VOL. 6 Nº2- 2024

UTN IBARRA - ECUADOR

Facultad de Ingeniería en Ciencias Aplicadas
Innovation & Development in Engineering and Applied Science

IDEAS

Advanced Optimization and Numerical Simulation of a Pump as a Turbine System

http://doi.org/10.53358/ideas.v6i2.1009

Ignacio Benavides[1], Paúl Hernández[1], Víctor Hidalgo[2], Carlos Mafla[1], Jorge Melo[1]
1: Facultad de Ingeniería en Ciencias Aplicadas, Universidad Técnica del Norte, Ibarra, Ecuador
2: Facultad de Ingeniería Mecánica, Escuela Politécnica Nacional, Quito, Ecuador
ibbenavides@utn.edu.ec, ephernandez@utn.edu.ec, victor.hidalgo@epn.edu.ec, cnmafla@utn.edu.ec, jlmelo@utn.edu.ec

Fecha de envío, febrero 7/2024 - Fecha de aceptación, marzo 9/2024 - Fecha de publicación, julio 15/2024

Abstract: This work aims to enhance the parametric analysis and numerical simulation of a centrifugal pump used in turbine applications. The primary objective is to optimize the pump's performance when used as a turbine, focusing on various design parameters. Key characteristics of the pump, such as power rating, head, and flow rate, are thoroughly examined. The specific type of turbine application and the critical design parameters considered for optimization are also highlighted. Advanced numerical simulations, including Computational Fluid Dynamics (CFD) and Multiphysics simulations, have been employed to provide a comprehensive view of the efficiency and operational characteristics of the pump-turbine system. The study adopts improved methodologies for a more extensive parametric analysis, thus enhancing the parametric analysis and numerical simulation of the pump in turbine applications. This provides valuable insights into the system's efficiency and operational characteristics. By incorporating advanced methodologies, the study enables a deeper understanding and potential performance improvements in turbomachinery.

Keywords: Simulation, efficiency, pump, turbine.

Resumen: Este trabajo tiene como objetivo mejorar el análisis paramétrico y la simulación numérica de una bomba centrífuga utilizada en aplicaciones de turbinas. El objetivo principal es optimizar el rendimiento de la bomba cuando se utiliza como turbina, centrándose en varios parámetros de diseño. Las características clave de la bomba, como la potencia nominal, la cabeza y el caudal, se examinan minuciosamente y se destacan el tipo específico de aplicación de la turbina y los parámetros de diseño críticos considerados para la optimización. Se han empleado simulaciones numéricas avanzadas, incluidas las simulaciones de Dinámica de Fluidos Computacional (CFD) y las simulaciones multifísicas, para proporcionar una visión integral de la eficiencia y las características operativas del sistema bomba-turbina. El estudio adopta metodologías mejoradas para un análisis paramétrico más extenso, de este modo mejora el análisis paramétrico y la simulación numérica de la bomba en aplicaciones de turbina, lo que aporta información valiosa sobre la eficiencia y las características operativas del sistema. Al incorporar metodologías avanzadas, el estudio permite una comprensión más profunda y mejoras potenciales en el rendimiento de la turbomaquinaria.

Palabras clave: Simulación, eficiencia, bomba, turbina.

Autor de correspondencia:
Ignacio Benavides, ibbenavides@utn.edu.ec

Introduction

In 2022, a staggering 20 million people still lacked access to electricity, as reported by the "International Electrification Agency" [1]. Addressing this disparity aligns with the central goal of the "United Nations 2030 Agenda," which emphasizes universal access to electricity through the adoption of clean and cost-effective technologies [2]. The increasing use of renewable sources, particularly hydropower, has gained importance due to its economic viability, minimal environmental footprint, and increased efficiency.

While large-scale hydro plants pose potential hydrological challenges and environmental impacts, micro and pico-hydro plants emerge as promising solutions for decentralized community electrification with negligible environmental consequences [3] [4] [5]. The conventional design process of micro and peak hydropower plants involves reducing the size of turbines and electromechanical components, which inevitably raises project costs. Pump-as-turbine (PAT) systems present an alternative in such scenarios, taking advantage of the abundance of affordable centrifugal pumps, high component reliability, and reduced maintenance costs [6] [7]. The main obstacle in the implementation of PAT is the absence of characteristic curves and the optimal point of efficiency when operating in turbine mode.

Numerous authors have proposed theoretical and experimental models for the prediction of PAT performance. Williams [8], in a comparative study of eight prediction models, identified Sharma's model [9] as the most accurate. Fontanella et al. [7] developed a highly reliable prediction model based on experimental results from 32 pumps operating as turbines under various conditions. Singh et al. [10] presented an optimized PAT routine, derived from the research of Derakshan and Nourbakhsh [11], achieving a maximum error of 2.6% at low specific speeds, but finding higher errors above 10% for higher specific speeds.

Computational fluid dynamics (CFD), an instrumental computer-based approach to the analysis of thermal physics, has proven invaluable in turbomachinery research and design [12]. Several authors [13],[14] have successfully simulated axial, mixed, and radial pumps that function as turbines, aligning well with experimental findings. Ferracotta et al. [15] concluded in their research that CFD simulations of PAT under specific operating conditions exhibit high accuracy in replicating fluid behavior, proving to be a cost-effective alternative to experimentation. However, each simulation requires validation through mathematical models or experimental data to ensure that the chosen parameters (turbulence models, boundary, and initial conditions, etc.) faithfully reproduce the behavior of the device.

This work aims to pioneer a novel methodology to characterize a commercial pump when operating as a turbine. Leveraging reverse engineering techniques, parameterization tools, and CFD simulations, the studio employs open-source tools such as FreeCAD and OpenFOAM. The aim is to develop a comprehensive understanding of pump performance under turbine conditions, contributing to the wider field of sustainable energy solutions.

Materials and methods

Pump and operating conditions

For the present study, a commercial centrifugal pump "SAER NCB-65/200N-A" has been purchased, in which the main operating conditions of the pump and the characteristics provided by the manufacturer are detailed in Table 1.

Table 1. Pump operating conditions given by the manufacturer

Pump Data Sheet			
Model	BCN -65/200N-A		
Dimensions	80/65/200		
Speed [rpm]	1800		
Flow	Nominal (Qd)	**[l/min]**	1050
	Maximum (Qmax)	**[l/min]**	1870
	Minimum (Qmin)	**[l/min]**	583
Preponderance	Nominal	**[m]**	21.8
	Maximum	**[m]**	23.9
	Minimal	**[m]**	14.1
Preponderance H(Q=0)		**[m]**	24.5
NPSH 3%		**[m]**	4.71
Maximum Working Pressure		**[bar]**	2.49
Power transmitted to the shaft		**[HP]**	6.37
Efficiency		**%**	80.8
Maximum power consumed		**[HP]**	7.4421

Initial conditions

Barbarelli et al. [16] introduced an approach to determine the initial operating parameters of a pump repurposed as a turbine. This method is based on empirical correlations derived from pump operating specifications, in particular the specific speed of turbomachinery in pump mode. Calculated as the flow, the specific velocity (η_{sp}) is determined by equation (1).

$$\eta_{sp} = \eta_P \cdot \frac{\sqrt{Q_P}}{H_P^{3/4}} \quad (1)$$

This specific velocity value is then used in equations (2) and (3) to determine the rotational speed of the impeller and establish the specific velocity of the pump operating as a turbine:

$$\eta_{sp} = 0.9867 \cdot \eta_{st} + 5.2818 \quad (2)$$

$$\eta_{st} = \eta_t \cdot \frac{\sqrt{Q_T}}{H_T^{3/4}} \quad (3)$$

The flow rate (Q_T) and head (H_T) of the pump operating as a turbine are determined through the c_H and c_Q conversion factors, calculated by equations (4) and (5).

$$c_H = -0.00003\eta_{sp}^3 + 0.00440\eta_{sp}^2 - 0.20882\eta_{sp} + 4.64293 \quad (4)$$

$$c_Q = 0.00029\eta_{sp}^2 - 0.02771\eta_{sp} + 2.01648 \quad (5)$$

The H_T and Q_T expressions are obtained by multiplying the conversion factors by the original values of the pump as shown in equations (6) and (7).

$$H_T = c_H \cdot H_P \quad (6)$$

$$Q_T = c_Q \cdot Q_P \quad (7)$$

Table 2 presents the calculated initial operating conditions for the SAER NCB-65/200N-A centrifugal pump. These values are derived from the equations, which provides a basis for understanding the pump's performance when operating as a turbine.

Table 2. PAT Operating Conditions

Variable	Value	Units
H_T	38.368	$[m]$
Q_T	$[m^3/s]$	$[m^3/s]$
η_{st}	18.19	$[rpm]$
c_H	1.76	-------------
c_Q	1.52	-------------
η_T	1 819	$[rpm]$

Where:

- η_{st} = Specific rotation speed of the pump operating as a turbine
- c_H = Turbine mode head conversion factor.
- c_Q = A flow conversion factor that works like a turbine.
- η_T = Rotation speed of the pump in turbine mode.
- H_T = Pump head that functions as a turbine.

Q_T = Pump operating flow rate in turbine mode.

Geometry extraction

To facilitate an accurate numerical simulation of the pump running like a turbine, advanced reverse engineering techniques, specifically 3D scanning, have been employed for geometry extraction. Considering the intricate nature of the pump design (Fig. 1), the silicone molds of the inner scroll and blades were meticulously crafted. The application of silicon to the surfaces is carried out uniformly to prevent the formation of air cavities, ensuring an accurate reproduction of the intricate geometry.

The HANDYSCAN3D-370 3D scanner was instrumental in capturing a point cloud of the molds, and the acquired data underwent meticulous post-processing to mitigate the noise and reflection points inherent in the 3D scanning process. This meticulous approach to geometry extraction lays the foundation for a reliable and detailed numerical simulation of the pump's operation as a turbine.

Computer-aided design (CAD)

After post-processing of the geometry extraction, the surfaces presented a suboptimal surface quality, attributed to the characteristics of the mold and the 3D scanning technique. To rectify this, the open-source parametric 3D modeler FreeCAD was employed to create a high-quality, cohesive pump model. The volute-casing dimensions provided by the manufacturer were integrated with those obtained from the scanned data, resulting in a realistic and accurate representation of the pump.

Blade characteristics, including discharge angle (β_2), inlet angle (β_1), height (b), and blade thickness (t), extracted from the scanned model, were critical to generating a complete 3D model of the impeller, as illustrated in Fig. 1. This meticulous CAD process ensures a reliable representation of the pump's complexities, laying the foundation for a robust numerical simulation of the turbine's operation.

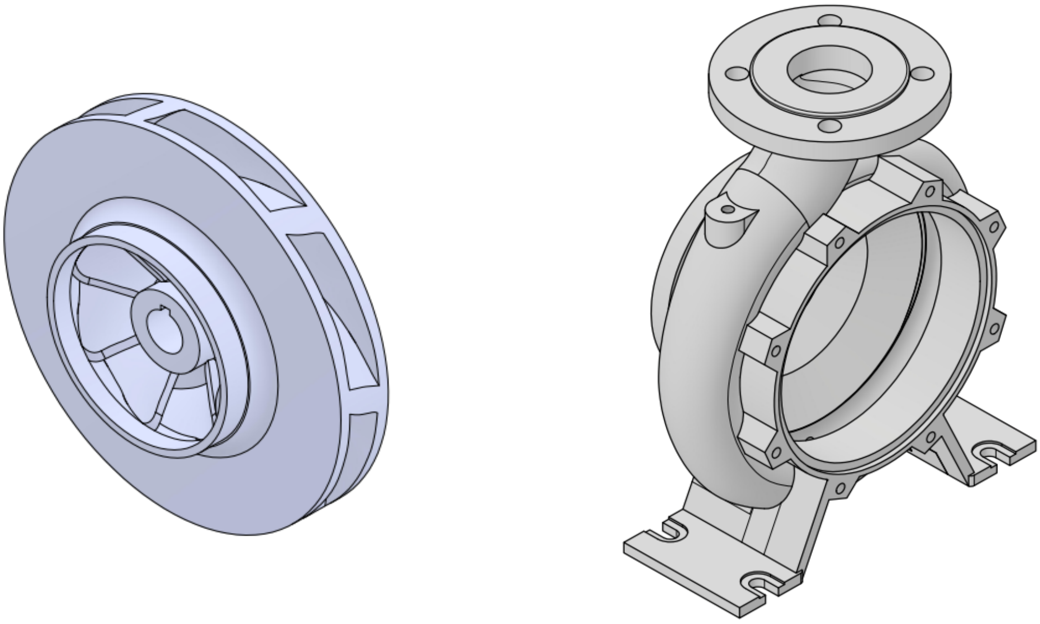

Fig. 1 Pump impeller obtained by reverse engineering technology.

Numerical simulation

The numerical simulations were carried out using OpenFOAM 7, a robust, open-source, open-source computational fluid dynamics (CFD) software available for free. The simulation process has been systematically divided into three stages: pre-processing, solver execution, and post-processing.

In the context of numerical simulations, it is crucial to thoroughly discuss the validation of the model utilized. This validation process serves to confirm the accuracy and reliability of the simulation results. Validation can be achieved through various means, including comparison with experimental data from previous studies, conducting new experiments specifically for validation purposes, or comparing results with similar simulations conducted by other researchers. By ensuring that the numerical model accurately represents the physical phenomenon under investigation, researchers can have confidence in the credibility of the simulation results and their applicability to real-world scenarios.

The computational domains for both volute and impeller were derived from the 3D CAD model, encompassing only the volume occupied by the working fluid during pump-as-turbine (PAT) operation. Given the complexities of the domain, the OpenFOAM cfMesh library was used to generate an unstructured hybrid mesh, incorporating hexahedral and tetrahedral elements. Figure 2 illustrates the application of mesh refinements near the blades and walls, which improves the accuracy of the solution. A meticulous mesh independence study was carried out to determine the optimal number of mesh elements, optimizing the computational resources required for the simulation process. This careful approach ensures the reliability and efficiency of the numerical simulation of the pump operating like a turbine.

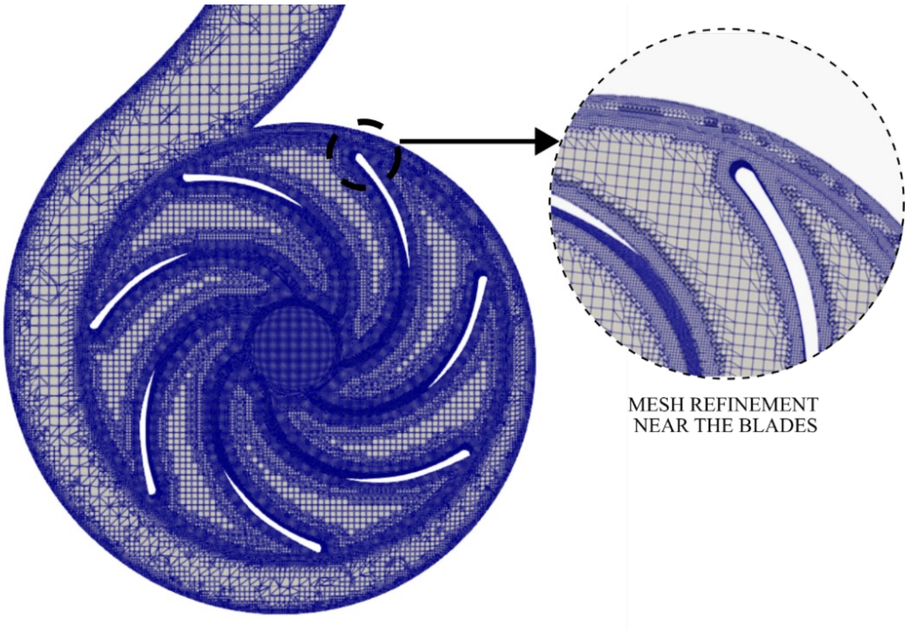

Fig. 2 Refining of the pump impeller blade wall made of cfMesh.

The process of refining the blade wall was carried out using cfMesh, improving the quality of the mesh near the impeller blades to improve the accuracy of the solution.

Computational Proficiency

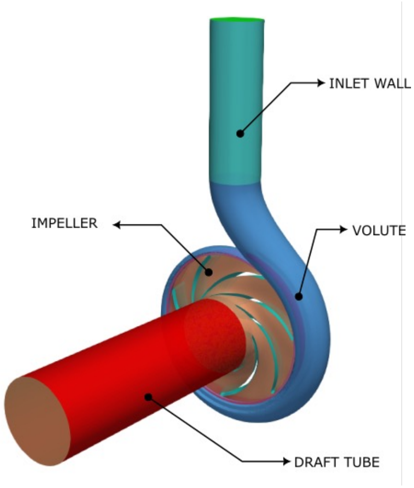

Fig. 3 Computational domain

The computational domain (Figure 3), established for the SAER NCB-65/200N-A centrifugal pump comprised four distinct components: the volute, the impeller, the draft tube and the inlet. To ensure a fully developed flow at the inlet and improve the accuracy of the simulation, the inlet tube was enlarged, following the recommendations of [17], [18], [19].

To faithfully replicate the phenomena, it is crucial to select the right turbulence solvers and models. For this study, the steady-state solver for incompressible flow, "simpleFoam", and the k-ω SST turbulence model were chosen. These selections were made due to their proven high accuracy in capturing flow dynamics near the impeller blades. In addition, a multiple frame of reference (MRF) approach was employed to simulate the rotational motion of the impeller within the volute. This was combined with AMI's cyclic boundary conditions, making it easier to exchange data between the two interconnected domains.

Processing

To determine the suitability of the SAER NCB-65/200N-A centrifugal pump for turbine operation, it is essential to determine its efficiency in turbine mode. For most turbomachinery, efficiency can be calculated as the ratio of the power generated to the power available. In the case of rotating turbomachinery such as PATs, the power generated can be calculated by multiplying the simulated torque by the rotational speed [19]:

$$\eta = \frac{T_T \cdot \eta_T}{\rho \cdot g \cdot H_T \cdot Q_T} \quad (8)$$

Where:

T_T = Simulated pair.

η = Efficiency of the pump working as a turbine.

ρ = Fluid density.

g = Acceleration by gravity.

H_T = Operational Head of the PAT.

Q_T = Operation flow of the PAT.

This efficiency calculation is critical to evaluating the pump's performance as a turbine and its energy conversion potential.

A comprehensive study of mesh independence was carried out, using five different mesh configurations, as summarized in the Table 3. In particular, the "very fine" mesh, comprising 4,969,665 elements, yielded a simulated torque of 41.73 [Nm], showing a minimum error of 0.5% compared to the theoretical torque. The convergence behavior of this mesh, as shown in Figure 5, shows residual values that fall below 10E-04 after 1500 iterations, ensuring accurate replication of the phenomenon (Figure 4). In addition, the maximum y+ value observed in this CFD analysis stands at 275, aligning well within the recommended range for the k-ω SST turbulence model (Table 4).

Table 3. Mesh Independence Study

	Number of Items	Torque [Nm]	%error
Thick mesh	1.317.699	39,77	5%
Medium Mesh	2.020.451	39,24	6%
Fine mesh	3.164.361	39,52	6%
Very fine mesh	4.969.665	41,73	0,5%

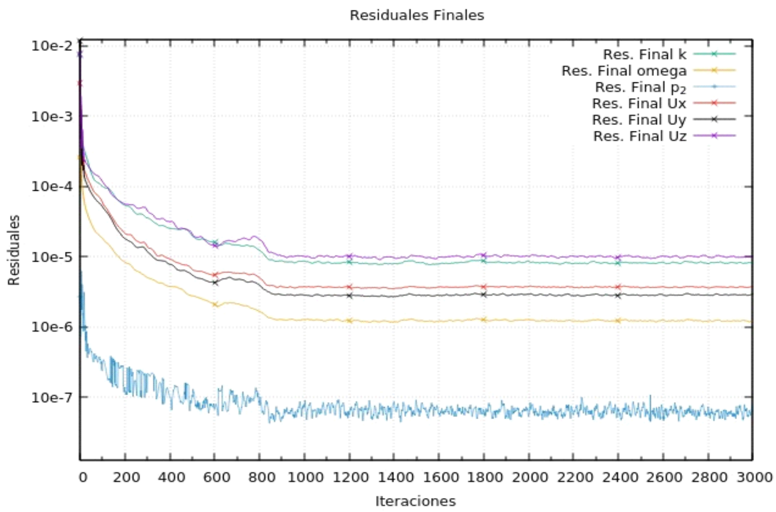

Fig. 4 Final residuals of the simulation

Table 4. Convergence mesh behavior

Zone	y^+_{max}
Impeller	275
Volute	150
Entrance wall	150
Outlet wall	100

An efficiency of 79.53% was achieved for the centrifugal pump "SAER NCB-65/200N-A" operating as a turbine, under the operational parameters outlined in Table 2. In particular, the Best Efficiency Point (BEP) specified by the manufacturer for the pump mode stands at 80.08%. This result corroborates the findings of the study by Nautiyal et al. [20], which postulates that the efficiency of a pump operating as a turbine cannot exceed its efficiency in pump mode. It is essential to bear in mind that the efficiency of LAPs does not depend solely on operating conditions; Rather, impeller geometry also plays a critical role in determining performance [21].

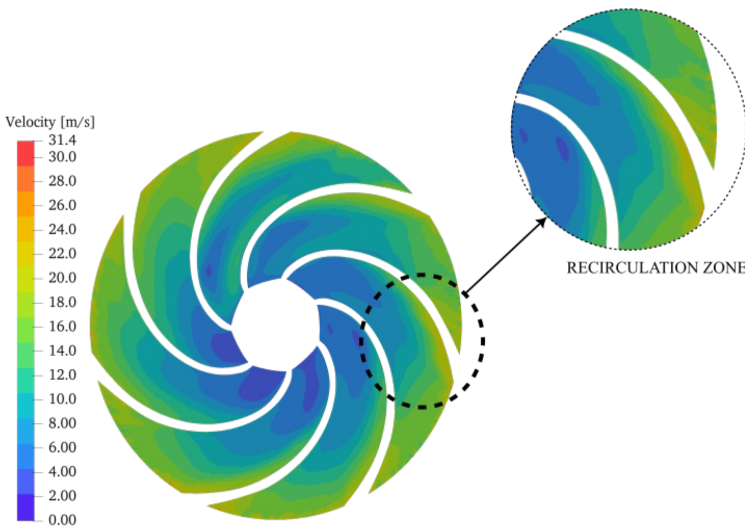

Fig. 5 Velocity contours on the impeller.

Figure 5 and 6 illustrates, respectively, the velocity and pressure contours in the impeller. These observations reveal a non-uniform distribution of velocity and eddy formation within the flow. This phenomenon is attributed to Coriolis acceleration in the impeller, which induces secondary flows and discrepancies in velocities across the blade surface. The maximum flow velocity is concentrated in the regions near the trailing edge, although less than the inlet velocity, due to the interaction between the flow and the impeller. This observation underscores the intricate fluid dynamics at play within the pump-turbine system, shedding light on the complex phenomena that influence its performance.

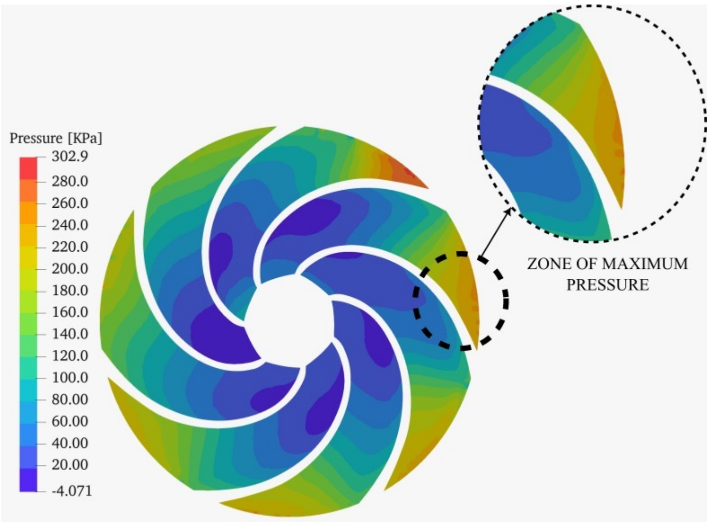

Fig. 6 Pressure contours

Results and Discussion

The discussion encompassed various aspects of the parametric analysis and numerical simulation of a centrifugal pump employed as a turbine. Notably, the utilization of reverse engineering techniques facilitated the accurate virtualization of the pump's geometry, essential for subsequent numerical simulations. This enabled a detailed examination of the pump's performance when operating as a turbine, considering specific design parameters such as impeller geometry and blade profile.

Advanced numerical simulations, conducted using OpenFOAM and incorporating Multiphysics analyses, provided valuable insights into the efficiency and operational characteristics of the pump-turbine system. Mesh independence studies revealed that a mesh comprising 4,969,665 elements offered a balance between computational efficiency and solution accuracy, exhibiting minimal error compared to theoretical results.

The simulations yielded an efficiency of 79.53% for the pump in turbine mode, indicating its suitability for turbine applications without necessitating geometric modifications. Additionally, the observation of non-uniform velocity distribution and pressure contours, along with the identification of high cavitation susceptibility zones, underscored the complexity of the fluid dynamics within the pump-turbine system.

Furthermore, discussions emphasized the importance of validating the numerical model, either through comparison with previous experiments or simulations, to ensure the reliability of the findings. This validation process enhances the credibility of the simulation results and their applicability to real-world scenarios.

Overall, the results obtained from the parametric analysis and numerical simulations provide valuable insights into the performance and operational characteristics of the pump-turbine system. These findings lay the groundwork for further research aimed at optimizing pump-as-turbine efficiency and exploring potential geometric optimizations to enhance performance in turbine applications.

Based on the information obtained in the research process, the results from the parametric analysis and numerical simulations of the pump operating as a turbine offer valuable insights into its performance and operational characteristics. The achievement of an efficiency of 79.53% in turbine mode, albeit slightly lower than the BEP efficiency in pump mode, underscores the pump's viability for turbine applications without requiring geometric modifications. Furthermore, the observation of non-uniform velocity distribution and pressure contours, as well as the identification of high cavitation susceptibility zones, highlights the intricate fluid dynamics and challenges associated with optimizing the pump's performance. The validation of the numerical model, through comparison with previous experiments or simulations, enhances the credibility of the findings and underscores the reliability of the simulation approach. These results pave the way for further exploration, including the potential effects of blade trimming and other geometric optimizations on pump-as-turbine efficiency, offering promising avenues for future research and development in this field.

Conclusions

The use of reverse engineering techniques facilitated the precise virtualization of the geometry of the SAER NCB-65/200N-A centrifugal pump. Leveraging reverse engineering tools along with computational fluid dynamics significantly improved the accuracy of the simulation.

The CFD-simulated flow behavior of the SAER NCB-65/200N-A centrifugal pump operating as a turbine aligns closely with the findings of analogous pump-as-turbine (PAT) studies. The observed pressure distribution and velocity contours closely resemble the operating principles expected of such systems.

The mesh independence analysis, using five different Cartesian meshes, determined that a mesh composed of 4,969,665 elements sufficiently replicates the phenomenon, exhibiting a negligible error of 0.5% compared to the theoretical results.

The efficiency achieved through numerical simulation in turbine mode stands at 79.53%, slightly lower than the efficiency at the Best Efficiency Point (BEP) when operated as a pump. This suggests that the SAER NCB-65/200N-A centrifugal pump can effectively operate as a turbine without the need for geometric alterations.

Further research is warranted to explore the effects of blade trimming and other geometric impeller optimizations on PAT efficiency. These studies hold promise for improving the performance and applicability of pump-turbine systems in various engineering applications.

References

1. Cozzi, L.; "IEA," 03 November 2022. [Online]. Available: https://www.iea.org/commentaries/for-the-first-time-in-decades-the-number-of-people-without-access-to-electricity-is-set-to-increase-in-2022.

2. "Universal Access to Electrification." https://unite.un.org/sites/unite.un.org/files/app-desa-electrification/index.html (accessed March 08, 2023).

3. Frey, G. W. and Linke, D. M. "Hydropower as a Renewable and Sustainable Energy Resource Responding to Global Energy Challenges in a Reasonable Manner", 2002.

4. Moran, E. F.; Lopez, M. C.; Moore, N.; Müller, N. and Hyndman, D. W. "Sustainable hydropower in the 21st century", Proc Natl Acad Sci U S A, vol. 115, no. 47, pp. 11891–11898, noviembre de 2018, doi: 10.1073/pnas.1809426115.

5. "Global Scale Environmental Effects of Hydrological Alterations: Introduction", 2000.

6. Carravetta, A.; Derakhshan, S.; Helena, H. and Ramos, M. "Springer Tracts in Mechanical Engineering Pumps as Turbines Fundamentals and Applications." [Online]. Available: http://www.springer.com/series/11693

7. Fontanella, S.; Fecarotta, O.; Molino, B.; Cozzolino, L. and Della Morte, R. "A Performance Prediction Model for Pumps as Turbines (PAT)", Water (Switzerland), Vol. 12, No. 4, April 2020, doi: 10.3390/W12041175.

8. Williams, A. A. "The Turbine Performance of Centrifugal Pumps: A Comparison of Prediction Methods," Proceedings of the Institution of Mechanical Engineers, Part A: Journal of Power and Energy, vol. 208, no. 1. pp. 59-66, 1994. doi: 10.1243/PIME_PROC_1994_208_009_02.

9. Sharma and K. R., "Small Hydropower Projects - Using Centrifugal Pumps as Turbines," Kirloskur Electric Co, 1985.

10. Singh, P. and Nestmann, F. "An Optimization Routine in a Prediction and Selection Model for Centrifugal Pump Turbine Operation," Exp Therm Fluid Sci, 2010.

11. Derakhshan, S. and Nourbakhsh, A. "Experimental Study of Characteristic Curves of Centrifugal Pumps Operating as Turbines at Different Specific Speeds," ElsevierJournal of Experimental Thermal and Fluid Science, 2008.

12. Plua, F.; Hidalgo, V.; Cando, E.; Pérez-Sánchez, M. and Amparo López-Jiménez, P. "Pumps as Turbines (PATs) by Analysis with CFD Models", vol. 12, núm. 3, 2022.

13. Rawal, S.; and Kshirsagar, J.T. "Numerical simulation in a pump operating in turbine mode," in Proceedings of the twenty-third international symposium of pump users, 2007.

14. Nautiyal, H.; Kumar, A. and Yadav, S. "Experimental Research of the Centrifugal Pump Operating as a Turbine for Small Hydropower Systems," ENERGY SCIENCE AND TECHNOLOGY, vol. 1, no. 1, pp. 79-86, 2011, [online]. Available: www.cscanada.net

15. Carravetta, A.; Fecarotta, O. and Ramos, H. "Numerical Simulation in Pump as Turbine: Mesh Reliability and Performance Issues", 2011.

16. Barbarelli, S.; Amelio, M.; Florio G. and Scornaienchi, N. M. "Selection Procedure for Pumps Operating as Turbines in Micro Hydropower Plants," in Energy Procedia, September 2017, vol. 126, pp. 549-556. doi: 10.1016/j.egypro.2017.08.282.

17. Tian, P.; Huang, J.; Shi, W. and Zhou, L. "Optimization of a Centrifugal Pump Used as a Turbine Impeller Using an Orthogonal Test Approach," Fluid Dynamics and Materials Processing, vol. 15, no. 2, pp. 139-151, 2019, doi: 10.32604/fdmp.2019.05216.

18. Jain, S. V. and Patel, R. N. "Investigations into Turbine Mode Pump Operation: A State-of-the-Art Review," Renewable and Sustainable Energy Reviews, vol. 30. Elsevier Ltd, pp. 841-868, 2014. doi: 10.1016/j.rser.2013.11.030.

19. Yang, H.; Zhu, L.; Xue, H.; Duan, J. and Deng, F. "A Numerical Analysis of the Effect of Impeller Rounding on the Centrifugal Pump as a Turbine," Processes, Vol. 9, No. 9, September 2021, doi: 10.3390/pr9091673.

20. Nautiyal, H.; Varun, G. and Kumar, A. "Analytical, Experimental, and Computational Study of Reverse Operation Pumps: A Review," Renewable and Sustainable Energy Reviews, vol. 14, no. 7, pp. 2059-2067, 2010, doi: 10.1016/j.rser.2010.04.006.

21. Plua, F.; Hidalgo, V.; E. Cando, M. Pérez-Sánchez and Amparo López-Jiménez, P. "Pumps as Turbines (PATs) by Analysis with CFD Models", vol. 12, núm. 3, 2022.

22. Zhang, J.; Appiah, D.; Zhang, F.; Yuan, S.; Gu, Y. and Asomani, S. N. "Experimental and Numerical Investigations on Pressure Pulsation in a Pump-Mode Operation of a Pump-Like Turbine," Energy Sci Eng, vol. 7, no. 4, pp. 1264-1279, 2019, doi: 10.1002/ese3.344.