

Experimental and Numerical Comparison of Hydraulic Parameters of Ski-Jump Bucket Energy Dissipator

Dr P R Modak¹, Dr S D Nagrale², Dr V S Chavhan³, Dr S P Khedekar⁴

¹Assistant Professor, Department of Civil Engineering, AISSMS COE Pune, Savitribai Phule Pune University, India.

²Assistant Professor, Department of Civil Engineering, AISSMS COE Pune, Savitribai Phule Pune University, India.

³Assistant Professor, Department of Civil Engineering, AISSMS COE Pune, Savitribai Phule Pune University, India.

⁴Assistant Professor, Department of Civil Engineering, AISSMS COE Pune, Savitribai Phule Pune University, India.

*Corresponding Email: vschavhan@aissmscoe.com

Abstract

Ski-jump spillways are the most important structures used when flow velocity exceeds 20 m/s. In the present study attempts have been made to study in detail the flow hydraulic parameters over ski-jump spillway. The development of computer science and different types of computational fluid dynamics (CFD) software, the behavior of ski-jump spillways can be studied in a short time and without paying high expenses. In this study, Flow 3D software has been used to simulate flow over ski-jump spillways and results were compared with experimental data. The model tests were carried out in a hydraulic flume for four distinct discharges. The volume of fluid (VOF) method is applied to obtain the free surface. The computational results showed a close agreement with experimental data obtained in the laboratory.

Keywords: *ski-jumps spillway, CFD model, Flow 3D, Fluent, VOF.*

1. INTRODUCTION

Energy dissipating structures are constructed at the end of spillways discharge channels to dissipate the extra energy. In general, hydraulic jump stilling basins, roller buckets, ski jump buckets or other energy dissipating structures are implemented for dissipating the extra energy. If the selected choice for the energy dissipation is a ski jump or a flip bucket, a curved circular surface called bucket will be founded at the end of spillway chute to cause the mild transition of the flow from the spillway surface to the downstream in the outlet channel. Flip buckets are used for energy dissipating and avoiding the river bed scour in the downstream [1]. The flip bucket is a kind of ski jump spillway usually used as an end to a chute's or tunnel spillway in the case of appropriate geological and topological conditions.

The flip bucket itself is not considered as an energy dissipator; however, it is an integral part of the energy dissipation system. The aim of the flip bucket is to guide the highvelocity flow (the jet) to the farther location rather than the dam, powerhouse, spillway and other parts of the dam structure. A small amount of water's energy is dissipated due to the friction through the bucket. In the hydraulic design of the flip buckets, those parameters which are of high importance for the designers include the bucket's geometry, pressure exerted on its boundaries, and the jet trajectory characteristics [2]. Ski jumps or flip buckets have been proposed as a successful hydraulic design in Dordogne hydraulic projects which have been performed in France in the middle of 1930s and also upon the experimentations carried out by Maitre and Obolenski on the jet flows in 1954 [3]. Rohne and Peterka [4] conducted a study on an improved design of flip buckets performed by US bureau of reclamation. The simplest flip bucket is in the shape of a cylindrical shell's sector tangent to the floor of the flood conduit (chute or tunnel). This kind of flip bucket was founded as the energy dissipating system after the studies conducted in 1993 [5] on Grand Coulee dam located in Washington. Then, in 1945, the modified version of this structure named the slotted bucket was founded in Angostura dam located in South Dakota. This structure has been also constructed for Brantley dam of New Mexico. Joun and Hager [6] evaluated the flip buckets both in the type of a prismatic rectangular channel and a bucket having a lateral flow deflector. In their research, the scale effects in the hydraulic models, pressure distribution through the bucket, flow projectile, shock-waves' creation conditions and governing choking relations have been investigated. Jorabloo et al. [7] compared the results of the numerical simulation of flow over the flip bucket with those obtained via the

experimental model. They implemented Fluent® software for their numerical analysis. They concluded that the pressure distribution and jet trajectory obtained via the numerical analysis were in close agreement with the experimental results. Yamini and Kavianpour [8] investigated the hydrostatic and dynamic pressures and their distributions on the flip bucket curve. In their study they examined the effects of geometry and flow characteristics on pressure fields. Yamini et al. [9] experimentally investigated the pressure fluctuation and also the effect of entrance flow conditions on the pressure fluctuation on the bed of compound flip buckets of the Gotvand dam in Iran. Flip buckets are specifically designed for a specific project and the designs are improved using the scale models [10]. Till now, the design of the flip bucket structures has been frequently based on the hydraulic models' studies and due to their site specifications. Therefore, limited design patterns are available in this regard.

Although, flip buckets have been recently used in many hydraulic designs, the number of studies conducted on their basic hydraulic specifications is scarce. A great number of observations are position-oriented and specially designed for a specific hydraulic project. For this reason, the design guides of the flip buckets are not complete at this time. Therefore, many of these hydraulic structures will be examined by physical models before approving the final design [12]. Physical models are constructed in order to study the spillways flow specifications in hydraulic laboratories. However, they are expensive, cumbersome, and timeconsuming [13]. Computational fluid dynamics (CFD) has been increasingly implemented by engineers for modelling and analyzing complicated problems related to the hydraulic design [14]. CFD models are capable of predicting many flow specifications on a spillway. Most of the investigations dealing with the CFD implementation for simulating the spillways have been using Flow-3D® code and this software has been successful at representing either the physical model's results or design curves of U.S. army corps of engineers (USACE) and U.S. bureau of reclamation (USBR) [15]. Savage and Johnson compared the Flow-3D® simulation results with those obtained via the physical model and USBR and USACE data for a standard ogee-crested spillway [16]. Their estimated values for pressure heads on the spillway surface and flow rates were in close agreement with the experimental achievements. The two-dimensional flow analysis at the toe of a dam were performed by Heidarinejad and Najibi [17]. Ho et al. [18] examined the behavior of a spillway with a standard WES profile under increased maximum flood. Eklund [19] investigated the jet trajectory of a ski-jump spillway in Stornnforsen using physical and CFD models.

The numerical model implemented here is constructed using the Flow-3D® code. The available obstacle in the calculation area is identified using fractional area/volume obstacle representation (FAVOR) technique. This approach which has been presented by Hirt et al. [20] uses values between 0 and 1 in order to identify the obstacle existence in each cell. The value of 1 indicates the case when the whole cell is filled with the obstacle. However, 0 points out to a completely empty cell [21, 22]. This program uses a finite volume calculation approach and the volume of fluid (VOF) method which has been previously improved by Hirt and Nichols [23] for computing free surface motion. Implementing this method lead to a significant decrease in the simulation time [24]. In the VOF method, it is assumed that the two fluids (water and air) do not interfere [25]. For each phase (fluid), a variable is defined called the volume fraction which represents the occupied amount of computational cells by that phase [26].

2. Experiment layout

Experiments were conducted in the fluid mechanics laboratory, in a hydraulic flume that was made of glass with a cross section 0.30 m wide, 0.30 m deep and 6 m long (Fig. 1). The conventional ski-jump bucket (Figure 1) was with continuous lip of angles 35° and a radius of 0.0915 m. The width and invert of the bucket was 0.3m and 0.015m respectively. Design parameters of the conventional ski-jump was as per IS 7365 (2010). A pump delivered the flow rate, enabling an accurate discharge adjustment in a closed-circuit system. Clear-water flow depths were measured on the channel centerline with a pointer gauge.

3. Flow-3d

The Flow-3D model developed by Flow Science Incorporated of Los Alamos, New Mexico, USA, is a Computational Fluid Dynamics (CFD) tool capable of simulating the dynamic and steady state behavior of liquids and gases in one, two or three dimensions. It is based on Navier Stokes equations of fluid dynamics. It is applicable to almost any type of the flow process and capable of simulating free surface flow, and the program utilizes specialized algorithms to track the location of the water surface over large and small spatial and temporal variations. These capabilities make the model well suited for simulating

the varied and complex flow conditions, which typically occur in a variety of hydraulic design and analysis problems. The continuity and Reynolds-averaged Navier-Stokes equations are for this case solved in a vertical plane (two dimensions), in order to compute the water motion for turbulent flow.

The continuity and Reynolds-averaged Navier-Stokes equations are for this case solved in a vertical plane (two dimensions), in order to compute the water motion for turbulent flow.

Where U is the Reynolds-averaged velocity over time t , x is the spatial geometrical scale, ρ is the water density, P is the Reynolds-average pressure, δ is the Kronecker delta and νT is the turbulent eddy-viscosity. The turbulence is predicted by the RNG $k-\epsilon$ model (turbulent kinetic energy k and its dissipation ϵ) using the constant empirical values by Launder and Spalding [23].

4. Computation of the free water surface

The free water surface represents a particular challenge in 3D numerical models. The selected computer programs used different methods. Flow-3D uses the Volume of Fluid Method (VOF). This is a two-phase approach where both the water and the air are modeled in each computational cell. The method is based on the concept that each cell has a fraction of water (F), which is 1 when the element is totally filled with water and 0 when the element is filled with air. If the value is between 1 and 0, the element contains the free water surface. The VOF method requires a fixed grid. The number of gridpoints in the vertical direction depends on the water depth and the initial settings. In this study, a width-averaged approach is used, so the components in the direction normal to the main flow direction are zero. The formula can also be used for three dimensions.

5. Grid types and generation

Flow-3D uses a structured and orthogonal grid with rectangular (2D) and hexahedral cells (3D). The non-adaptive grid is fixed and does not move during the calculation. The border between the geometry and the water is defined by the Fractional Area Volume Obstacle Representation (FAVOR) method. Figure 1 shows a longitudinal profile of the grid used in Flow-3D. When the water surface moves, the grid will also move vertically. Hence, only the water phase will be calculated, not the air phase. The grid is regenerated during the computation with a varying number of grid cells over the depth.

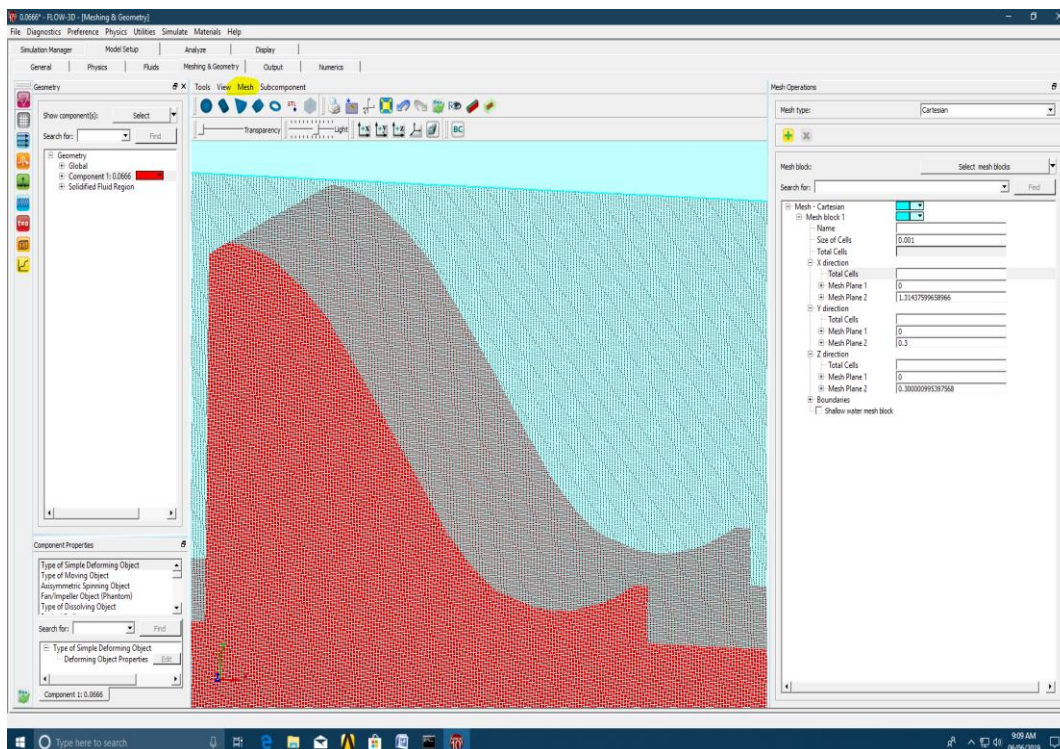


Fig. 1- View of mesh

6. Boundary conditions

To have an accurate result, an appropriate condition should be selected for boundary based on the nature of the flow. In the present study, the computational domain is divided into two sequential zones with

overlapping boundaries, which enable the calculations to efficiently make use of meshes as well as to avoid the time step limitations due to high infiltration at the module. Inlet condition of the weir is set as specified pressure. Also, outflow Boundary condition is used for outlet. Boundary condition for both inlet and outlet has set as continuative because to allow water infiltrate. For the top, specified pressure condition selected as to visualize experiment model.

7. RESULTS AND DISCUSSION

Four different discharges were used to model ski-jump bucket. All turbulence methods were utilized to model water surface profile over a weir, a symmetric plan was chosen to compare observation and simulation results. Table 1 indicate a comparison of results between experimental data and predicted value obtained from the simulation model. As is shown in table 1, CFD result has the best similarity with experimental data. The percentage of error between CFD and experimental results varies from 3.69% to 15.69%.

Sr.no	Discharge (Q) (m ³ /s)	Depth of flow on lip (m)	Tail Water Depth (m)	CFD Energy Dissipation	Experimental Energy Dissipation	% error
1.	0.00962	0.0013	0.025	54.47	48.45	15.69%
2.	0.00735	0.012	0.015	41.13	46.61	11.75%
3.	0.00595	0.011	0.09	55.72	53.67	3.69%
4.	0.00431	0.010	0.08	55.22	52.01	5.81%

Table 1: Comparison of Experimentl and CFD Results

8. Conclusion

Basic experiments were conducted on a ski-jump bucket to investigate the trajectory features and energy dissipation. 3-D numerical scheme provided by models was used to predict the water surface and streamlines. The computed values were then compared with the experimental values. The computational results showed a good agreement with experimental data obtained in the laboratory. The presented results in this study can encourage the researchers further in studying ski-jump bucket using CFD.

REFERENCES

- [1] A. K. Bhaga, G. Sudhamsu, S. Sharma, I. S. Abdulrahman, R. Nittala and U. D. Butkar, "Internet Traffic Dynamics in Wireless Sensor Networks," 2023 3rd International Conference on Advance Computing and Innovative Technologies in Engineering (ICACITE), Greater Noida, India, 2023, pp. 1081-1087, doi: 10.1109/ICACITE57410.2023.10182866.
- [2] Umakant Dinkar Butkar, Manisha J Waghmare. (2023). Hybrid Serial-Parallel Linkage Based six degrees of freedom Advanced robotic manipulator. *Computer Integrated Manufacturing Systems*, 29(2), 70-82. Retrieved from <http://cims-journal.com/index.php/CN/article/view/786>
- [3] Butkar, M. U. D., Mane, D. P. S., Dr Kumar, P. K., Saxena, D. A., & Salunke, D. M. (2023). Modelling and Simulation of symmetric planar manipulator Using Hybrid Integrated Manufacturing. *Computer Integrated Manufacturing Systems*, 29(1), 464-476.
- [4] T.J. Rhone, A. J. Peterka, Improved tunnel spillway flipbuckets, *Journal of Hydraulic Engineering*, ASCE 126 (1959)1270-1291.
- [5] Umakant Butkar, "Review On- Efficient Data Transfer for Mobile devices By Using Ad-Hoc Network", *International Journal of Engineering and Computer Science*, vol 5, Issue 3, 2016
- [6] R. Joun, W.H. hager, Flip bucket without and with deflectors, *Journal of Hydraulic Engineering*, ASCE 126 (2000) 837-845.
- [7] Umakant Dinkar Butkar, Manisha J Waghmare. (2023). Novel Energy Storage Material and Topologies of Computerized Controller. *Computer Integrated Manufacturing Systems*, 29(2), 83-95. Retrieved from <http://cims-journal.com/index.php/CN/article/view/787>
- [8] O. A. Yamini, M. R. Kavianpour, Experimental study of static and dynamic pressures over simple flip bucket, 5th symposium on advances in science and technology, Khavaran Higher education Istitute, Mashhad, Iran. May 12-14 (2011).
- [9] O.A. Yamini, M.R. Kavianpour, S.H. Mousavi, A. Movahedi, Bavandpour, Experimental investigation of pressure fluctuation on the bed of compound flip buckets, *ISH Journal of Hydraulic Engineering* 1 (2017) 1-8.
- [10] N. V. A. Ravikumar, R. S. S. Nuvvula, P. P. Kumar, N. H. Haroon, U. D. Butkar and A. Siddiqui, "Integration of Electric Vehicles, Renewable Energy Sources, and IoT for Sustainable Transportation and Energy Management: A Comprehensive Review and Future Prospects," 2023 12th International Conference on Renewable Energy Research and Applications (ICRERA), Oshawa, ON, Canada, 2023, pp. 505-511, doi: 10.1109/ICRERA59003.2023.10269421
- [11] M.H. Chaudhry, *Open-Channel Flow*, Second Edition, Springer, 2008.
- [12] L. Schmocker, M. Pfister, W.H. Hager, H. E. Minor, Aeration characteristics of ski jump jets, *Journal of Hydraulic Engineering*, ASCE 134 (2008) 90-97.
- [13] M. R. Bhajantri, T.I. Eldho, P.B. Deolalikar, Hydrodynamic modelling of flow over a spillway using a two-dimensional finite volume-based numerical model, *Sadhana* 31 (2006)743-754.

- [14] D. S. Thosar, R. D. Thosar, P. B. Dhamdhere, S. B. Ananda, U. D. Butkar and D. S. Dabhade, "Optical Flow Self-Teaching in Multi-Frame with Full-Image Warping via Unsupervised Recurrent All-Pairs Field Transform," 2024 2nd DMIHER International Conference on Artificial Intelligence in Healthcare, Education and Industry (IDICAIEI), Wardha, India, 2024, pp. 1-4, doi: 10.1109/IDICAIEI61867.2024.10842718.
- [15] P. B. Dhamdhere, D. S. Thosar, S. B. Ananda, R. D. Thosar, D. S. Dabhade and U. D. Butkar, "A Semantic Retrieval System Using Imager Histogram Computation to reduce Trademark infringement based on the conceptual similarity of text," 2024 2nd DMIHER International Conference on Artificial Intelligence in Healthcare, Education and Industry (IDICAIEI), Wardha, India, 2024, pp. 1-6, doi: 10.1109/IDICAIEI61867.2024.10842701.
- [16] B. M. Savage, and M. C. Johanson, Flow over ogee spillway: physical and numerical model case study, J. of Hydraulic Engineering, ASCE 127 (2001) 640-649.
- [17] G. Heidarinejad, and A. Najibi, Two-dimensional analysis of flow at the toe of a dam, Proceedings of Intl. Conf. on Hydraulic Structures, Shahid Bahonar Kerman University, Kerman, (2001).
- [18] D.K. Ho, H.K.M. Boyes, S.M. Donohoo, Investigation of spillway behaviour under increased maximum flood by computational fluid dynamics technique, 14th Australian Fluid Mechanics Conference, Adelaide University, Adelaide, Australia, (2001).
- [19] S. Eklund, CFD modelling of ski-jump spillway in Stornforsen, Master's Thesis, Royal Institute of Technology, Sweden, (2017).
- [20] C.W. Hirt, B.D. Nichols, N.C. Romero, SOLA: A numerical solution algorithm for transient fluid flows (No. LA-5852). Los Alamos Scientific Lab., N. Mex.(USA), (1975).
- [21] Flow-3D® Manual, Version 8.1 [22] C.W. Hirt, J.N.A. Silcilian, Porosity technique for the definition of obstacles in rectangular cell meshes, Proc., 4th Int. Conf. Ship Hydro., National Academy of Science Washington, D.C., (1985) 1-19.
- [23] C.W. Hirt, B.D. Nichols, Volume of fluid (VOF) method for the dynamics of free boundaries, Journal of computational physics 39 (1981) 201-225.
- [24] W. Choi W. and M. H. Kim, Free-surface fluid flows over spillway, 4th International Conference Hydro-Science and Engineering, Seoul, 2000.
- [25] B. Margeirsson, Computational modeling of flow over a spillway in Vatnsfallsstífla dam in Iceland, Master's Thesis, Department of Applied Mechanics, Chalmers University of Technology, Sweden, (2007).
- [26] M. Sartaj, K. Beirami, A.M. Fooladgar, Analysis of two-dimensional flow over standard ogee spillway using RNG turbulence model, 7th International Congress on Civil Engineering, Tarbiat Modares University, Tehran, Iran. (2006).