Alexandria Engineering Journal (2016) 55, 467–473

HOSTED BY

ELSEVIER

Alexandria University

Alexandria Engineering Journal

www.elsevier.com/locate/aej
www.sciencedirect.com

ORIGINAL ARTICLE

Numerical modeling of flow pattern in dam spillway's guide wall. Case study: Balaroud dam, Iran

CrossMark

Sadegh Dehdar-behbahani [a,*], **Abbas Parsaie** [b]

[a] *Shahid Chamran University, Ahvaz, Iran*
[b] *Department of Water Engineering, Lorestan University, Khorramābād, Iran*

Received 2 June 2015; revised 11 January 2016; accepted 19 January 2016
Available online 3 February 2016

KEYWORDS

Balaroud dam;
Cross waves;
Optimal form of structure;
Guide wall;
Approach channel

Abstract Flow pattern recognition using the physical modeling and numerical simulation helps designers to propose optimal shape for hydraulics structures. Optimal shape of hydraulic structure causes increasing their performance. In this study, the effect of geometries of guide walls on the flow pattern and rating curve of Balaroud dam spillway's (Iran) was simulated using numerical and physical simulation. Numerical simulation of flow pattern through the guide walls was carried out using Flow 3D software as computational fluid dynamic (CFD) tool. To this task, three plans for the guide walls were evaluated. The results showed that the plan (3) has best performance for removing cross waves and smoothly passing the flow through the approach channel and over the spillway. Evaluating the turbulence models showed that the RNG-K-epsilon becomes the best performance to show the cross waves.

1. Introduction

Study on the flow through the hydraulic structures usually is conducted using physical modeling. Physical modeling is based on the basic fluid mechanic equations. Physical modeling of hydraulic structures means that a scaled laboratory model from the prototype is constructed. This approach is a safe way to analyze the flow through or over the hydraulic structures [9,12,25]. Due to high cost of laboratory experiments, researchers have attempted to use numerical simulation along the physical modeling. In the field of numerical simulation, the governing equations of the flow which are usually Navier–Stokes equations together with turbulence models are solved with powerful numerical methods such as finite volume and finite elements [17,18,19,20,26]. Several turbulence models have been proposed for solving the turbulent flow. In this regard the K-epsilon, RNG K-epsilon and large eddy simulation (LES) are the advanced turbulence models that recently have been widely used in the hydraulic engineering [4,5,15,22,23,26]. Several commercial software and open source codes have been proposed for comfortable use of the numerical modeling in the hydraulic engineering studies. In this regard, the Flow 3D and Fluent as commercial software and Open-FOAM as free access code can be mentioned [16]. Numerical modeling of turbulent flow

* Corresponding author.
E-mail addresses: sadegh.dehdar@gmail.com (S. Dehdar-behbahani), Abbas_Parsaie@yahoo.com (A. Parsaie).

Peer review under responsibility of Faculty of Engineering, Alexandria University.

http://dx.doi.org/10.1016/j.aej.2016.01.006
1110-0168 © 2016 Faculty of Engineering, Alexandria University. Production and hosting by Elsevier B.V.

over hydraulic structures helps to define flow properties including velocity and pressure distribution through the structure. In addition to define hydraulic characteristics, numerical models have high ability to show the flow pattern which some time is important as flow properties [21,24]. Defining the flow pattern through the spillway's guide wall is categorized as this subject and guide wall shape has meaningful effect on the flow pattern [1,2,6,7,8,10]. There are few articles about flow pattern in the guide wall but in this regard, Wang and Chen [27,28] studied the spillway of Yutang dam to remove vortex flow and separation on the guide wall. They proposed that the guide wall should be redesigned and present a new shape for the guide wall to elimination remove vortex front the gate. Parsaie et al. [21] numerically studied the effect of geometry of guide walls of Kamal-Saleh (Iran) on the flow pattern passing through the approach channel and over the ogee weir. They stated that the geometry of the right side wall causes cross wave creation through the spillway specifically at the entrance of approach channel. In this paper, physical modeling and numerical simulation of flow through the guide wall and approach channel of the Balaroud dam spillway are considered, whereas the main focus of this paper was on the numerical simulation. To this purpose, the Flow 3D as CFD tool is used. The aim of this paper was finding an optimal shape for the guide wall which removes the cross waves in the maximum flood design. Successfully using the Flow 3D for modeling the hydraulic properties has been reported in the several studies such as souring, waving, cavitation, and energy dissipation [3,11,13]. Using this software for modeling the flow characteristics on the board crest weir was applied by Maghsoodi et al. [14] and they stated that this package has high performance for modeling the flow over this type of weirs.

2. Material and methods

Computational fluid dynamic (CFD) is an advanced numerical approach which is used along the physical modeling for modeling the hydraulic phenomena. Using the CFD tools for modeling the flow through the hydraulic structures needs to define 2D or 3D geometry of structure. Because this paper focuses on the flow pattern through approach channel, the 3D computer model of the Balaroud dam spillway was planned in the AutoCAD software (Fig. 1).

The Balaroud dam is an earth dam located at the west south of Iran (Khozestan Province). Fig. 2 shows the scaled laboratory model which was constructed for the study of the flow properties through the spillway. To assess the geometry of guide wall on the flow pattern, three plans were proposed for the guide wall of approach channel. Figs. 3–5 show the computer models and their geometrical properties which were proposed for plans of the guide wall. In this figures the radius of guide wall and length of approach channel are shown. The scale of constructed laboratory model is equal to 1:110 and the plexiglass was used as material for constructing the flume.

CFD modeling by Flow 3D includes few steps which are explained as follows: preparing the 3D computer model by AutoCAD software and exporting the suitable format (stl format) for uploading in the Flow 3D, Summoning the STL file from the Flow 3D, meshing the computational domain, defining the flow characteristics, boundary and initial condition and then running the software. Figs. 6 and 7 show the Balaroud dam spillway in the Flow 3D software. Fig. 6 shows the mashing process and Fig. 7 shows the boundary conditions. In Fig. 6, W shows the wall, S is symmetry and O is outflow boundary conditions. The Discharge of flow is considered for upstream boundary condition.

3. Overview on governing equation on the Flow 3D

The continuity equation at three dimensional cartesian coordinates is given as in Eq. (1).

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

where u, v, z are velocity component in the x, y, z direction. A_x, A_y, A_z are cross-sectional area of the flow, ρ is the fluid density, $PSOR$ is the source term, v_f is the volume fraction of the fluid and three-dimensional momentum equations are given in the Eq. (2).

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

where P is the fluid pressure, G_x, G_y, G_z are the acceleration created by body fluids, f_x, f_y, f_z are viscosity acceleration in

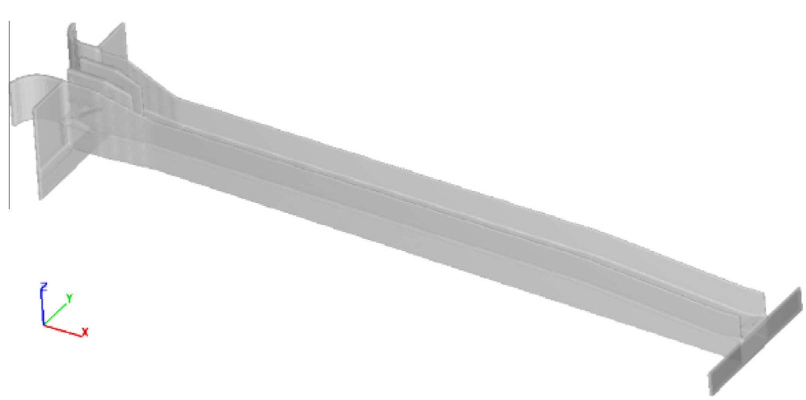

Figure 1 Sketch of 3D model of spillway of Balaroud dam.