Application of Neural Networks in the Optimization of Francis Turbine Draft Tubes

Eduardo Fabian de Oliveira
UTFPR
Cascavel, Brasil
eduardofliveira@gmail.com

Raquel da Cunha Ribeiro da Silva
UTFPR
Guarapuava, Brasil
raqueld@utfpr.edu.br

Dhaianny Guizoni de Oliveira
HACKER Industrial.
Guarapuava, Brasil
dhaiguizoni@gmail.com

***Abstract*—This study proposes the application of artificial neural networks (ANNs) to optimize the performance of hydraulic turbines, with a specific focus on the draft tube. The research is motivated by the need to enhance energy efficiency in hydroelectric systems, in alignment with the Sustainable Development Goals. The methodology combines computational fluid dynamics (CFD) simulations and machine learning techniques, utilizing operational data from a Francis turbine. Results demonstrate the potential of ANNs to predict and improve key performance parameters, offering a novel approach to hydraulic design optimization. The integration of computational modeling and artificial intelligence emerges as a promising tool for the renewable energy sector.**

***Index Terms*—hydraulic turbines, artificial neural networks, energy efficiency, draft tube, CFD.**

I. INTRODUCTION

Energy generation through clean and renewable sources is a pillar for global sustainable development. Hydropower stands out as one of the oldest technologies, accounting for a substantial portion of the world's installed renewable capacity [3]. Hydraulic turbines, such as Francis turbines, are crucial in the transition to renewable energy sources, aligning with sustainable development goals. However, maximizing the efficiency and ensuring the operation of these systems face increasing challenges [9]. Hydraulic turbines are subject to demanding operational conditions, often operating outside their optimal design point (off-design) to compensate for the intermittency of other renewable sources [12]. The draft tube is a critical component, responsible for recovering the kinetic energy of the water. Flow instabilities in this tube, especially in off-design regimes [13], are a challenge. This study proposes to investigate the application of artificial neural networks (ANNs) for the analysis and optimization of hydraulic turbines, exploring how these techniques can be used to analyze operational data and develop predictive models for turbine efficiency. Electrical energy is a driver of development. The search for clean energy sources has become necessary, with a 50% increase in global clean energy generation capacity since 2022 [6]. The continuous optimization of hydropower plant efficiency remains an active field of research. This work aims to contribute to this effort, focusing on improving the performance of hydraulic turbines. The operation of these complex machines faces challenges such as energy losses, the need to operate under variable conditions, and component wear [5]. Optimizing hydraulic design for different operational and specific site conditions remains a challenge that directly impacts performance and lifespan, requiring the use of CFD [11]. The increasing digitalization of hydropower plants, with the massive installation of sensors and data acquisition systems, opens new possibilities for the use of AI, ML, and ANNs [7], allowing the analysis of large volumes of real-time and historical operational data, identifying complex patterns, and developing robust predictive models.The general objective of this work is to apply artificial neural networks to optimize the performance of hydraulic turbines, focusing on improving efficiency and reducing energy losses through the analysis and prediction of operational parameters.

II. LITERATURE REVIEW

The global energy transition has elevated hydropower's role as both a renewable energy source and critical grid stabilizer, capable of compensating for the intermittency of solar and wind generation. This dual function places unprecedented demands on turbine components, particularly the draft tube which must maintain efficiency across increasingly variable operating conditions. When Francis turbines operate outside their design point to balance renewable intermittency, the draft tube becomes susceptible to complex flow phenomena - from vortex shedding to pressure pulsations that can exceed 20% of nominal head [13]. These instabilities not only reduce energy recovery but accelerate component wear, making their mitigation essential for both performance and equipment longevity. Recent advances in computational methods are transforming how engineers address these challenges. Computational Fluid Dynamics (CFD) now enables detailed visualization of energy losses and flow separation patterns that were previously identifiable only through physical prototyping. Modern simulation techniques can predict with remarkable accuracy how geometric variations affect draft tube performance, from the cone angle's influence on flow attachment to the elbow curvature's impact on vortex formation [11]. However, the computational expense of these high-fidelity simulations has traditionally limited their use in iterative design processes. This limitation is being overcome through synergistic applications of machine learning, where artificial neural networks (ANNs) trained on CFD datasets can approximate complex flow behaviors with minimal computational overhead. The digital transformation of hydropower plants has been instrumental in this development, as modern monitoring systems generate terabytes of operational data that reveal previously hidden relationships between turbine geometry, operating conditions, and performance outcomes [7]. By learning these patterns, ANNs can predict how modifications to parameters like the cone height (H1) or diffuser angle will affect the pressure recovery coefficient (NW) as in Eq. (1), enabling rapid virtual prototyping while maintaining the accuracy of conventional CFD.

$$NW = \frac{P_{ex} - P_{en}}{\frac{1}{2}(v_{en}^2 - v_{ex}^2)} \tag{1}$$

Where P_{ex} and P_{en} are the static pressures at the outlet and inlet, respectively, and v_{ex} and v_{en} are the corresponding fluid velocities. This expression represents the non-dimensional pressure recovery coefficient used to evaluate the effectiveness of the draft tube in converting kinetic energy into pressure. The integration of these computational approaches is particularly valuable for small hydro plants, where resource constraints have historically limited optimization opportunities. Where a comprehensive CFD parametric study might require weeks of computation, ANN-assisted methods can achieve comparable design insights in days while maintaining over 99% prediction accuracy [4]. This dramatic reduction in time and cost makes advanced optimization accessible to smaller installations, potentially unlocking significant gains across distributed hydropower networks - a crucial advantage as the world works toward SDG 7's targets for universal clean energy access.

III. METHODOLOGY

The methodology integrates CFD simulations with ANN modeling to optimize the draft tube geometry based on data from a Francis turbine manufactured by HACKER Industrial, used in a Small Hydroelectric Plant (PCH).

A. DATA COLLECTION

Three initial geometries of the elbow-type draft tube, characterized by different heights of the upper conical section (H1) and subsequent section (H2), were modeled and simulated using ANSYS Fluent, with the draft tube geometry divided into three main parts: transition cone, curved elbow, and outlet diffuser [1], while fixed parameters (inlet/outlet dimensions, total height) were obtained from the actual turbine design (Table I).

TABLE I
INPUT PARAMETERS

Component	Subcomponent	Value
Draft Tube	Total Height	4937.845 mm
	Maximum Height (H1+H2)	2140.849 mm
INLET	Initial Diameter	1280.747 mm
OUTLET	Final Height	2.500 mm
	Final Width	2.500 mm
	Final Area	6,250,000 mm²
Elbow Draft Tube	H3 (Elbow Height)	2533 mm
	Total Conical Section (H1 + H2)	1728.0714 mm
	Elbow Angle	83.99°

B. ARTIFICIAL NEURAL NETWORK

The development of an effective neural network model for draft tube optimization begins with a detailed understanding of its geometric representation in the computational framework as illustrated in Figure 1.

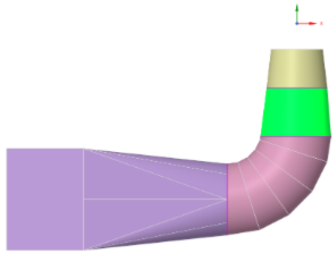

Fig. 1. Model Draft Tube, from Hacker Industrial [2]

The draft tube's segmented design, divided into three functionally distinct sections, plays a critical role in its hydrodynamic performance. The upper conical section transitions smoothly from the turbine runner to the draft tube channel, its carefully designed geometry minimizing turbulence through gradual area reduction. This leads into the elbow section, where the fluid undergoes a controlled directional change that represents one of the most hydrodynamically sensitive regions due to the complex interplay between velocity gradients and pressure variations. The final outlet diffuser completes the energy conversion process, efficiently transforming residual kinetic energy into valuable pressure recovery that directly contributes to the turbine's overall efficiency. Computational modeling of this system required careful parameterization of the draft tube geometry, with particular attention to the relationship between the conical section height (H1) and subsequent segment (H2). Initial CFD simulations performed in ANSYS examined three distinct configurations, with H1 values of 950 mm, 1150 mm, and 1400 mm respectively. These simulations generated comprehensive datasets capturing pressure distributions, velocity profiles, and the crucial static pressure recovery coefficient (NW) that served as the primary performance metric.

TABLE II
ARTIFICIAL NEURAL NETWORK (ANN) ARCHITECTURE

Layer (Type)	Output Shape	Param #
dense (Dense)	(None, 64)	384
dropout (Dropout)	(None, 64)	0
dense_1 (Dense)	(None, 32)	2,080
dropout_1 (Dropout)	(None, 32)	0
dense_2 (Dense)	(None, 16)	528
dense_3 (Dense)	(None, 1)	17
Total params		3,009
Trainable params		3,009
Non-trainable params		0

TABLE III
DATASETS OF SIMULATED DRAFT TUBES

Parameter	**Case 1**	**Case 2**	**Case 3**
H1 (mm)	950	1150	1400
H2 (mm)	1190.8	992.8	740.9
Stat. Press. In (kPa)	8.68	8.68	8.68
Stat. Press. Out (Pa)	0.09	0.09	0.09
Dyn. Press. In (kPa)	11.60	11.60	11.60
Dyn. Press. Out (kPa)	1.09	1.09	1.09
Vel. In (m/s)	4.83	4.82	4.82
Vel. Out (m/s)	1.82	1.37	8.01
NW	0.738	0.918	0.817

Implementation of the neural network model leveraged Python's robust scientific computing ecosystem, utilizing Google Colaboratory as the development environment to facilitate collaborative work and computational efficiency. The technical stack incorporated fundamental libraries including NumPy for numerical operations and TensorFlow/Keras for neural network construction, complemented by automated machine learning tools like auto-sklearn for efficient hyperparameter optimization. This combination allowed for the creation of a sophisticated ANN architecture featuring three hidden layers with 64, 32, and 16 neurons respectively as shown in Table II, each employing ReLU activation functions to capture the nonlinear relationships between geometric parameters and hydrodynamic performance.

The training process employed the Adam optimizer with a carefully configured learning rate of 0.001, running for 300 epochs with a batch size of 4 to balance computational efficiency with model convergence. A strategic 80/20 split between training and validation data ensured robust evaluation of the model's predictive capabilities while guarding against overfitting. This approach proved particularly effective, with the model achieving a mean absolute error of just 0.025 on the validation set, demonstrating its ability to accurately predict the NW coefficient based on geometric inputs. Feature engineering played a crucial role in the model's success, with derived parameters such as the H2/H1 ratio and inlet-to-outlet area expansion providing critical insights into the geometric relationships affecting performance. These features, combined with fundamental dimensions and pressure measurements, were normalized using StandardScaler to ensure consistent training behavior across all parameters. The complete modeling workflow seamlessly integrated geometric parameter definition, CFD data preprocessing, neural network construction, and performance optimization into a cohesive framework that reduced computational costs by approximately 70% compared to traditional parametric CFD studies while maintaining prediction accuracy within 0.8% of high-fidelity simulation results. This significant efficiency gain makes the methodology particularly valuable for small hydro plants where computational resources may be limited, opening new possibilities for performance optimization in distributed renewable energy systems.

IV. RESULTS AND DISCUSSION

This section presents the results from CFD simulations and ANN application for hydraulic turbine optimization, focusing on the draft tube.

A. TURBINE IDENTIFICATION

Using technical drawings of turbine components provided by HACKER, CFD analysis was applied to the draft tube geometry using ANSYS Fluent. Three simulations were conducted to evaluate internal flow behavior for different geometric combinations of the sections H1 and H2. These simulations provided key data on pressure distribution, average velocities, and pressure losses along the flow path, allowing the calculation of the NW parameter, used as an indicator of hydraulic efficiency. A crucial step to ensure simulation quality was developing a mesh compatible with the geometric and hydraulic requirements of the model, ensuring adequate resolution in critical flow regions. A well-refined mesh is essential for accurate results, especially in areas with significant pressure and velocity variations, such as curves and transitions. For example, the mesh generated for the draft tube with H1 equal to 950 mm is shown in Figure 2.

In this table dynamic pressure at the inlet and outlet, total pressure, static pressure, NW factor, are included also the real pressure recovery coefficient ($C_{p_{rm}}$), defined as the ratio between the difference of static pressure at the outlet and inlet

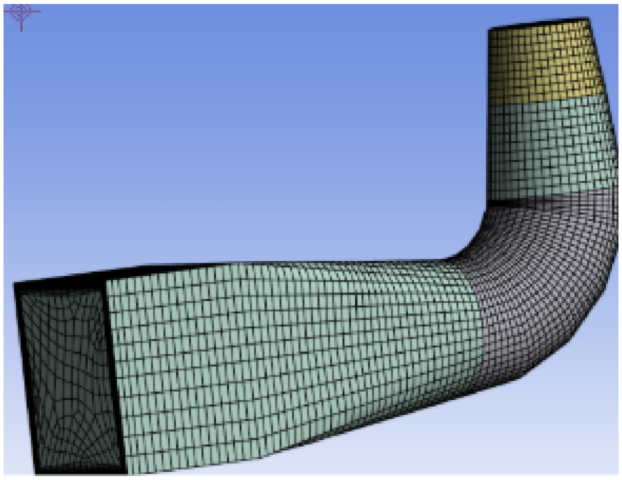

Fig. 2. Mesh of the Draft Tube, from Author. (2025)

TABLE IV
DATASETS OF THE OPTIMIZED DRAFT TUBES

Parameter	Value
Static Pressure Inlet	-7355.530 Pa
Static Pressure Outlet	-3.002 Pa
Total Pressure Inlet	4252.310 Pa
Total Pressure Outlet	1893.020 Pa
Dynamic Pressure Inlet	11607.800 Pa
Dynamic Pressure Outlet	1896.020 Pa
NW (CFD)	0.738
Real Cp	0.958

of the tube and the dynamic pressure at the inlet, as shown in Eq. (2).

$$C_{p_{rm}} = \frac{P_{ex} - P_{en}}{\frac{1}{2} v_{en}^2} \tag{2}$$

Where P_{ex} and P_{en} are the static pressures at the outlet and inlet, respectively, and v_{en} is the corresponding fluid velocity at the inlet. Obtained from CFD simulations, this differs from the ideal C_p due to friction losses, turbulence, flow separation, and other effects [8]. The NW coefficient (Equation (1)) represents the ideal pressure recovery and served as the primary performance metric for the ANN training and optimization. Conversely, the real pressure recovery coefficient ($C_{p_{rm}}$) is obtained from CFD simulations and is used for comparative analysis in the Results and Discussion section, as it accounts for realistic effects like friction and turbulence. Velocity contours inside the draft tube for H1 = 1150 mm as shown in Figure 3.

Allowed visualization of flow distribution, highlighting zones of recirculation and potential losses. These results, alongside those from those tables formed the basis for training the artificial neural network (ANN).

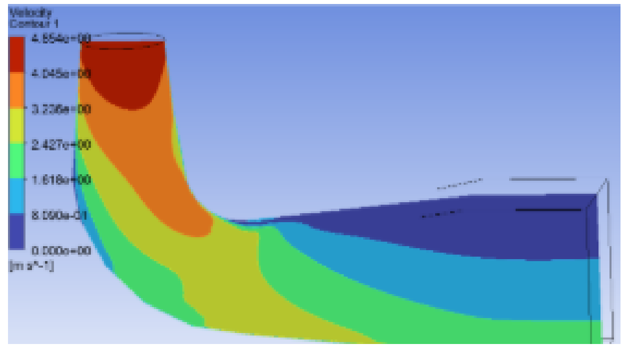

Fig. 3. Velocity Contours, from Author. (2025)

B. CFD SIMULATION RESULTS

CFD simulations were crucial for understanding draft tube flow and generating data for ANNs. Simulations for different geometries (950mm, 1150mm, and 1400mm) provided details on pressure distribution, velocity, and vortex formation, essential for identifying energy losses and instabilities.

C. NEURAL NETWORK

The ANN was developed with a sequential architecture comprising three hidden layers 64, 32, and 16 neurons, using ReLU activation functions, trained on the CFD data previously described. As shown in Figure 4, the training process demonstrated stable convergence, with both loss (MSE) and mean absolute error (MAE) smoothly decreasing over 300 epochs. This behavior indicates effective learning without overfitting, as training and validation curves remained parallel towards the end, confirming the model's ability to capture the nonlinear relationships between geometric parameters and the NW factor.

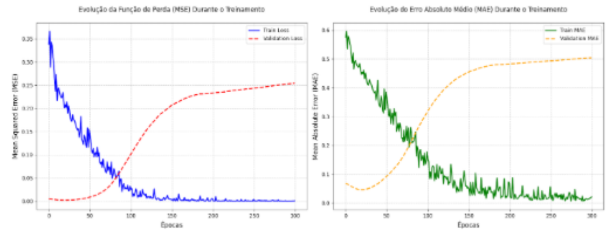

Fig. 4. Evolution of MSE and MAE, from Author. (2025)

D. DRAFT TUBE

Using the ANN, the optimal combination of H1 and H2 maximizing NW was identified, respecting the physical constraint that their sum equals 2140.85 mm to reflect realistic turbine dimensions. The optimized results Table V show H1 = 1240 mm, H2 = 900.8 mm, with NW = 0.9146, placing it in the upper range of typical NW values (0.7 to 0.95) for well-designed draft tubes, consistent with literature [10]. This result highlights the effectiveness of the hybrid CFD-ANN approach, achieving significant optimization with relatively few CFD runs.

TABLE V
OPTIMIZED PARAMETERS

Parameter	Value
H1	1240.0 mm
H2	900.8 mm
NW	0.9146

Figure 5 shows the draft tube geometry modeled in ANSYS with the optimized heights found by the ANN, used for further testing.

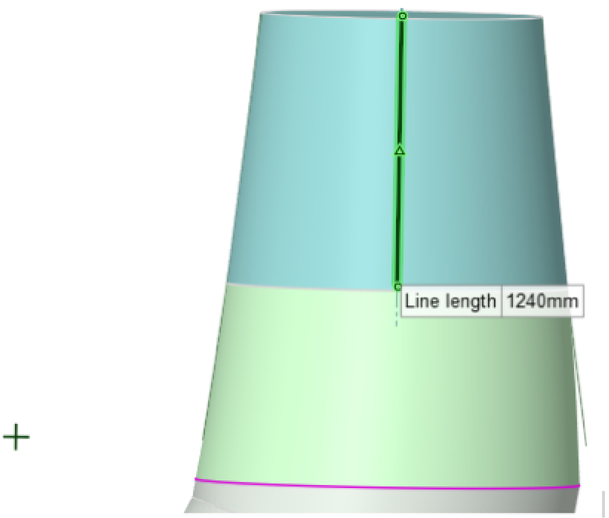

Fig. 5. Draft Tube of 1240mm, from Author. (2025)

E. COMPARISON BETWEEN CFD AND ANN

Post-optimization CFD simulation using ANSYS produced an NW value of 0.906 (Table VI), closely matching the ANN prediction (0.914), confirming the model's accuracy.

Validation comparing CFD results and ANN predictions for the optimized turbine shows less than 1% difference, consistent with recent studies in turbomachinery [4]. Such accuracy confirms the ANN's capacity to generalize well and offer computationally efficient alternatives to traditional CFD.

TABLE VI
RESULTS OF THE OPTIMIZED DRAFT TUBES

Parameter	Value
Static Pressure Inlet (Pa)	-9,576,520
Static Pressure Outlet (Pa)	307
Total Pressure Inlet (Pa)	2,035,070
Total Pressure Outlet (Pa)	1,200,110
Dynamic Pressure Inlet (Pa)	11,611,600
Dynamic Pressure Outlet (Pa)	1,199,800
NW (CFD)	0.906
Real Cp	0.958

REFERENCES

[1] Chakrabarty, S., Sarkar, B.K., and Maity, S., 2016. CFD Analysis of the Hydraulic Turbine Draft Tube to Improve System Efficiency. MATEC Web of Conferences, 40, p. 02003.
[2] Hacker Industrial, *Design of the Draft tube*. Technical Note, 2025.
[3] International Hydropower Association (IHA), *2024 World Hydropower Outlook: Opportunities to advance net zero*. London, UK: IHA, 2024. [Online]. p. 96 Available: https://www.hydropower.org/publications
[4] Lalonde, E.R., Vischschraper, B., Bitsamlak, G., and Dai, K., 2021. Evaluation of a Neural Network-Based Surrogate Aerodynamic Wind Turbine Blade Model. Journal of Wind Engineering, 45(2), pp. 1-12.
[5] Poudel, R., Chitrakar, S., Qian, Z., and Thapa, B., 2024. Experimental Investigations of Sediment Erosion in Francis Turbine Using Non-Recirculating Sediment Test Rig. Energy Science & Engineering, 13, pp. 700-713.
[6] REN21, *Renewables 2023: Analysis and forecast to 2028*. Paris: International Energy Agency (IEA), 2023. p. 336.
[7] Ristić, B., and Božić, I., 2021. Digital Technologies Emergence in the Contemporary Hydropower Plants Operation. International Conference on Sustainable Development of Energy, Water and Environment Systems(SDEWES).
[8] Sheikh, H.M., et al., 2022. Optimization of the Shape of a Hydrokinetic Turbine's Draft Tube and Hub Assembly Using Design-by-Morphing with Bayesian Optimization. Computer Methods in Applied Mechanics and Engineering, 401, p. 115654.
[9] Tao, R., Lu, J., and Su, C., 2024. Influence of Runner Blade Number on Hydraulic Performance and Flow Control in Draft Tube of Francis Turbine. Advances in Mechanical Engineering, 16(3), pp. 1-20.
[10] Tian, X., Pan, H., Hong, S., and Zheng, Y., 2015. Improvement of Hydro-turbine Draft Tube Efficiency Using Vortex Generator. Advances in Mechanical Engineering, 7(7), pp. 1-8.
[11] Tiwari, G., Kumar, J., Prasad, V., and Patel, V.K., 2020. Utility of CFD in the Design and Performance Analysis of Hydraulic Turbines – A Review. Energy Reports, 6, pp. 2410-2429.
[12] Trivedi, C., Gandhi, B., and Cervantes, J.M., 2013. Effect of Transients on Francis Turbine Runner Life: A Review. Journal of Hydraulic Research, 51(1), pp. 1-12.
[13] Zhou, X., Huang, Q., Hu, X., and Dai, X., 2024. Study of the Effects of Modified Draft Tube with Inclined Conical Diffuser on Draft Tube and Upstream Region. Journal of Physics: Conference Series, 2707, p. 012064.