA_z w\right)\right]=0 \tag{3}$$

Turbulence models

Flow 3D offers 5 types of Turbulence models: Prantl mixing length, K-ε equation, RNG models, Large eddy simulation model. the simplest model would be a transform equation for the kinetic energy along with fluctuation of turbulence velocity and another parameter which defines some other characteristics of turbulence. Turbulence models have been proposed recently is based on Reynolds-averaged Navier–Stokes equations. These approaches involves statistical methods to extract an averaged equation related to the turbulence quantities .

Steps of solving a problem in Flow3D software

Preparing the 3D model by AutoCAD software. 2. Defining the problem in the software and check the final mesh.3.choosing the basic equations that should be solved. 4. Defining the characteristics of fluid. 5. Defining the boundary conditions. 6. Initialization of the flow field. 7. Adjusting the output. 8. Adjusting the control parameters, choice of the calculation method and solution formula. 9 .start of calculation. Defining of Boundary Conditions(BC) are important stage of simulation with Flow 3D software. This software has a wide range of boundary conditions.

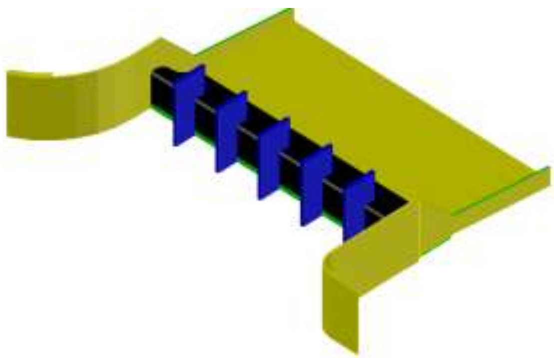

Figure 2. Three-dimensional shape of the, Kamal Saleh spillway

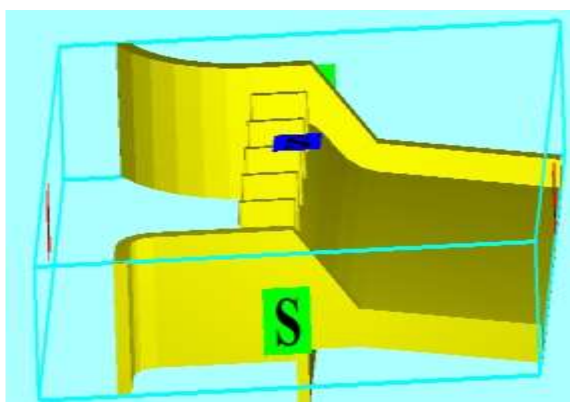

Figure 3. meshing the geometry and boundary conditions of the model Flow 3D

Results and discussion

The behavior of water in spillway is strongly affected by the flow pattern at the entrance of the spillway, the flow pattern formation at the entrance is affected by the guide wall, and choice of an optimized form for the guide wall has a great effect on rising the ability of spillway to easily the PMF, so any turbulence in approach channel can cause turbulence in the spillway, flow reduction, and even probability of cavitation. Optimize the flow guiding walls (in terms of length, angle and radius) can cause the loss of turbulence and flow disturbances on spillway. For this purpose, initially proposed for model for the discharge of spillway dam, Kamal Saleh, 80, 100, 120 liters per second were surveyed. The reform proposals are presented. Geometric properties of the conducting Guidance wall are given in the following table.

Table 1. Characteristics and dimensions of the guidance walls tested

Case	Direct Length(m)	Radius of curvature(m)	Cobb angles(Degree)
1	0.200	0.325	90
2	0.375	0.175	120
3	0.100	0.400	110

The pattern of flow in the guidance wall

According to (4), (5) and (6) and the observed streamline the right wall is suitable performance but the left wall was not suitable performance and the left wall of the geometric design creates a secondary flow in the first part of the wall guidance direct input to spillway. The causes of transverse waves at beginning of spillway. By increasing the flow rate at the inlet spillway Rotational flows gone But the lines can cause severe distortion. The results of the software implementation to increase the flow in the figures (7) to (9) and (11) to (14) are given. As the forms (8) and (9) is observed The right wall is good But the left wall is not suitable And perversion flow lines are clearly marked. Transverse waves produced severe curvature of the flow lines in Figure (14) and (10) are given. After checking the suitability of the proposed project and the right wall, It has been suggested that the geometry of the left wall, right wall should be like. Three-dimensional model was built by AutoCAD software spillway again Flow 3D software was inside, The software was Ron for spillway modified. The results of the software implementation for maximum flow and Modified wall The figures (18) and (19) are shown. Modifying effect on the geometry of the left wall of the spillway inflow patterns were studied. According to the figures (18) and (19) Correction the left wall Cause the current guidance And Elimination of a severe curvature of the flow lines. Eventually Makes it easy to cross the stream and Transverse waves are eliminated. This geometry increases the efficiency of the spillway when design flood will occur.

Conclusions and recommendations

the right wall is good But the left wall was not good and This geometric design creates a Rotating flow At the beginning of spillway and The transverse waves are caused.

increasing the flow rate at the inlet weir on the left wall of waves and Rotational flows gone, But the lines can cause severe distortion.

Modify the left wall, causing the current guidance and Severe curvature of the flow lines is eliminated and Eventually lead to easy passage and eliminates the cross waves is.

large radius and angle of the curve, causing widening of the canal wall approach (Approach-Channel) and to reduce the influence of the wall is conducted.

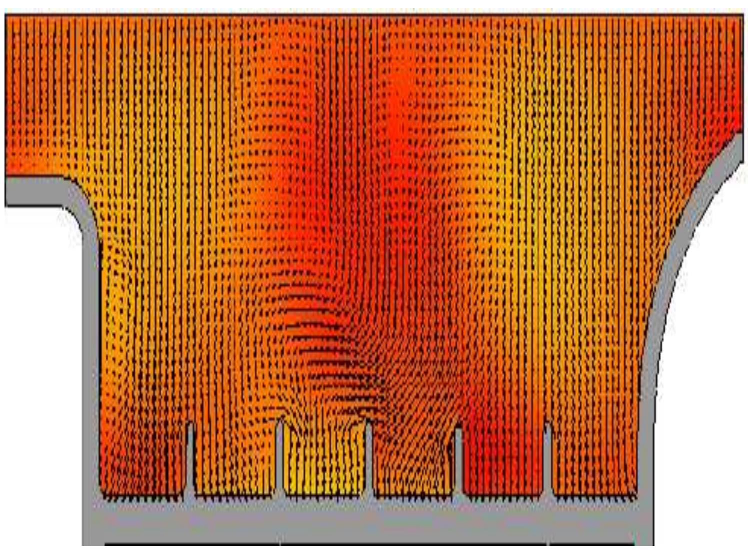

Figure 4. in the model proposed for the flow of 80 liters per second

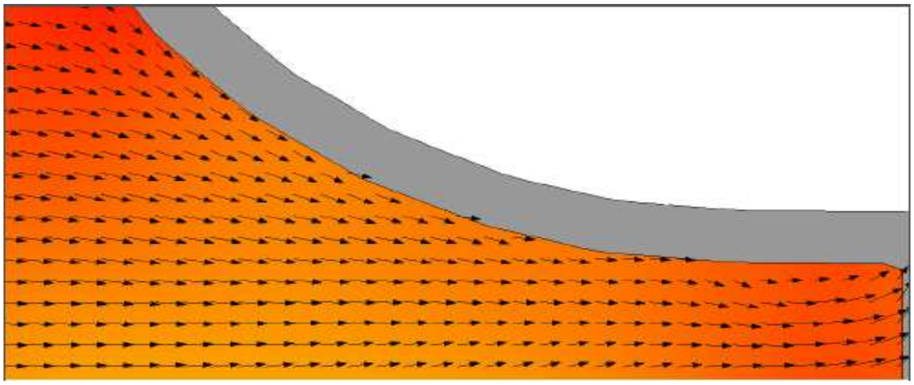

Figure 5. flow pattern down the right wall

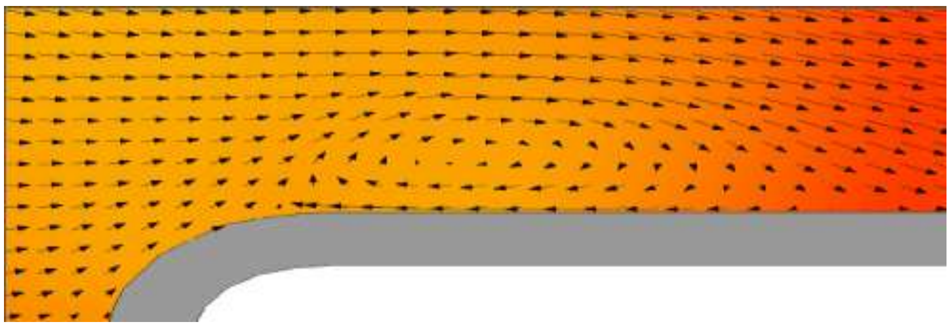

Figure6. the pattern of flow through the side wall of the left

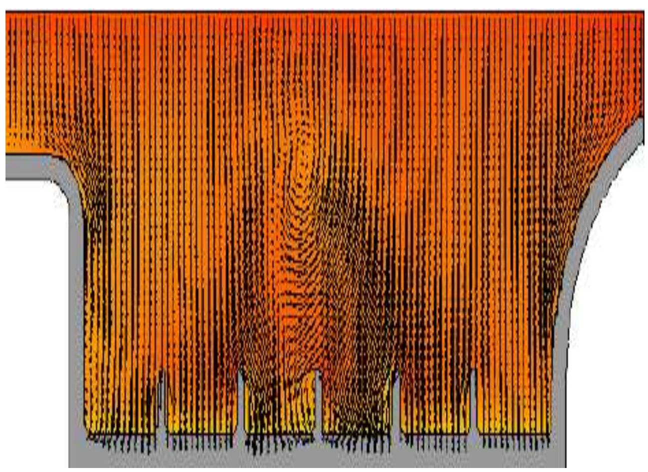

Figure 7. proposed to model the flow rate of 100 liters per second

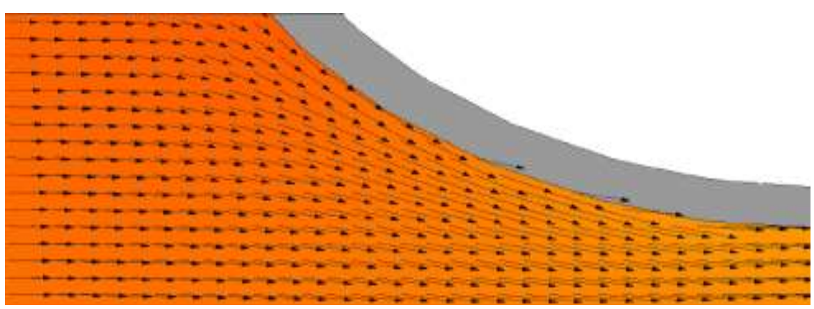

Figure 8. flow pattern down the right wall

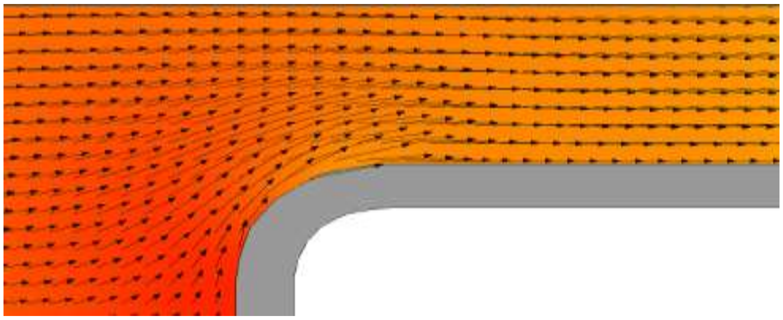

Figure 9. pattern of flow through the side wall of the left

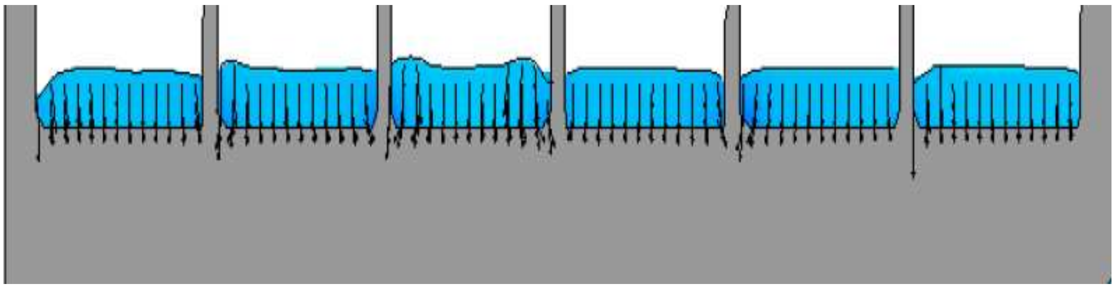

Figure 10. two-dimensional flow pattern on the spillway crest

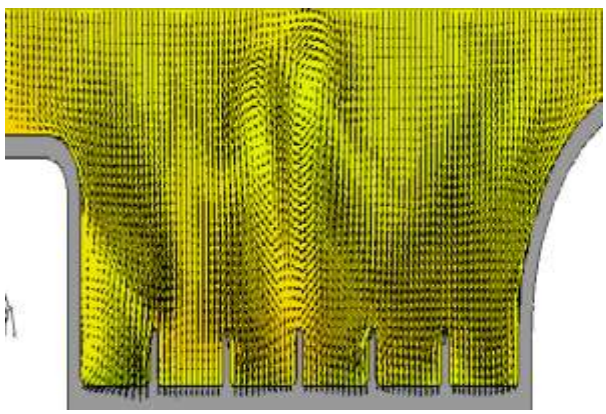

Figure 11. presents the flow pattern in the design flow of 120 liters per second

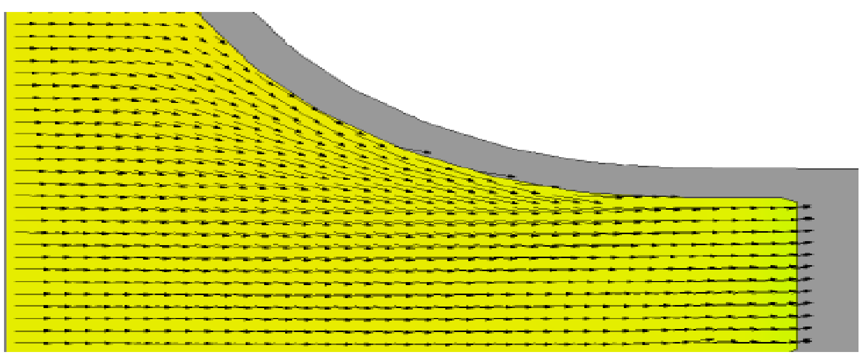

Figure 12. shows the flow pattern near the wall in right direction

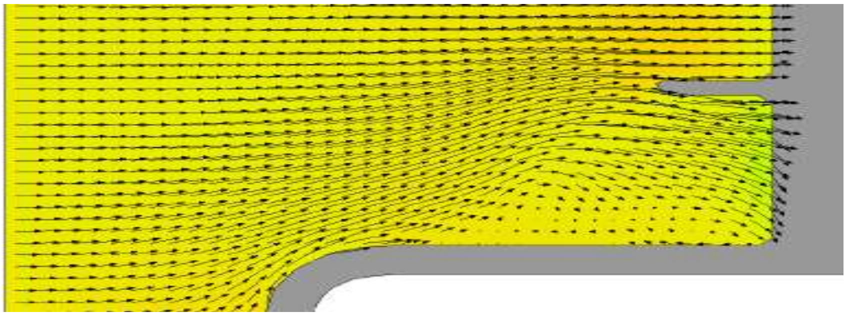

Figure 13. to guide the flow pattern near the left wall.

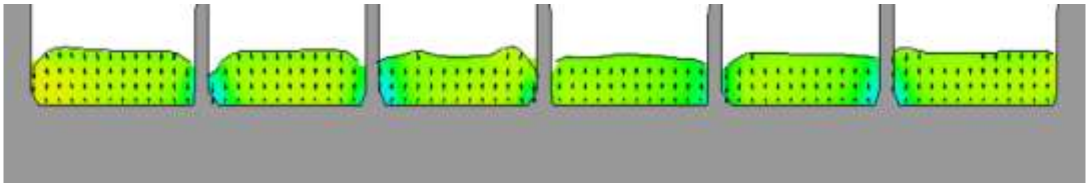

Figure 14. two-dimensional flow pattern on the wall of the spillway crest guidance

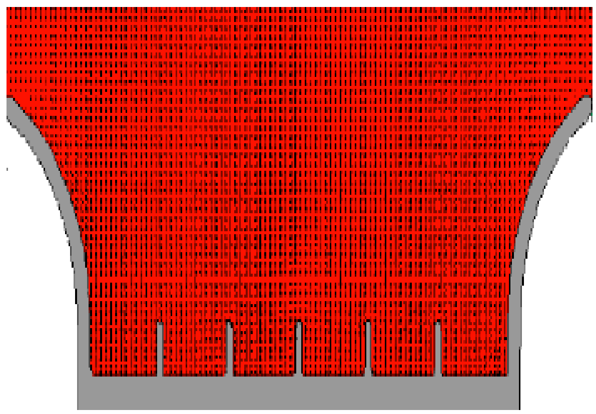

Figure 15. proposed a model for the flow rate of 120 liters per second

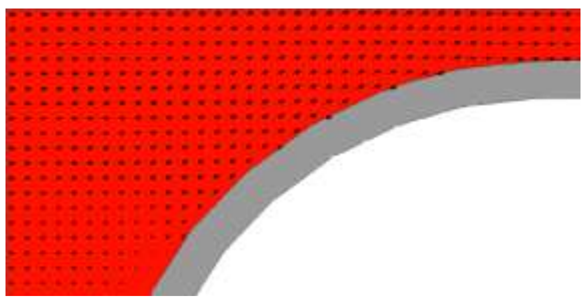

Figure 16. (16) to guide the flow pattern near the wall.

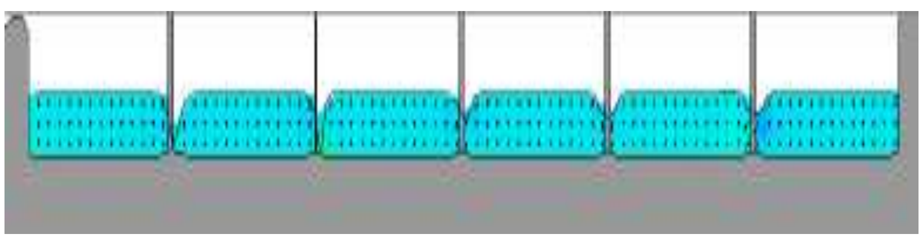

Figure 17. two-dimensional flow pattern on the wall of the spillway crest

References

Abbasi S, Kamanbedast A.2012. Investigation of Effect of Changes in Dimension and Hydraulic of Stepped Spillways for Maximization Energy Dissipation. World Applied Sciences Journal, 18(2), 261-267.

Chanel PG, Doering JC. 2007. “An Evaluation of Computational Fluid Dynamics for Spillway Modelling “.16th Australasian Fluid Mechanics Conference :1201-1206.

Daneshfaraz R, Zogi N. 2013. Investigation ofCavitation in Stepped Spillway of Siah-Bishe Dam by Using Flow-3d Model. International Research Journal of Applied and Basic Sciences, 4(11), 3382-3388.

Flow Science, Inc. Flow-3D User’s Manuals, Version 9.3, Santa Fe, NM. 2007.

Henderson FM.1966.Open channel flow. Macmillan.

Hirt CW. “ Modeling Turbulent Entrainment of Air at a Free Surface “. Flow Science, Inc . FSI-03-TN61.

Ho D, Boyes K, Donohoo S, Cooper B.2003.” Numerical Flow Analyses for Spillways “ . 43rd ANCOLD Conference Hobart, Tasmania, 24-29 October 2003 .

Ho D, Boyes K, Donohoo S.2001.” Investigation of Spillway Behaviour under Increased Maximum Flood by Computational Fluid Dynamics Technique “.14th Australasian Fluid Mechanics Conference, Adelaide University, Adelaide, Australia.

Ikegawa M, Washizu K. 2005. " Finite element method applied to analysis of flow over a spillway crest". International Journal for Numerical Methods in Engineering,Volume 6, Issue 2 , Pages 179 – 189.

Kim S, Lee H, An S.2010. “ Improvement of hydraulic stability for spillway using CFD model “ . J. of the Physical Sciences Vol. 5(6), pp. 774-780.

Kim SD, Lee HJ, An SD.2010. Improvement of hydraulic stability for spillway using CFD model. International Journal of the Physical Sciences, 5(6), 774-780.

Nikseresht AH, Talebbeydokhti N, Rezaei MJ.2013. "Numerical simulation of two-phase flow on step-pool spillways." Scientia Iranica, 20(2), 222-230.

Novak P, Moffat A, Nalluri C.2007. Hydraulic Structures. USA and Canada: Taylor & Francis.

Sabbagh-Yazdi SA, Rostami F.2007. " Numerical Investigation of Pressure Load over Spillway Chutes with Vertical Curvatures ". 5th IASME / WSEAS International Conference on Fluid Mechanics and Aerodynamics, Athens, Greece, August 25-27, 2007.

Savage BM, Johnson MC. 2006.” Physical and Numerical Comparison of Flow over Ogee Spillway in the Presence of Tailwater “ . J Of Hydraulic Engrg, ASCE , December 2006 : 1353- 1357.

Savage BM, Johnson MC.2001.” Flow over Ogee Spillway: Physical and Numerical model case study ” . J Of Hydraulic Engrg, ASCE , Vol. 127, No. 8 : 640-649.

Song CG, Seo W. 2008. " Analysis of Discharge of Dam Spillway according to weir shape " . Seoul National University, Seoul, Korea.

Wang J, Chen H.2009. "Experimental Study of Elimination of Vortices Along Guide Wall of Bank Spillway." Advances in Water Resources and Hydraulic Engineering, Springer Berlin Heidelberg, 2059-2063.

Yoo S, Hong K, Hwang M.2002. “ A 3-dimensional numerical study of flow patterns around a multipurpose dam” . Fifth International Conference on Hydroinformatics, Cardiff, UK

Zhenwei MU, Zhiyan Z, Tao Z.2012. "Numerical Simulation of 3-D Flow Field of Spillway based on VOF Method." Procedia Engineering, 28(0), 808-812.