Advanced Engineering Days

aed.mersin.edu.tr

A study on the flow velocities and energy dissipation potential of flow separator placed in spillway flip bucket

Ali Emre Ulu*[1], Mehmet Cihan Aydın[2], Ercan Işık[3]

[1]Bitlis Eren University, Engineering and Architecture Faculty, Department of Civil Engineering, Bitlis, Turkey,
aliemreulu@gmail.com; mcaydin@gmail.com; ercanbitliseren@gmail.com

Cite this study: Ulu, A. E., Aydin, M. C., & Isık, E. (2021). A study on the flow velocities and energy dissipation potential of flow separator placed in spillway flip bucket. 1st Advanced Engineering Days, 38-40

Keywords
Spillway
CFD
Energy dissipater
Flow separator

ABSTRACT

The design of the energy dissipating pools at the downstream of the spillway is an important issue in terms of controlling the downstream flow regime, jet velocities and the scour problem that may occur in this region. In this context, there are many designs for flip-bucket pools for both energy dissipation and minimizing scour problems. In this study, the energy dissipation performance of a flow separator structure placed in an energy breaker pool was investigated numerically by increasing the shear stresses. The results obtained were interpreted and suggestions were made for future studies.

Introduction

Spillways are water structures that are widely used in hydraulic engineering. These structures are used to safely transfer the excess water accumulated in the dam reservoirs to the downstream part. In cases where the spillway design is not done correctly, at the downstream some problems may occur due to scouring, cavitation and energy breakdown at the downstream of the spillway and regulator [1-2]. There are a number of experimental and numerical studies in the literature to dissipate this energy of the high-speed flow that occurs just downstream of the spillway structures [3-7]. In this study, velocities, jets form and the energy dissipation potential of a flow separator structure placed in the flip bucket of a 2D Ogee-style spillway structure was investigated numerically. Possible scour in the downstream region was not taken into account in the study.

Material and Method

FLOW-3D, a powerful computational fluid dynamics (CFD) program, was used as a method in the study. Flow-3D is a widely used reliable method that can simulate fluid motions [8]. The program determines fluid motion by solving equations such as conservation of mass and momentum. The mass conservation equation used in Flow-3D is presented below [9].

$$V_F\frac{\partial\rho}{\partial t}+\frac{\partial}{\partial x}(\rho uA_x)+R\frac{\partial}{\partial y}(\rho vA_y)+\frac{\partial}{\partial z}(\rho wA_z)+\xi\frac{\rho uA_x}{x}=R_{\mathrm{DIF}}+R_{\mathrm{SOR}} \quad (1)$$

Here; V_f: Volume ratio of the fluid, ρ: Density of the fluid, R_{dif}: Turbulent diffusion term, R_{sor}: Mass source, u, v, w are the velocity components in the coordinate directions (x, y, z).

Numerical modeling and analysis

In the study, a two-dimensional ogee type spillway with a crest elevation of 5.28 m and a flip-bucket diameter of 2.0 m was used. In the study, hexahedral 0.05 m cell size was used. Total mesh count is calculated as 130,000 (Figure 1). Analyzes were continued until the flow became completely stable (120 seconds). Analyzes performed using the Renormalized Group (RNG) turbulence model were solved by giving fluid elevation. As the fluid elevation, the water level started from 5.38 m and the fluid elevation was increased by 0.1 m to 6.48 m. Since the analyzes were carried out in two dimensions, the channel boundary conditions were determined as Symmetry. The channel floor boundary condition has been assigned as Wall, and the inlet and outlet parts have been assigned as Specified Pressure. 1 m downstream water inlet is defined in order to provide downstream conditions and to observe hydraulic jump.

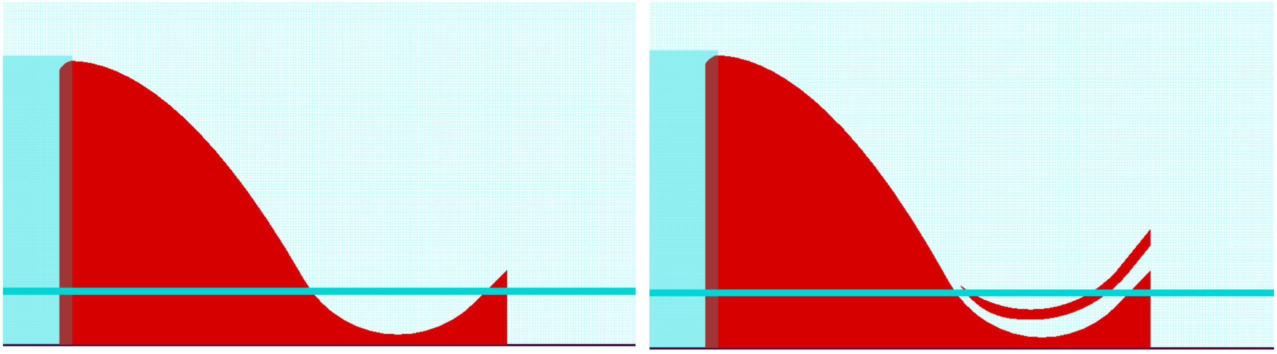

Figure 1. Mesh structures of the models

Results

In the study, the performance of a flow separator structure placed to increase the energy breaking potential of the spillway flip bucket at different upstream water loads was investigated in the numerical environment. From the results obtained, velocity contours with H = 5.68 m and H = 6.38 m water height, starting from the bottom of the spillway, are given in Figure 2. The head loads corresponding to these heights are H_o = 0.40 m and H_o = 1.10 m, respectively.

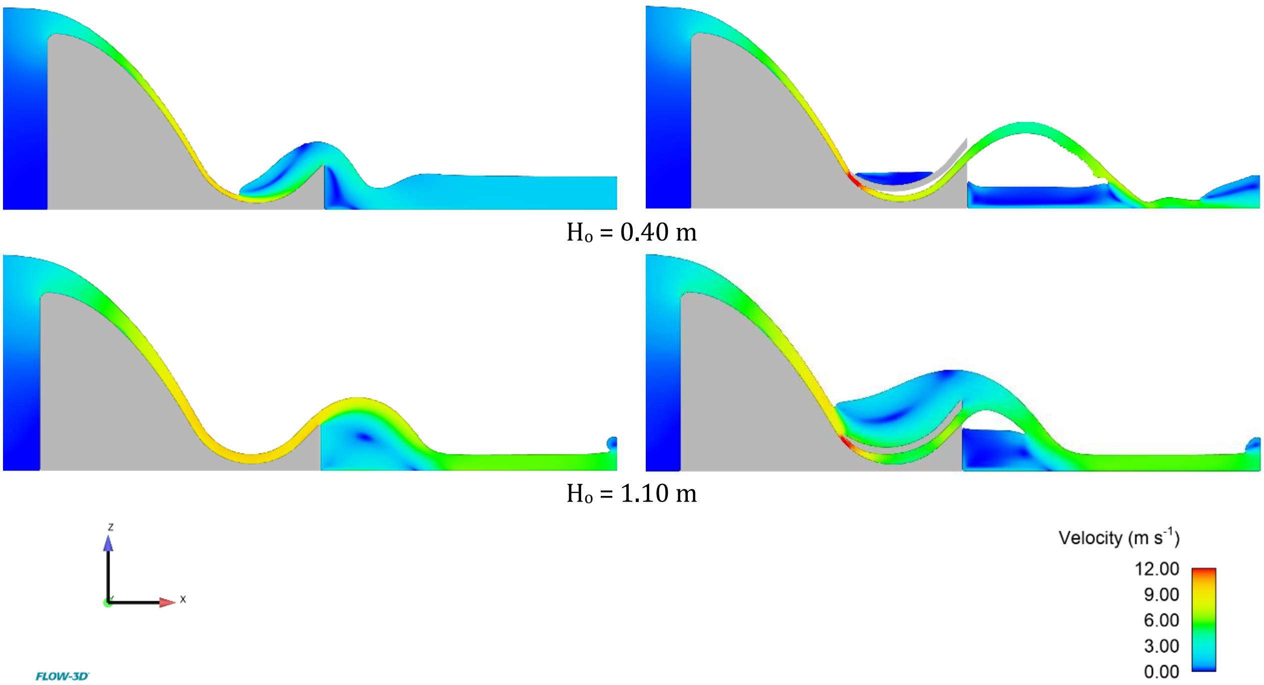

Figure 2. Velocity profiles of the flow

Discussion

In the light of the data and figures obtained from the study, it has been observed that the flow rises by jumping from the flip-bucket structure in the model using a flow separator at low weir loads. At low head loads, a hydraulic jump has occurred just after the jet throwing region of the model with the flow separator. This situation was not observed in the model without the separator. In the case where the head load is increased, while the energy dissipation potential is better in the case with a flow separator, it will be appropriate to make a cavitation calculation since an increase in velocity is detected at the end section where the separator starts.

Conclusion

In the study, the effect of a flow separator placed in the energy breaking pools of the spillway structures, which has an important place in the field of hydraulic engineering, on the flow velocities and energy dissipation status was investigated. In future studies, the performance of the flow separator can be evaluated by comparing the different dimensions, the cavitation risk it will create and how it will behave in case of a possible scour in the downstream region compared to the normal situation.

Acknowledgement

Acknowledgements of support for the paper are welcome.

Funding:

This research received no external funding.

Author contributions:

Ali Emre Ulu: Methodology, Software **M. Cihan Aydın:** Model design, Validation. **Ercan Işık:** Visualization, Writing-Reviewing and Editing.

Conflicts of interest:

The authors declare no conflicts of interest.

References

[1] Aydin, M. C., & Ulu, A. E. (2018). Effects of different shaped baffle blocks on the energy dissipation and the downstream scour of a regulator. Bitlis Eren University Journal of Science and Technology, 8(2), 69-74.
[2] Aydin, M. C., Ulu, A.E., & Karaduman, Ç., (2017). Investigation of Energy Dissipation and Sediment Scour Over a Regulator Using CFD. CEPPIS 2017, Poland.
[3] Chatila, J., & Tabbara, M. (2004). Computational modeling of flow over an ogee spillway. Computers & structures, 82(22), 1805-1812.
[4] Daneshfaraz, R., & Ghaderi, A. (2017). Numerical investigation of inverse curvature ogee spillway. Civ Eng J, 3(11), 1146-1156.
[5] Daneshfaraz, R., Ghaderi, A., Akhtari, A., & Di Francesco, S. (2020). On the effect of block roughness in ogee spillways with flip buckets. Fluids, 5(4), 182.
[6] Hurtig, K., Croockewit, J., & Vasquez, J. A. (2013). Verification of CFD simulations of flip bucket spillway performance using physical model results. CANADIAN DAM ASSOCIATION 2013 Annual Conference, Montréal, Québec, Canada.
[7] Yavuz, C., Dincer, A. E., & Aydin, I. (2016). Head Loss Estimation for Water Jets from Flip Buckets. The International Journal of Engineering and Science (IJES), 5(11), 48-57.
[8] Aydın, M. C., & Işık E. (2015). Using CFD in Hydraulic Structures. International Journal of Scientific and Technological Research. 1(5):7-13.
[9] Flow 3D, (2014). Manual, Sediment Scour Model. Flow-3D User Manual, v11.0.3. Flow Science, Inc.