International Journal on
Advanced Science
Engineering
Information Technology

Vol.12 (2022) No. 3
ISSN: 2088-5334

Pumps as Turbines (PATs) by Analysis with CFD Models

Frank Plua [a,b*], Victor Hidalgo [c,d], Edgar Cando [c,d], Modesto Pérez-Sánchez [e] , P. Amparo López-Jiménez [e]

[a] Departamento de Ingeniería Civil y Ambiental, Escuela Politécnica Nacional, Ladrón de Guevara E11.253, Quito, 170525, Ecuador
[b] Doctorado en Ingeniería del Agua y Medioambiental, Camí de Vera, Valencia, 46022, España
[c] Departamento de Ingeniería Mecánica,Escuela Politécnica Nacional, Ladrón de Guevara E11.253, Quito, 170525, Ecuador
[d] Laboratorio Informática-Mecánica, Ladrón de Guevara E11.253, Quito,170525, Ecuador
[e] Departamento de Ingeniería Hidráulica y Medio Ambiente, Universitat Politècnica de València, Camí de Vera, Valencia, 46022, Spain
*Corresponding author: *frank.plua@epn.edu.ec*

Abstract—**Pumps as turbines (PATs) are the typical solution for electrification using micro hydropower plants (MHP) in the rural sector. Other engineering applications where lately the use of PATs have increased are irrigation, water supply, and energy recovery systems due to their availability, short delivery time, long service life, economic feasibility, construction, and maintenance advantages. However, selecting the suitable pump(s) is difficult because manufacturers only provide performance curves when operating in pump mode; therefore, there is no universal method to predict that issue. For this reason, theoretical, analytical, experimental, and numerical simulation research have been made to predict these curves and the PATs' performance. The present paper analyzes PATs with Computational Fluid Dynamics (CFD) based on advanced research. For this aim, information from a wide range of types of pumps with different rotation speeds was classified to examine case approaches, computational domains, mesh generation, boundary conditions, optimization of elements, and CFD package used to establish the effectiveness of this tool and to find characteristics which have not been enough investigated at present. Most studies used CFD simulations with ANSYS code and K-ε turbulence closure model, which presented adequate results. Finally, this paper shows that numerical simulations with CFD analysis were successfully carried out to determine pump performance and predict curves in direct and reverse mode, improving certain components and conducting more profound research on certain specific issues.**

Keywords—**Pump as Turbine (PATs); CFD (Computational Fluid Dynamics); Best Efficient Point (BEP); efficiency prediction; mycro-hydropower.**

Manuscript received 21 May 2021; revised 9 Sep. 2021; accepted 24 Dec. 2021. Date of publication 30 Jun. 2022.

I. INTRODUCTION

The use of PATs in systems is the new trend in applications such as irrigation[1]–[4], hydroelectricity[5], drinking water supply [6], [7] and, energy recovery systems[8]–[12]. The main advantages of PATs are availability[1], short delivery time, long service life, and economic feasibility, construction and, maintenance advantages. Different types of analysis have been carried out to predict the characteristics curves of pumps working in turbine mode to select the machine to be placed in the systems[13], [14]. However, this task has not yet been solved for the existing range of possibilities. The main problems related to the use of PATs are that these cannot handle the flow, and their performance is low in variable flow rates. Computational Fluid Dynamics is a tool that allows complex modeling phenomena in fluids and is an alternative to understand the main aspects that govern the flow behavior in PATs. The application of CFDs in PATs applications is to predict the characteristic curves, their performance in specific case studies and optimize their design according to requirements[15].

To carry out an adequate study of numerical simulation with CFDs is important to define and create, with the help of specialized software, a high-quality mesh that reproduces the flow's geometry and allows optimal use of computational resources. Once the computational domain is defined, it is necessary to establish the equations and turbulence models used to solve the Navier Stokes equations [16], [17]. The boundary conditions are necessary to run the model and simulate permanent or transient flow types with this information. In this research, it is required to get the main modeling conditions used by the authors and the results of their studies.

Despite the considerable amount of research carried out, it has not been possible to accurately predict the characteristic curves of pumps that work as turbines, so their selection and implementation within power generation systems is a complicated task. Therefore, the present paper seeks to establish the analysis of different methodologies regarding the use of CFDs in PATs numerical modeling to verify whether it is a reliable element to predict the behavior of these elements to propose expressions that allow generating the operating curves reliably.

II. MATERIALS AND METHOD

The main activities where the study focuses are: Search for research reports that have made CFD models in PATs - this information is better if another analysis complements the research, whether experimental, theoretical, or empirical-; Analysis of the data of the main parameters related to the simulation: computational domain, grid analysis, boundary conditions, used CFD package; and, analysis of results. The results will try to determine which parameters have been the most used, verify the reliability of the application of the CFD tool and analyze particular research cases. For executing the analysis of numerical CFD simulations in PATs, the main tasks are indicated in the flowchart of the methodology used. See Fig. 1

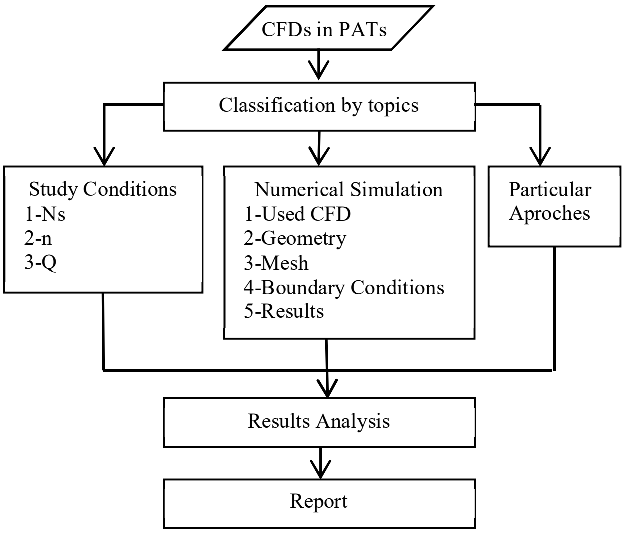

Fig. 1 Investigation Flowchart

III. RESULTS AND DISCUSSION

A. General approach

As indicated in section II, an initial number of 65 documents related to PATs were checked and classified by topics. Types of pumps analyzed were: axial flow pump; mixed flow pump[6]; conventional centrifugal pump,[18]–[21]; radial flow multistage pump [22]; mixed flow pump with mixed flow diffuser and axial diffuser[23]; and centrifugal pumps with a horizontal and vertical axis and single-stage and multistage[24]. In general terms, numerical simulations were carried out aiming to determine the performance in the pump[9], execute analysis of flow in turbine mode [22]; and predict head and flow values and extrapolate [18]. Other objectives related to specific investigations are explained at the end of this section. The consulted researches focused on two types of analysis: comparison between CFD numerical simulations and experimental test,[6], [20], [21], [25], [26];and, comparison between theoretical analysis, CFD numerical simulations and experimental test [21], [27], [28].

B. Flow rates, specific speed, and rotational speed

Tables I,II, III show values of specific speed (Ns), rotational speed (n) and flow rate (Q) used in investigations and their respective references. Most studies are between 10 and 25 for Ns and from 1000 to 2000 rpm for n with the nine times repeated value of 1500 rpm. In [6], studies were carried out to assess PAT behavior in a range between 60 percent and 140 percent from BEP.

TABLE I
SPECIFIC SPEED

Ns	Reference
0,8	[29]
1,0	[29]
12,9	[30]
18,0	[30]
18,1	[28]
18,5	[19]
20,5	[31]
20,5	[21]
21,0	[32]
23,0	[30]
23,0	[22]
26,1	[20]
36,4	[28]
37,6	[31]
37,6	[21]
38,2	[20]
52,8	[28]
64,0	[31], [21]
239,0	[33]
257,0	[23]
260,0	[25]
306,0	[23]

TABLE II
ROTATIONAL SPEED

n rpm	Reference #
250	[34]
260	[35]
600	[34]
680	[34]
780	[34]
800	[6]
810	[36]
850	[34]
880	[34]
900	[6]
930	[36]
1000	[6], [22]
1020	[34]
1050	[36]
1140	[34]
1170	[36]
1275	[36], [34]
1450	[37]
1480	[32]
1500	[18] [19], [26],[27], [28], [33], [36], [34]
1750	[5]
1800	[23]
1850	[23]
2880	[24]
3900	[32]

TABLE III.A
FLOW RATE

Q l/s	Reference #
2,9	[36]
5,5	[36]
7,7	[29]
10,0	[38]
13,8	[29]
25,0	[26]
25,0	[28]
25,2	[28]
26,5	[28]
34,3	[18]
35,0	[39]
50,0	[24]
85,0	[23]
168,0	[32]
300,0	[23]

TABLE III.B
FLOW RATE

Q kg/s	Reference #
13,0	[20]
8,9	[38]
11,7	[38]
14,8	[38]
17,8	[38]

C. *Numerical simulations*

Below is the summary of the main parameters related to the numerical simulation of PATs using CFD numerical modeling. Turbulence and closure equation models: To solve Navier Stokes equations, RANS (Reynolds Average Navier-Stokes)[6], [18], [22]–[28], [30], [32]–[36], [38], [40] and URANS (Unsteady RANS) [11], [41], [42] turbulence models were used. Closure turbulence models used in the investigations were k-ε [6], [19], [21], [23], [24], [26], [27], [32],[34], [43], [37], [44], [45] Menter's SST [22], [23], Spalart Allmaras [18],k-ω [25], [29] , k-ω SST [11], [29], [41], [42] , k-ε with wall functions[36], [40] and RNG k-ε[31] .

1) CFD Packages and Conditions: ANSYS-CFX software was the most used for CFD modelling and numerical computations. ANSYS was used in 20 studies [19], [22], [25]–[33], [35], [38], [46]–[52]; CFD Code Fluent was used in 1[18]; Open foam in 3 [11], [41], [42]; FloEFD in 3 [34], [36], [40]; Ansys Fluent in 2,[24], [43]; and CFX [20], ASTROE [25], Pump Linx [21], CFTurbo [30], in 1 study. Incompressible Navier Stokes where analyzed by steady [24], [39], and unsteady condition [23], [24], [39].

2) Mesh Generation: Next tables present tools, type of elements, and the number of cells with their respective reference compiled in this research.

TABLE IV
MESH TOOL

Mesh tool -	Reference #
ICEM CFD	[20],[22], [24], [25], [26], [27], [28],[33], [32],[38][41], [31]
GGI	[23]
SolidWorks	[36], [34], [31],[33]
Floworks	[36]
ANSYS Workbench	[35]
BladeGen	[19], [28]
PumpLinx	[21]
Ansys Turbogrid	[30]

TABLE V
NUMBER OF CELLS

#cells -	Reference #
1,0x10^{6}-1,5x10^{6}	[39]
2,6x10^{6}	[27]
1,0x10^{6}	[26]
2,5x10^{6}	[23]
1,2x10^{5}	[32]
12x10^{6}	[19]
1,2x10^{6}	[28], [35]
1,8x10^{6}	[20]
1,0x10^{5}	[36], [34]
11x10^{6}	[11], [43]
2,1x10^{5}	[38]
1,5x10^{5}	[38]
1,4x10^{5}	[38]
4,2x10^{6}	[31]

TABLE VI
TYPE OF ELEMENT

mesh -	Reference #
unstructured tetrahedral	[6], [18], [24], [38], [29]
block structured	[22]
mixed	[53]
prismatic cells, tetrahedrons	[26]
structured hexaedral grid	[19], [23], [24], [26], [27], [33], [29]
pyramids	[29]

3) Boundary conditions: Next table shows the boundary conditions used in this research and their respective reference.

TABLE VII.A
BOUNDARY CONDITIONS

Inlet -	Reference #
Total pressure	[6], [20]
Mass flow rate	[19], [23], [30], [35], [41]
Stagnation pressure	[18]
Constant total pressure	[48]
Static pressure	[26],[27], [33]
Volumetric flow	[21], [36], [34], [29]
Flowrate	[40]

TABLE VII.B
BOUNDARY CONDITIONS

Outlet	**Reference**
-	#
Mass flow rate	[6], [20], [27], [33]
static pressure	[19],17, [23], [30], [36],33, [29]
constant static pressure	[5]
pressure difference	[18]
pressure	[21], [35]
pressure recorded	[34]
uniform pressure	[32], [41],*36*

4) Results: The CFD methodology is an increasingly used tool to effectively predict the performance of a pump working as a turbine because comparison with experimental tests gives results with acceptable accuracy.[19], [20], [24], [25], [27], [32]; CFD presented satisfactory results for values of Head and efficiency in regions near to BEP [6]. Results between numerical analysis and manufacturer data in pump mode are in good coincidence. Regards studies of operation in reverse mode, hydraulic efficiency can be higher [22], similar [23], or lower concerning direct mode. Also, flow in BEP in that mode is attained at higher flows respect pump mode[6]. From an economic viewpoint[30], the use of PATs represents lower payback time than a turbine in low capacities.

With regards to difficulties in some studies, some researchers reported that there was a considerable error between experimental and numerical analysis [6], [22], and the results of unsteady simulations are very different from steady classical 1-D [39]. With regards to the operation of PATs, in reverse mode, machines generate pulsations; if the rotating speed increases, the effects of pulsations will also be increased [39]; and internal recirculation will appear in the impeller in high and low flowrates.

Some researchers established explanations about the difference between results from experimental and numerical analysis. The principal reasons were: CFD and pump geometry are not identical; estimation of losses is inaccurate [22], and more experience in the computational analysis is needed.

D. Particular approaches

Through CFD numerical studies, some researchers have presented different analysis and approach proposals aiming to improve performance in PATs or their elements. Kerschberger [25] has developed an inverse design method to optimize and redesign the blade profile. First, an analysis with a 3D-Euler frictionless CFD code to investigate the velocity field and pressure phenomenon at the blade channels. Then, after an optimization based on the first results, a fully 3D-RANS simulation with a commercial Solver ANSYS CFX5 v1.2 was performed. The comparison of CFD and experimental results showed a good level of agreement and a significant pump improvement.

Páscoa *et al.* [18] considered results from direct operation in a pump working as a turbine, presented correlations to predict head and flow values extrapolated (using RANS code with Spalart-Allmaras) of PATs. This approach could be useful to provide an initial approximation to the range of pumps to use as PATs. Another approach was developed to search for the most efficient rotational speed at each constant head.

Yang *et al.* [27] compared three methods to predict the behavior of PATs: a theoretical and empirical analysis, CFD; and experimental. In theoretical analysis, a new formula was created in the function of tested PATs and then compared with Stepanoff and Sharma predictions. Then, in the experimental setup, a single-stage centrifugal pump (n=1500 rpm) was made and tested at Jiangsu University. The measure devices were torque meter, turbine flow meter, and pressure gauge. The numerical investigation was developed with ANSYS CFX code, k-ε turbulence model with the ICEM-CFD software, the structured hexahedral grid was generated for a mesh with blocks for different parts of the pump. The achieved results were that the numerical method's agreement could predict PAT performance and BEP with acceptable accuracy. Another research conducted at Jiangsu University analyzed the influence of the number of blades on the presence of pulsation pressures and performance on PATs [26]. The numerical investigation was validated by comparing its results with the experimental results. Conditions of numerical analysis were similar to the mentioned investigation. The research concludes that the increase in the number of blades decreases the pressure pulsations, and the highest efficiency can be achieved in the PATs if an optimal number of blades is found (8 in this case). These factors also depend on the radial gap between the volute tongue and impeller tips [33]. The results showed an optimal radial relation with the BEP, and if the radial gap is increased, the high-frequency pressure pulsation is reduced.

Wang *et al.* [19] presented a numerical simulation using the ANSYS CFX 12.0 package, k-ε closure model, and structured hexahedral mesh to analyze the influence of forwarding curved blades pumps, which was tested with experiments. The same author[28] presented in 2017 a special impeller to use in PATs (Ns=18.1) through theoretical, experimental, and numerical study. The designed impeller improved notoriously performance and was recommended as a convenient alternative. It was verified in other centrifugal pumps with different specific speeds (Ns=36.4 and 52.8). Numerical simulation was slightly similar to previous analysis with the difference in grid elements (structured hexahedral) and boundary conditions.

Su *et al.* [20] focus the study on obtaining the rules of the flow rate distribution analyzing the characteristics of the internal flow rate. A centrifugal pump single-suction was simulated using numerical simulation. The computational domain was divided into five components. CFX code was used to solve the equations with k-ω SST turbulence model in unsteady CFD simulation. The study's principal conclusion is that average time velocity diminishes between the outlet and inlet section of the PAT. The simulation was validated with an experiment in Guanyi Pump Co. Ltd, Foshan, China, and CFD results are suitable with experiments for tested Ns.

Frosina *et al.* [21] developed a new method for PATs with numerical simulation. Three centrifugal pumps with various specifics speeds (Ns: 20.5, 37.6, 64.0) were simulated in direct mode and compared with manufacturers' data. Then, the numerical model was validated with the results of tests executed at the Federico II University of Naples. Finally, the results of this methodology were compared with other prediction methods. There is a dispersion because some methods present high deviations, and the others are similar. PumpLinx CFD commercial code was used in the analysis.

Bahreini [30] applied CFturbo V.9 software, designed three centrifugal pumps (Ns: 12.85, 17.96, 22.98). The performance of the pumps was obtained with ANSYS CFX 16 code, and calibration presented acceptable results. Then, turbine mode was investigated numerically, and efficiency in turbine mode was compared with pump mode, so the first mode of operation has lower efficiencies than the second. Economic analysis was carried out, and the study concludes that PATs have lower payback time than turbines in low capacities.

Pugliese *et al.* [24] evaluated the performance of different models of centrifugal PATs (48 rps): Horizontal and vertical axis; single-stage and multistage; and further motor class efficiency. The study aimed to analyze analytical relationships from the literature that predict the performance of PATs at the BEP and propose a new model. This model resulted in predicting characteristic curves although, in some cases, underestimates head and power numbers. A CFD model was carried out for Horizontal Axis Single Stage PAT with ANSYS Code and steady and unsteady conditions. CFD model was reliable to predict characteristics curves.

Three numerical and experimental analyses were conducted in similar conditions. For the numerical study of PATS, the FloEFD tool was used, and meshes were built with Solid Works CAD System after an optimization process. The boundary conditions were volume flow rate at the inlet and static and recorded pressure at the outlet. Experimental developments were accomplished at the University of Lisbon. Pérez-Sanchez *et al.* [36] found different machines and rotational speeds the head drop, and Simao *et al.* [34] the influence of rotation speed and rotation variation on PATs, and the same authors[40] executed analysis for PATs installed in parallels systems. These studies have contributed to a better understanding of the PATs behavior in systems.

Capurso *et al.* [41] and Capurso *et al.* [42] investigated the factor slip phenomenon on the prediction of performance in PATs. They introduced a new coefficient that improved the accuracy of a 1D model for predicting characteristic curves in reversible pumps. Numerical simulation in PATs was analyzed by OpenFoam code to solve U-RANS equations with $\kappa-\omega$ SST turbulence model. The grid was generated with ICEM-CFD, and the number of cells after a sensitive analysis was 11 000 000. The same author used equal code, domain, and boundary conditions from numerical simulation [11] to design a novel impeller. The new impeller presented a significant improvement regarding the original pump, because the novel element optimizes the geometry because it increases the slip factor and reduces secondary losses in the PAT.

Lal [38] carried out an analysis of cavitation and NPSHr at PATs under different operating conditions. ANSYS code was used, and the grid was generated in ANSYS ICEM CFD for three blocks: Impeller, Casing, and Inlet Pipe with 1380152, 213813, and 156862 tetrahedral elements, respectively. Simulations showed that efficiency in PAT is higher 2% than pump mode, cavitation reduces about 2% efficiency in direct and inverse modes, and NPSH values vary by 3% if loading goes from 60% to 120% at 1 atm.

Rossi *et al.* [29] presented a model to obtain PATs performance in in-and out-of-design conditions based on a predicting model derived from experimental data from 32 PATs and numerical simulation using CFD analysis. ANSYS-CFX package was used to resolve RANS equations with $\kappa-\omega$ τηε closure model. The predictive method has excellent concordance with CFD analysis near to BEP points, and for far regions, there is a slight mismatch.

Capurso *et al.* [11] analyzed cavitation and NPSHr at PATs under different operating conditions. ANSYS code was used, and the grid was generated in ANSYS ICEM CFD for three blocks: Impeller, Casing, and Inlet Pipe with 1380152, 213813, and 156862 tetrahedral elements, respectively. Simulations showed that efficiency in PAT is higher 2% than pump mode, cavitation reduces about 2% efficiency in pump and pump as turbine modes and NPSH values vary 3% if loading goes from 60% to 120% at 1 atm.

Renzi et al. [35] conducted a case study of an axial flow pump in pump and PAT mode in a wastewater sewer to place it in a treatment plant. Analysis was executed with ANSYS Workbench to assess the technical and economic viability of the solution. Energy recovery and economic savings were achieved.

Jemal [31] presented a review based on theoretical, numerical, and experimental research conducted on PATs. Numerical simulation is used to predict performance curves, optimize their elements, perform case studies, know the performance in parallel systems, and select suitable pumps. Most simulations were performed in ANSYS, and sensitivity studies were performed to obtain the optimal grid for the simulation.CFD might be an excellent tool for running PATs studies, but it is not enough to determine the exact solution. Koswara *et al.* [43] analyzed PATs changing the angle of impeller blade tip using ANSYS FLUENT software. Results show that the optimal impeller blade angle is 25°.

IV. CONCLUSION.

This paper presents an analysis of the development of numerical simulations using pump as turbines with CFD techniques. This tool has been used to predict characteristic curves and performance of pumps in direct and reverse modes, improve certain machine elements, simulate its operation in different study cases, and be an effective solution in the various research approaches. Better simulations will be done according to the researcher's experience, considering all the losses that occurred in the phenomena and with a good quality of the mesh. However, at the moment, the prediction and selection of suitable pumps to operate in reverse mode for a wide range is still an unsolved issue.

NOMENCLATURE

PATs	Pump as turbine
BEP	Best efficient point
CFD	Computational Fluid Dynamics
k	Turbulent kinetic energy
SST	Shear stress transport

Greek letters

ε	turbulent dissipation
ω	specific dissipation

ACKNOWLEDGMENT

The authors gratefully acknowledge the financial support provided by Escuela Politécnica Nacional for the development of the project: PIS 19-06, PII-DIM-2019-06, and Process Simulation Laboratory of Mechanical Engineering Faculty.

REFERENCES

[1] M. Pérez- Sánchez, F. J. Sánchez-Romero, H. Ramos M, and P. A. López-Jiménez, *Bombas operando como turbinas (PAT) : principios de funcionamiento y selección*. Universidad Politécnica de Valencia, 2020.

[2] M. Pérez-Sánchez, J. F. P. Fernandes, P. J. C. Branco, P. A. López-Jiménez, and H. M. Ramos, "PATs Behavior in Pressurized Irrigation Hydrants towards Sustainability," *Water*, vol. 13, no. 10, p. 1359, 2021, doi: 10.3390/w13101359.

[3] J. W. Kim *et al.*, "Simultaneous efficiency improvement of pump and turbine modes for a counter-rotating type pump-turbine," *Adv. Mech. Eng.*, vol. 8, no. 11, pp. 1–14, 2016, doi: 10.1177/1687814016676680.

[4] S. Lee, C. Pomeroy, and S. Burian, "Setting Future Water Rates for Sustainability of a Water Distribution System," *J. Water Resour. Plan. Manag.*, vol. 147, no. 2, p. 04020108, 2021, doi: 10.1061/(asce)wr.1943-5452.0001313.

[5] M. Binama, W. T. Su, X. Bin Li, F. C. Li, X. Z. Wei, and S. An, "Investigation on pump as turbine (PAT) technical aspects for micro hydropower schemes: A state-of-the-art review," *Renew. Sustain. Energy Rev.*, vol. 79, no. April 2016, pp. 148–179, 2017, doi: 10.1016/j.rser.2017.04.071.

[6] S. Rawal, "Numerical Simulation on a Pump Operating in a turbine mode," *Proc. 23rd Int. users Symp.*, 2007.

[7] C. A. M. Ávila, F. J. Sánchez-Romero, P. A. López-Jiménez, and M. Pérez-Sánchez, "Leakage management and pipe system efficiency. Its influence in the improvement of the efficiency indexes," *Water (Switzerland)*, vol. 13, no. 14, 2021, doi: 10.3390/w13141909.

[8] M. Pérez-Sánchez, F. J. Sánchez-Romero, H. M. Ramos, and P. A. López-Jiménez, "Improved planning of energy recovery in water systems using a new analytic approach to PAT performance curves," *Water (Switzerland)*, vol. 12, no. 2, 2020, doi: 10.3390/w12020468.

[9] L. E. C. Rosado, P. A. Lopez, F. Sanchez, and P. Conejos, "Applied Strategy to Characterize the Energy," *Water*, vol. 12, no. 6, pp. 1–22, 2020.

[10] S. Ahmadi, S. Yadollah, and Ali Vakili, "Frameworks, quantitative indicators, characters, and modeling approaches to analysis of energy system resilience: A review," *Renew. Sustain. Energy Rev.*, vol. 144, p. 110988, Jul. 2021, doi: 10.1016/J.RSER.2021.110988.

[11] T. Capurso, - L Bergamini, -S M Camporeale, - B Fortunato, and - M Torresi, "CFD Analysis of the Performance of a Novel Impeller for a Double Suction Centrifugal Pump Working as a Turbine," in *13 th European Conference on Turbomachinery Fluid Dynamics & Thermodynamics*, 2019, pp. 1–11.

[12] M. Stefanizzi, T. Capurso, G. Balacco, M. Binetti, S. M. Camporeale, and M. Torresi, "Selection, control and techno-economic feasibility of Pumps as Turbines in Water Distribution Networks," *Renew. Energy*, vol. 162, pp. 1292–1306, 2020, doi: 10.1016/j.renene.2020.08.108.

[13] F. A. Plua, F. J. Sánchez-Romero, V. Hidalgo, P. A. López-Jiménez, and M. Pérez-Sánchez, "New expressions to apply the variation operation strategy in engineering tools using pumps working as turbines," *Mathematics*, vol. 9, no. 8, pp. 1–17, 2021, doi: 10.3390/math9080860.

[14] C. A. M. Ávila, F. J. Sánchez-Romero, P. A. López-Jiménez, and M. Pérez-Sánchez, "Definition of the operational curves by modification of the affinity laws to improve the simulation of pats," *Water (Switzerland)*, vol. 13, no. 14, pp. 1–17, 2021, doi: 10.3390/w13141880.

[15] F. Plua, V. Hidalgo, P. A. López-Jiménez, and M. Pérez-Sánchez, "Analysis of applicability of cfd numerical studies applied to problem when pump working as turbine," *Water (Switzerland)*, vol. 13, no. 15, pp. 1–18, 2021, doi: 10.3390/w13152134.

[16] V. Hidalgo *et al.*, "Scale-adaptive simulation of unsteady cavitation around a Naca66 hydrofoil," *Appl. Sci.*, vol. 9, no. 18, 2019, doi: 10.3390/app9183696.

[17] E. Cando *et al.*, "Unsteady numerical analysis of the liquid-solid two-phase flow around a step using Eulerian-Lagrangian and the filter-based RANS method," *J. Mech. Sci. Technol.*, vol. 31, no. 6, pp. 2781–2790, 2017, doi: 10.1007/s12206-017-0521-6.

[18] J. C. Páscoa, F. J. Silva, J. S. Pinheiro, and D. J. Martins, "A new approach for predicting PAT-pumps operating point from direct pumping mode characteristics," 2012.

[19] T. Wang, F. Kong, S. Yang, and Y. Fu, "Numerical Study on Hydraulic Performances of Pump as Turbine with Forward-Curved Blades," in *Fluids Engineering Division Summer Meeting*, 2014, pp. 1–6, doi: https://doi.org/10.1115/FEDSM2014-21347.

[20] X. Su, S. Huang, X. Zhang, and S. Yang, "Numerical research on unsteady flow rate characteristics of pump as turbine," *Renew. Energy*, vol. 94, pp. 488–495, Aug. 2016, doi: 10.1016/j.renene.2016.03.092.

[21] E. Frosina, D. Buono, and A. Senatore, "A Performance Prediction Method for Pumps as Turbines (PAT) Using a Computational Fluid Dynamics (CFD) Modeling Approach," *Energies*, vol. 10, no. 1, 2017, doi: 10.3390/en10010103.

[22] M. Sedlář, J. J. Jiříšoukal, and M. Komárek, "CFD analysis of middle stage of multistage pump operating in turbine regime.," in *Engineering Mechanics*, 2009, vol. 16, no. 6, pp. 413–421.

[23] P. Hlbočan and M. Varchola, "Numerical Simulation on a Mixed-Flow Pump Operating in a Turbine Mode," *Eng. Mech.*, vol. 20, no. 2, pp. 97–105, 2013.

[24] F. Pugliese, F. De Paola, N. Fontana, M. Giugni, G. Marini, and J. F. Francos, "Experimental and numerical investigation of centrifugal Pumps As Turbines," in *Proceedings of the 10th International Conference on Energy Efficiency in Motor Driven System*, 2017, pp. 6–7.

[25] P. Kerschberger and A. Gehrer, "Hydraulic development of high specific-speed pump-turbines by means of an inverse design method, numerical flow-simulation (CFD) and model testing," *IOP Conf. Ser. Earth Environ. Sci.*, vol. 12, p. 012039, Aug. 2010, doi: 10.1088/1755-1315/12/1/012039.

[26] S. S. Yang, F. Y. Kong, X. Y. Qu, and W. M. Jiang, "Influence of blade number on the performance and pressure pulsations in a pump used as a turbine," *J. Fluids Eng. Trans. ASME*, vol. 134, no. 12, 2012, doi: 10.1115/1.4007810.

[27] S. S. Yang, S. Derakhshan, and F. Y. Kong, "Theoretical, numerical and experimental prediction of pump as turbine performance," *Renew. Energy*, vol. 48, pp. 507–513, Dec. 2012, doi: 10.1016/j.renene.2012.06.002.

[28] T. Wang, C. Wang, F. Kong, Q. Gou, and S. Yang, "Theoretical, experimental, and numerical study of special impeller used in turbine mode of centrifugal pump as turbine," *Energy*, vol. 130, pp. 473–485, 2017, doi: 10.1016/j.energy.2017.04.156.

[29] M. Rossi, A. Nigro, and M. Renzi, "A predicting model of PaTs' performance in off-design operating conditions," in *Energy Procedia*, 2019, vol. 158, pp. 123–128, doi: 10.1016/j.egypro.2019.01.056.

[30] A. Bahreini and A. Sattari, "Numerical and Economic Study of Performance of Centrifugal Pump as Turbine," *J. Comput. Appl. Mech.*, vol. 48, no. 2, pp. 151–160, 2017, doi: 10.22059/jcamech.2017.232024.137.

[31] A. N. Jemal and M. G. Haile, "Comprehensive Review of Pump as Turbine," *Renew. Energy Sustain. Dev.*, vol. 5, no. 2, p. 68, Dec. 2019, doi: 10.21622/resd.2019.05.2.068.

[32] J. B. Bogdanovic´-Jovanovic´, D. R. Milenkovic´, D. M. Svrkota, B. Bogdanovic´, and Z. T. Spasic´, "Pumps used as turbines: Power recovery, energy efficiency, CFD analysis," *Therm. Sci.*, vol. 18, no. 3, pp. 1029–1040, 2014, doi: 10.2298/TSCI1403029B.

[33] S. S. Yang, H. L. Liu, F. Y. Kong, B. Xia, and L. W. Tan, "Effects of the radial gap between impeller tips and volute tongue influencing the performance and pressure pulsations of pump as turbine," *J. Fluids Eng. Trans. ASME*, vol. 136, no. 5, 2014, doi: 10.1115/1.4026544.

[34] M. Simão, M. Pérez-Sánchez, A. Carravetta, P. López-Jiménez, and H. M. Ramos, "Velocities in a centrifugal PAT operation: Experiments and CFD analyses," *Fluids*, vol. 3, no. 1, Mar. 2018, doi: 10.3390/fluids3010003.

[35] M. Renzi, P. Rudolf, D. Štefan, A. Nigro, and M. Rossi, "Energy recovery in oil refineries through the installation of axial Pumps-as-Turbines (PaTs) in a wastewater sewer: A case study," in *Energy Procedia*, 2019, vol. 158, pp. 135–141, doi: 10.1016/j.egypro.2019.01.058.

[36] M. Pérez-Sánchez, M. Simão, P. A. López-Jiménez, and H. M. Ramos, "CFD Analyses and experiments in a pat modeling: pressure variation and system efficiency," *Fluids*, vol. 2, no. 4, Dec. 2017, doi: 10.3390/fluids2040051.

[37] F. X. Shi, J. H. Yang, and X. H. Wang, "Analysis on the effect of variable guide vane numbers on the performance of pump as turbine,"

Adv. Mech. Eng., vol. 10, no. 6, pp. 1–9, 2018, doi: 10.1177/1687814018780796.

[38] B. Lal and T. S. Deshmukh, "Performance Analysis of Centrifugal Pump at Different Operating Mode," vol. 4, no. 11, 2018, doi: 10.24113/ijoscience.v5i7.170.

[39] H. Carravetta, A., Fecarotta, O., & Ramos, "Numerical simulation on Pump As Turbine: mesh reliability and performance concerns.," in *2011 International Conference on Clean Electrical Power (ICCEP)*, 2011, pp. 169–174.

[40] M. Simão, M. Pérez-Sánchez, A. Carravetta, and H. M. Ramos, "Flow conditions for PATS operating in parallel: Experimental and numerical analyses," *Energies*, vol. 12, no. 5, 2019, doi: 10.3390/en12050901.

[41] T. Capurso *et al.*, "How to Improve the Performance Prediction of a Pump as Turbine by Considering the Slip Phenomenon," *Proceedings*, vol. 2, no. 11, p. 683, Jul. 2018, doi: 10.3390/proceedings2110683.

[42] T. Capurso *et al.*, "Slip factor correction in 1-D Performance prediction model for paTs," *Water (Switzerland)*, vol. 11, no. 3, Mar. 2019, doi: 10.3390/w11030565.

[43] E. Koswara, H. Budiman, and N. Fikri, "Flow Analysis in Pump As Turbines (PATs)Using Ansys Fluent Software," *Sintek J.*, vol. 14, no. 1, pp. 1–13, 2020.

[44] S. Fengxia, Y. Junhu, M. Senchun, and W. Xiaohui, "Investigation on the power loss and radial force characteristics of pump as turbine under gas–liquid two-phase condition," *Adv. Mech. Eng.*, vol. 11, no. 4, pp. 1–10, 2019, doi: 10.1177/1687814019843732.

[45] A. S. Aidhen, S. Malik, and C. D. Kishanrao, "Theoretical, numerical and experimental research of single stage, radial discharge centrifugal pump operating in turbine mode," *Int. J. Innov. Technol. Explor. Eng.*, vol. 8, no. 12, pp. 1265–1270, 2019, doi: 10.35940/ijitee.L3910.1081219.

[46] A. Maleki, M. M. Ghorani, M. H. S. Haghighi, and A. Riasi, "Numerical study on the effect of viscosity on a multistage pump running in reverse mode," *Renew. Energy*, vol. 150, pp. 234–254, 2020, doi: 10.1016/j.renene.2019.12.113.

[47] S. Miao, J. Yang, F. Shi, X. Wang, and G. Shi, "Research on energy conversion characteristic of pump as turbine," *Adv. Mech. Eng.*, vol. 10, no. 4, pp. 1–10, 2018, doi: 10.1177/1687814018770836.

[48] H. X. Shi, L. P. Chai, X. Z. Su, and R. Jaini, "Performance optimization of energy recovery device based on pat with guide vane," *Int. J. Simul. Model.*, vol. 17, no. 3, pp. 472–484, 2018, doi: 10.2507/IJSIMM17(3)443.

[49] H. M. P. Rosa and B. S. Emerick, "Revista Brasileira de Engenharia Agrícola e Ambiental CFD simulation on centrifugal pump impeller with splitter blades Simulação CFD em rotor de bomba centrifuga com pás intermediárias," pp. 3–7, 2020.

[50] M. Liu, L. Tan, and S. Cao, "Theoretical model of energy performance prediction and BEP determination for centrifugal pump as turbine," *Energy*, vol. 172, pp. 712–732, 2019, doi: 10.1016/j.energy.2019.01.162.

[51] J. Du, H. Yang, Z. Shen, and J. Chen, "Micro hydro power generation from water supply system in high rise buildings using pump as turbines," *Energy*, vol. 137, pp. 431–440, 2017, doi: 10.1016/j.energy.2017.03.023.

[52] X. Wang, X., Yang, J., Xia, Z., Hao, Y., & Cheng, "Effect of Velocity Slip on Head Prediction for Centrifugal Pumps as Turbines," *Math. Probl. Eng.*, doi: doi:10.1155/2019/5431047 (https://doi.org/10.1155/2019/5431047).

[53] M. Rossi, A. Nigro, and M. Renzi, "Experimental and numerical assessment of a methodology for performance prediction of Pumps-as-Turbines (PaTs)operating in off-design conditions," *Appl. Energy*, vol. 248, no. April, pp. 555–566, 2019, doi: 10.1016/j.apenergy.2019.04.123.