

Article

Mitigating Particle Erosion in Axial-Flow Turbines Through Air Injection at the Inlet Rotor Section

José Gustavo Coelho ^{1,*} , Rafael de Almeida ², Hermeson Conceição Wanzeler ³
and André Luiz Amarante Mesquita ³ 

¹ Department of Mechanical Engineering, Federal University of Triângulo Mineiro, Uberaba 38025-180, Brazil

² Department of Mechanical Engineering, Federal University of Espírito Santo, Vitória 29075-910, Brazil

³ Laboratory of Fluid Dynamics and Particulate, FluidPar, Federal University of Pará, Tucuruí 68455-901, Brazil; andream@ufpa.br (A.L.A.M.)

* Correspondence: jose.gustavo@uftm.edu.br

Abstract

This study presents a computational analysis of degradation caused by cavitation and hydro-abrasive erosion in a low-head axial microturbine ($H = 4$ m), incorporating strategic air injection as a passive mitigation technique. Using Computational Fluid Dynamics (CFD) within ANSYS CFX 2025 R2, the study investigates hydrodynamic performance and the spatial distribution of surface wear across the runner blades. The turbine geometry was developed from aerofoil profiles mapped onto cylindrical coordinates, using a structured three-dimensional mesh with localized refinement to ensure grid independence. Physical modeling employed the Shear Stress Transport (SST) turbulence model, with cavitation dynamics governed by the Rayleigh–Plesset equation and sediment transport modeled using a Lagrangian framework incorporating the Finnie erosion model. The numerical framework showed good agreement with reference characteristic curves, confirming its predictive accuracy. The results indicate that vapor cavities form predominantly on the suction side, whereas solid particle erosion highly concentrated on the pressure side of the blades, where the outer 20% of the span accounts for over 91% of the total erosion intensity. Parametric assessments of controlled air injection revealed a highly non-linear mitigation response, identifying IAVF 2 as the optimal air-injection case. This configuration reduced integrated erosion by 0.95% and maximum localized erosion by 6.17%. In contrast, excessive air volumes accelerated material removal due to localized flow distortion. The findings indicate that carefully controlled air injection is a viable strategy for extending the operational lifespan of small-scale hydropower assets.

Keywords: axial flow turbomachinery; multiphase flow CFD; cavitation-erosion synergy; air injection mitigation



Academic Editor: Krzysztof Rogowski

Received: 30 May 2026

Revised: 25 June 2026

Accepted: 29 June 2026

Published: 7 July 2026

Copyright: © 2026 by the authors.

Licensee MDPI, Basel, Switzerland.

This article is an open access article distributed under the terms and conditions of the [Creative Commons Attribution \(CC BY\)](https://creativecommons.org/licenses/by/4.0/) license.

1. Introduction

Axial turbines play a pivotal role in global energy generation, distinguished by their effectiveness in handling large water volumes under low-head conditions [1]. However, the integrity of these assets is challenged by cavitation, which, not only compromises hydraulic performance, but also triggers instabilities and structural damage [2]. Furthermore, the presence of suspended sediments intensifies abrasive wear, leading to increased maintenance costs [3]. Given this complexity, geometric optimization and flow analysis are vital for mitigating energy losses [4], particularly in sustainable energy recovery applications

within existing infrastructure—such as Wastewater Treatment Plants (WWTPs)—which can yield attractive financial returns [5].

Wear in these machines is a multifaceted phenomenon in which cavitation and erosion often coexist, accelerating surface degradation. According to Shrestha et al. [6], erosion removes material and alters the natural frequencies of the blades, inducing critical vibrations. This scenario is exacerbated in particle-laden flows, where the simultaneous action of these phenomena modifies surface roughness and compromises hydrodynamic stability [7]. Consequently, experimental studies are essential for predicting wear rates under varying sediment concentrations [8], while localized pressure drops promote material embrittlement through the implosion of vapor bubbles [9].

Computational Fluid Dynamics (CFD) has significantly advanced this field by enabling the analysis of phenomena that are difficult to observe experimentally while reducing prototyping costs [1]. Current models allow investigations ranging from inter-blade cavitation vortices and their associated pressure oscillations [10] to severe multiphase flows. In this regard, implementing modified drag and erosion models is fundamental to capturing particle trajectories and identifying zones of severe impact [11], thereby providing crucial data on material loss when experimentally validated [12].

The dynamics of particulate transport are intrinsically linked to vortex structures and secondary flows. Inter-blade vortices act as sediment-trapping mechanisms, directing particles toward the runner and intensifying cutting erosion [13]. This behavior depends on particle inertia (Stokes number), which dictates how particles deviate from streamlines to collide with solid boundaries [14]. In clearance regions, tip-leakage vortices concentrate erosion and alter pressure distributions [15], creating a complex flow regime that degrades overall system performance [7].

To mitigate the combined effects of surface degradation, active research has focused on flow-control strategies. Air injection acts as an effective passive erosion control mechanism through two primary physical phenomena [16,17]. First, it leverages a cushioning effect, wherein the presence of a dispersed gas phase near the wall reduces the effective mixture density. This localized density reduction significantly increases the drag force acting on incoming sand particles attempting to penetrate the fluid layer, thereby reducing their terminal impact kinetic energy (v_p^n) and subsequent indentation capability. Second, the mechanism provides hydrodynamic shielding by modifying the local pressure gradient (∇p) adjacent to the blade boundaries. According to the Lagrangian trajectory equations, the establishment of a positive pressure gradient normal to the wall generates an adverse force that diverts sediment trajectories away from the metallic substrate, drastically reducing the overall particle collision frequency (N) [16]. Furthermore, recent numerical simulations demonstrate that air injection significantly modifies near-wall flow characteristics, effectively suppressing sediment accumulation and reducing hydro-abrasive wear on critical runner components [18]. Experimental evidence also suggests that injecting compressed air into low-pressure regions disrupts localized vacuum conditions and increases pressure, providing a substantial cushioning effect that suppresses cavitation-induced noise and reduces structural vibration by up to 30% [19].

To ensure maximum mitigation coverage across the blade surfaces, the geometric location of the air admission is critically important. Recent advance in multiphase turbomachinery flow control, such as those demonstrated by Lu et al. [20], have shown that positioning air injection ports upstream of the runner, specifically within the adjustable guide vane and rotor inlet interface, substantially reduces in bubble attachment and local vapor volume fractions while simultaneously stabilizing pressure recovery fields [21,22]. While this upstream injection technique has been successfully used by Lu et al. to suppress cavitation dynamics, its application to hydro-abrasive sediment erosion has not yet been

explored in the literature. Guided by this rationale, the present numerical framework positions strategic air injection directly at the inlet rotor section, aiming to optimize the introductory volume fraction of the gaseous phase right at the threshold of the high-gradient hydrodynamic conversion zone. This strategic placement is designed to maximize particle trajectory deflection and cushioning efficiency before the multi-component mixture propagates into the critical peripheral regions of the runner blades. Furthermore, this approach is intended to evaluate mitigation effectiveness against solid particle wear, filling a gap in the literature while addressing the significant implementation challenges associated with alternative methods. Unlike direct air injection through blade surfaces—which involves prohibitive computational costs and high implementation complexity—the use of an inlet-based point source offers a streamlined, novel, and economically viable framework for assessing practical applicability in sediment-laden flows.

In view of these mechanisms and the challenges associated with accelerated wear in small hydropower plants, this work presents a comprehensive numerical investigation of concurrent cavitation and sediment erosion phenomena within a low-head ($H = 4$ m) axial microturbine. The primary novelty of this investigation lies in the geometric and operational decoupling of the air injection mechanism from the rotating elements of the turbine. While historical flow-control and cavitation-mitigation methodologies rely on injection air through intricate channels machined directly into the runner blades or draft tube surfaces [1,10], the framework developed herein utilizes discrete point-source injectors positioned entirely within the stationary inlet rotor domain. This upstream configuration shifts the mitigation mechanism from local blade-surface pressure modulation to macro-scale hydrodynamic deflection of the discrete phase. By establishing targeted localized flow perturbations upstream of the runner inlet, incoming solid particles are systematically deflected away from the critical blade leading edges. Consequently, this approach eliminates the need for complex structural or geometric modifications to the rotor blades, presenting an alternative engineering strategy that addresses hydro-abrasive wear through manufacturing simplicity and reduced installation costs.

2. Mathematical Model

2.1. Governing Equations

The fluid flow is modeled using a three-phase mixture approach (liquid water, water vapor, and air), governed by the conservation equations for mass and momentum [18,19]. The continuity and momentum equations for the multi-phase mixture are defined as follows:

$$\frac{\partial \rho_m}{\partial t} + \nabla \cdot (\rho_m \mathbf{v}_m) = 0 \quad (1)$$

$$\begin{aligned} \frac{\partial (\rho_m \mathbf{v}_m)}{\partial t} + \nabla \cdot (\rho_m \mathbf{v}_m \mathbf{v}_m) = & -\nabla p + \nabla \cdot [\mu_m (\nabla \mathbf{v}_m + \nabla \mathbf{v}_m^T)] + \rho_m \mathbf{g} \\ & + \mathbf{F} + \nabla \cdot \left(\sum_{k=1}^n \alpha_k \rho_k \mathbf{v}_{dr,k} \mathbf{v}_{dr,k} \right) \end{aligned} \quad (2)$$

where ρ_m represents the mixture density [kg/m^3], $\mathbf{v}_m$ is the mass-averaged velocity vector [m/s], p is the static pressure [Pa], μ_m is the mixture dynamic viscosity [$\text{Pa}\cdot\text{s}$], $\mathbf{g}$ is the gravitational acceleration, and $\mathbf{F}$ is the body force [18,19]. The term $\mathbf{v}_{dr,k}$ represents the drift velocity for the k -th phase and account for the relative slip between the dispersed bubbles and the continuous liquid phase [18].

The dynamics of the secondary phases, specifically water vapor and injected air, are governed by their respective volume fraction transport equations [18,19]:

$$\frac{\partial(\alpha_k \rho_k)}{\partial t} + \nabla \cdot (\alpha_k \rho_k \mathbf{v}_m) = -\nabla \cdot (\alpha_k \rho_k \mathbf{v}_{dr,k}) + S_k \quad (3)$$

For the vapor phase, the source term S_k corresponds to the net mass transfer rate, $R_e - R_c$, representing the evaporation and condensation processes associated with cavitation [18]. For the air phase, S_k accounts for the air injection source implemented via the point-source boundary condition in the runner domain [18,19].

The mixture properties, namely density (ρ_m) and dynamic viscosity (μ_m), are determined from the volume fractions weighted contributions (α_k) of the liquid (l), vapor (v), and air (a) phases:

$$\rho_m = \alpha_v \rho_v + \alpha_a \rho_a + (1 - \alpha_v - \alpha_a) \rho_l \quad (4)$$

$$\mu_m = \alpha_v \mu_v + \alpha_a \mu_a + (1 - \alpha_v - \alpha_a) \mu_l \quad (5)$$

where α_k satisfies the condition $\sum \alpha_k = 1$ [18,19]. The variables ρ_k and μ_k denote the density and dynamic viscosity of each phase, respectively.

To close the transport equations and model turbulence effects within the multiphase flow, the Shear Stress Transport (SST) model was adopted. This two-equation model combines the robustness and accuracy of the $k - \omega$ formulation in near-wall regions with the free-stream independence characteristic of the $k - \epsilon$ model away from solid boundaries. The selection of the SST model is justified because of its superior ability to predict the onset and extent of flow separation under adverse pressure gradients, which commonly occur in turbomachinery operating under cavitating and sediment-laden conditions. Consequently, this approach provides a more reliable prediction of viscous stresses and the resulting velocity field [1].

The continuous flow field and multi-component transport equations are resolved using a steady-state formulation to achieve numerical convergence across the four coupled phases: continuous liquid water, water vapor, discrete solid sediments, and the injected gas phase. Although cavitation dynamics and particle impacts are inherently transient, this algebraic configuration provides a necessary baseline control for managing the high computational demands of the multiphase system. By filtering highly localized, time-dependent fluctuations, the model isolates the macro-scale hydrodynamic trajectories required to evaluate the physical viability of the upstream air-injection technique, while the resolution of micro-scale, transient interactions is left for future studies.

2.2. Boundary Conditions

The full computational domain and its respective boundary conditions are illustrated in Figure 1. The boundary types were assigned according to their physical roles within the system. For all solid surfaces, impermeability and no-slip condition were enforced, ensuring that the fluid velocity at the wall was zero and that interaction occurred exclusively through viscous shear stresses. A total pressure condition was prescribed at the domain inlet, while a mass flow rate condition was imposed at the outlet to ensure consistency with the experimental test bench conditions. The local gravitational acceleration was set to 9.782 m/s^2 , corresponding to the experimental site in Itajubá.

To provide closure for the turbulence transport equations, the inlet flow was specified with a turbulence intensity of 5%, a standard value for internal subsonic flows that enables accurate calculation of the production and dissipation terms. Cavitation phenomena were modeled using the Rayleigh–Plesset model, with water vapor incorporated as a secondary phase. The physical properties were calculated based on a fluid temperature of 25°C , with

the saturation pressure defined at 3169 Pa. Phase change dynamics were controlled by a condensation coefficient of 0.01 and a vaporization coefficient of 50, which regulate the growth and collapse rates of the vapor cavities. The vapor phase density was maintained at 0.02308 kg/m^3 .

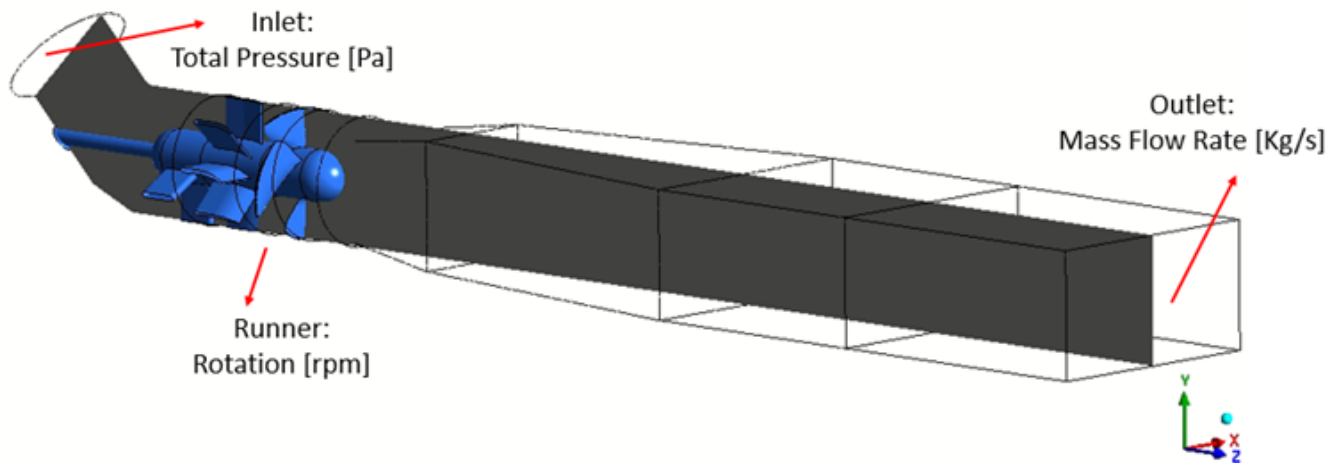


Figure 1. Full computational domain and boundary condition setup. The model features a runner with an outer diameter of 0.2888 m, an inner diameter of 0.112 m, a length of 0.125 m, and a nose cone (ogive) length of 0.0715 m.

The transport of particulate matter was modeled as a dispersed phase, with sand acting as the solid material. A material density of 2650 kg/m^3 was assigned to the particles to match the sediment properties found in the Madeira River (Porto Velho region, Brazil), as reported by Pires [23]. Based on the seasonal river data documented by Pires [23], a realistic sediment concentration of 0.85 g/L was adopted to model the interaction between the continuous water flow and the dispersed solid phase. Finally, Table 1 outlines the parametric evaluation conducted as a function of the total number of tracked particles to ensure statistical convergence of the erosion predictions.

Table 1. Sensitivity study of particle injection on maximum erosion depth.

Number of Particle Injections (—)	Maximum Erosion (10^{-4} m)
2000	19.3
300,000	2.69
600,000	1.83
900,000	1.42
1,200,000	1.35

Due to the direct correlation between the number of particle injections and the overall computational overhead, a sensitivity analysis was performed to determine an optimal injection count. An injection quantity of 9×10^5 was selected for the final simulations, as the resulting erosion data exhibited a variance of less than 5% compared to higher injection counts, indicating a stabilized numerical behavior. Consequently, this value ensures a rigorous characterization of the erosive process while optimizing computational resources.

2.3. Cavitation Model

The Rayleigh–Plesset model is employed to simulate cavitation using a homogeneous multiphase approach, with mass transfer rates estimated from bubble dynamics. The model

describes the growth and collapse of a spherical vapor bubble in a liquid, as defined by Equation (6):

$$R_B \frac{d^2 R_B}{dt^2} + \frac{3}{2} \left(\frac{dR_B}{dt} \right)^2 + \frac{2\sigma}{\rho_l R_B} = \frac{p_v - p}{\rho_l} \quad (6)$$

where R_B is the bubble radius, ρ_l is the liquid density, and σ is the surface tension coefficient between the liquid and vapor phases. This equation is derived from a mechanical force balance. Furthermore, the mass transfer rates for vaporization (R_e) and condensation (R_c) are determined as follows:

$$R_e = F_e \frac{3\alpha_{nuc}(1 - \alpha_v)\rho_v}{R_B} \sqrt{\frac{2}{3} \frac{|p_v - p|}{\rho_l}} \quad \text{if } p < p_v \quad (7)$$

$$R_c = F_c \frac{3\alpha_v \rho_v}{R_B} \sqrt{\frac{2}{3} \frac{|p_v - p|}{\rho_l}} \quad \text{if } p > p_v \quad (8)$$

where p_v is the saturation vapor pressure, α_{nuc} is the nucleation site volume fraction, and F_e and F_c are empirical constants for evaporation and condensation, respectively [24,25].

2.4. Particle Dynamics and Erosive Wear

The transport of the solid phase, comprising sand sediments, was modeled using a Lagrangian framework within the Discrete Phase Model (DPM) approach [24]. Individual particle trajectories were determined by integrating the force balance per unit mass, as formulated in Equation (9):

$$\frac{du_p}{dt} = F_D(u - u_p) + \frac{g(\rho_p - \rho)}{\rho_p} \quad (9)$$

where u_p denotes the particle velocity, $F_D(u - u_p)$ represents the unit drag force, and g is the gravitational acceleration. Given the dilute nature of the mixture (1% mass concentration), a one-way coupling scheme was implemented. In this regime, the continuous phase dictates the particle motion, while the influence of the discrete phase on the main flow field remains negligible [26]. The reliability of this coupling was assessed via the Stokes number (Stk), defined in Equation (10):

$$Stk = \frac{\rho_p d_p^2 u}{18\mu D} \quad (10)$$

In this expression, d_p is the particle diameter and D corresponds to the characteristic rotor diameter [27]. In the present study, the maximum Stokes number obtained was 1.6×10^{-3} for a particle diameter of 27 μm . This result ($Stk \ll 1$) confirms that the sediment particles closely follow the fluid streamlines [14], characterizing a regime dominated by velocity field alignment.

Surface degradation due to erosion was predicted using the Finnie model, which attributes material removal to the kinetic energy of impacting particles and their impingement angle [28]. The erosion rate E is quantified by Equation (11):

$$E = kv_p^n f(\gamma) \quad (11)$$

The impact angle function, $f(\gamma)$, is defined in Equation (12):

$$f(\gamma) = \begin{cases} \sin(2\gamma) - 3\sin^2(\gamma) & \text{if } \tan(\gamma) < 1/3 \\ \frac{1}{3}\cos^2(\gamma) & \text{if } \tan(\gamma) \geq 1/3 \end{cases} \quad (12)$$

Within ANSYS CFX, Equation (11) was rewritten as Equation (13) to obtain a dimensionless erosion factor:

$$E = \left(\frac{v_p}{v_0} \right)^n f(\gamma) \quad (13)$$

where v_0 represents the empirical reference velocity [m/s] used to scale the magnitude of the impact kinetic energy, effectively absorbing the material-dependent calibration constants [24]. In this investigation, the reference velocity was defined as $v_0 = 2010$ m/s, and the exponent was set to $n = 2.0$.

Finnie's empirical model was selected to balance predictive accuracy and computational efficiency. Given that the coupled continuous flow and cavitation phase-change equations require substantial numerical resources, adopting this simplified model is justified.

3. Validation of Erosive Wear Modeling

3.1. Case Study

The numerical methodology was validated by replicating the experimental conditions established by [29]. The computational domain was designed to replicate the discharge nozzle and test specimen under a 90° direct impingement configuration (Figure 2). The specific operational parameters used in the validation study are summarized in Table 2.

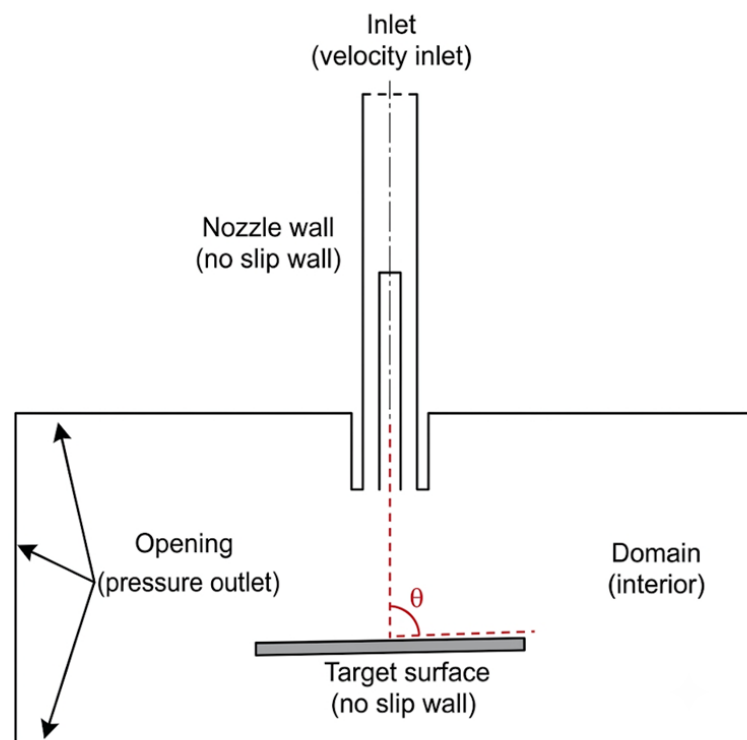


Figure 2. Boundary conditions.

Table 2. Operational parameters for submerged erosion testing validation.

Parameter	Value
Nozzle diameter	7 mm
Nozzle-to-sample distance	12.7 mm
Mean nozzle velocity	14 m/s
Specimen material	Stainless steel 316
Nominal sand diameter	300 μ m
Sand concentration (by mass)	1%
Impact angle	90°

3.2. Numerical Procedure

The numerical simulation employed a Lagrangian particle tracking approach, using particles with a diameter of 300 μm and a density of 2650 kg/m^3 . Given the dilute nature of the mixture (1% mass concentration), a one-way coupling scheme was implemented [26]. This setup ensured that fluid–solid interactions were accurately captured while maintaining computational efficiency.

The geometry was discretized using ANSYS CFX. A mesh independence test was performed across three nodal densities (0.77, 2.9, and 6.8 million nodes). A comparison of the thickness loss (μm) is presented in Figure 3. To optimize computational cost, the 2.9-million-node mesh was adopted. RMS residuals were set to 10^{-4} . Near-wall refinement ensured y^+ values < 5 , maintaining the validity of the SST model wall functions and ensuring that the velocity profile responsible for transporting particles to the impact region was accurately captured.

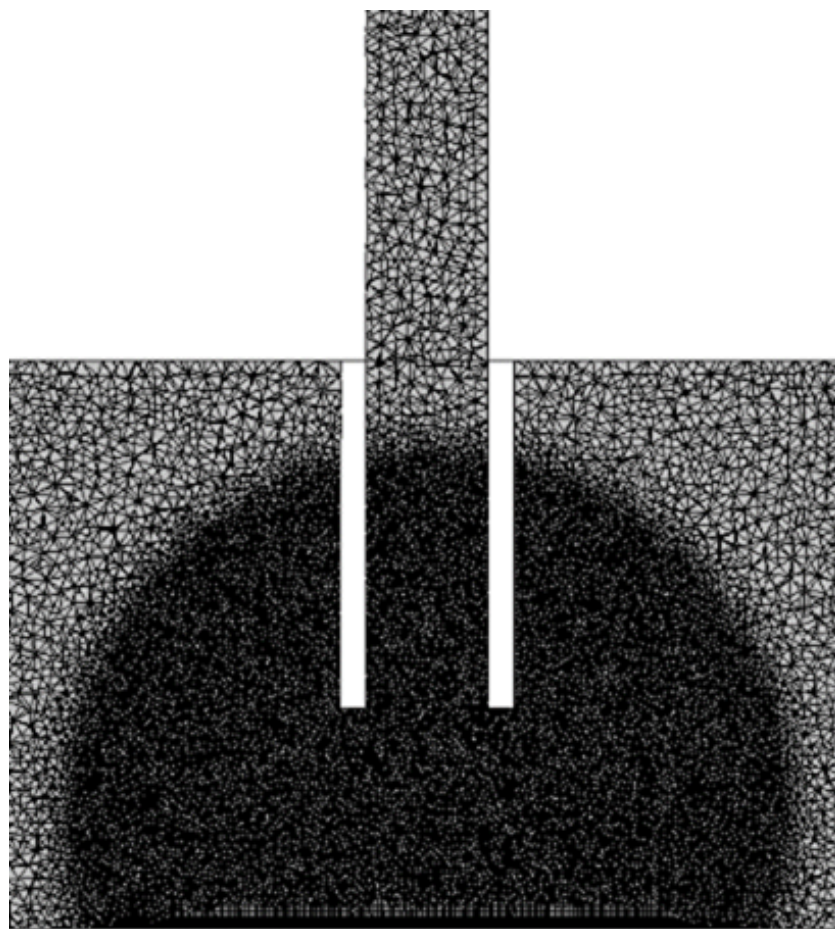


Figure 3. Computational mesh implemented for the solid particle transport.

To ensure that the erosion predictions were statistically independent of sample size, a sensitivity analysis of the number of injected particles was performed. Three distinct populations were evaluated: 150,000, 300,000, and 350,000 particles. The numerical results demonstrated that the maximum deviation in predicted thickness loss between the 300,000 and 350,000 particle sets was below 5%, indicating asymptotic convergence. Consequently, a population of 300,000 particles was established as the optimal threshold, providing a statistically representative characterization of the erosive phenomenon while maintaining computational cost-effectiveness. These sensitivity metrics are compiled and presented in Table 3.

Table 3. Sensitivity study of particle injection on maximum erosion depth.

Number of Particle Injections (–)	Maximum Erosion (10^{-4} m)
150,000	2.84
300,000	2.69
350,000	2.61

3.3. Validation Results

The fidelity of the computational framework was verified by benchmarking the numerical predictions against the experimental measurements reported by [29]. To quantify erosive degradation, the depth of material removal was calculated as a function of localized mesh parameters, as defined in Equation (14):

$$h_i = \frac{E_{rate,i} \cdot t}{\rho_{mat} \cdot A_i} \quad (14)$$

where h_i represents the localized erosion depth at mesh element i , $E_{rate,i}$ denotes the mass flow rate of particles impacting the surface [kg/s]; t is the exposure duration [s]; ρ_{mat} is the density of the AISI 316 stainless steel specimen [kg/m³]; and A_i is the face area of the corresponding element [m²]. Since ANSYS CFX outputs wear data through the erosion rate density (E) variable [kg/(m²·s)], Equation (14) was simplified to facilitate direct depth profiling, as shown in Equation (15):

$$h_i = \frac{E \cdot t}{\rho_{mat}} \quad (15)$$

The correlation between the experimental data and the numerical results is illustrated in Figure 4. The simulation successfully replicated the physical phenomenon, accurately predicting both the peak erosion magnitude (approximately 140 μ m) and the characteristic “W-shaped” wear topography.

This characteristic “W shaped” profile is a direct consequence of the hydrodynamic influence on sediment trajectories. As the jet approaches the target, the fluid velocity decreases sharply, reaching a minimum at the stagnation point (the center of the impact zone). Consequently, the particle kinetic energy is dissipated in this region, leading to a localized reduction in thickness loss, as evidenced by the velocity field shown in Figure 4.

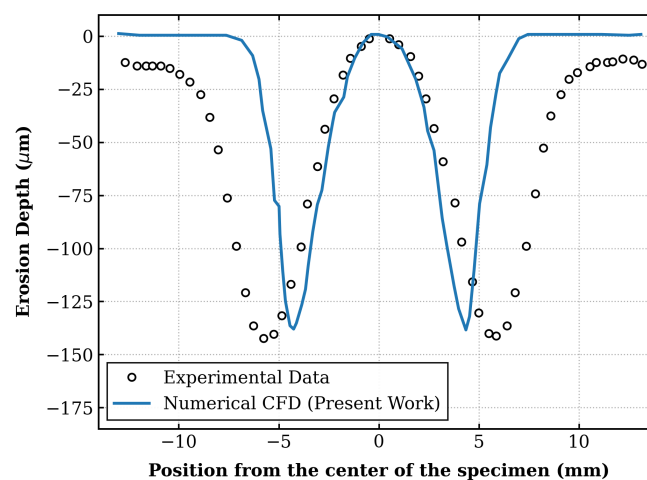


Figure 4. Validation of the numerical results through comparison with experimental data obtained from Mansouri [29].

To quantify numerical accuracy, Table 4 presents the normalized the wear distribution profile and details the relative percentage errors at critical hydrodynamic regions. Within the areas of maximum material degradation, the mathematical framework demonstrates predictive capability, yielding localized relative errors of only 0.44% and 1.97% at the left and right erosion peaks, respectively. This localized accuracy is critical because these peaks represent the regions accounting for more than 80% of the total active wear volume (Profile Share). In the central stagnation zone, where particle impact velocity approaches zero due to the localized high-pressure fluid cushion, the relative discrepancy remains low at 0.83%. Globally, the normalized Mean Absolute Error (MAE) and Root Mean Square Error (RMSE) remain within single-digit percentages at 4.38% and 6.37%, respectively. Combined with a global active profile error of 1.58% and a coefficient of determination (R^2) indicating that the model captures 97.40% of the experimental variance, these metrics demonstrate that the discrete phase framework can map particle–surface interactions.

Table 4. Quantitative validation metrics and local relative errors across the wear profile.

Region/Location	Coordinate x (mm)	Profile Share (% of y_{max})	Relative Error (%)
Left Erosion Peak	−4.85	92.44%	0.44%
Central Stagnation Zone	0.52	0.84%	0.83%
Right Erosion Peak	4.67	81.24%	1.97%
Global Statistical Index	Value (%)	Description	
Normalized MAE	4.38%	Mean absolute error scaled by y_{max}	
Normalized RMSE	6.37%	Root mean square error scaled by y_{max}	
Active Profile Mean Error	1.58%	Global relative error (active wear zone)	
Determination Coefficient	97.40%	Variance prediction capacity ($R^2 = 0.974$)	

As the flow accelerates radially outward from the stagnation zone, the increased particle momentum results in higher material removal rates in adjacent regions. This hydrodynamic behavior accounts for the wear peaks observed in Figure 5, confirming the model’s ability to capture the complex fluid–solid interactions inherent to impinging jets. This level of fidelity demonstrates that the proposed framework is a reliable tool for forecasting the operational lifespan of industrial components in which impingement is the primary wear mechanism, such as flow control devices and turbomachinery components.

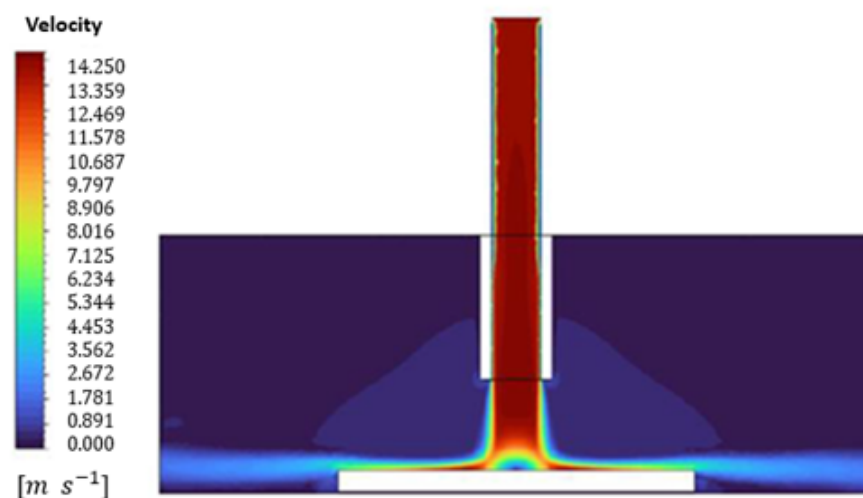


Figure 5. Velocity field contours across the fluid domain cross-section.

4. Case Study and Air Injection Setup

4.1. Case Study Description

The geometry selected for this study is based on the axial turbine design pioneered by [30]. Developed under guidelines emphasizing robustness, structural simplicity, and reduced maintenance and operational requirements, this machine serves as the structural and functional baseline for the present simulations. The prototype's experimental trials were conducted at the Hydromechanical Laboratory for Small Hydroelectric Power Plants (LHPCH) in Itajubá, Brazil, providing a reliable performance benchmark for this investigation. The technical specifications and operational parameters are summarized in Table 5. The hydro-mechanical assembly features a compact linear arrangement, with horizontal flow from left to right (Figure 6).

Table 5. Technical specifications and design parameters of the axial microturbine.

Parameter	Symbol	Value	Unit
Net head	H	4	m
Flow rate	Q	286	L/s
Rotational speed	n	1145	rpm
Design power	P	9.8	kW
Outer runner diameter	D_{ext}	0.2888	m
Hub diameter (inner)	D_{int}	0.112	m
Runner length	L	0.125	m
Nose cone (ogive) length	L_{og}	0.0715	m

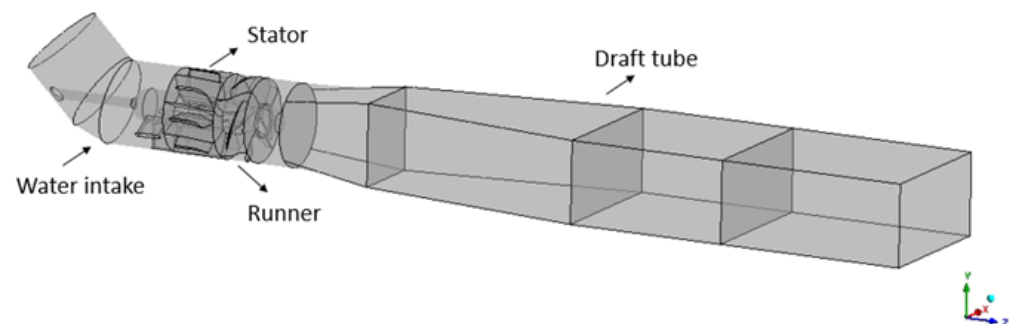


Figure 6. Schematic representation of the Sulci axial turbine assembly.

The numerical simulations encompassed the entire turbine domain. For mesh generation, Ansys Meshing was used for the stationary components, while TurboGrid was employed specifically to discretize the guide vane and runner regions. Within the rotor domain, local refinement was applied at the leading and trailing edges of the blades; a high nodal density was maintained in these regions to accurately capture geometric curvature and resolve steep pressure and velocity gradients.

The finalized computational grid comprised 12,904,645 nodes and 15,920,892 elements, consisting of a hybrid arrangement of 3,187,060 tetrahedra and 12,280,000 hexahedra. To satisfy the requirements of the SST turbulence model, the near-wall resolution was carefully controlled, resulting in an average y^+ value of 2.48 across the rotor surfaces.

To verify spatial accuracy, a mesh independence study was performed using four distinct grids with increasing levels of refinement. This sensitivity analysis was conducted at a fixed operating condition, focusing on established quality metrics and the convergence of primary variables. Figure 7 illustrates the global mesh topology.

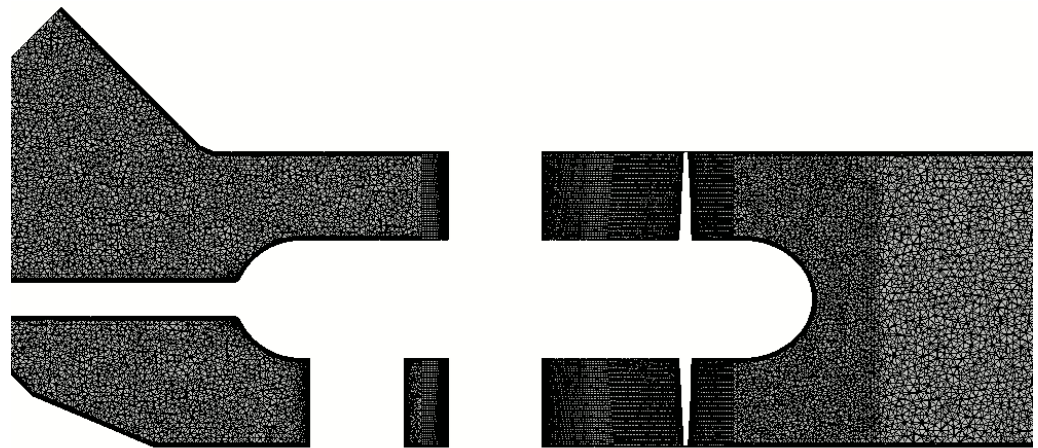


Figure 7. Details of the computational mesh utilized in this study, highlighting the higher refinement within the stator and runner regions.

The results of the convergence study are presented in Table 6, which compares the predicted shaft power (P_s) with the experimental reference value (P_{ref}). Mesh 3 was selected for subsequent analyses, as it provided the optimal compromise between numerical accuracy and computational cost, with a relative error of only 2.58%.

Table 6. Mesh independence study at ($H = 3.307$ m).

Mesh No.	P_s (kW)	P_{ef} (kW)	Relative Variation (%) (P_s/P_{ef})
1	7.407	—	6.01%
2	7.643	7.881	3.03%
3	7.678	—	2.58%
4	7.683	—	2.52%

Following discretization of the computational domains, boundary conditions were defined within the ANSYS CFX environment to initialize the numerical solver. These specifications included inlet and outlet conditions, no-slip wall constraints, and the definition of frame-change interfaces (stator-rotor) between stationary and rotating components.

4.2. Air Injection Setup

To model the mitigation of hydro-abrasive erosion via aerodynamic cushioning and reduce surface wear, a strategic air injection mechanism was implemented by introducing a localized secondary gaseous phase into the continuous hydrodynamic domain. The air inlet was positioned upstream of the runner blades, targeting critical transition zones where local static pressure is most likely to fall below the water saturation threshold. By introducing a controlled gaseous mass flow rate at this entry point, the numerical framework evaluates how local fluid density and void fraction distributions alter the cavitation inception velocity. This pre-runner air admission modifies the multi-component flow topology, cushioning micro-scale shockwaves generated during transient vapor cavity collapse while also influencing carrier phase velocity triangles that govern sediment transport. In this phase of the study, the air-injection system is represented as a mathematical point source, and its spatial coordinates and implementation are illustrated in Figure 8. To provide an absolute spatial reference, the origin of the computational coordinate system (0,0,0) is located at the center of the turbine rotor. Relative to this reference, the injection point is located at $(x, y, z) = (0, 0.065, -0.05)$ m, marked in red and defined without the need for physical surface faces. The injection vector is prescribed to be strictly axial and aligned with the local bulk flow velocity to ensure consistent momentum transfer between phases.

Furthermore, the injected air is modeled under standard atmospheric conditions at 25 °C, with a constant density of 1.185 kg/m³ and a dynamic viscosity of 1.831 × 10^{−5} Pa · s. This modeling strategy was adopted to evaluate mitigation effectiveness while avoiding the high computational costs, geometric complexity, and numerical instability often associated with face-integrated injection schemes through blade surfaces.

The inlet air volume fraction (IAVF) is typically defined as a ratio of volumetric discharge, expressed as follows [20]:

$$\text{IAVF} = \frac{q}{Q_{\text{BEP}}} \quad (16)$$

where q denotes the gaseous volume flow rate (kg/s), and Q_{BEP} is the turbine discharge in the operating regime, corresponding to 286 kg/s. Accordingly, Table 7 provides a structured summary of the complete set of operational parameters utilized.

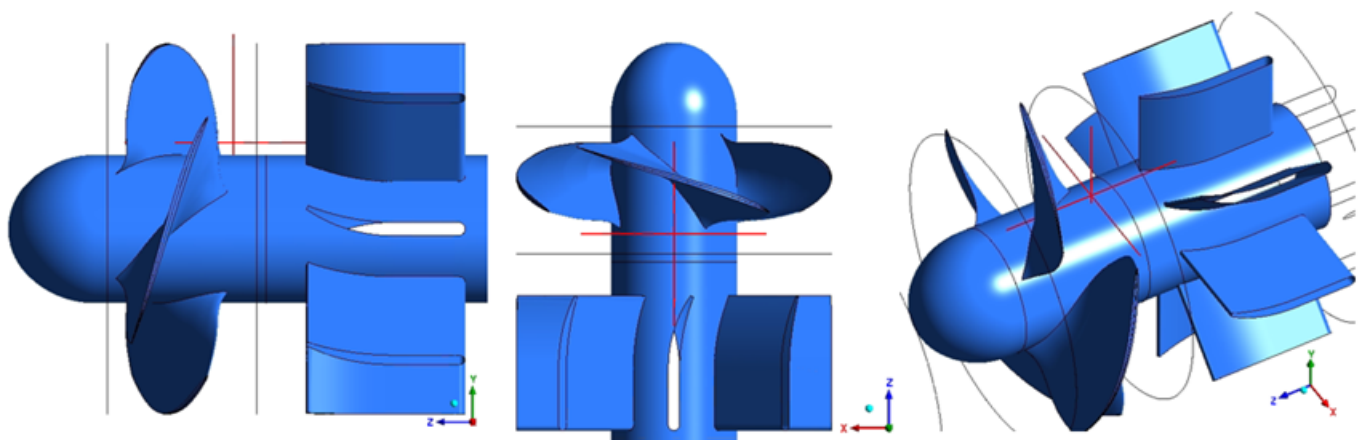


Figure 8. The spatial location of the air injection point source within the runner domain, presented in three orthogonal views. The precise coordinate of the air admission is highlighted by the red marker located at $(x, y, z) = (0, 0.065, -0.05)$ m relative to the rotor center.

Table 7. Mass flow rate specifications for the simulated air injection scenarios.

IAVF	q (kg/s)	Q_{BEP} (kg/s)	q/Q_{BEP}
IAVF 0	no injection	—	—
IAVF 1	1.850×10^{-6}	286	6.47×10^{-9}
IAVF 2	1.850×10^{-5}	286	6.47×10^{-8}
IAVF 3	5.500×10^{-5}	286	1.92×10^{-7}
IAVF 4	1.850×10^{-4}	286	6.47×10^{-7}
IAVF 5	1.850×10^{-3}	286	6.47×10^{-6}

5. Results and Discussion

5.1. Validation at Nominal Conditions

The accuracy of the numerical model was assessed by evaluating deviations between the computational results and experimental measurements recorded during bench testing. While the simulation successfully captured torque trends and the fundamental fluid dynamics within the turbine, this comparison is essential to confirm the precision of the numerical approach.

To visualize this correlation, power output was plotted as a function of rotational speed ($P \times n$), enabling a comprehensive assessment of turbine performance across its operational envelope. As shown in Figure 9, the analysis focuses on a net head of 3.307 mwc to ensure consistency between the numerical and experimental environments.

The global statistical validation of the hydrodynamic power curve was conducted by normalizing the absolute residuals against the maximum operating power (P_{max}). As summarized in Table 8, the numerical framework demonstrates predictive accuracy across the full operational range. The normalized Mean Absolute Error (MAE) and Root Mean Square Error (RMSE) are limited to low single-digit values of 1.80% and 2.35%, respectively. Furthermore, the peak local discrepancy (maximum local deviation) reaches 4.58%. Combined with a global average relative error of 1.92% and a coefficient of determination (R^2) indicating that the numerical reproduces 95.93% of the experimental power variance, these indicators confirm the reliability of the hydrodynamic model.

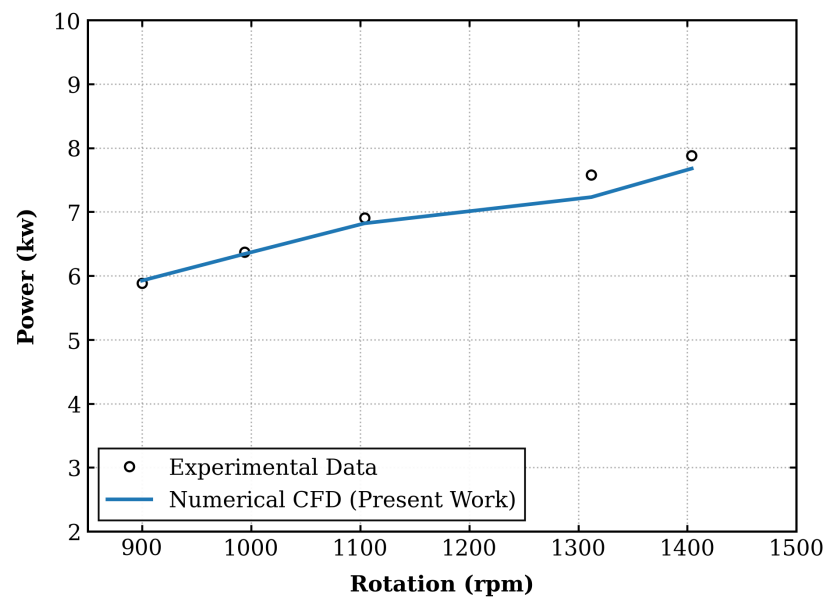


Figure 9. Comparison of power output (P) versus rotational speed (rpm) between CFD and experimental data at 3.307 mwc, with experimental datasets obtained from Souza [30].

Table 8. Global statistical validation metrics and normalized error indices for the turbine power performance curve.

Statistical Metric/Index	Value (%)	Turbomachinery Reference Scale
Normalized Mean Absolute Error (MAE)	1.80%	Scaled by maximum operating power (P_{max})
Normalized Root Mean Square Error (RMSE)	2.35%	Scaled by maximum operating power (P_{max})
Global Average Relative Error	1.92%	Integrated point-to-point discrepancy
Maximum Local Deviation	4.58%	Peak localized relative error
Coefficient of Determination (R^2)	95.93%	Explained performance variance capacity

Furthermore, Figure 10 presents a cross-sectional view of the velocity fields within the runner domain, providing insight into internal hydrodynamic behavior across the blades. The velocity magnitude contours reveal progressive flow acceleration within the inter-blade channels, driven by geometric restriction and energy transfer to the shaft. High-velocity gradients are well-resolved near the blade surfaces, particularly along the suction side, which correlates with localized static pressure reduction that triggers cavitation onset. Conversely, localized deceleration zones and low-velocity wakes are observed immediately downstream of the trailing edges. This wake development highlights the presence of shear layers and minor flow separation, characteristic of the three-dimensional flow topology governing this axial microturbine configuration under operating conditions.

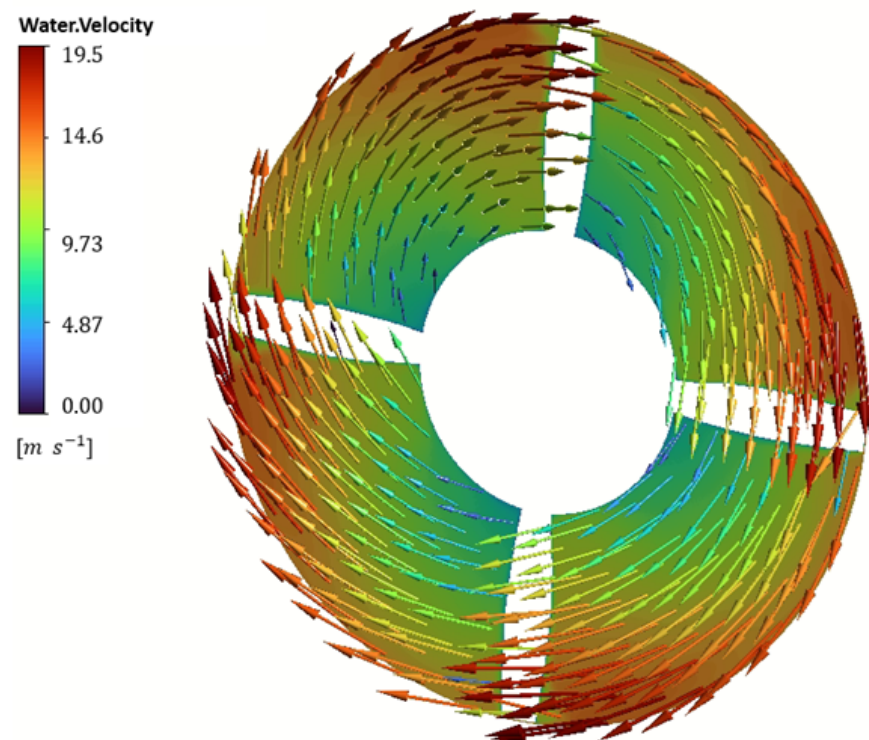


Figure 10. Velocity field in the rotor of the turbine.

5.2. Validation in the Cavitation Region

To align the numerical study with the experimental data, the simulation focused on an operating condition characterized by a cavitation coefficient of $\sigma = 1.141$. This parameter serves as a benchmark for evaluating the impact of phase change on turbine performance. The dimensionless coefficient was calculated following the analytical framework proposed by [31], defined in Equation (17):

$$\sigma = \frac{P_{suc} - P_v}{\rho g H} \quad (17)$$

where P_{suc} represents the static pressure at the suction side, P_v is the saturation vapor pressure (3169 Pa), ρ is the fluid density, g denotes the gravitational acceleration, and H is the operating head.

The numerical results obtained for this operating point show strong agreement with the experimental measurements, confirming the reliability of the modeling approach and its ability to replicate cavitating flow conditions. The comparative data for this validation case are presented in Table 9.

Table 9. Comparison between experimental and numerical results for the cavitation validation case ($\sigma = 1.141$).

Parameter	Experimental	Numerical	Relative Error (%)
Power (kW)	9.28 kw	8.95	3.6
Efficiency (%)	78.50	81.44	3.7

To evaluate the predictive capability of the multiphase framework, localized hydrodynamic patterns and phase transitions were investigated. Figure 11 visualize cavitation inception regions alongside the surface streamlines projected onto the blade geometry. The numerical solution captures the development of low-pressure cores near the blade leading edge, where vapor volume fraction contours define the boundaries of cavitation inception. By overlaying surface streamlines, the model reveals a complex topology dominated by

vortical structures and recirculation zones. Although these local flow topologies provide a quantitative representation of the simulated physics, their validity is supported by the macro-scale validation data reported in Table 9. The agreement between the CFD predictions and experimental measurements for operating power (P), hydraulic efficiency (η), and the Thoma cavitation parameter (σ) provides the quantitative basis supporting the observed local flow behavior. Therefore, the adherence of computed streamlines to blade curvature and the development of detached vapor structures indicate that both the SST turbulence model and Rayleigh–Plesset cavitation kinetics are consistent with the verified operating conditions of the runner.

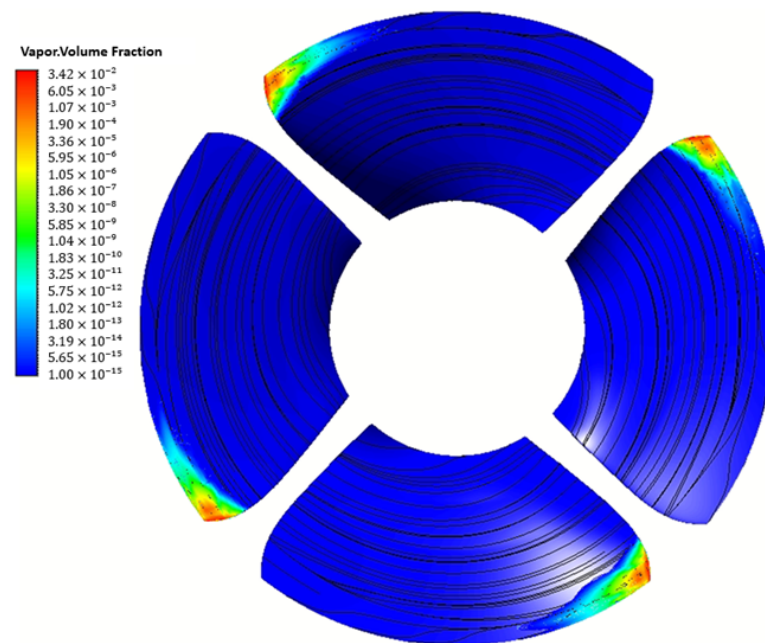


Figure 11. Visualization of cavitation inception regions and surface streamlines projected onto the blade geometry.

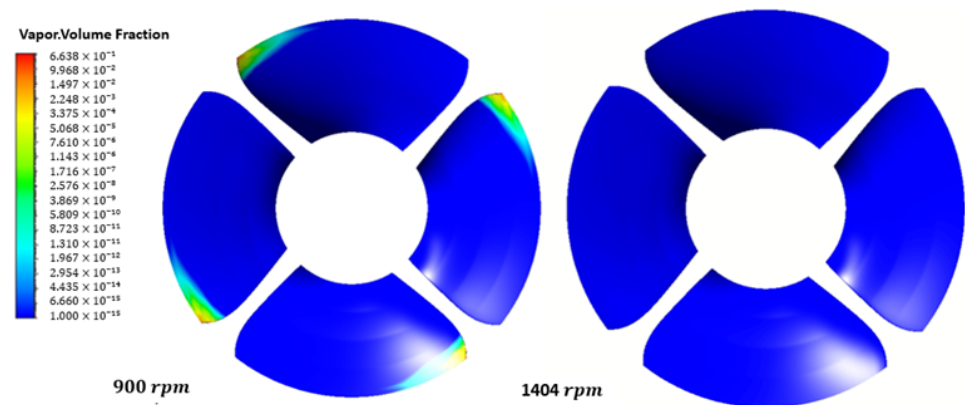
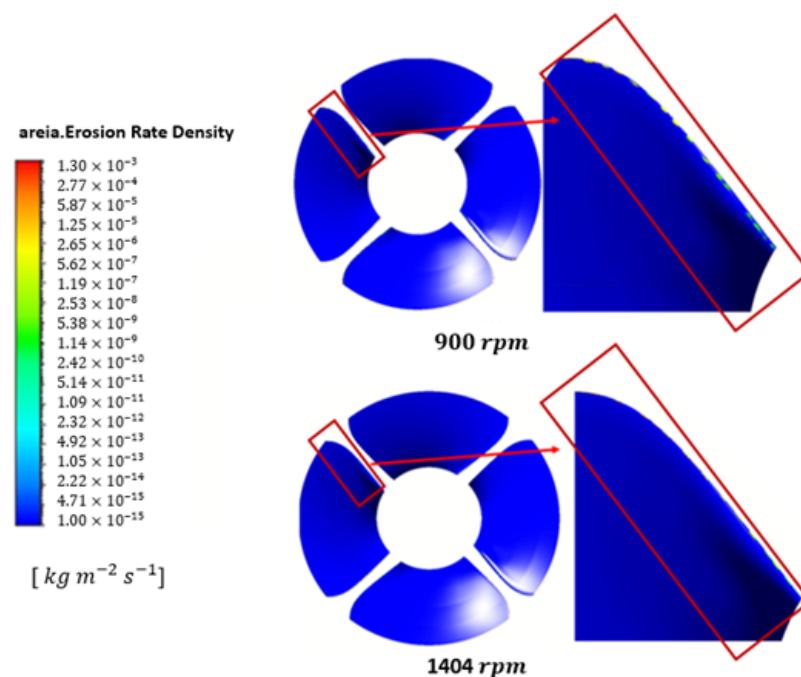
5.3. Influence of Cavitation on Particle Region

To investigate the combined influence of cavitation inception and sediment transport on runner surface degradation, two operating scenarios were evaluated. The first defines a non-cavitating baseline at a high cavitation coefficient ($\sigma = 2.5$), whereas the second represents a cavitating condition at a lower coefficient ($\sigma = 1.15$). The numerical parameters governing these cases are detailed in Table 10. In this context, the multiphase cavitating regime ($\sigma = 1.15$) corresponds to a rotational speed of 900 rpm, with an inlet total pressure of 31,299.0 Pa and an outlet mass flow rate of 229.0 kg/s. Conversely, the non-cavitating baseline ($\sigma = 2.5$) is maintained at higher rotational speed of 1404 rpm, with an inlet total pressure of 34,289.7 Pa and an outlet mass flow rate of 302.48 kg/s.

The spatial distribution and evolution of vapor volume fraction under both operating conditions are illustrated in Figure 12, showing the multi change phase phenomena near critical surface regions. Quantitatively, the vapor distribution exhibits a global average volume fraction of 5.98×10^{-4} , while the localized phase change reaches a maximum of 6.60×10^{-1} within low-pressure cores. Complementing this hydrodynamic mapping, the corresponding erosion and material removal contours are presented in Figure 13. The integrated mass loss varies with rotational speed; the global average wear intensity decreases from $\text{Erosion Rate}_{\text{mean}} = 2.110 \times 10^{-8} \text{ kg} \cdot \text{m}^{-2} \cdot \text{s}^{-1}$ at 9.00 rpm to $\text{Erosion Rate}_{\text{mean}} = 1.914 \times 10^{-8} \text{ kg} \cdot \text{m}^{-2} \cdot \text{s}^{-1}$ at 1.404 rpm, indicating that particle trajectories and impact dynamics depend on the velocity field.

Table 10. Boundary conditions for the evaluated alternative operational scenarios.

Boundary Condition	Operational Values	
Rotational speed	900 rpm	1404 rpm
Inlet condition (Total Pressure)	31,299.0 Pa	34,289.7 Pa
Outlet condition (Mass Flow Rate)	229.0 kg/s	302.48 kg/s

**Figure 12.** Visualization of cavitation inception and vapor distribution under the two analyzed operational conditions: 900 rpm scenario (left) and 1404 rpm scenario (right).**Figure 13.** Overall view of the runner blade wear distribution for the 900 rpm and 1404 rpm operational scenarios.

The numerical contours show that vapor volume fractions are concentrated along the suction side of the runner blades, consistent with localized low-pressure regions in this area. Conversely, sediment-induced wear is predominantly localized on the pressure side of the blades. This distribution aligns with hydrodynamic trajectories of the dispersed phase, as sand particles impinge on the concave surface under inertial and centrifugal forces. When evaluating total structural degradation, both mechanisms are considered together, meaning that the erosion footprint expands across the blade geometry. This combined wear behavior results in simultaneous material removal on both sides of the blade: cavitation-induced pitting on the suction side and particle abrasion on the pressure side, shown in Figure 14.

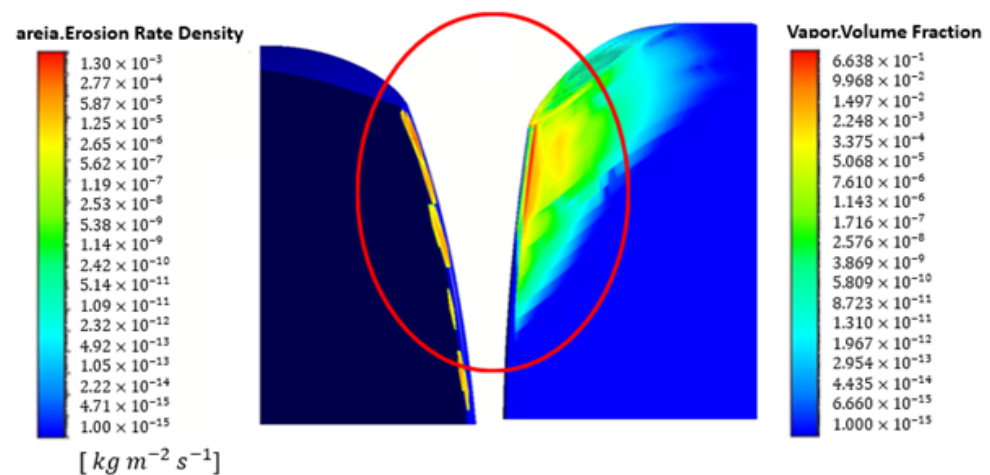


Figure 14. Cavitation wear (left) on the blade suction side and erosion wear (right) on the blade pressure side.

A comparison of the erosion distribution at 900 rpm (left) and 1404 rpm (right) is shown in Figure 15. Although both scenarios exhibit a similar wear footprint on the pressure side, the absolute erosion rate is strongly influenced by the rotor angular velocity and mechanical loading. At 900 rpm, the turbine operates at a higher torque of 63.11 Nm, compared to 51.63 Nm at 1404 rpm. This higher torque reflects increased blade loading under altered velocity triangles, which increases the flow incidence angle and alters flow development. Consequently, this shift increases the particle residence time and expand the primary impingement region near the mid-span. At 1404 rpm, the lower torque corresponds to more streamlined inlet conditions, resulting in reduced erosion for this operating case.

Interestingly, the numerical results indicate that the low-speed scenario (900 rpm) exhibits a slightly higher erosion depth than the high-speed baseline. From a fluid dynamics perspective, it remains unclear whether this difference is primarily driven by macroscopic operating conditions namely, reduced volumetric flow rate and altered velocity triangles at 900 rpm or indirect effects associated with the special structure of the cavitation cloud on the opposite blade surface.

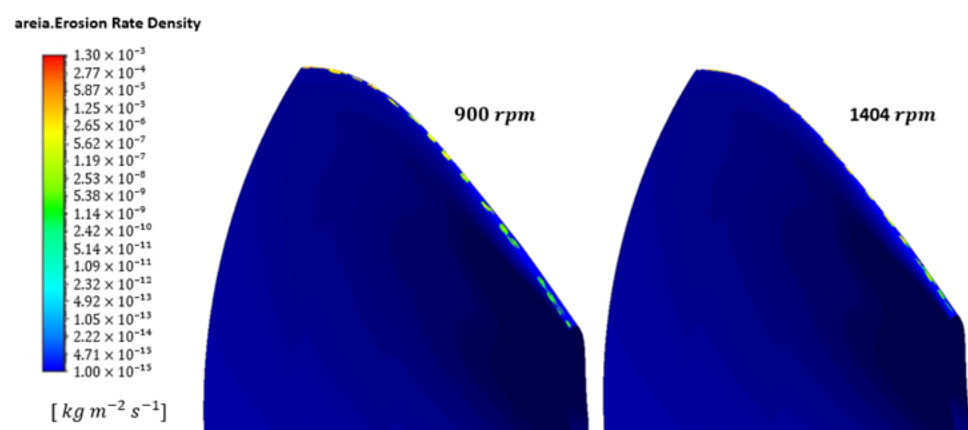


Figure 15. Erosion wear distribution under different operational conditions: 900 rpm scenario (left) and 1404 rpm scenario (right).

It should be explicitly stated that the present numerical framework does not directly resolve transient micro-scale interactions between cavitation bubble collapse and particle impacts. Because this study represents an initial phase of a broader parametric investigation involving four coupled phases, a one-way fluid–solid coupling approach was adopted to

manage computational cost and ensure numerical convergence. This formulation isolates macro-scale particle deflection under varying cavitation numbers while filtering highly localized transient phenomena such as micro-jets and shockwaves generated during vapor collapse, which may otherwise alter sediment trajectories within the boundary layer. Consequently, the cavitation cloud does not significantly alter the primary trajectory or impact velocities on the pressure side; therefore, the reported trends reflect largely independent wear mechanisms. While the stability of these trajectories suggests adequate statistical sampling within the Lagrangian framework, extension to a fully coupled transient model an important direction for future work.

5.4. Strategic Air Injection for Mitigating Solid Particle Erosion

To establish a baseline for the mitigation analysis, the solid sand phase ($\dot{m}_s = 0.2431 \text{ kg/s}$) was introduced into the high-flow operating regime ($\dot{m}_w = 286.0 \text{ kg/s}$). This sediment loading increases the shaft power to $P_{\text{shaft}} = 11,166.6 \text{ W}$ and raises the total pressure drop to $\Delta p = 47,108 \text{ Pa}$, defined as the difference in total pressure between the inlet and outlet regions of the computational domain. This increase directly reflects the higher hydraulic loading on the runner blades. Consequently, hydraulic efficiency decreases slightly to $\eta = 0.8263$ compared with the clean-flow baseline. Hereafter referred to as the erosion reference, this configuration serves as the benchmark against which all subsequent air-injection scenarios are evaluated, forming the basis for all erosion reduction indices reported in the following. For completeness, the operational parameters governing this reference case are detailed in Table 11.

Table 11. Summary of simulated cases: inlet mass flow rates and global performance parameters.

Case	$\dot{m}_s$ (kg/s)	P_{shaft} (W)	P_{hyd} (W)	η (–)	ΔP (Pa)
IADF 0	0.24310	11,166.60	13,513.40	0.8263	47,108.00
IADF 1	0.24310	11,165.60	13,531.10	0.8252	47,170.70
IADF 2	0.24310	11,164.30	13,529.10	0.8252	47,165.90
IADF 3	0.24310	11,161.60	13,525.60	0.8252	47,158.90
IADF 4	0.24310	11,149.60	13,476.30	0.8275	46,994.30
IADF 5	0.24310	10,993.00	13,310.80	0.8259	46,656.50

The erosion rate density distribution on the runner blade surface is highly localized. Out of 122,000 surface nodes exported from CFD after processing, non-zero erosion values are detected at 491 nodes (approximately 0.40% of the total surface area) for the erosion reference case. This strong spatial concentration indicates that solid particle impingement is confined to a narrow peripheral region near the blade outer radius on the pressure side, where centrifugal acceleration drives sand phase outward and increases their normal impact velocity.

The localized impacts and macroscopic trends of sediment-induced damage across the evaluated mitigation scenarios are analyzed using spatial and integrated metrics. The spatial distribution of the erosion rate density (ERD) is shown in Figure 16, illustrating how the extent of the abrasive footprint responds to varying air admission levels. Quantitatively, the first injection configuration (Figure 16 left) yields a global integrated wear intensity of $\text{Erosion Rate}_{\text{mean}} = 2.40 \times 10^{-11} \text{ kg} \cdot \text{m}^{-2} \cdot \text{s}^{-1}$, with degradation concentrated near the blade-hub junctions. Conversely, the second scenario (Figure 16 right) demonstrates the effect of enhanced aerodynamic shielding; a larger volumetric air fraction provides coverage over the hydrodynamic lifting surfaces, acting as a buffer that reduces particle impacts. Consequently, this cushioning effect reduces the global mass loss indicator by approximately one order of magnitude, resulting in an average wear intensity of

Erosion Rate_{mean} = $1.10 \times 10^{-12} \text{ kg} \cdot \text{m}^{-2} \cdot \text{s}^{-1}$ and indicating the development of more targeted aeration.

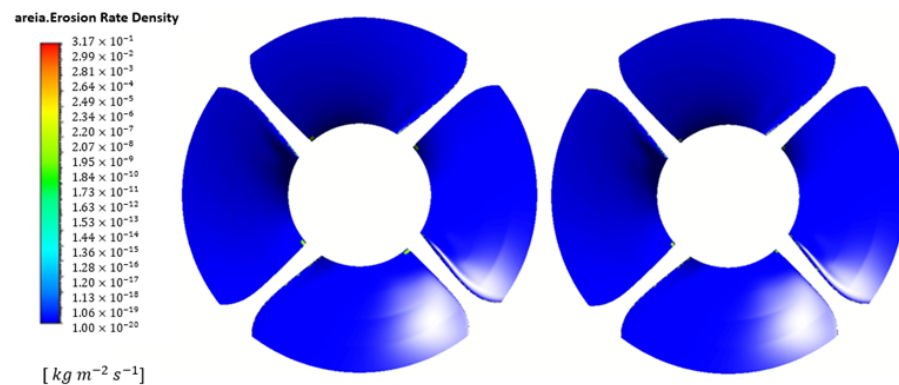


Figure 16. Sand erosion rate density on the runner blade surface for all simulated cases: IAVF 0 (left) and IAVF 2 (right).

To quantify net mitigation effectiveness, Figure 17 presents the integrated total erosion rate ($\dot{E}_{\text{total}}$) per case, obtained by summing the product of the local ERD and node area over all blade surface nodes. Interestingly, the comparison reveals a strongly non-linear relationship between air injection volume and total material removal. Low air mass flow rates (IAVF 1 to IAVF 3) initially increase $\dot{E}_{\text{total}}$ relative to the reference case. Optimal mitigation is achieved at the intermediate injection rate (IAVF 2), where global wear returns to the baseline level. Conversely, a further increase to IAVF 5 leads to a sharp increase in the integrated erosion rate, as shown on the logarithmic vertical axis. This behavior indicates that although strategic air admission can cushion particle impingement, excessive gas volumes likely increase the local carrier phase velocity or alter particle trajectories, thereby concentrating sediment impacts along pressure side boundaries.

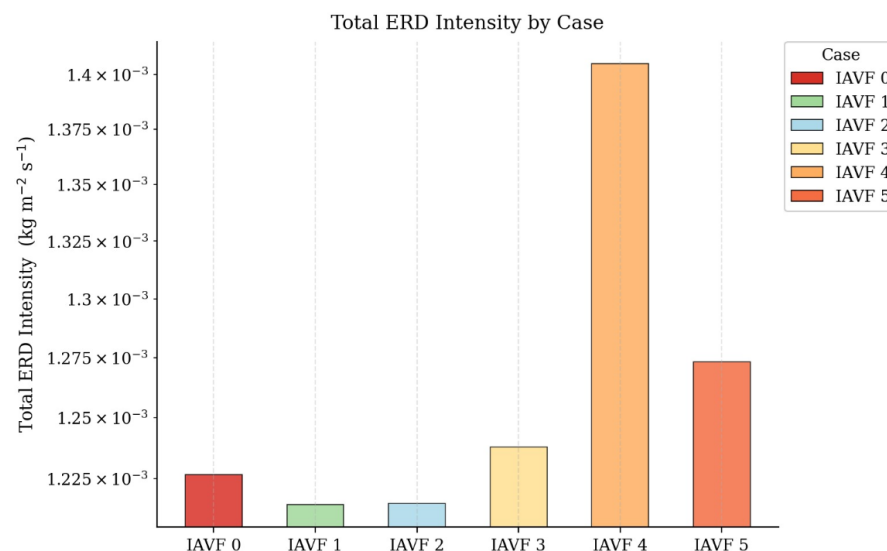


Figure 17. Integrated total erosion rate by case, obtained by summing the product of local ERD and node area over all blades surface nodes.

The global aerodynamic effects of strategic air admission on turbine performance are quantified through shaft power variations, as shown in Figure 18. For low injection volumes (IAVF 2), the absolute reduction in shaft power relative to the erosion reference case remains below 0.05%, while the corresponding variation in hydraulic efficiency does not

exceed -0.14% . These deviations fall within the numerical uncertainty of the multiphase CFD framework, indicating that low air injection levels do not meaningfully compromise the machine's hydrodynamic conversion capability. As the admission rate increases to (IAVF 4), the shaft power deficit reaches a minor threshold of -0.152% , which remains negligible in engineering terms. Conversely, at the highest injection rate (IAVF 5), corresponding to an inlet void fraction of 5.3×10^{-3} , the mechanical shaft power decreases by -1.55% , marking the only scenario in which aerodynamic performance degradation becomes practically relevant.

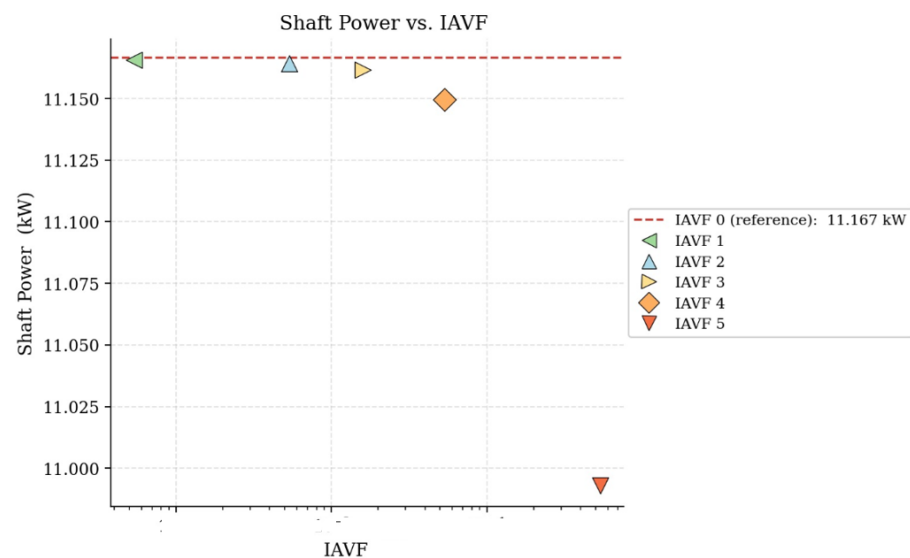


Figure 18. Shaft power P_{shaft} as a function of IAVF. The horizontal dashed line marks the erosion reference, IAVF 0 ($\dot{m}_{\text{air}} = 0$).

Figure 19 illustrates the spatial distribution of the air fraction on the blade surfaces. In the first configuration (Figure 19 left), the gaseous phase remains localized near the injection coordinate, resulting in an integrated global rotor average of $\alpha_{\text{air,mean}} = 5.00 \times 10^{-9}$. In contrast, the second scenario (Figure 19 right) promotes a spatial expansion along the surface. Although the maximum local peak remains unchanged due to the contour scale limit, the broader distribution and phase transport toward the adjacent lower blade raise the cumulative volume, increasing the global average to $\alpha_{\text{air,mean}} = 1.60 \times 10^{-8}$. This indicates that the second configuration produces a more distributed fluid shield over the lifting surfaces.

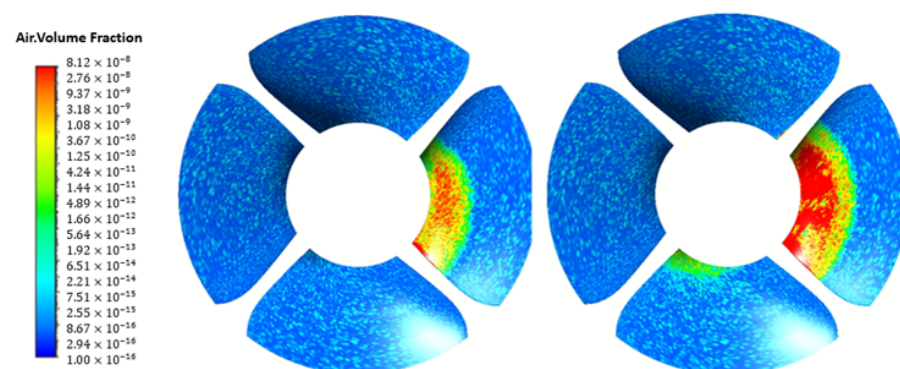


Figure 19. Air volume fraction distribution on the runner blade surface for IAVF 2 (left) and IAVF 5 (right).

Across all scenarios, the gaseous phase remains concentrated near the hub region (inner radius). This behavior results from two coupled mechanisms: the localized injection configuration (Figure 8), which introduces air directly at the hub radius, and centrifugal buoyancy effects in rotating frame, which prevent lighter bubbles from migrating outward. Furthermore, as each runner blade passes the discrete injection location, it is intermittently exposed to the jet. Consequently, air remains confined near the hub region, whereas denser sand particles migrate toward the outer radius under centrifugal forces.

Because of this phase separation, the air film does not directly overlap the erosion hot spot at the outer radius. Instead, mitigation occurs through alteration of the flow field within the inter-blade passage. At low injection rates, this perturbation produces a slight reduction in the particle impact flux at the outer radius. However, at higher injection rates, air accumulation near the hub increases, altering the velocity and pressure distributions across the entire passage. With increasing discharge, this distortion increases both normal impact velocity and particle flux at the outer radius, accounting for the increase in peak ERD and total erosion. A multi-nozzle configuration (e.g., four symmetrically distributed injectors) may offer improved phase uniformity compared with a single point source.

Figure 20 illustrates the spatial distribution of both phases along the normalized blade span. The outer 20% of the blade span ($0.8 \leq r^* \leq 1.0$) accounts for more than 91% of the cumulative erosion intensity, reflecting a sharp increase in wear due to the elevated relative velocity near the blade tip, rather than a uniform radial distribution. In plot (a) the cumulative ERD fraction for the reference case; highlight a strong concentration of damage within the outer span region. All air-injection scenarios follow a similar spatial trend, shifting downward in magnitude without altering peak wear locations or radial distribution patterns. Complementarily, plot (b) shows the mean air void fraction across the normalized span for all five injection scenarios on a logarithmic scale. The results indicate that the air phase remains largely confined to the hub region ($r^* < 0.2$), defined based on the inner runner geometry. For injection rates up to IAVF4, this confinement is maintained. However, the highest injection rate (IAVF5, dashed line) induces radial migration toward the mid-span, indicating a transition in the flow regime at elevated void fractions.

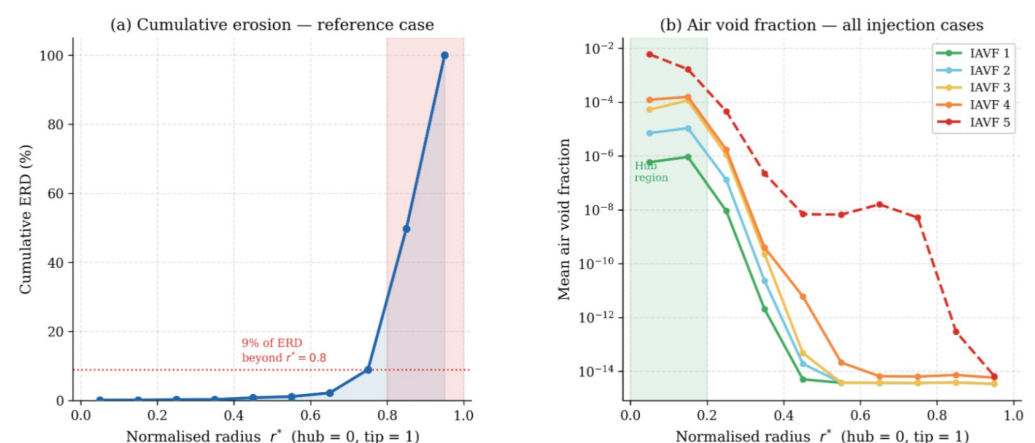


Figure 20. Radial distributions along the normalized blade span $r^* = (R - R_{\text{hub}})/(R_{\text{tip}} - R_{\text{hub}})$. (a) Percentage of total cumulative erosion rate density (ERD) for the reference case, and (b) mean air void fraction per radial bin for all five injection cases presented on a logarithmic scale.

The numerical results show that no simple proportional relationship exists between air admission volumes and the resulting erosion reduction; instead, the mitigation performance is non-linear, with an optimum at IAVF 2 followed by rapid deterioration at higher injection levels. Beyond this threshold, excessive air accelerates material removal due to flow-field distortion, indicating a critical limit for stable operation. Within this framework, the optimal

scenario (IAVF 2) yields mitigation indices of $\zeta_{\text{sum}} = 0.95\%$ for average integrated wear reduction and $\zeta_{\text{max}} = 6.17\%$ for maximum localized wear reduction. These results confirm that measurable reductions in both global and localized degradation are achievable through strategic air injection. However, further parametric investigations across a wider range of operating conditions and runner geometries are required to identify configurations with greater erosion reduction potential.

To contextualize these results relative to numerical uncertainty, the localized mitigation trend are considered alongside the grid and particle-tracking sensitivities. The mesh discretization error acts as a systematic bias across all cases because the mesh and numerical schemes remain identical; therefore, relative comparisons are independent of mesh resolution uncertainty. Additionally, particle-tracking sensitivity tests show erosion-rate variations of less than $\pm 0.1\%$, confirming that stochastic noise in the Lagrangian model is smaller than the reported mitigation effects. From an engineering perspective, achieving a 6.17% reduction in peak wear at the blade leading edges using a non-invasive upstream air injection configuration demonstrates practical potential as an alternative to costly rotor modifications, validating the practical relevance of this initial framework.

To contextualize the practical impact of the proposed mitigation technique on asset management, standard industrial inspection and overhaul schedules for hydraulic turbines operating in sediment-laden flows [32] were considered. Routine inspections and localized assessments are typically performed every 1 to 2 years, particularly to monitor critical regions, such as the blade leading edges where the 6.17% reduction was observed. Major overhauls and surface restoration, which are typically scheduled every 5 to 7 years under baseline conditions, may be extended. By introducing upstream air injection to continuously modify particle trajectories, the accumulation of localized micro-damage is reduced. Consequently, the interval for major runner refurbishment may be extended to 8 to 10 years, reducing downtime and long-term maintenance costs.

6. Conclusions

This study presented a numerical investigation into the complex degradation mechanisms caused by cavitation and solid-particle erosion in a low-head axial microturbine. By using a Computational Fluid Dynamics (CFD) framework within ANSYS CFX, coupled with the SST turbulence model and Rayleigh–Plesset cavitation kinetics, it was possible to replicate hydrodynamic performance and map the spatial distribution of surface wear. The numerical framework was validated against external experimental datasets from the literature, using power and efficiency benchmarks from Souza (1989) [30] and wear rates from Mansouri (2015) [29].

The spatial mapping of material degradation reveals a distinct asymmetric footprint governing each wear phenomenon across the runner blade geometry, featuring an outer diameter of 0.2888 m, an inner diameter of 0.112 m, a length of 0.125 m, and a nose cone length of 0.0715 m. Vapor volume fraction contours indicate that cavitation inception and bubble development are predominantly concentrated along the suction side of the blades, driven by localized severe low-pressure fields. Conversely, solid particle erosion, quantified using the Lagrangian framework and the Finnie model, is localized on the pressure side and casing components, consistent with particle trajectories that impinge on concave boundaries under inertial and centrifugal forces. Furthermore, spatial analysis along the normalized blade span demonstrates that erosion is not uniformly distributed but is heavily concentrated within the outer 20% of the span ($0.8 \leq r^* \leq 1.0$), an interval that accounts for more than 91% of the total cumulative erosion intensity due to the steep increase in relative velocity near the blade tip.

The parametric analysis conducted under different operational scenarios provided insight into the macroscopic-hydrodynamic coupling of the system. While the total material damage is intensified by the simultaneous action of cavitation on the suction side and particle abrasion on the pressure side, isolated evaluation of sediment-induced wear rates indicates that maximum erosion depths remain in close agreement between the cavitating ($\sigma = 1.15$) and non-cavitating ($\sigma = 2.5$) baselines. Within the current numerical framework, this stability suggests that the macroscopic structure of the cavitation cloud on one face does not significantly alter the primary trajectories or velocity magnitude profiles of dispersed sand particles on the opposing face. However, the slightly higher erosion depth observed in the low-speed scenario (900 rpm) underscores the strong influence of altered velocity triangles and reduced discharge volumes on localized particle impingement mechanics.

Ultimately, the evaluation of strategic air injection represents a significant advancement toward a simplified solution for erosion reduction. In this stage of the research, it is represented as a discrete mathematical point source located at the coordinates $(x, y, z) = (0, 0.065, -0.05)$ m relative to the rotor center. This modeling strategy proved highly efficient, enabling tracking of phase distribution while avoiding the prohibitive computational cost and numerical instabilities inherent in face-integrated injection schemes.

The radial profiles demonstrate that all air-injection scenarios follow a similar qualitative trend to the reference case, shifting erosion curves downward in magnitude without altering the overall trajectory or peak wear locations. The results show that the air film remains predominantly confined to the inner hub region ($r^* < 0.2$), a threshold defined by the geometric architecture of the inner runner. However, the peak injection rate (IAVF5) triggers radial migration toward the mid-span. The numerical results confirm a non-linear relationship between air admission and wear reduction, with the optimal case (IAVF2) yielding mitigation indices of $\xi_{\text{sum}} = 0.95\%$ for integrated wear reduction and $\xi_{\text{max}} = 6.17\%$ for localized wear reduction, demonstrating that a measurable decrease in degradation is achievable.

However, there remains scope to further improve and refine this technique, defining the next milestones for continuation of this research. Future work will focus on optimizing both injector positions and the total number of injection points, as well as exploring peripheral air admission via the casing, given that hydro-abrasive erosion is heavily concentrated at the outer blade extremities ($r^* \geq 0.8$). Methodologically, the next phase of the project will transition from steady-state evaluations to fully transient multiphase simulations to capture the dynamic behavior of vortex structures and transient vapor-cavity collapse. Additionally, a fully coupled Eulerian–Lagrangian coupling framework will be implemented to assess how the secondary gaseous phase alters the local continuous velocity field and directly influences sediment particle trajectories. Finally, although this mitigation strategy was evaluated on a small bench-scale axial microturbine, it provides a mathematically grounded framework that can be scaled and tested in larger installations, such as low-head bulb turbines, where greater geometric flexibility allows improved placement of injection nozzles.

Author Contributions: J.G.C.: Supervision, conceptualization, writing – review and editing; R.d.A.: Conceptualization, methodology, investigation, data curation, formal analysis, writing; H.C.W.: Conceptualization, methodology, investigation, data curation, formal analysis, writing—original draft; A.L.A.M.: Supervision, methodology, validation, writing—review and editing. All authors have read and agreed to the published version of the manuscript.

Funding: This research was funded by National Council for Scientific and Technological Development (grant no. 408034/2022-0).

Data Availability Statement: The original contributions presented in this study are included in the article. Further inquiries can be directed to the corresponding author.

Acknowledgments: The authors would like to thank the Conselho Nacional de Desenvolvimento Científico e Tecnológico (CNPQ) for supporting this research through the CNPq/MCTI/FNDCT N° 25/2022 project 408034/2022-0. The authors would like to express their gratitude to Zulcy de Souza and the Federal University of Itajubá (UNIFEI) for kindly providing the experimental data utilized for the validation of this study.

Conflicts of Interest: The authors declare no conflicts of interest.

Abbreviations

Parameter/Variable Name	Variable/Symbol [Unit]
Air injection source coordinates	$(x, y, z)_{inj}$ [m]
Injection direction vector	—
Turbine hydraulic head	H [m]
Local gravitational acceleration	g [m/s ²]
Rotational speed	N [rpm]
Maximum operating power baseline	P_{max} [kW]
Fluid domain thermal baseline	T [C] (°C)
Mixture density	ρ_m [kg/m ³]
Mixture dynamic viscosity	μ_m [Pa·s]
Mass-averaged velocity vector	v_m [m/s]
Phase drift velocity vector	$v_{dr,k}$ [m/s]
Static/local pressure	p [Pa]
Liquid phase volume fraction	α_l
Liquid phase density (water)	ρ_l [kg/m ³]
Liquid phase dynamic viscosity	μ_l [Pa·s]
Air volume fraction	α_a
Air density at standard atmosphere	ρ_{air} [kg/m ³]
Air dynamic viscosity	μ_{air} [Pa·s]
Vapor volume fraction	α_v
Vapor phase density	ρ_v [kg/m ³]
Vapor saturation pressure	p_v [Pa]
Nucleation site volume fraction	α_{nuc}
Liquid-vapor surface tension coefficient	σ [N/m]
Vaporization empirical coefficient	F_e
Condensation empirical coefficient	F_c
Spherical vapor bubble radius	R_B [m]
Solid particle tracking count	—
Nominal sediment diameter	d_p [μm]
Solid particle density (sand)	ρ_p [kg/m ³]
Sediment inlet concentration	C_s [g/L]
Characteristic rotor diameter	D [m]
Stokes number	Stk
Particle velocity vector	u_p [m/s]
Particle impact velocity magnitude	v_p [m/s]
Particle impingement angle	γ [rad] (°)
Finnie velocity power exponent	n
Target material erosion constant (steel)	k
Normalized Mean Absolute Error	MAE
Normalized Root Mean Square Error	RMSE
Global Average Relative Error	—
Maximum Local Deviation	—

Coefficient of Determination	R^2
Air fraction average	α
Integrated erosion rate average	Erosion Rate [$\text{kg}\cdot\text{m}^{-2}\cdot\text{s}^{-1}$]

References

1. Tiwari, G.; Kumar, J.; Prasad, V.; Patel, V.K. Utility of CFD in the design and performance analysis of hydraulic turbines—A review. *Energy Rep.* **2020**, *6*, 2410–2429. [\[CrossRef\]](#)
2. Luo, X.W.; Ji, B.; Tsujimoto, Y. A review of cavitation in hydraulic machinery. *J. Hydrodyn.* **2016**, *28*, 335–358. [\[CrossRef\]](#)
3. Poudel, R.; Chitrakar, S.; Qian, Z.; Guo, Z.; Thapa, B. Experimental Investigations of Sediment Erosion in Guide Vanes of Francis Turbine. *IOP Conf. Ser. Earth Environ. Sci.* **2024**, *1411*, 012017. [\[CrossRef\]](#)
4. Ramalho, R.V.C.; Filho, M.J.M.P.; Sena, M.J.d.S.; Mendes, R.L.S.G.; Neto, S.U.; Souza, D.E.S.e.; Coelho, J.G.; Furtado, G.C.d.; Mesquita, A.L.A. Design of Low-Cost Axial-Flow Turbines for Very Low-Head Micro-Hydropower Plants. *Processes* **2025**, *13*, 1865. [\[CrossRef\]](#)
5. Coelho, J.G.; Oliveira Fávero, V.; Oliveira, G.A.; Pinho, F.A.A. Investigation on reuse of energy at a sewage treatment plant and payback period. *Environ. Qual. Manag.* **2023**, *33*, 235–244. [\[CrossRef\]](#)
6. Shrestha, R.; Pradhan, S.S.; Gurung, P.; Ghimire, A.; Chitrakar, S. A review on erosion and erosion induced vibrations in Francis turbine. *IOP Conf. Ser. Earth Environ. Sci.* **2022**, *1037*, 012028. [\[CrossRef\]](#)
7. Rakibuzzaman, M.; Suh, S.H.; Kim, H.H.; Islam, M.D.; Zhou, L.; El-Emam, M.A. Cavitation and erosion effects on hydraulic performances of a submersible drainage pump. *Alex. Eng. J.* **2025**, *113*, 431–450. [\[CrossRef\]](#)
8. Kapali, A.; Chitrakar, S.; Shrestha, O.; Neopane, H.P.; Thapa, B.S. A Review on Experimental Study of Sediment Erosion in Hydraulic Turbines at Laboratory Conditions. *IOP Conf. Ser. J. Phys. Conf. Ser.* **2019**, *1266*, 012016. [\[CrossRef\]](#)
9. Kan, K.; Binama, M.; Chen, H.; Zheng, Y.; Zhou, D.; Su, W.; Muhirwa, A. Pump as turbine cavitation performance for both conventional and reverse operating modes: A review. *Renew. Sustain. Energy Rev.* **2022**, *168*, 112786. [\[CrossRef\]](#)
10. Sun, L.; Guo, P.; Luo, X. Numerical investigation on inter-blade cavitation vortex in a Francis turbine. *Renew. Energy* **2020**, *158*, 64–74. [\[CrossRef\]](#)
11. Chen, M.; Tan, L. Solid-liquid multiphase flow and erosion in the energy storage pump using modified drag model and erosion model. *J. Energy Storage* **2023**, *73*, 108859. [\[CrossRef\]](#)
12. Shrestha, S.; Bijuckchhe, P.L.; Roudel, R.; Chitrakar, S.; Thapa, B.; Guo, Z.; Qian, Z. Numerical Investigation of Sediment Erosion in Guide Vanes of Francis Turbine with Experimental Validations. *IOP Conf. Ser. Earth Environ. Sci.* **2024**, *1385*, 012008. [\[CrossRef\]](#)
13. Zhao, Z.; Shi, J.; Zhong, Q.; Luo, Y.; Qian, Z. Sediment erosion on the runner of a Pump turbine in turbine mode. *J. Energy Storage* **2026**, *141*, 119273. [\[CrossRef\]](#)
14. Lira, S.H.; Torres, M.J.; Silva, R.G.; Viñuela, J.Z. Numerical Characterization of the Solid Particle Accumulation in a Turbulent Flow through Curved Pipes by Means of Stokes Numbers. *Appl. Sci.* **2021**, *11*, 7381. [\[CrossRef\]](#)
15. Han, W.; Wang, J.; Kang, J.B.; Li, L.Y.; Peng, G.Y. Erosion characteristics of hydraulic turbine guide-vane end clearance in sediment water flow: A simplified model analysis. *J. Flow Control Meas. Vis.* **2017**, *5*, 111–126. [\[CrossRef\]](#)
16. Dhiman, P.; Singh, V.P.; Karn, A. Experimental and computational analysis of air injection as a mitigation technique for silt erosion in hydro turbines. *J. Renew. Energy Sustain. Dev.* **2024**, *10*, 345. [\[CrossRef\]](#)
17. Hilo, A.K.; Go, Y.J.; Hong, J.W.; Ahn, B.K.; Park, C.; Kim, G.D.; Moon, I.S. Cavitating flow control and noise suppression using air injection. *Phys. Fluids* **2024**, *36*, 087146. [\[CrossRef\]](#)
18. Liu, H.; Yang, X.; He, Y.; Lu, J.; Jiang, L. Effects of air injection on the dynamic characteristics of sediment deposition and erosion distribution in pump-turbine runners. *Sensors* **2025**, *25*, 6192. [\[CrossRef\]](#) [\[PubMed\]](#)
19. Praveen, S.; Marimuthu, S.; Alqahtani, B.; Bharathiraja, G.; Gokilakrishnan, G. Enhancing bulb turbine performance assessing air injection for vibration mitigation and hydrofoil cavitation. *J. Appl. Fluid Mech.* **2024**, *17*, 2313–2332. [\[CrossRef\]](#)
20. Lu, J.; Pan, Y.; Zhou, Y.; Zhu, B.; Zhang, C.; He, Y. Study on the influence of air-injection on cavitation characteristics of pump-turbine under turbine condition. *Flow Meas. Instrum.* **2025**, *104*, 102917. [\[CrossRef\]](#)
21. Krishna, K.; Kumar, J.; Patel, V.K. Design modifications in hydraulic turbine runner blades to mitigate sediment erosion: A review. *Renew. Sustain. Energy Rev.* **2016**, *65*, 248–264. [\[CrossRef\]](#)
22. Van Wijngaarden, L. Mechanics of collapsing cavitation bubbles. *Ultrason. Sonochem.* **2016**, *29*, 524–527. [\[CrossRef\]](#) [\[PubMed\]](#)
23. Pires, G.R.M.; Siqueira, I.B.A.F.; de Sousa, M.J.; Lozovei, J.C.; Capanema, E.R.S.; Barghouthi, M.R.; Moreira, C.C. Physical characterization of sediments from the Madeira River in the Porto Velho region. In Proceedings of the XXIII Brazilian Symposium on Water Resources, Foz do Iguaçu, PR, Brazil, 24–28 November 2019.
24. Ansys Inc. *ANSYS CFX-Solver Theory Guide, Release 2025*; Ansys, Inc: Canonsburg, PA, USA, 2025.
25. Ghahramani, E.; Ström, H.; Bensow, R.E. Numerical simulation and analysis of multi-scale cavitating flows. *J. Fluid Mech.* **2021**, *922*, A22. [\[CrossRef\]](#)

26. Hankeln, F.; Riedelbauch, S. Comparison of different models for determination of erosion wear in centrifugal pumps. In Proceedings of the 13th European Conference on Turbomachinery Fluid Dynamics & Thermodynamics (ETC13), Lausanne, Switzerland, 8–12 April 2019. Available online: <https://www.euroturbo.eu/paper/ETC2019-037.pdf> (accessed on 28 June 2026).
27. Beck, A.D.; Vagnoni, E. Effect of Stokes number on erosion on Pelton buckets for sediment-laden flows. *IOP Conf. Ser. Earth Environ. Sci.* **2025**, *1561*, 012042. [[CrossRef](#)]
28. Kannojiya, V.; Deshwal, M.; Deshwal, D. Numerical Analysis of Solid Particle Erosion in Pipe Elbow. *Mater. Today Proc.* **2018**, *5*, 5021–5030. [[CrossRef](#)]
29. Mansouri, A.; Arabnejad, H.; Karimi, S.; Shirazi, S.A.; McLaury, B.S. A combined CFD/experimental methodology for erosion prediction. *Wear* **2015**, *332*, 1090–1097. [[CrossRef](#)]
30. Souza, Z. *Test and Operation Reports for the MEP Tube-Type Hydraulic Turbine*; FUPAI Internal Report; Federal University of Itajubá: Itajubá, Brazil, 1989. Available online: <https://fupai.org.br> (accessed on 28 June 2026).
31. Brijkishore; Khare, R.; Prasad, V. Prediction of cavitation and its mitigation techniques in hydraulic turbines—A review. *Ocean Eng.* **2021**, *221*, 108512. [[CrossRef](#)]
32. Dorji, U.; Ghomashchi, R. Hydro turbine failure mechanisms: An overview. *Eng. Fail. Anal.* **2014**, *44*, 136–147. [[CrossRef](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.