Flow Pattern Simulation in Intake of Ilam Reservoir Dam Using Flow3D

Mehrdad Safarian and Ali Afrous

1Department of Water Engineering, Shoushtar Branch, Islamic Azad University, Shoushtar, Iran.
2Department of Engineering, Andimeshk Branch, Islamic Azad University, Andimeshk, Iran.

Abstract: Characteristics and behavior of flows through river are complex phenomenon that study of it sometimes has so much cost. The study of the phenomena of the physical and mathematical models (one-dimensional, two-dimensional, and three-dimensional) and the experience of experts is used. In this study is used FLOW3D model to study the flow pattern in the rear intake of Ilam dam. This dam plays a crucial role in supplying water to downstream areas as well. The results indicate that the flow stream lines have been towards to the western intake from the beginning, so that the construction of dams and reduce the flow velocity; the sedimentation trend of the middle stream service has increased. Efficiency of the dewatering should be minimized from the intake and corrective actions to be taken to resolve existing problems.

Introduction

To understand the current situation and potential morphological study of natural variation or river engineering projects including plants, water diversion and impoundment of the river is inevitable.

Yıldırım conducted investigations for the critical submergence in a rectangular intake. In the study, theoretical model setup was chosen as Rankine half body, which can be obtained by superposing line sink and uniform flow [1]. Aghamajidi et al. simulated the Flow in T-Shaped Elbow Intake Station Using Flow-3D Result showed that in case of no elbow channel, turbulence energy variations from the beginning of intake’s opening until its end was increased about 60% and turbulence energy dissipation was also increased by 31% from the beginning of intake’s opening (in the river) till its end [2]. Kuroush Vahid et al. discussed experimental formation of water surface profiles in a novel intake method by bottom intake with porous environment and showed that flow profile uniformity in low flow is less that uniformity in high flow. In case of low flow due to low velocity on porous surface, more flow is passed through [3].

Esmaeily modeled experimentally and numerically flow pattern of cylindrical spillway and compared it by using Fluent Software. The results of this research showed a good coordination between experimental and fluent results in flow pattern of spillway [4]. Tukur et al. examined the rate of sedimentation in the conveyance canals of Kano River Irrigation Project. In this research, five canals were sampled from the total number of canals using systematic sampling technique [5]. Since Ilam dam, the main source of drinking water in Ilam and surrounding villages in this paper, three-dimensional mathematical model of FLOW-3D is used to investigate flow pattern in Ilam dam.

Materials and Methods

Geographic location: Ilam dam is located at a distance of 22 km south of Ilam (in Iran) downstream of the confluence of two rivers Dehbaraftab and Chavyz been constructed.

Properties of the dam:

- Dam type: rock fills with clay core
- Operated elevation: 960 m above sea level
- Dam crest elevation: 965 m above sea level.
- Crown Length: 162 meters
- Foundation height: of 65 m

Corresponding Author: Ali Afrous, Department of Water Engineering, Andimeshk Branch, Islamic Azad University, Andimeshk, Iran.
E-mail: ali.afrous@gmail.com.
Foundation width: 250 m
Dam crest width: 10 m
Intakes of this dam are located in 923 and 940 meter from sea level.

**Fig. 1:** Scheme of Ilam Dam.

*Description of FLOW-3D:*

FLOW-3D’s unique strengths in modeling 3D free-surface flows make it a natural platform for advanced water and environmental applications. The software has been used for decades by industry professionals to predict complex flow dynamics, flow/structure interactions, and environmental fate and transport. Applications of FLOW-3D include municipal, river planning and restoration, dam safety, hydroelectric and spillway operations, design, and optimization, reservoir maintenance and planning, and coastal and estuary engineering [6]. FLOW-3D addresses a wide range of design problems in hydraulic engineering.

Users can increase the capacity of existing infrastructure in hydropower plants, develop novel approaches to fish passages, design intake that minimize head loss, develop improved for bay designs and tailrace flows, analyze scour and deposition and evaluate air entrainment.

*Geometry of model:*

To model the desired overflow, simultaneous Flow3D and AutoCad software is used. Since the software Flow3D ability to open files with the extension Cad AutoCad does, so that the three-dimensional structure is drawn in AutoCad software using the Export command to convert the Stl format. Then in Section Meshing and Geometry software Flow3D button, and then click on the desired file earlier AutoCad software Stl file is created, selected and recalled.

In software, there’s a section titled Physics in the area, the force of gravity, the viscosity of the flow and turbulence model is determined. Air Entrainment by the density of air entering the first option in this case, its value is equal to 0.0001225. Then, enter the options gravity, in which case its value is set to 9.8106. Then, enter Viscosity and Turbulence and in part of Viscosity Option because water is a Newtonian fluid selected this option. After that it must be selected from the Turbulence model Option [7].

**Fig. 2-a:** Modeling in Flow3D.

**Fig. 2-b:** Modeling in Flow3D.

**RESULTS AND DISCUSSION**

Based on the information and conditions and according to the boundary conditions that mentioned previously, as it can be seen in Figures 4 to 6, for the three discharge rates of 100, 200 and 280 CMS stream flow lines as follows (figures 3-a to 3-c):
As it can be seen in the figures, the stream lines have fairly regular arrangement relative to each other and oriented in the stream lines to the overflow of the intake. With increasing the discharge rate, stream flow lines turmoil is more in the behind of weir, about the discharge rates of 200 and 280 CMS stream flow lines closer to each other and their density is higher.

Longitudinal velocity changes:
The variation of the longitudinal velocity in discharge of 200 CMS (Figure 4-a), a tendency of flow lines toward weir are shown on the left and on the right, low velocities amounts can be observed, that the low velocity indicating for sedimentation potential in this area. With increasing in discharge rate, creating eddies around the intake can be seen in Figures 4-b and 4-c.

Results of pattern flow simulation of river show that sedimentation potential around the intake was predictable from the beginning of operation and with decreasing of discharge this trend will be exacerbated. The results show that the velocity in the middle is high and close to the weir, flow rate decreased and finally reached the edge of the dam is less than 1 m/s.

Changes of vertical and lateral velocity components:
Study of the lateral velocity component, in terms of their impact is important on the conduct of flow to side of intake. Into components along the axis (y) coordinates is called lateral velocity component that are divided into two categories the positive and negative velocities. The lateral component of the velocity, the more negative closer to the intake, there is a negative component indicates that the component is erodible.

As it can be seen in the figures, in discharge rate of 280 CMS lateral velocity at the edge of the weir is positive and negative in bottom of dam weir, which would indicate a potential for sedimentation in the intake.
Comparison the chart of 100 and 280 CMS discharges we see that by increasing the discharge vectors with negative values for the lateral velocity increases and reduced the potential for sedimentation. In Figure 5-b plan of lateral velocity change to the flow of 280 CMS showed.

Fig. 4-a: Stream lines toward weir in Q 200 cms.

Fig. 4-b: Longitudinal velocity changes in Q 200 cms.

Fig. 4-c: Section of longitudinal velocity changes in Q 200 cms.

Fig. 5-a: Lateral velocity changes in 100 cms Q.
Fig. 5-b: Plan of lateral velocity changes in 280 cms Q in 200 cms Q. Change of kinetic energy of turbulence (k) and the turbulence energy loss (ε)

For the study of turbulence flow in FLOW3D used k- ε model that it can demonstrate high changes of discharge rates. As can be seen in Figure 12, the amount of kinetic energy along the river and the lake is low, indicating a high potential for fouling the path of the river.

Figure 13 is related to a section near the intake shows that changes the kinetic energy is in the range of 0.0001 to 0.0258 that represents the kinetic energy is very low and close to the dam, where the energy changes into kinetic energy decreases to be less than 0.0044.

Fig. 6-a: Changes of kinetic energy in Q 100 cms.

Fig. 6-b: Plan of kinetic energy in Q 100 cms.

REFERENCES


