Technical Strategies in Water Systems
https://sanad.iau.ir/journal/tsws
ISSN (Online): 2981-1449
Summer 2024: Vol 2, Issue 2
https://doi.org/10.30486/TSWS.2024.1191046

Research Article

Numerical analysis of flow hydraulics in a culvert-weir system with a circular crest and gate

Donya Heidari [1], Elham Izadinia[2*]

1 Master's Student, Department of Civil Engineering, Faculty of Engineering and Technology, Iran University of Science and Technology, Tehran, Iran.

2 Assistant Professor, Department of Civil Engineering, Faculty of Engineering and Technology, Shahid Ashrafi Esfahani University, Isfahan, Iran.

Corresponding Author email: elham.izadinia@gmail.com

Received: 16 Oct 2024 Accepted: 09 Dec 2024 Published: 24 Dec 2024

Extended Abstract

Introduction

Circular-crested weirs are crucial hydraulic structures in water engineering projects, playing a significant role in managing surface flow. These weirs are favored for their simple design and high flow capacity, making them widely used in various applications. The research aims to investigate the hydraulic characteristics of flow in a culvert-circular weir system, focusing on the impact of key parameters such as weir radius and gate opening on the discharge coefficient. The study utilizes computational fluid dynamics (CFD) simulations to analyze velocity profiles, water surface levels, and other flow parameters. By modeling different configurations of the culvert-circular weir system in Gambit and transferring these models to Fluent for precise CFD simulations, the research evaluates the discharge coefficients of the weir, gate, and the entire structure under varying water head conditions. The findings indicate that the discharge coefficient increases with the presence of a gate compared to a weir without a gate, and that reducing the weir radius while increasing the gate opening enhances the discharge coefficient. Furthermore, the comparison of numerical simulation results with reliable experimental data demonstrates a good agreement, validating the accuracy of the numerical modeling. This research aims to provide precise data and practical analyses to improve the design and performance of combined hydraulic structures.

Materials and Method

The study employs CFD simulations to analyze the flow characteristics of the culvert-circular weir system. The simulation process follows several key steps. First, the geometry of the flow is defined, including the fluid and its path, using software like ANSYS Design Modeler or Gambit. Next, a mesh or element is generated to facilitate numerical solutions of the fluid equations, utilizing the finite volume method in Fluent. Boundary conditions are applied to specify flow characteristics, fluid properties, and dynamic mesh settings. The k-ε turbulence model is chosen for its rapid convergence and high accuracy, particularly for turbulent flow simulations. This model allows for detailed analysis of velocity profiles, pressure, and turbulence parameters across the flow domain. The study evaluates two groups of models: Group A focuses on varying the weir radius,

Technical Strategies in Water Systems
https://sanad.iau.ir/journal/tsws
ISSN (Online): 2981-1449
Summer 2024: Vol 2, Issue 2
https://doi.org/10.30486/TSWS.2024.1191046

while Group B examines different gate openings. The geometric specifications of the models are summarized in a table, detailing the radius, gate opening, and discharge rates.

Results and Discussion

The results reveal significant insights into the hydraulic performance of the culvert-circular weir system. For Group A, as the weir radius decreases, the discharge coefficient increases, with the lowest coefficient observed at a radius of 15 cm and the highest at 5 cm. In Group B, the maximum discharge coefficient corresponds to a gate opening of 10 cm, while the minimum is for a 3 cm opening. The water surface profile analysis shows that the flow depth upstream increases with a larger weir radius, while downstream levels remain consistent across different radii. Additionally, the velocity profiles indicate that flow velocity peaks near the water surface and decreases downstream. The study also illustrates how varying the gate opening affects flow characteristics, with higher openings leading to increased flow rates and velocity near the weir.

Conclusion

The findings of this research demonstrate that the discharge coefficient in gated weirs is greater than that in ungated weirs. Additionally, reducing the weir radius results in an increased discharge coefficient. The analysis emphasizes the importance of both weir radius and gate opening in optimizing hydraulic designs. The results provide valuable insights for engineers in designing more efficient hydraulic structures, contributing to improved water management practices. Future work could explore additional parameters and configurations to further enhance the understanding of flow dynamics in circular-crested weirs and their applications in hydraulic engineering.

Keywords: Circular weir, Computational Fluid Dynamics (CFD), Culvert-weir combination, Flow hydraulics, Numerical simulation