ISSN: 2326-9865
DOI: https://doi.org/10.17762/msea.v71i1.2131

Modelling the Velocity Distribution around the Hydraulic Structure

Mahesh Chandra Shah
Department of Civil Engineering, Graphic Era Hill University, Dehradun, Uttarakhand, India 248002

Article Info
Page Number: 401-407
Publication Issue:
Vol. 71 No. 1 (2022)

Article History
Article Received: 02 February 2022
Revised: 10 March 2022
Accepted: 25 March 2022

Abstract: Intense pressure and velocity oscillations, as well as extensive air entrainment, make the investigation of the hydraulic leap created in stilling basins very difficult. Its intricacy, coupled with the real-world need for stilling basins to dissipate energy, puts physical modelling front and center. Despite stilling basins' essential role in engineering, bibliographic research has often focused primarily on the more familiar case of the classical hydraulic leap. Thus, a physical model of a typical USBR II stilling basin was used to investigate the features of the hydraulic leap in this work. Under this framework, the model investigated the hydraulic jump's free surface profile and velocity distribution. A series of experiments were conducted to compare the efficacy of modern tools like the time-of-flight camera with that of more conventional ones like the Pitot tube. Notwithstanding some of the stated constraints, the findings demonstrated a good depiction of the free sample surface and the velocity distribution. In addition, the used instruments demonstrated the substantial impact the energy dissipation devices had on the flow parameters. In comparison to traditional hydraulic leaps, significant variations in hydraulic jump form and maximum velocity locations within recorded vertical profiles were discovered.

Keywords: velocity profile, physical model, and USBR II stilling basin

Introduction

This study use hydraulic structures to examine sediment dynamics and hydraulic movements inside a managed reservoir. Typically, three-dimensional flow patterns that are very turbulent surround hydraulic structures. In addition, a two-dimensional (2-D), depth-averaged numerical model is insufficient for regions with considerable changes in sediment concentration along the contour, such as those near intakes and sluice gates. In order to comprehend the impact of hydraulic structures on hydrodynamic, sediment transport, and morphological changes in managed reservoirs, a three-dimensional analysis of these phenomena is often necessary. This research aims to enhance an analytical or numerical (3-D) framework and propose a mathematical scheme to start predicting bed level changes near earth dams, where factors like complex geometry, flow pattern, downdrafts, and sediment density distribution have a major effect on simulation results. [1].

This study is an example of an application study, in which mathematical knowledge is used to create an accurate simulation of a natural process. Both physical and numerical models are employed in the construction of a genuine, sophisticated water controlled reservoir. In order to learn more about the aforementioned processes, physical models are still commonly employed today. In general, physical models are time-consuming to build and resource-intensive to maintain, especially at larger scales. For the future of the controlled reservoir's

ISSN: 2326-9865
DOI: https://doi.org/10.17762/msea.v71i1.2131
functioning, we'll be using numerical models for their ability to evaluate situations that would be impossible to test in a physical model. In addition, the project's budget may be lowered and further alternatives explored. When water moving at high speeds (supercritical flow) discharges into an area of lower speeds (subcritical flow), a phenomena known as a hydraulic leap occurs, causing a relatively sudden increase in velocity the top of the ocean. Significant energy loss, air entrainment, surface waves, and spray are all results of hydraulic leaps[2].

The extra kinetic energy is dissipated or reduced by the hydraulic jump's energy dissipator function. Once the flow reaches the downstream channel, it has been calmed using a variety of energy dissipation devices. These techniques are used to mitigate the effects of scour below overflow spillways, chutes, and sluices by transforming the kinetic energy of flow into potential energy. Energy dissipators are designed using hydraulic information gleaned from model experiments. For a certain range of Froude numbers, we can determine the size of the stilling basins[3-4].

While the hydraulic leaps are more effective as free jumps, there are times when they are unable to be used because of differences in the depth of the tail water. So, the stilling basin's projected operating Froude numbers must be estimated based on the basin's predicted size and accessories[5]. Several scholars have studied hydraulic jump and energy dissipators in a stilling basin in the past. In addition, prior research has mostly analyzed hydraulic jumps on both smooth and uneven surfaces. The experimental studies of drag force (FD) operating on sills and baffles in open channels, with a focus on free and submerged hydraulic leaps on smooth and bumpy bed surface (artificial grass). Due to a rise in the depth of the tail water, the hydraulic leaps may include both types of landings. Vegetation grows, making the channel bed uneven[6-8].

The jump's behavior must be studied in both surface- and depth-water circumstances, on both smooth and uneven beds.The stilling basin's sills and baffles hold the leap in place and prevent it from spreading.For both free and submerged flows, the drag force acting on sills and baffles must be calculated[9]. The drag coefficient is a measure of the force of drag. Surface currents of varying sizes and lifetimes emerge in a turbulent flow and engage in complicated dynamical interactions. It comes as no surprise that a sizable body of research has focused on the prevention and enhancement of turbulence for use in engineering applications.Much time and energy is spent on developing numerical approaches to capture the major impacts linked to turbulence in research. The techniques may be broken down into the three categories below[10]..

Turbulence models for Reynolds-averaged Navier-Stokes (RANS) equation

The model and the Reynolds stress model are two of the most well-known classical turbulence models that account for these ancillary components. During the past three decades, this method has been the standard for engineering flow calculations while only requiring moderate computer resources to achieve reasonable accuracy[11].

Large eddy simulation: This method of calculating turbulence is an intermediate step in determining how huge eddies behave. The unstable Navier-Stokes equations are space-

ISSN: 2326-9865
DOI: https://doi.org/10.17762/msea.v71i1.2131

filtered before being used in the calculations, such that only the big eddies are used.Sub-grid scale models account for the impact of the 12 smallest, unresolved eddies on the resolved flow (mean flow plus big eddies).While (as of this writing) this method is only beginning to tackle CFD issues with complicated geometry, the requirements for computational resources in terms of storage and volume of computations are high due to the need to solve unsteady flow equations[12.-13]

Literature Review

Ivo Baselt, et.al 2021 When totally submerged by overflowing water, ring meshes arranged vertically act as barriers against explosive hazards. Surface Structure Imaging Velocimetry was used to estimate the flow rate across the structures, and the water layer thickness was calculated by factoring in discharges between 1.0 and 4.5 L/s. With lesser discharges, the water moves through over the mesh to generate surface surges. With larger flows, the surges vanish and are replaced by simple water curtains. A theoretical method for describing the flow dynamics was developed using a momentum balance equation. The free parameters were examined and compared with experimental data, including the machined surface of the metal plates and also the deflection angle at the rings. The primary result of this research was that for any given discharge, the thickness of the water rose linearly from 1.0 to 4.6 mm, but the flow velocity stayed constant at w0.54 m/s. Thus, rings should be provided with the maximum discharge to accomplish the best attenuation of blast wave dangers feasible.

Christopher Valela et.al 2021 Due to its tendency to erode pier foundations, bridge scour is a serious problem for public safety or riverine communities. As an upgrade to the standard Flat Plate Collar, a new collar style is presented (FPC). The riverbed near the pier's footing is shielded from the passing rapid flow thanks to the collar's three-dimensional design, which also serves to confine the horseshoe vortex and safely direct it downstream. This prevents the riverbed from being subjected to high shear loads, which significantly lessens scour. We used OpenFOAM (CFD) software and conducted experimental experiments to iteratively refine the collar design. Shapes for the collars were optimized by numerical testing and iteration. The next step included building and testing an FPC and physical replica of the new collar. The collar has been shown to minimize scour more than an FPC, but it still needs work to become a practical scour remedy.

Maoxing Wei et.al 2020 In this study, we provide findings from a comprehensive statistical analysis of experimental data for three hydraulic leaps in a narrow flume at low Froude numbers (1.5-1.9). The undular behavior of the three produced leaps increases at first and subsequently decreases as the Froude number rises. Particle image velocimetry (PIV) is a method for measuring turbulent velocity fields with great spatial and temporal resolution. The free surface profile of the undular leap may be calculated by carefully evaluating the correct velocity vectors from the PIV observations. The pressure fluctuation along the undulating streamlines provides a visual representation of the undulating flow patterns, which are in turn understood in terms of the redistribution of velocity between consecutive crests and troughs.

ISSN: 2326-9865
DOI: https://doi.org/10.17762/msea.v71i1.2131

Methodology

Both of the study's foci are addressed via its technique. The first goal involves the flow field during both the free and underwater leaps. Three-dimensional acoustic Doppler velocity (ADV) measurements were taken utilizing a manually traversed device[14]. Drag force measurements on sill and baffle blocks constitute the second aim. Strain-gauge force balances were calibrated to quantify these drag forces (load cell). Figure 1 depicts the methodology's flowchart. Bed level changes in rivers, reservoirs, estuaries, and other bodies of water where fluid flows interact with sediments fluxes are at the heart of morphodynamics.Coupling between a hydrodynamic model, a geological model, and a Models of bed level change and sediment transport may be used to represent the dynamic equilibrium between sediment volume and its redistribution through time[.

It is frequently required to explore these processes in three dimensions when modeling reservoirs and rivers in order to understand the influence of hydraulic structures on the hydrodynamic and sediment transport processes and morphological changes. The primary objective of this study is to provide a system for calculating bed-level changes around hydraulic structures, where such factors as complicated shape, flow pattern, and sediment concentration distribution have a significant impact on simulation results.

To this day, physical models remain a vital resource for studying these processes. In general, physical models are time-consuming to build and resource-intensive to maintain, especially at larger scales. The advantages of using a numerical model include the ability to save money and time and to explore a wider range of possibilities than would be possible with a physical model as shown in figure 2 and figure 3

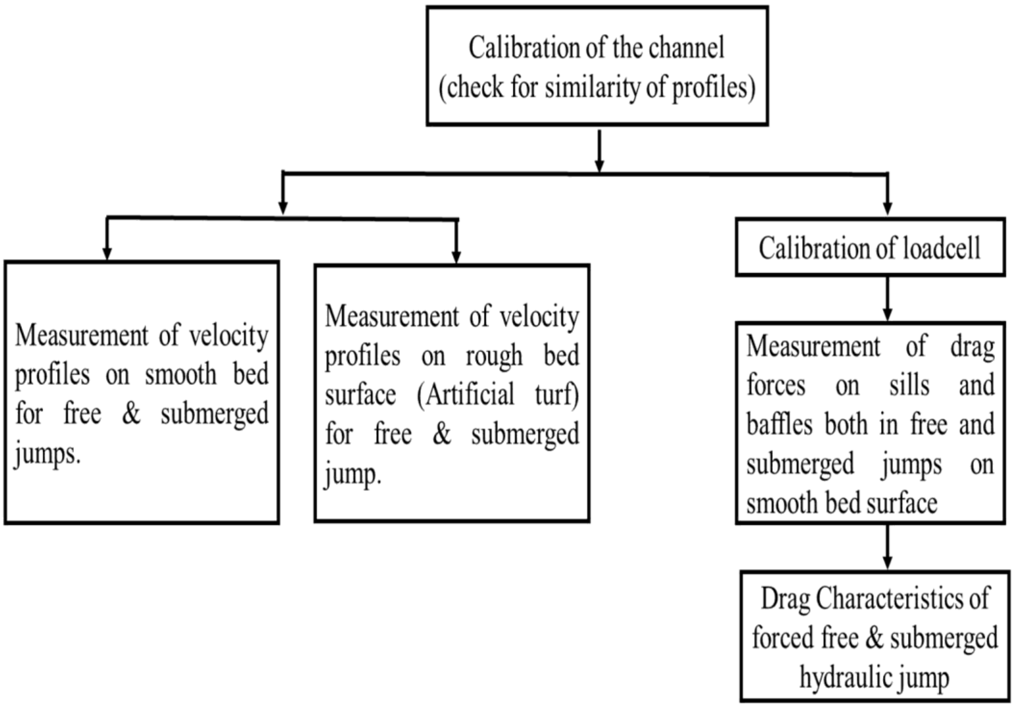

Figure 1. Methodology for the velocity distribution

ISSN: 2326-9865
DOI: https://doi.org/10.17762/msea.v71i1.2131

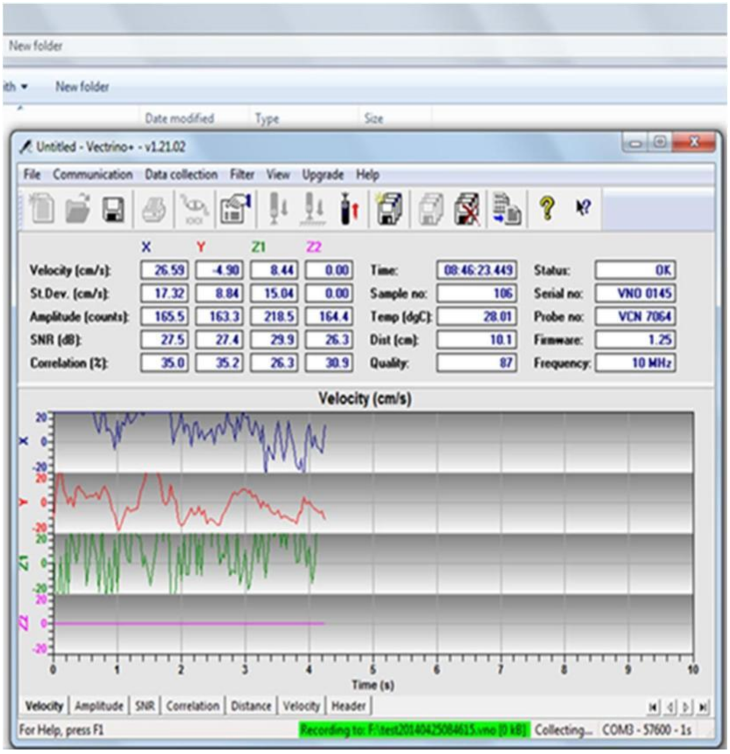

Figure 2. Velocity reading recorded in Vectrino plus software

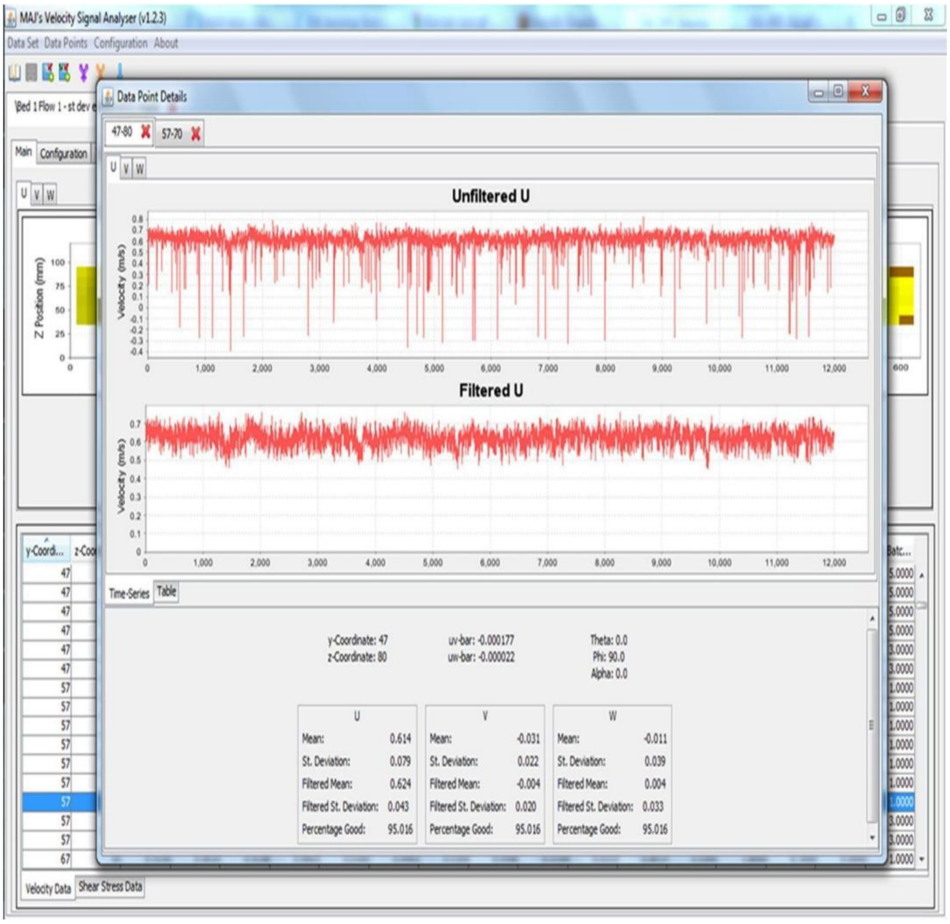

Figure 3 Velocity Signal Analyzer software (De-spiking velocity readings)

Table 1 Experiments for the present study on velocity distribution and drag characteristics

S. No.	Details of Experiments conducted	
1	Velocity profiles for free hydraulic jump on smooth bed surface	F_1 - 4.06,5.28,6.18,7.14
2	Velocity profiles for submerged hydraulic jump on smooth bed surface	F_1 - 4.22,5.42,6.17,6.82
3	Drag characteristics of sill for free hydraulic jump on smooth bed surface	F_1 - 6.06,6.44,7.23,7.47
4	Drag characteristics of sill for submerged hydraulic jump on smooth bed surface	F_1 - 5.03,6.11,6.18,7.12
5	Drag characteristics of baffles for free hydraulic jump on smooth bed surface	F_1 - 6 to 7.12
6	Drag characteristics of baffles for submerged jump on smooth bed surface	F_1 - 5 to 7.11
7	Velocity profiles for free hydraulic jump on rough bed (artificial turf) surface	F_1 - 4.90,5.91,6.44,7.77
8	Velocity profiles submerged hydraulic jump on rough bed (artificial turf) surface	F_1 - 4.62,5.15,6.60,7.17
9	Calibration of force balance with cylinder and sill	F_d - 0.714 and 0.412

The precise methodological framework for the goals was developed in this paper. Next, the specifics of the experiment were described, including the Froude number ranges tested and the instruments used to take the readings. Table 1 depicted the experimental values **.**

Conclusion

The purpose of this research was to examine the flow characteristics after a hydraulic construction using a 3D model. First, the reliability of the analytical framework was established and by contrasting the findings of the study with those of the field discharge experiments. The flow characteristics of the downstream hydraulic structures were then determined by applying different boundary conditions and gate operating circumstances to the analytical model that had already been confirmed.

References

1. Baselt, I. (2021). Flow Velocity and Water Layer Thickness at Vertical Ring Mesh Structures. Journal of Hydraulic Engineering, 147(8). https://doi.org/10.1061/(asce)hy.1943-7900.0001899
2. Bai, Z., Bai, R., Tang, R., Wang, H., & Liu, S. (2021). Case Study of Prototype Hydraulic Jump on Slope: Air Entrainment and Free-Surface Measurement. Journal of Hydraulic Engineering, 147(9). https://doi.org/10.1061/(asce)hy.1943-7900.0001916

3. Valela, C., Nistor, I., Rennie, C. D., Lara, J. L., & Maza, M. (2021). Hybrid Modeling for Design of a Novel Bridge Pier Collar for Reducing Scour. Journal of Hydraulic Engineering, 147(5). https://doi.org/10.1061/(asce)hy.1943-7900.0001875
4. Wei, M., Chiew, Y.-M., & Emadzadeh, A. (2020). Flow Patterns and Turbulent Kinetic Energy Budget of Undular Jumps in a Narrow Flume. Journal of Hydraulic Engineering, 146(9). https://doi.org/10.1061/(asce)hy.1943-7900.0001788
5. Bagheri, S., & Kabiri-Samani, A. (2020). Overflow characteristics of streamlined weirs based on model experimentation. Flow Measurement and Instrumentation, 73, 101720. https://doi.org/10.1016/j.flowmeasinst.2020.101720
6. Sharafati, A., Haghbin, M., Motta, D., & Yaseen, Z. M. (2019). The Application of Soft Computing Models and Empirical Formulations for Hydraulic Structure Scouring Depth Simulation: A Comprehensive Review, Assessment and Possible Future Research Direction. Archives of Computational Methods in Engineering, 28(2), 423–447. https://doi.org/10.1007/s11831-019-09382-4
7. Saad, N. Y., & Fattouh, E. M. (2017). Hydraulic characteristics of flow over weirs with circular openings. Ain Shams Engineering Journal, 8(4), 515–522. https://doi.org/10.1016/j.asej.2016.05.007
8. Eng, J., & Mater. (n.d.). Investigating LES Turbulence Model in Modeling the Velocity Distribution Around the Hydraulic Structure Using ANSYS. Journal of Civil Engineering and Materials Application Is Published by Lexis Publisher; Journal, 2017. https://doi.org/10.15412/J.JCEMA.12010201
9. Aneesh Kumar, J., Krishnakumar, K., & Savithri, S. (2015). Computer Simulation of Centrifugal Casting Process Using FLOW-3D. Materials Science Forum, 830-831, 53–56. https://doi.org/10.4028/www.scientific.net/msf.830-831.53
10. Salamatian, S. A., Forghani, M., & Karimaee Tabarestani, M. (2016). Flow Pattern and Stress Distribution around Three Spur Dike in Ninety inety Degree Bend. International Journal of Engineering and Technology, 8(6), 462–467. https://doi.org/10.7763/ijet.2016.v8.934
11. VAGHEFI, M., SHAKERDARGAH, M., & AKBARI, M. (2014). Numerical investigation of the effect of Froude number on flow pattern around a submerged T-shaped spur dike in a 90° bend. TURKISH JOURNAL of ENGINEERING and ENVIRONMENTAL SCIENCES, 38, 266–277. https://doi.org/10.3906/muh-1405-2
12. Nikora, N., Nikora, V., & O'Donoghue, T. (2013). Velocity Profiles in Vegetated Open-Channel Flows: Combined Effects of Multiple Mechanisms. Journal of Hydraulic Engineering, 139, 1021-1032. https://doi.org/10.1061/(ASCE)HY.1943-7900.0000779
13. Stoesser, T., Kim, S. J., & Diplas, P. (2010). Turbulent Flow through Idealized Emergent Vegetation. Journal of Hydraulic Engineering, 136, 1003-1017. https://doi.org/10.1061/(ASCE)HY.1943-7900.0000153
14. Tang, X., & Ali, S. (2013). Evaluation of Methods for Predicting Velocity Profiles in Open Channel Flows with Submerged Rigid Vegetation. In Proceedings of the 35th IAHR World Congress (Vol. 4, B1, pp. 1744-1755). Beijing: Tsinghua University Press