Uluslararası İleri Doğa Bilimleri ve Mühendislik Araştırmaları Dergisi
Sayı 8, S. 10-15, 8, 2024

Araştırma Makalesi

IJANSER

International Journal of Advanced Natural Sciences and Engineering Researches
Volume 8, pp. 10-915, 8, 2024

Research Article

https://as-proceeding.com/index.php/ijanser
ISSN: 2980-0811

ANALYSIS OF ENERGY DISSIPATION PHENOMENON AT VERTICAL FALL USING COMPUTATIONAL FLUID DYNAMICS MODELING - A CASE STUDY OF LOWER GOGERA BRANCH CANAL PAKISTAN

Muhammad Haseeb*, Muhammad Adnan [2], Ali Ejaz [3] and Moiz Ali

[1]*Assistant Executive Engineer, Water and Power Development Authority WAPDA, Paksitan*
[2]*Structure Engineer, Faisal and Fahad Associates, Bahria Town Rawalpindi, Pakistan.*
[3]*Assistant Executive Engineer, Water and Power Development Authority WAPDA, Paksitan*
[4]*Assistant Executive Engineer, Water and Power Development Authority WAPDA, Paksitan*

(haseebm147@gmail.com) Email of the corresponding author

(Received: 03 September 2024, Accepted: 18 September 2024)

(3rd International Conference on Scientific and Innovative Studies ICSIS 2024, September 11-12, 2024)

ATIF/REFERENCE: Haseeb, M., Adnan, M., Ejaz, A. & Ali, M. (2024). ANALYSIS OF ENERGY DISSIPATION PHENOMENON AT VERTICAL FALL USING COMPUTATIONAL FLUID DYNAMICS MODELING - A CASE STUDY OF LOWER GOGERA BRANCH CANAL PAKISTAN. *International Journal of Advanced Natural Sciences and Engineering Researches*, 8(8), 10-15.

Abstract **–** In this work experimental and numerical investigations were carried out to study the energy dissipation at the downstream of the vertical fall of Lower Gogera Branch Canal (LGBC) using CFD software and compared the results with a comprehensive physical model constructed in the model tray hall of Centre of Excellence in Water Resource Engineering, UET, Lahore for validation of numerical model. The study has considered two case scenarios. The first model was the original vertical drop having a drop of 3.69 m having stilling basin at the downstream and the second model was prepared by providing the baffle blocks at the downstream to dissipate the energy. The discharge passing through this fall is 63.73 m3/s. The baffle blocks were designed according to the specifications provided in the USBR manual for small Hydraulic structures. To simulate the free flow surface, the volume of fluid (VOF) method, and the Renormalization Group (RNG) were adopted in the FLOW-3D software. The results highlighted that the numerical model shows proper coordination with experimental results and the difference between the amount of energy calculated using physical and numerical model is within 5%. Moreover, the use of baffle blocks at the downstream of the fall was found to be effective to dissipate energy and reduce the velocity to some extent resulting in the safety of structure.

Keywords – Numerical Modelling; Canal Fall; Hydraulic Jump; Energy Dissipation; Flow-3D.

I. INTRODUCTION

The design and maintenance of hydraulic structures are critical for effective water management in irrigation systems and the safety of the structure. Canal falls, also referred to as drop structures, grade controls, or over-falls, facilitate the controlled descent of water from higher to lower elevations, thereby mitigating the risk of bed and bank erosion in irrigation canals and open channels. Proper design of these structures is essential to ensure that the excess energy from falling water is efficiently dissipated, preventing potential damage to canal infrastructure and maintaining the stability of the canal.

The initial design and evaluation of canal falls is majorly relied on physical model studies. These involve constructing scaled-down versions of the proposed hydraulic structures and conducting experiments to observe their behavior under various flow conditions. Physical models provide valuable insights into the performance of hydraulic structures but come with several limitations. These include high costs associated with model construction and experimentation, time-consuming processes, and the need for specialized technical expertise and materials. As a result, physical modeling may not always be feasible for extensive design evaluations or for exploring a broad range of design alternatives.

Advancements in numerical modeling techniques, particularly Computational Fluid Dynamics (CFD), offer a promising alternative to traditional physical models. The improvements in computational speed and storage capacity, coupled with the development of advanced turbulence modeling approaches, have made CFD a viable and complementary investigation tool in hydraulic engineering. CFD allows for the simulation of fluid flow and energy dissipation processes with a high degree of accuracy and flexibility. By utilizing CFD, engineers can evaluate and optimize hydraulic designs at an early stage, reducing the need for extensive physical testing and potentially lowering overall project costs. This approach enables the rapid screening of various design options and provides detailed insights into the behavior of hydraulic structures under different conditions.

In the context of canal falls, particularly vertical falls, CFD modeling plays a crucial role in analyzing energy dissipation. Vertical falls are designed to manage the energy of water as it descends from a higher elevation to a lower one, effectively preventing erosion and damage to the hydraulic structure, canal bed and banks. The efficiency of a vertical fall in dissipating energy is essential for maintaining the structural integrity of the canal and ensuring its effective operation. Numerical simulations can provide a detailed understanding of how water interacts with the fall structure, including the distribution of energy and the resulting impact on the downstream channel.

This research focuses on the application of CFD modeling to analyze the energy dissipation phenomena at vertical falls, specifically using a case study of the Lower Gogera Branch Canal in Pakistan. The Lower Gogera Branch Canal is a critical component of the region's irrigation infrastructure, and optimizing the design of its vertical falls is essential for preventing erosion and ensuring efficient water management. By employing CFD, this study aims to simulate the flow characteristics and energy dissipation processes associated with the canal’s vertical falls, offering valuable insights into their performance and effectiveness.

Several previous studies have demonstrated the effectiveness of CFD in analyzing hydraulic structures. For example, Kabiri-Samani et al. (2017) investigated grid drop-type dissipators and found that varying supercritical flows can generate complex patterns of jet dispersion [1]. Fleit et al. (2018) validated CFD simulations against experimental data for ogee crested dams, showing that numerical models can accurately predict flow conditions [2]. Valero et al. (2016) compared smooth and stepped chutes using CFD, highlighting differences in turbulence and energy dissipation [3]. Similarly, Mishra (2015) explored energy dissipation in flexible aprons using CFD, confirming that numerical models can effectively simulate flow dynamics and turbulence [4]. Sobeih (2012) focused on mitigating local scour downstream of hydraulic structures by using openings in weirs and introducing a system of floor water jets to reduce scour depth [5]. Farouk and Elgamal (2012) examined the hydraulic performance of both single and multi-drop structures through empirical equations and numerical simulations. Their work, utilizing ANSYS CFD and the volume of fluid (VOF) method, confirmed that numerical simulations effectively replicated experimental conditions and provided satisfactory results for understanding energy dissipation in these structures [6]. Khan (2011) used a high-resolution CFD model with FLOW-3D™ to evaluate the energy dissipation capabilities of the Powell Butte Reservoir overflow structure. The study revealed that the existing Type VI USBR impact basin and Type IX USB baffle apron were insufficient for handling the expected maximum overflow rate of 170 mgd, necessitating an increase in the wing wall height by one foot to prevent overtopping and erosion [7]. These studies underscore the growing acceptance of CFD as a reliable tool for hydraulic modeling.

The present study aims to 1. utilize Flow 3D software to analyze the energy dissipation characteristics of vertical falls at the Lower Gogera Branch Canal by simulating flow conditions and assessing how effectively different design configurations manage and dissipate energy. 2. Compare the results of CFD simulations with the physical model results to evaluate the accuracy and reliability of numerical models in predicting the behavior of vertical falls, thus validating the effectiveness of CFD as a tool for hydraulic analysis. 3. Provide insights and recommendations for optimizing the design of vertical falls in the Lower Gogera Branch Canal.

In summary, while physical models have traditionally been used for hydraulic structure design, CFD modeling offers a more efficient, versatile, and cost-effective alternative. This research will leverage CFD to investigate the energy dissipation characteristics of vertical falls, providing valuable information for optimizing canal designs and enhancing irrigation system performance. The findings will contribute to advancing hydraulic engineering practices and improving the management of water resources.

II. MATERIALS AND METHOD

Lower Gogera Branch canal is located in Punjab, Pakistan. It is one of the oldest irrigation canals in Pakistan. The estimated terrain elevation above sea level is 196 meters. It originates from the Lower Chenab Canal (LCC) at Buchiana head works which starts from Head Khanki at Chenab River in Gujrat District. The system's command area is served by a network of major canals, branch canals, distributaires, and minors covering 3.75 million acres. The height of the fall 65+000 is 3.69 m. The width of the abutment is 32.93m and the length of stilling basin is 5m. There are 4 bays at the downstream and no baffle blocks are installed at the downstream. The discharge passing through this fall is 63.37 cumecs. Physical Model was constructed according to the scale (12:1) in the Model Tray Hall of Centre of Excellence in Water Resources Engineering (CEWRE), UET, Lahore. The model was constructed with bricks and plastered with cement, sand mixture while bed of model was prepared by using soil collected from site. The length of model was 8.82 ft.

A. Flow-3D

FLOW 3D is a 3-dimensional software developed by FLOW Science for free surface modeling. FLOW 3D utilizes the Volume of Fluid (VOF) technique having a well-defined grid and solve the Reynold Averaged Navier-Stokes equation. The program has a different turbulent model. The RNG and K-ε models are important. Renormalization-Group (RNG) model is an advanced form of the k-ε model. Both models have the same equations. The distinction between the two models is that the constants of the k-ε model are empirically computed and constants in the RNG model are computed explicitly. The RNG model is more robust as compared to the k-ε model. The small-scale intensity turbulent flows are delineated by the RNG model with good accuracy. (Flow Science 2017).

B. Numerical model setup

The vertical fall drawings obtained from the irrigation department are utilized in the preparation of 3D geometry in stereolithography format using sketch up. The geometry is imported in FLOW 3D. The Renormalized group (RNG) model is selected in the turbulent model. Afterward, the meshing of the model is done. The uniform meshes size of 0.25m, 0.5m and 0.75m respectively are used in free flow simulation of the model. The sensitivity analysis of the Mirani dam spillway has been done using the following three sets of boundary conditions as shown in Table 1 below.

C. Model validation

Validation of the numerical model is significantly important to precisely predict the hydraulics of flow. The model has been validated by comparing the physical model and numerical model result. The simulated water level values obtained from FLOW 3D numerical model are quite agreeable to physical model water surface elevation at a discharge of 66.26 m3/s as shown in Fig 1. The maximum percentage average water level error of 0.6 % between the physical and numerical model shows the successful validation of a numerical model.

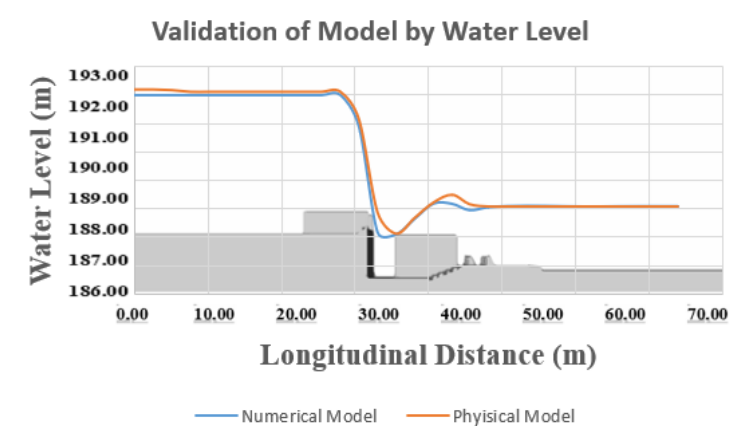

Fig 1: Validation of Model by Water Level

D. Meshing of Geometry

The uniform meshes size of 0.25 m, 0.5 m and 0.75m respectively are used in free flow simulation of the model. The simulation with 0.25 m uniform mesh predicted water depth accurately along the channel as compared to other mesh sizes. The uniform mesh size of 0.25 m. is adopted in further research.

E. Velocity

The water surface elevation of the numerical model and the physical model shows great agreement with each other in model validation. The velocity of the water at fall without baffle blocks is about 6 m/s which becomes constant to 1.36 m/s after a length of 5m from stilling basin. After installation of the baffle blocks at the downstream of the fall, velocity of the water becomes constant to 1.04 m/s after striking with the blocks. The hydraulic jump at 63.37 cumecs has been well contained inside the structure. The velocity profiles for both the models are shown in fig 2 and 3.

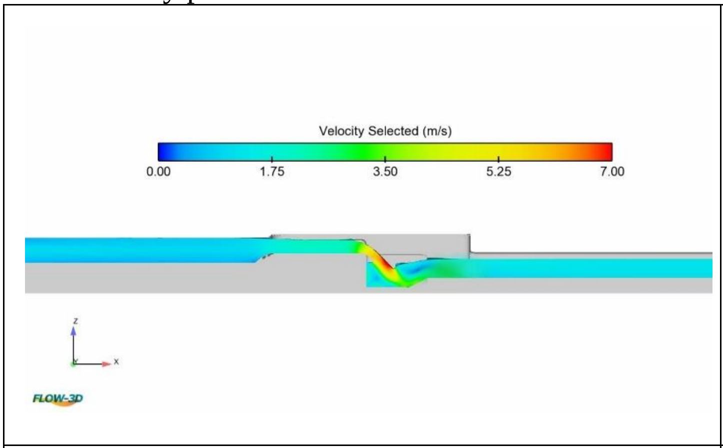

Fig 2: Velocity Contour Profile without d/s Blocks

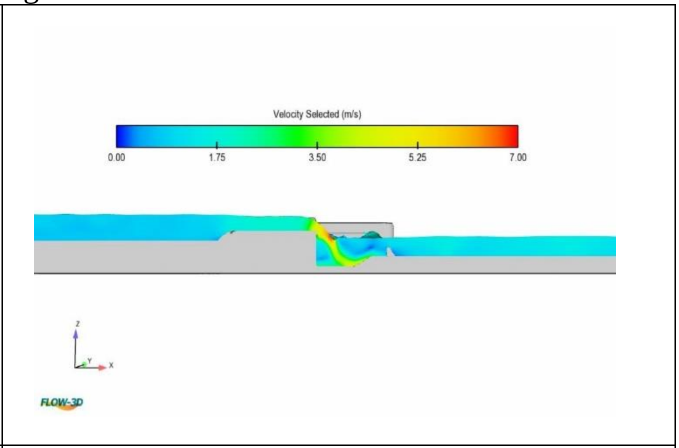

Fig 3: Velocity Contour Profile after installing d/s Blocks

III. RESULTS & DISCUSSIONS

The simulation has been run for two models to calculate the amount of energy dissipated. The first model is the original constructed structure and the second model is the modification in the original structure by providing baffle blocks of 0.3 m height designed according to the according to the specifications provided in the USBR manual for small Hydraulic structures. The water surface elevation of the numerical and the physical model shows great agreement with each other in model validation. The hydraulic gradient profiles and specific energy profiles for both the falls are shown in fig 4 and fig 5. The energy dissipation is computed by the difference of upstream specific energy at the entrance of the stilling basin and exit of the stilling basin which is shown in the table 1.

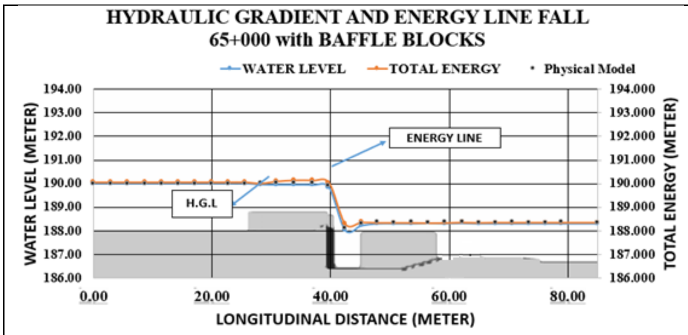

Fig 4: Water surface and specific energy profile along Fall 65+000 at discharge of 63.37 cumecs without blocks

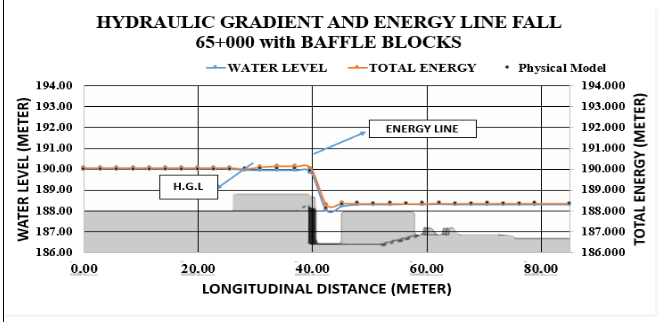

Fig 5: Water surface and specific energy profile along Fall 65+000 at discharge of 63.37 cumecs with blocks

Table 1. Comparison of Physical and Numerical model of different fall structures

Scenario	**Discharge**	**U/S specific energy**		**D/S specific energy**		**Energy absorbed**		**% Difference**
		Physical	**Numerical**	**Physical**	**Numerical**	**Physical**	**Numerical**	
	m^3/sec	m	m	m	m	%	%	%
Without baffle blocks	63.37	3.90	3.95	2.30	2.40	39.24	41.03	4.03
With baffle blocks	63.37	3.90	3.95	2.50	2.20	35.90	44.30	5.79

IV. DISCUSSION

The study demonstrates that CFD modeling can accurately replicate physical experiments, with a discrepancy of less than 5% in energy calculations, underscoring its reliability for hydraulic structure analysis. The effectiveness of baffle blocks in dissipating energy highlights their practical value in enhancing the safety and efficiency of vertical falls in canal systems.

V. CONCLUSION

The conclusions based on CFD modeling of the vertical fall at RD 65+000 of Lower Gogera Branch Canal by installing baffle blocks at the downstream are as follows:

1. The comparison of the upscale result of a physical model of Vertical fall in MT hall and Numericalmodel structure shows a great agreement with each other. The difference between the amount of energy calculated using physical and numerical model is within 5%.
2. The use of baffle blocks in fall 65+000 was found to be effective to dissipate energy and reduce the velocity to some extent at downstream of the structure.
3. The more finer size mesh should be used to improve results keeping in view the time limitation and hardware system requirement.

ACKNOWLEDGMENT

We are grateful to Irrigation Department Lahore and UET Lahore for providing comprehensive engineering drawings and great cooperation incompletion of Physical model testing.

REFERENCES

[1] Sharif, M., & Kabiri-Samani, A. (2018). Flow regimes at grid drop-type dissipators caused by changes in tail-water depth. Journal of Hydraulic Research, 56(4), 505-516.

[2] Valero Huerta, Daniel, et al. "Numerical investigation of USBR type III stilling basin performance downstream of smooth and stepped spillways." 6th International Symposium on Hydraulic Structures. Utah State University, 2016.

[3] Pedersen, Ø., Fleit, G., Pummer, E., Tullis, B. P., & Rüther, N. (2018). Reynolds-averaged Navier-Stokes modeling of submerged ogee weirs. Journal of Irrigation and Drainage Engineering, 144(1), 04017059.

[4] Mishra, K. (2015). 3D Numerical Modelling of Energy Dissipation in Flexible Apron of Barrages. Journal of The Institution of Engineers (India): Series A, 96(1), 47-56.

[5] Sobeih, M. F., Helal, E. Y., Nassralla, T. H., & Abdelaziz, A. A. (2012). Scour depth downstream weir with openings. International Journal of Civil & Structural Engineering, 3(1), 259-270.

[6] Farouk, M., & Elgamal, M. (2012). Investigation of the performance of single and multi-drop hydraulic structures. International Journal of Hydrology Science and Technology, 2(1), 48-74.

[7] Khan, L. A. (2011). Computational fluid dynamics modeling of emergency overflows through an energy dissipation structure of a water treatment plant. In World Environmental and Water Resources Congress 2011: Bearing Knowledge for Sustainability (pp. 1484-1493).