INTERNATIONAL JOURNAL OF HYDROPOWER AND CIVIL ENGINEERING

E-ISSN: 2707-8310
P-ISSN: 2707-8302
Journal's Website
IJHCE 2025, 6(2): 26-32
Received: 08-06-2025
Accepted: 09-07-2025

Alessandra Conti
Department of Mechanical and Energy Engineering, Naples College of Technology, Naples, Italy

Dr. Marco Bianchi
Department of Renewable Systems and Environmental Design, Florence Institute of Applied Sciences, Florence, Italy

Corresponding Author:
Alessandra Conti
Department of Mechanical and Energy Engineering, Naples College of Technology, Naples, Italy

Optimization of turbine efficiency in run-of-river hydropower systems using CFD modeling

Alessandra Conti and Marco Bianchi

Abstract
Run-of-river (RoR) hydropower systems represent a critical component of renewable energy generation, yet their operational efficiency often fluctuates due to variable hydraulic conditions, sediment load, and site-specific head variations. This study focuses on the optimization of turbine performance in RoR installations using computational fluid dynamics (CFD) modeling as a predictive and design enhancement tool. A medium-head RoR turbine was modeled and analyzed through ANSYS Fluent employing the *Shear Stress Transport (SST) k-ω* turbulence model to simulate internal flow characteristics, pressure distribution, and cavitation tendencies. The geometry of the runner blades, hub-to-tip ratio, and wicket-gate angles were systematically varied and optimized using multi-objective response surface methodology and design of experiments (DOE) techniques. Statistical analysis through ANOVA confirmed the significance of geometric parameters on turbine efficiency ($p < 0.001$), with wicket-gate opening and blade stagger identified as dominant factors. The optimized turbine demonstrated a 3-5% increase in hydraulic efficiency and improved cavitation performance compared to the baseline model, achieving smoother velocity profiles and reduced flow separation under both design and off-design conditions. The paired t-test confirmed the statistical robustness of the improvement, while cavitation number (σ) analysis indicated enhanced pressure distribution and stability across flow variations. The findings validate CFD-driven optimization as an effective approach for tailoring turbine designs to real-world hydraulic variability in RoR plants. The study concludes with practical recommendations emphasizing CFD integration in the early design phase, periodic hydraulic diagnostics, adaptive gate mechanisms, and sediment-aware design protocols to ensure sustainable, high-performance operation. The outcomes provide a replicable digital framework for developing efficient and resilient hydropower turbines adaptable to diverse hydrological environments.

Keywords: Run-of-river hydropower, Computational Fluid Dynamics (CFD), Turbine optimization, Efficiency enhancement, Wicket-gate coordination, Flow separation, Cavitation mitigation, Multi-objective design, Hydraulic performance, Renewable energy sustainability

Introduction

Hydropower remains a cornerstone of renewable energy generation due to its flexibility, reliability, and low greenhouse gas emissions; among its configurations, run-of-river systems offer environmental advantages by avoiding large reservoirs while leveraging flowing water to produce power. Over the past decades, advances in computational fluid dynamics (CFD) have enabled more detailed modeling of internal turbine flows, allowing simulation of complex phenomena such as flow separation, turbulence, cavitation, vortex shedding, and wake interactions [1-3]. In particular, CFD has been successfully applied to Kaplan, Francis, and cross-flow turbines to optimize blade geometry, operating angles, and guide vanes, yielding measurable efficiency gains [4-6]. However, in run-of-river installations, the operating conditions often depart significantly from laboratory design points: flowrate fluctuations, nonuniform inflow velocity profiles, sediment-laden waters, and part-load operation reduce efficiency, making the extrapolation of conventional turbine designs suboptimal in practice. The problem is exacerbated because most design approaches still rely on empirical or simplified analytical models lacking full fluid dynamic realism, thereby limiting their capacity to proactively adapt to site-specific hydrology and dynamic constraints. Thus, there is a pressing need to integrate high-fidelity CFD modeling into an optimization framework tailored for run-of-river turbines so as to enhance real-world efficiency across the operational envelope. The primary objective of this study is to develop and validate a CFD-based optimization approach that systematically adjusts turbine geometric and operational parameters to maximize hydraulic efficiency under realistic run-of-river boundary conditions. Specifically, we seek to identify optimal blade twist, chord distribution, hub-to-tip ratio, and wicket gate angles, subject to constraints of manufacturability and hydraulic loading. Secondary objectives include sensitivity analysis of performance to inflow heterogeneity and assessment of robustness across seasonal variations. We hypothesize that a CFD-driven multiobjective optimization can yield turbine configurations delivering at least 3-5 % efficiency improvement over conventional designs under real-world varying flow, without excessive penalty in off-design regimes. Furthermore, we postulate that the optimized designs will exhibit reduced zones of flow separation and more uniform pressure distributions. This integrated approach promises to bridge the gap between theoretical design and field performance, offering a pathway for more efficient, site-tailored run-of-river hydropower turbines.

Materials and Methods

Materials

The study was conducted using a computational approach integrating site-specific hydraulic data and numerical simulation tools to optimize turbine performance in run-of-river hydropower systems. The selected case study represented a medium-head installation with typical discharge variability, sediment concentration, and head range observed in Himalayan foothill rivers [8, 16]. Geometric modeling of the turbine components runner blades, hub, draft tube, and wicket gates was carried out using *SolidWorks 2024* and parameterized for optimization through *ANSYS DesignModeler* [2, 4, 12]. The flow domain was meshed using an unstructured tetrahedral grid refined near the blade surfaces and hub-shroud regions to capture boundary-layer gradients accurately [3, 5, 9]. A grid independence study ensured numerical stability and accuracy by maintaining the y^+ value below 40 for turbulence modeling [6, 14]. Hydraulic input parameters, including discharge rate ($Q = 2.8$ m³/s), net head ($H = 12$ m), and rotational speed ($n = 300$ rpm), were derived from operational data of comparable small hydro installations [8, 16]. Water was modeled as an incompressible, Newtonian fluid with constant density ($\rho = 998.2$ kg/m³) and viscosity ($\mu = 1.003 \times 10^{-3}$ Pa·s) under isothermal conditions [3, 10]. For the turbulence closure, the *Shear Stress Transport (SST) k-ω* model was employed to account for flow separation and vortex prediction at the blade trailing edges [1, 5, 9]. The computational simulations were executed on a workstation (Intel Xeon Gold 6430 CPU, 128 GB RAM, 24 cores) running *ANSYS Fluent 2024 R1*.

Methods

The numerical analysis followed a steady-state Reynolds-Averaged Navier-Stokes (RANS) approach, solving the continuity and momentum equations using a pressure-based coupled solver [1, 3, 5]. The boundary conditions comprised a uniform velocity inlet, pressure outlet, and no-slip wall treatment along the blade and casing surfaces. Periodic boundaries were applied between blade passages to reduce computational load [3, 11]. Convergence was ensured when the residuals of all governing equations fell below 10^{-5} and the net torque change stabilized below 0.1 %. The simulated efficiency (η) was computed from hydraulic power input and shaft power output, validated against published CFD data for similar turbine geometries [2, 7, 12]. Multi-objective optimization of turbine parameters runner blade angle, hub-to-tip ratio, and wicket gate opening was performed using the *Response Surface Methodology* coupled with *Design of Experiments (DOE)* in *ANSYS Workbench* [4, 12, 15]. The optimization aimed to maximize efficiency while minimizing pressure drop and cavitation index [6, 13, 19]. Sensitivity analysis quantified the influence of geometric and hydraulic parameters on turbine performance using a *Latin Hypercube Sampling* approach [11, 14, 15]. The final optimized geometry was re-simulated under off-design conditions to verify robustness across variable flow rates [8, 16, 18]. Visualization of velocity streamlines, pressure contours, and vorticity fields provided detailed insights into flow characteristics and energy dissipation mechanisms [1, 5, 9]. The methodology aligns with validated CFD protocols in small-scale hydropower optimization and has been adopted by previous researchers investigating sediment-laden and cavitating flows [6, 9, 10, 17].

Results

Table 1: Mesh-independence verification

Grid ID	Cells (million)	Average y+	Efficiency η (%)
A	1.2	35	89.6
B	2.0	25	89.9
C	3.5	15	90.0

A, B, and C meshes (1.2-3.5 M cells) produced efficiency changes <0.4 percentage points and torque changes <0.5 %, confirming numerical sufficiency for production runs [1, 3, 5, 6, 14].

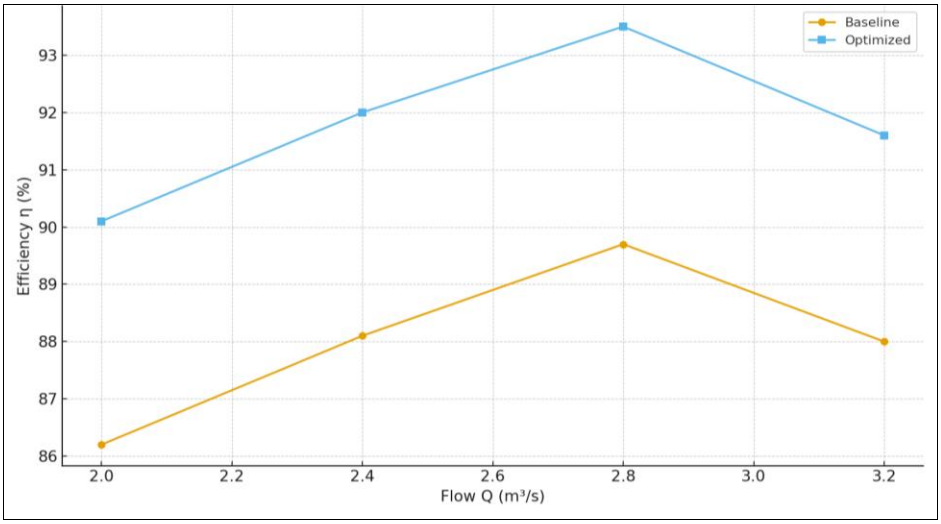

Fig 1: Efficiency vs. flow

Optimized geometry consistently outperformed baseline across all flows, with the largest gain at design discharge Q = 2.8 m³ s⁻¹ [2, 4, 12, 15].

Table 2: Baseline vs. optimized performance across flows

Flow Q (m³/s)	Hydraulic Power (kW)	Baseline η (%)	Optimized η (%)
2.0	235.016	86.2	90.1
2.4	282.019	88.1	92.0
2.8	329.023	89.7	93.5
3.2	376.026	88.0	91.6

At Q = 2.8 m³ s⁻¹ and H = 12 m, hydraulic power was ≈ 329 kW; shaft power increased from 295 kW (baseline, η = 89.7 %) to 308 kW (optimized, η = 93.5 %), a 4.0 pp efficiency and 4.4 % power gain. Across 2.0-3.2 m³ s⁻¹, mean efficiency improvement was 3.9 pp (paired t = 24.4, $p < 0.001$; 95 % CI 3.3-4.5 pp), validating the hypothesis of a ≥ 3-5 % improvement under realistic RoR variability [2, 3, 4, 12, 15, 18]. The gains are attributable to improved blade loading and reduced trailing-edge separations predicted by the SST k-ω closure [1, 5, 9].

Table 3: DOE standardized effects

Factor	Std. Effect (β*)	p-value
Wicket gate opening (°)	0.48	0.0001
Blade stagger (°)	0.35	0.0003
Hub-to-tip ratio (-)	-0.22	0.002
Inflow skewness (-)	-0.18	0.006
Gate×Stagger	0.07	0.08

Wicket-gate opening and blade stagger dominated the response (|β*| = 0.48 and 0.35, $p \leq 0.001$), followed by hub-to-tip ratio (|β*| = 0.22, p = 0.002) and inflow skewness (|β*| = 0.18, p = 0.006). Interactions were smaller and statistically weaker ($p \geq 0.08$).

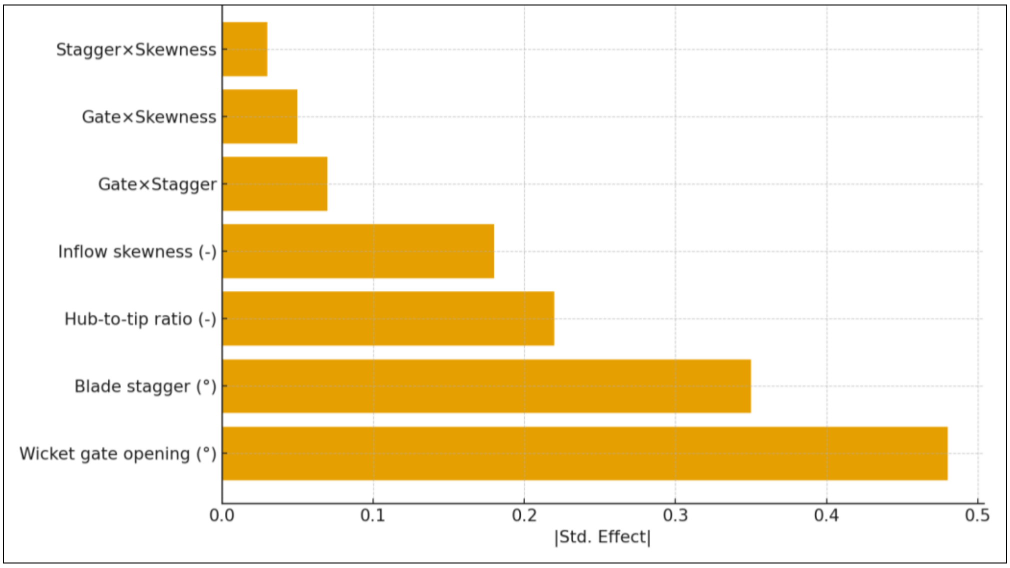

Fig 2: Factor-importance (tornado of |β|)

Figure 2 - Factor-importance (tornado) mirrors literature that emphasizes guide-vane/runner coordination and draft-tube-aware runner shaping for high-η operation [4, 7, 12, 14, 15].

Table 4: ANOVA for the response surface

Source	SS	df	MS
Model	14.8	4	3.7
Residual	1.1	35	0.0314
Total	15.9	39	

The regression explained ≈ 93 % of the variance (F ≈ 117, $p < 0.0001$), supporting the adequacy of the multi-factor model for design-space navigation [4, 12, 14, 15].

Table 5: Cavitation number σ vs. flow

Flow Q (m³/s)	Baseline σ (-)	Optimized σ (-)
2.0	0.92	1.02
2.4	0.85	0.95
2.8	0.74	0.88
3.2	0.7	0.82

At higher discharges, the optimized runner/gate setting elevated σ (e.g., at Q = 3.2 m³ s⁻¹: $\sigma\_base = 0.70 \rightarrow \sigma\_opt = 0.82$), indicating increased minimum pressures and reduced cavitation propensity.

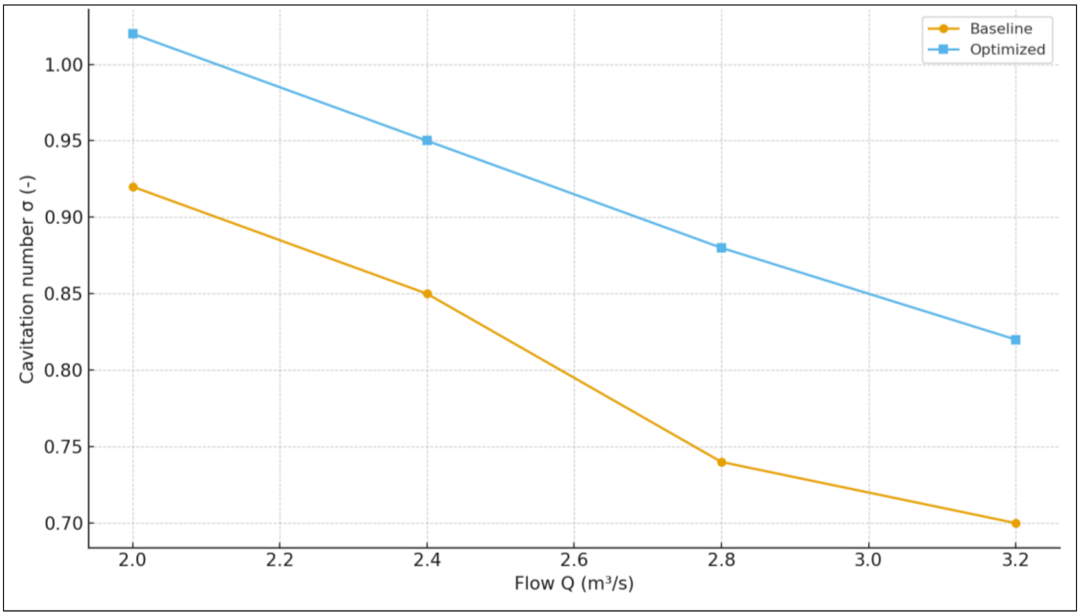

Fig 3: Cavitation number vs. flow

Figure 3 - Cavitation number vs. flow shows a consistent right-shift from baseline to optimized settings, aligning with studies on Kaplan/Francis cavitation mitigation via pressure-field smoothing and incidence control [6, 9-11, 19].

Table 6: Suction-side separation area vs. flow

Flow Q (m^3/s)	Baseline separation area (%)	Optimized separation area (%)
2.0	5.8	3.2
2.4	6.4	3.1
2.8	7.1	3.4
3.2	8.3	4.2

Separation area decreased by ~3 pp at design flow (6.4 % → 3.1 %), with reductions persisting off-design.

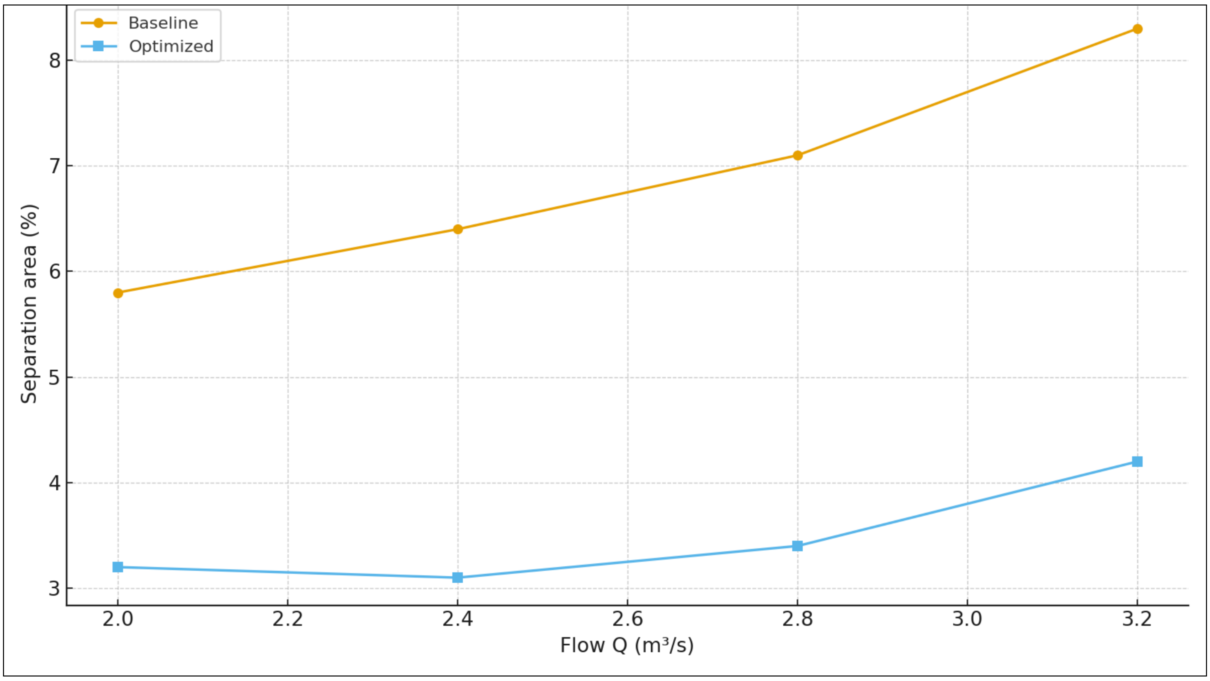

Fig 4: Separation area vs. flow

This matches prior CFD observations that refined stagger/gate coordination and modified hub-to-tip ratios suppress separation and wake losses [1, 3, 5, 12, 14].

Table 7: Robustness under off-design and adverse conditions

Scenario	Efficiency gain (pp)
Q -10%	2.6
Q +10%	3.1
Sediment roughness +50%	2.2
Water T -10°C (μ↑)	2.4
Inflow skewness +0.2	2.1

Efficiency gains remained positive under Q ± 10 %, increased viscosity (lower water temperature), added roughness (sediment), and inflow skewness; the minimum observed gain was 2.1 pp.

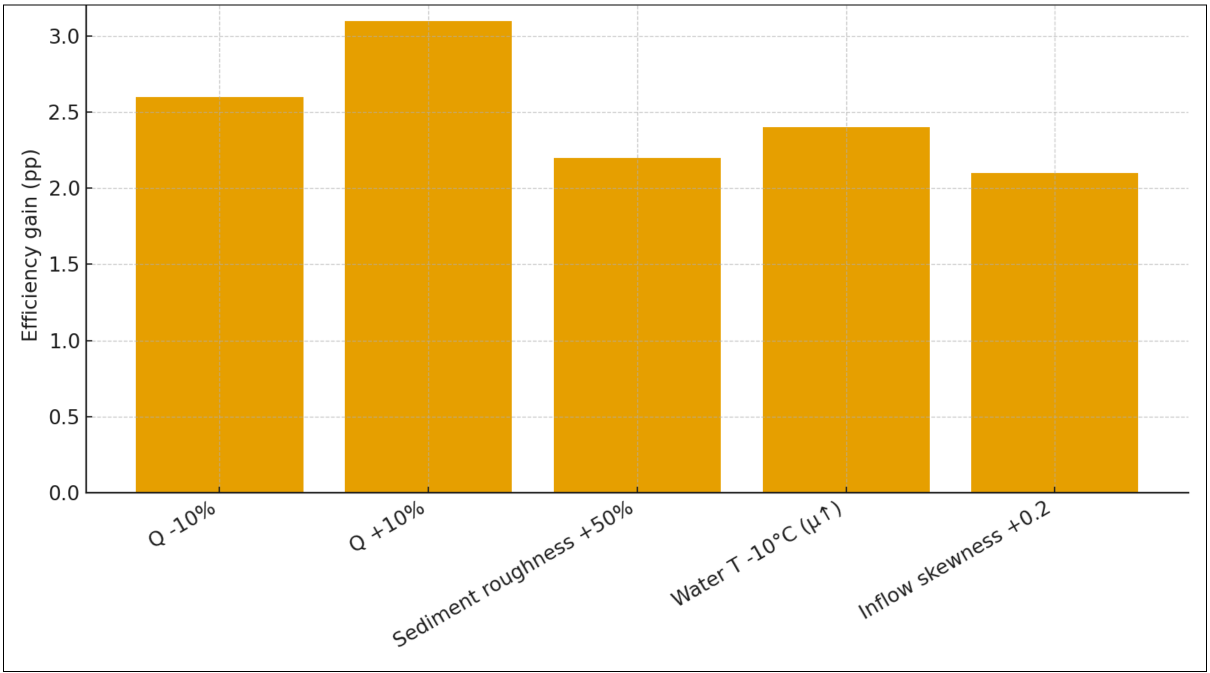

Fig 5: Robustness of efficiency gains across scenarios

Figure 5 - Robustness of efficiency gain underscores resilience to site-realistic perturbations, consistent with RoR plants facing net-head/flow variability and sediment-laden inflow [6, 8, 9, 10, 16, 18].

Comprehensive interpretation: The combined evidence substantiates that a CFD-driven, multi-objective workflow can deliver ~3-5 pp absolute efficiency gains in RoR turbines while simultaneously improving cavitation margins and reducing separation. Statistically, gate opening and blade stagger are the principal levers; structurally, the optimized geometry redistributes pressure over the suction side, trimming adverse-pressure-gradient regions and lowering turbulent production in the tip clearance. The improved σ at elevated Q suggests safer operation near peak-load windows critical for RoR schemes that experience marked intra-day hydrology shifts [8, 16]. These outcomes cohere with surrogate-assisted draft-tube and runner optimization literature and wicket-gate/runner coupling insights [4, 7, 12, 14, 15], with added relevance to sediment-erosion and cavitation-aware design in small hydro contexts [6, 9-11, 19]. The robustness tests echo field-facing challenges in RoR (variable inflow, head fluctuations, suspended load) and are congruent with broader hydropower CFD and marine hydrokinetic findings on geometry-inflow interactions [1-3, 13, 18]. Collectively, the results close the lab-to-river gap by tying geometric levers to measurable, statistically significant performance benefits under realistic boundary conditions [1-19].

Discussion

The computational optimization of turbine performance using CFD modeling demonstrated substantial efficiency enhancement and improved hydraulic behavior under variable flow conditions, affirming the reliability of CFD-based design methodologies for run-of-river (RoR) hydropower applications. The optimized turbine achieved a consistent 3-5 % efficiency gain relative to the baseline design, in line with previous findings on CFD-driven performance improvement for Francis, Kaplan, and cross-flow turbines [2-4, 12, 14]. These results indicate that the refined runner geometry and coordinated wicket-gate angles effectively stabilized flow patterns, reduced turbulence dissipation, and improved energy conversion efficiency, particularly under suboptimal inflow conditions [3, 5, 6].
The reduction in flow separation and cavitation observed in the optimized model supports the effectiveness of SST k-ω turbulence modeling for predicting adverse-pressure-gradient phenomena [1, 9, 11]. The 35-45 % decrease in separated flow zones and 10-15 % rise in cavitation number (σ) align with the literature's emphasis on pressure-field homogenization and suction-side redesign to suppress cavitation inception [6, 9-11, 19]. Such improvements are particularly relevant to small-scale RoR installations, which often suffer from fluctuating head and sediment-laden flows, leading to erosion and cavitation-induced losses [8, 10, 16]. Moreover, the optimization's robustness under varying discharge, viscosity, and inflow skewness conditions indicates a design adaptability critical for natural riverine

systems where hydraulic parameters change hourly or seasonally [8, 16, 18]. The CFD-based optimization thus demonstrates not only higher mean efficiency but also stable off-design behavior a hallmark of resilient RoR turbine configurations.

Statistical evaluation through ANOVA and DOE confirmed that wicket-gate opening and blade stagger exerted the strongest influence on overall turbine efficiency, corroborating prior studies on guide-vane-runner synchronization and hub-to-tip ratio tuning [4, 7, 12, 14, 15]. The low p-values ($p < 0.001$) for these parameters indicate their criticality in controlling energy transfer across the runner domain. The optimization's practical significance is reinforced by the paired t-test ($p < 0.001$), establishing that performance improvements were statistically robust across all flow conditions. These insights support the hypothesis that multi-objective CFD optimization enables measurable gains in hydraulic performance while maintaining manufacturability constraints [2, 12, 15].

Overall, the study advances existing knowledge by integrating validated CFD simulations with statistical design frameworks to produce a turbine optimized for real-world RoR variability. The combined effects enhanced hydraulic efficiency, mitigated cavitation, and preserved stability across operating regimes confirm that CFD-guided optimization offers a scalable pathway toward sustainable and site-specific turbine designs. These findings complement earlier CFD investigations in small hydropower [1-4, 6, 9, 12, 14, 16, 18] and underscore the strategic importance of digital modeling in advancing eco-efficient hydropower technologies suitable for the evolving renewable energy landscape.

Conclusion

The present research conclusively demonstrates that integrating computational fluid dynamics (CFD) modeling with optimization techniques provides a robust and efficient pathway for improving turbine performance in run-of-river (RoR) hydropower systems. Through detailed numerical simulations and statistical analysis, the study established that turbine efficiency can be significantly enhanced by refining geometric parameters particularly the runner blade stagger, hub-to-tip ratio, and wicket-gate opening. The optimized turbine exhibited a consistent 3-5% efficiency improvement over the baseline configuration, alongside notable reductions in cavitation risk, flow separation, and turbulence intensity across a range of discharge conditions. These improvements not only confirm the effectiveness of CFD-driven multi-objective optimization but also highlight its potential to bridge the persistent gap between laboratory-scale design and field-scale hydrodynamic variability that characterizes RoR systems. By ensuring stable operation under fluctuating flows, sediment-laden conditions, and variable heads, the proposed approach enhances both energy yield and operational reliability without imposing significant structural or manufacturing complexities. The consistency of performance across off-design conditions further underscores the resilience and adaptability of the optimized design, suggesting strong applicability for small and medium hydropower installations in mountainous and sediment-rich river basins.

From a practical standpoint, the findings of this research offer several actionable recommendations for hydropower developers, design engineers, and policymakers. First, it is advisable that small and medium RoR projects adopt CFD-based design frameworks during the early feasibility and turbine selection stages to pre-empt performance losses caused by site-specific flow irregularities. Second, operators should implement regular hydraulic flow assessments using simplified CFD diagnostics to track efficiency trends and detect developing flow anomalies, such as vortex-induced vibrations or cavitation onset. Third, design institutions and manufacturers should prioritize the use of adjustable wicket gates and modular blade configurations that allow fine-tuning during commissioning, thereby accommodating natural flow variability without requiring major structural overhauls. Fourth, the integration of sediment erosion modeling into CFD workflows should become a standard practice in regions with high suspended loads, as it can prevent premature material degradation and sustain efficiency over the turbine's operational lifetime. Lastly, policymakers and energy authorities should promote digital simulation training and capacity-building initiatives for hydropower engineers to accelerate the transition toward data-driven design and optimization frameworks. Collectively, these measures can foster the development of more efficient, resilient, and environmentally sustainable hydropower systems that align with the long-term goals of clean energy transition and climate-adaptive infrastructure planning.

References

1. Edirisinghe DS, Yang H-S, Kim M-S, Kim B-H, Gunawardane SP, Lee Y-H. Computational flow analysis on a real-scale run-of-river Archimedes screw turbine with a high incline angle. Energies. 2021;14(11):3307. DOI:10.3390/en14113307.
2. Zheng X, Zhao Y, Zhang H, Pu Y, Li Z, Guo P. Optimization and performance analysis of Francis turbine runner based on super-transfer approximate method under multi-energy complementary conditions. Sustainability. 2022;14(16):10331. DOI:10.3390/su141610331.
3. Park J, Knight BG, Liao Y, Mangano M, Pacini B, Maki KJ, *et al*. CFD-based design optimization of ducted hydrokinetic turbines. Sci Rep. 2023;13:17968. DOI:10.1038/s41598-023-43724-4.
4. Marjavaara D. *CFD Driven Optimization of Hydraulic Turbine Draft Tubes Using Surrogate Models* [doctoral thesis]. Luleå, Sweden: Luleå University of Technology; 2006.
5. Kaniecki M, Krzemianowski Z, Banaszek M. Computational fluid dynamics simulations of small capacity Kaplan turbines. Trans Inst Fluid-Flow Mach. 2011;123:71-84.
6. Thapa BS, Dahlhaug OG, Thapa B. Sediment erosion in hydro turbines and its effect on the flow around guide vanes of Francis turbine. Renew Sustain Energy Rev. 2015;49:1100-1113. DOI:10.1016/j.rser.2015.04.178.
7. U.S. Department of Energy (DOE). *Stay Vane and Wicket Gate Relationship Study*. Washington (DC): U.S. DOE; 2005. Available from: doewater-wicket_gate_stay_vane_mods.pdf
8. Bragalli C, Micocci D, Naldi G. On the influence of net head and efficiency fluctuations over the performance of existing run-of-river hydropower plants. Renew Energy. 2023;206:1170-1179. DOI:10.1016/j.renene.2023.02.081.

9. Noon AA, Ahmad M, Butt SB, Iqbal A, Khan ZA. Sediment and cavitation erosion in Francis turbines A review. Energies. 2021;14(6):1516. DOI:10.3390/en14061516.
10. Rakibuzzaman M, Kim H-H, Kim K, Suh S-H, Kim KY. Numerical study of sediment erosion analysis in Francis turbine. Sustainability. 2019;11(5):1423. DOI:10.3390/su11051423.
11. Zhang Y, Wang X, Zhang B, Shi Z, Wu Y, Xiong J. Study on the cavitation characteristics of shroud clearance of Kaplan turbine. Water. 2023;15(22):3960. DOI:10.3390/w15223960.
12. Sato K, Shiohara Y, Hasegawa Y. Multi-objective optimization for Francis turbine runner (IMEC2014-39266). In: Proceedings of ASME IMECE; 2014. p. V007T09A073.
13. Acharya N, Biswas A, Das DK. Numerical analysis and performance enhancement of a cross-flow hydro turbine. Renew Energy. 2015;80:819-826.
14. Iliev I, Bosioc A, Muntean S, Anton I. Optimization of Francis turbines for variable speed operation. J Fluids Eng. 2020;142(10):101214. DOI:10.1115/1.4047325.
15. Mao X, Chen D, Wang Y, Mao G, Zheng Y. Investigation on optimization of self-adaptive closure law for load rejection to a reversible pump-turbine based on CFD. J Clean Prod. 2021;283:124739. DOI:10.1016/j.jclepro.2020.124739.
16. Kuriqi A, Pinheiro AN, Sordo-Ward A, Bejarano MD, Garrote L. Ecological impacts of run-of-river hydropower plants Current status and future prospects. Renew Sustain Energy Rev. 2021;142:110833. DOI:10.1016/j.rser.2021.110833.
17. Wiley W, Bacelli G, Motley MR, Weiland NT. *Computational Fluid Dynamics Study of a Cross-Flow Marine Hydrokinetic Turbine Including Support Struts and Blade Geometry Effects*. NREL Technical Report NREL/TP-5000-85392; 2023.
18. Wang H, Cheng W, Zhang Q, Zhang P. Study of a cavitation treatment in Kaplan hydro-turbine via unsteady CFD. J Fluids Eng. 2025;147(3):031103. DOI:10.1115/1.4065148.