NUMERICAL MODELING OF WATER FLOW THROUGH HYDRAULIC GATES AND ITS INFLUENCE ON THE EFFICIENCY OF BANKI TURBINES IN MICRO-HYDROPOWER SYSTEMS

Ochilov Obid Boymurod ugli
Doctoral student (PhD)
ORCID: 0000-0002-1172-4747, E-mail: obidbekochilov@bk.ru
Karshi state technical university, Karshi city, Uzbekistan
https://doi.org/10.5281/zenodo.18383756

ARTICLE INFO

Received: 24th January 2026
Accepted: 25th January 2026
Online: 26th January 2026

KEYWORDS

water flow; pressure; velocity; sluice gate; Banki turbine; micro-hydropower plant; numerical simulation; SolidWorks Flow Simulation; cavitation; hydraulic efficiency

ABSTRACT

This paper presents a numerical investigation of the velocity and pressure distribution of water flow through a hydraulic sluice gate and its influence on the performance of a Banki turbine used in micro-hydropower systems. The study is based on the fundamental equations of fluid mechanics, including the Bernoulli and Navier–Stokes equations, and considers different flow regimes characterized by the Reynolds number. Numerical simulations were carried out using the SolidWorks Flow Simulation module to analyze variations in flow velocity and pressure under different gate opening conditions.

The results demonstrate an inverse relationship between velocity and pressure, confirming the principle of energy conservation in hydraulic flows. Regions of increased velocity are accompanied by significant pressure reduction, which may lead to cavitation under certain operating conditions. In addition, the nonlinear behavior of turbulent flow is revealed, indicating the necessity of numerical modeling for accurate prediction of hydraulic parameters.

Furthermore, the study discusses the main hydraulic, geometric, operational, and natural factors affecting the efficiency of the Banki turbine. The rational selection of turbine geometry, blade number, and inlet flow direction is shown to reduce hydraulic losses and improve energy conversion efficiency. The integration of flow modeling with turbine performance analysis provides a scientific basis for optimizing the design and operation of micro-hydropower plants and contributes to the sustainable utilization of small-scale hydropower resources.

Introduction

The global demand for electrical energy continues to grow due to population increase, industrial development, and the expansion of technological infrastructures. In this context, renewable energy sources play a crucial role in ensuring sustainable development and reducing environmental impacts. Among renewable sources, hydropower remains one of the most reliable and efficient technologies for electricity generation. In recent years, special attention has been paid to micro-hydropower plants (micro-HPPs), which utilize small water resources such as rivers, irrigation canals, and hydraulic structures.

A key challenge in micro-HPP design is the efficient utilization of water flow energy under low-head and variable flow conditions. This requires a detailed understanding of hydraulic processes, particularly the distribution of velocity and pressure in flow passages and their interaction with turbine components. Hydraulic gates (sluice gates) are widely used in water regulation systems and often serve as flow control elements in micro-HPPs. The flow passing through such gates is characterized by complex velocity and pressure fields, which significantly influence turbine performance and operational safety.

The Banki turbine (cross-flow turbine) is commonly applied in micro-HPPs due to its simple construction, low cost, and adaptability to medium and low heads. However, its efficiency strongly depends on hydraulic conditions, turbine geometry, blade configuration, and flow direction. Therefore, combining flow modeling through hydraulic gates with an analysis of Banki turbine efficiency factors represents an important research direction for improving micro-HPP performance.

This study aims to investigate the velocity and pressure distribution of water flow through a sluice gate using numerical simulation and to analyze the main factors affecting the efficiency of Banki turbines. The integration of computational fluid dynamics (CFD) modeling with turbine performance analysis provides a scientific basis for optimizing hydraulic and mechanical parameters in micro-hydropower systems.

Theoretical Background

The motion of water in open channels and hydraulic structures is governed by the fundamental laws of fluid mechanics. The Bernoulli equation expresses the conservation of mechanical energy along a streamline:

$$\frac{p}{\rho g} + \frac{v^2}{2g} + z = \text{const}$$

where p is pressure, v is velocity, ρ is fluid density, and z is elevation head. This equation indicates an inverse relationship between velocity and pressure, which is particularly important in regions where flow accelerates, such as under sluice gates.

For real viscous flows, the Navier–Stokes equations provide a more complete description:

$$\rho\left(\frac{\partial \vec{v}}{\partial t} + \vec{v} \cdot \nabla \vec{v}\right) = -\nabla p + \mu \nabla^2 \vec{v}$$

where $\mu$is the dynamic viscosity of water. These equations account for inertial, pressure, and viscous forces and are essential for modeling turbulent flow behavior.

The Reynolds number is used to characterize the flow regime:

$$Re = \frac{vD}{\nu}$$

where $D$is the hydraulic diameter and $\nu$is the kinematic viscosity. In most hydraulic structures and natural channels, Reynolds numbers are typically above 10^5, indicating turbulent flow conditions.

For Banki turbines, the hydraulic efficiency depends on the interaction between the incoming jet and turbine blades. The optimal conversion of kinetic energy into mechanical rotation requires proper alignment of the jet with the blade curvature and appropriate selection of blade number and turbine diameter.

Methodology

In this study, numerical simulations were carried out using the Flow Simulation module of SolidWorks. A geometric model of a sluice gate was created, and water flow through the gate opening was analyzed for different upstream head conditions. Boundary conditions were defined by specifying inlet velocity and outlet pressure, while turbulence effects were included using standard turbulence models available in the software.

The simulations focused on evaluating velocity and pressure distributions downstream of the gate for different gate opening heights. These results were used to assess flow acceleration zones, pressure drops, and potential cavitation regions. Cavitation risk was estimated by comparing local pressure values with the vapor pressure of water.

In parallel, the performance of the Banki turbine was analyzed by considering the main influencing factors, which can be grouped into four categories:

1. Hydraulic factors – water head, flow velocity, and flow direction at the turbine inlet.
2. Geometrical and конструктив factors – turbine diameter, blade width, blade curvature, and number of blades.
3. Operational factors – maintenance conditions, bearing lubrication, and flow regulation stability.
4. Natural factors – sediment concentration, debris, and biological elements such as fish and aquatic plants.

Parametric studies were performed by varying the number of blades and the turbine diameter to identify optimal configurations for given hydraulic conditions. The turbine rotational speed and power output were estimated using standard hydropower equations:

$$P = \rho g Q H \eta$$

where $Q$is discharge, $H$is head, and $\eta$is efficiency.

Results

The numerical simulations revealed that as the gate opening increases, flow velocity downstream of the gate rises significantly, while static pressure decreases accordingly. This confirms the theoretical inverse relationship between velocity and pressure predicted by the Bernoulli equation. High-velocity zones were observed near the gate lip, accompanied by low-pressure regions that may cause cavitation under certain conditions.

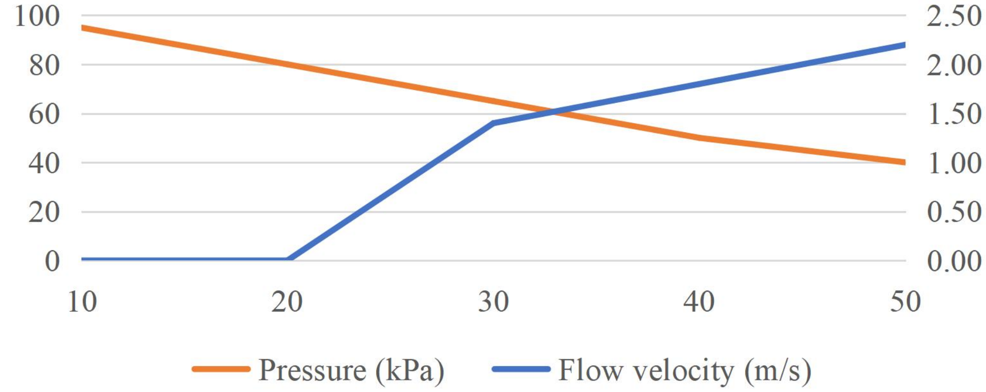

Figure. Relationship between gate opening, flow velocity, and pressure

The flow field exhibited nonlinear behavior, with turbulence intensity increasing as the gate opening and flow rate increased. This indicates that simple analytical models are insufficient for accurately predicting flow characteristics in such configurations, and numerical modeling is necessary for reliable design.

The analysis of the Banki turbine showed that turbine efficiency is highly sensitive to blade number and flow alignment. A low number of blades leads to incomplete energy extraction, whereas an excessive number of blades increases hydraulic resistance and mechanical losses. Simulation results indicated that an intermediate blade number provides the best compromise between energy capture and flow resistance.

Hydraulic factors, particularly inlet velocity and jet angle, were found to be dominant in determining turbine performance. Improper alignment of the jet with turbine blades resulted in flow separation and energy losses. конструктив optimization of blade curvature and spacing improved the effective use of kinetic energy.

Natural factors such as sediments and debris were identified as significant contributors to performance degradation and mechanical wear. Protective measures such as trash racks and sediment filters were shown to be necessary for long-term operation.

Discussion

The combined analysis of sluice gate flow and Banki turbine performance demonstrates the importance of integrated design in micro-HPP systems. Flow conditions created by hydraulic gates directly influence turbine efficiency and operational safety. If gate geometry and opening are not properly selected, unfavorable velocity and pressure distributions may occur, leading to increased cavitation risk and reduced turbine lifespan.

The results highlight the advantage of numerical simulation tools in predicting complex hydraulic phenomena. CFD modeling allows designers to visualize flow patterns, identify critical regions, and test different configurations without costly experimental setups. This approach supports the development of optimized turbine designs adapted to specific site conditions.

Moreover, the study confirms that turbine efficiency cannot be maximized solely by increasing flow rate or head. Instead, an optimal balance between hydraulic, mechanical, and operational parameters must be achieved. This aligns with modern engineering principles emphasizing system-level optimization rather than isolated component improvement.

From an environmental perspective, the use of Banki turbines in micro-HPPs offers a relatively low-impact solution compared to large dams. However, attention must be paid to fish protection and sediment management to ensure ecological compatibility.

Conclusion

This study demonstrates that numerical modeling of water flow through hydraulic gates combined with an analysis of Banki turbine efficiency factors provides a comprehensive framework for optimizing micro-hydropower systems. The simulations confirmed the inverse relationship between velocity and pressure, revealed nonlinear turbulent flow behavior, and identified potential cavitation zones.

The performance of Banki turbines was shown to depend strongly on hydraulic conditions, turbine geometry, and blade configuration. By selecting optimal parameters such as blade number, turbine diameter, and flow direction, hydraulic losses can be minimized and energy conversion efficiency improved.

The integration of CFD-based flow analysis with turbine performance evaluation supports the development of reliable and efficient engineering solutions for micro-HPPs. These findings contribute to the sustainable utilization of small-scale hydropower resources and provide practical guidance for the design and operation of micro-hydropower installations.

References:

1.White, F. M. (2011). Fluid Mechanics (7th ed.). McGraw-Hill, New York.
2.Munson, B. R., Young, D. F., Okiishi, T. H., & Huebsch, W. W. (2013). Fundamentals of Fluid Mechanics. John Wiley & Sons, Hoboken.
3.Çengel, Y. A., & Cimbala, J. M. (2014). Fluid Mechanics: Fundamentals and Applications. McGraw-Hill Education, New York.
4.Streeter, V. L., Wylie, E. B., & Bedford, K. W. (1998). Fluid Mechanics. McGraw-Hill, New York.
5.Bazhenov, L. B., & Dubrovsky, V. A. (2005). Hydraulics and Hydraulic Machines. Vysshaya Shkola, Moscow.
6.Mirzaev, N. T. (2018). Fundamentals of Hydraulics. National University of Uzbekistan Press, Tashkent.
7.Sidorov, A. P. (2010). Open Channel Hydraulics. Lan Publishing House, Saint Petersburg.
8.Mavlonov, M. M. (2017). Mathematical Modeling of Hydraulic Processes. Fan va Texnologiya, Tashkent.
9.Chow, V. T. (1959). Open-Channel Hydraulics. McGraw-Hill, New York.
10.Henderson, F. M. (1966). Open Channel Flow. Macmillan Publishing, New York.
11.Quaranta, E., Perrier, J. P., & Revelli, R. (2022). Optimal design process of cross-flow Banki turbines: Literature review and novel expeditious equations. Ocean Engineering, 257, 111582. https://doi.org/10.1016/j.oceaneng.2022.111582
12.Acharya, N., Kim, C. G., Thapa, B., & Lee, Y. H. (2015). Numerical analysis and performance enhancement of a cross-flow hydro turbine. Renewable Energy, 80, 819–826. https://doi.org/10.1016/j.renene.2015.01.064