International Journal of Research Publication and Reviews

Journal homepage: www.ijrpr.com ISSN 2582-7421

CENTRIFUGAL PUMP EFFICIENCY USING CFD TOOLS: A REVIEW

Vivekanand Singh [1], *Prof. Vikas Bakshi* [2]

[1] PG scholar, Department of Mechanical Engineering, SAGE University, Indore
[2] Asst. Professors, Department of Mechanical Engineering, SAGE University, Indore

ABSTRACT :

The efficiency of centrifugal pumps plays a crucial role in minimizing energy consumption and improving overall system performance in industrial and domestic fluid handling applications. This review paper presents a comprehensive investigation of design parameters influencing the efficiency of centrifugal pumps using Computational Fluid Dynamics (CFD) tools. Key factors such as impeller geometry, blade number, blade angle, volute design, clearance gap, and flow characteristics have been critically analyzed from existing literature. The application of CFD techniques enables detailed visualization of internal flow behavior, pressure distribution, and velocity patterns, offering deeper insights into hydraulic losses and performance optimization. Various studies employing software such as ANSYS CFX, FLUENT, and OpenFOAM have been reviewed to highlight advancements in numerical modeling, mesh refinement strategies, and turbulence models. The paper also discusses the challenges in correlating numerical results with experimental data and proposes potential directions for future research focused on improving pump design, reducing cavitation, and enhancing energy efficiency. Overall, the review emphasizes the significance of CFD as a powerful design and diagnostic tool for optimizing centrifugal pump performance.

Keywords- ANSYS CFX, Pump Efficiency, Impeller Design, Flow Analysis, Performance Optimization, Hydraulic Losses, Cavitation

1. INTRODUCTION

Centrifugal pumps are among the most widely used fluid machines in industrial, agricultural, and domestic applications due to their simple construction, continuous flow capability, and ease of operation. However, their efficiency largely depends on the design and operating parameters such as impeller geometry, blade number, blade outlet angle, volute design, and flow conditions. Even minor variations in these parameters can significantly influence hydraulic performance, energy consumption, and reliability. With the increasing global emphasis on energy conservation and sustainable engineering, improving the efficiency of centrifugal pumps has become a critical area of research. Traditional design methods based on empirical correlations and experimental testing provide limited insight into internal flow phenomena, such as secondary flows, recirculation zones, and cavitation. These complex flow behaviors often lead to hydraulic losses, vibration, and noise, which reduce the overall efficiency of the pump. The emergence of Computational Fluid Dynamics (CFD) has revolutionized the pump design process by enabling detailed analysis of flow behavior within the pump components. CFD simulations allow for precise visualization and quantification of parameters such as velocity distribution, pressure variation, turbulence, and energy losses, offering a deeper understanding of performance characteristics.

In recent years, researchers have utilized advanced CFD tools such as ANSYS CFX, FLUENT, and Open FOAM to analyze and optimize centrifugal pump designs. These tools facilitate virtual prototyping, reducing the time and cost associated with experimental testing. Various studies have focused on optimizing impeller blade angles, modifying volute geometries, and improving surface finishes minimizing losses and enhancing efficiency. Despite these advancements, challenges remain in accurately predicting cavitation behavior and correlating numerical predictions with experimental outcomes. This review paper aims to compile and analyze recent research efforts on centrifugal pump design optimization using CFD. The objective is to identify critical design parameters influencing pump efficiency and to assess how CFD-based methodologies have contributed to understanding and improving pump performance. The review further highlights the limitations of current modeling approaches and outlines future research directions for developing more efficient and reliable centrifugal pump systems.

2. PROBLEM IDENTIFICATION

Centrifugal pumps are extensively used in industrial processes, water supply systems, power plants, and irrigation due to their reliability and versatility. However, one of the persistent challenges in pump operation is low overall efficiency caused by complex internal fluid flow phenomena. Despite continuous improvements in design and manufacturing, a significant portion of energy is still lost due to hydraulic losses, friction, leakage, and cavitation effects. These losses not only reduce efficiency but also increase operational costs and maintenance requirements. Traditional pump design approaches rely heavily on experimental testing and empirical correlations, which often fail to capture the intricate flow behavior inside the impeller and volute. Such limitations lead to suboptimal designs and restricted scope for performance enhancement. Moreover, due to the interaction of multiple design variables such as impeller blade angle, number of blades, volute cross-section, and clearance gaps, it becomes difficult to predict their combined impact on efficiency through conventional methods. Another critical issue is the inadequate understanding of flow turbulence and pressure distribution within the pump, which directly affects energy conversion and cavitation characteristics. Without accurate visualization and quantitative analysis of these parameters, optimizing the pump design remains a challenging task. Therefore, there is a pressing need to adopt Computational Fluid Dynamics (CFD) techniques for a detailed investigation of internal flow characteristics and their influence on efficiency. CFD-based analysis offers a cost-effective, non-intrusive, and reliable method to evaluate and optimize pump design parameters before physical prototyping. This review paper identifies the core problem as the lack of integrated design analysis using CFD tools to understand and optimize the parameters that govern centrifugal pump performance. Addressing this problem will contribute to developing high-efficiency pumps with improved hydraulic performance, reduced energy losses, and enhanced operational life.

3. RESEARCH OBJECTIVES

The primary aim of this review paper is to investigate the influence of various design parameters on the efficiency and performance of centrifugal pumps using Computational Fluid Dynamics (CFD) tools. To achieve this aim, the following specific objectives have been identified:

1. To review and analyze the key design parameters such as impeller geometry, blade number, blade angle, volute design, and clearance gap that affect centrifugal pump efficiency.
2. To study the application of CFD-based tools like ANSYS CFX, FLUENT, and OpenFOAM in evaluating pump performance and internal flow characteristics.
3. To identify the relationship between geometric modifications and variations in hydraulic losses, pressure distribution, and cavitation behavior.
4. To compare numerical simulation results from existing studies with experimental data to assess the accuracy and reliability of CFD models.
5. To summarize optimization methods adopted by researchers for improving pump efficiency and reducing energy losses.
6. To highlight the limitations, challenges, and future research opportunities in CFD-based centrifugal pump design and analysis.

4. LITRATURE REVIEW

The performance and efficiency of centrifugal pumps have been widely studied by researchers using both experimental and computational approaches. The advancement of Computational Fluid Dynamics (CFD) tools has enabled detailed investigation of internal flow behavior, pressure distribution, and hydraulic losses, which are otherwise difficult to measure experimentally. Despite these advancements, discrepancies between simulated and experimental results remain due to mesh dependency, turbulence modeling limitations, and uncertainties in boundary conditions. Therefore, continuous refinement of CFD models and validation with experimental data are essential for achieving more accurate predictions of centrifugal pump behavior.

Kumar and Sharma (2024) conducted a comprehensive study on the optimization of a single-stage centrifugal pump using Computational Fluid Dynamics (CFD) techniques in ANSYS CFX. The main objective of their research was to improve pump efficiency through detailed analysis of flow characteristics and design parameter variations. They focused on critical parameters such as impeller blade outlet angle, number of blades, and volute geometry, which directly influence head, discharge, and overall efficiency. The CFD simulations provided detailed insights into velocity distribution, pressure variation, and turbulence zones inside the pump casing. By systematically modifying the impeller design and volute dimensions, they achieved a significant improvement in hydraulic performance, with an efficiency increase of approximately 8%. The authors validated their numerical results with experimental test data and observed close agreement, confirming the reliability of the CFD model. They also highlighted that optimization of blade curvature reduces recirculation and hydraulic losses, thereby improving energy conversion. Additionally, the study discussed the influence of mesh quality and turbulence model selection on result accuracy. Kumar and Sharma concluded that CFD-based optimization provides a powerful tool for enhancing centrifugal pump performance while minimizing the need for extensive physical prototyping.

Li and Chen (2023) conducted an advanced numerical investigation combining Response Surface Methodology (RSM) with Computational Fluid Dynamics (CFD) simulations to optimize the impeller geometry of a centrifugal pump. Their primary aim was to identify the most influential geometric parameters affecting head, discharge, and overall efficiency. The study considered key variables such as blade thickness, exit width, and outlet angle, which were systematically varied using design of experiments (DOE) techniques. CFD simulations were performed to evaluate the impact of these parameters on flow patterns, pressure distribution, and energy losses within the pump. The results demonstrated that both blade thickness and exit width play a crucial role in governing flow smoothness and hydraulic performance. Through RSM-based optimization, the authors developed a predictive model correlating geometric changes with efficiency outcomes. The optimized impeller configuration achieved nearly a 6% improvement in energy performance compared to the baseline design. Furthermore, validation with experimental data confirmed the accuracy of the numerical approach. Li and Chen concluded that integrating RSM with CFD offers a powerful, time-efficient method for multi-parameter optimization in centrifugal pump design, reducing dependence on traditional trial-and-error methods.

Patel et al. (2023) carried out an in-depth numerical investigation on the effect of impeller–volute interaction in a centrifugal pump using the k–ω SST turbulence model within ANSYS FLUENT. The study aimed to understand how the geometric alignment between the impeller outlet and the volute tongue influences overall hydraulic performance. Detailed CFD simulations were performed under different flow conditions to analyze pressure distribution, velocity fields, and turbulence intensity across the pump passage. The authors observed that improper alignment leads to flow separation, non-uniform velocity distribution, and increased secondary losses within the volute casing. By optimizing the impeller–volute positioning, smoother flow transitions and reduced energy losses were achieved. The numerical results indicated that a small adjustment in tongue clearance can substantially enhance head and efficiency without altering the impeller geometry. Additionally, Patel et al. validated their CFD results with available experimental data, confirming close correlation and model reliability. They also highlighted that the k–ω SST turbulence model provided superior accuracy in capturing boundary layer effects compared to the standard k–ε model. The study concluded that precise impeller–volute matching is a key factor in minimizing hydraulic losses and improving centrifugal pump efficiency..

Singh and Verma (2022) conducted a detailed investigation on cavitation behavior in centrifugal pumps using transient Computational Fluid Dynamics (CFD) analysis. Their study aimed to identify the onset, development, and effects of cavitation under varying operating conditions. By employing a multiphase flow model and the Rayleigh–Plesset cavitation model, they simulated vapor bubble formation and collapse within the impeller passage. The results revealed that cavitation inception predominantly occurs near the impeller inlet where local pressure drops below the vapor pressure of the fluid. This low-pressure zone leads to the formation of vapor bubbles, which subsequently collapse as they move toward regions of higher pressure, causing noise, vibration, and material erosion. The authors observed that reducing inlet pressure fluctuations and maintaining a higher Net Positive Suction Head (NPSH) can effectively delay cavitation onset. The transient analysis also showed that impeller blade design and inlet geometry significantly affect cavitation intensity. Furthermore, Singh and Verma highlighted that CFD provides a powerful visualization tool for predicting cavitation zones, enabling early detection and design optimization. They concluded that controlling inlet conditions and optimizing impeller geometry can enhance operational stability and extend pump service life.

Zhang et al. (2022) performed a comprehensive three-dimensional numerical simulation to evaluate the impact of impeller blade number on the hydraulic performance of a centrifugal pump. The study was conducted using ANSYS CFX to analyze flow behavior, pressure distribution, and velocity profiles under steady-state conditions. By systematically varying the number of blades while keeping other design parameters constant, the authors assessed how blade count influences head, efficiency, and energy losses. The results indicated that an increase in the number of blades enhances the pump head due to improved energy transfer from the impeller to the fluid. However, this improvement was accompanied by higher frictional and blockage losses caused by reduced flow area and increased turbulence. The study also revealed that excessive blade numbers can lead to flow stagnation and secondary circulation, which adversely affect hydraulic efficiency. Through parametric analysis, Zhang et al. identified an optimal blade count that provides a balanced performance between head and efficiency. The pressure and velocity contour plots clearly demonstrated smoother flow patterns at the optimal configuration. The authors concluded that careful selection of impeller blade number is essential to achieving efficient pump design, minimizing hydraulic losses, and ensuring stable operation across varying flow rates.

Pandey and Patel (2021) conducted a comparative numerical study to evaluate the performance of different turbulence models in predicting internal flow characteristics of a centrifugal pump. Using ANSYS FLUENT, they analyzed three commonly used models—standard k–ε, RNG k–ε, and SST k–ω—to determine their effectiveness in capturing pressure, velocity, and turbulence intensity distributions within the impeller and volute passages. The simulations were carried out under various flow conditions to assess model accuracy and computational efficiency. The results showed that the standard k–ε model tends to overpredict turbulence levels near the impeller outlet, while the RNG k–ε model provides moderate accuracy but struggles in regions of strong adverse pressure gradients. In contrast, the SST k–ω model demonstrated superior performance by accurately resolving near-wall flow behavior and predicting velocity and pressure profiles closer to experimental observations. The authors validated their CFD predictions with available test data, confirming that the SST k–ω model yielded the smallest deviation in head and efficiency values. Pandey and Patel emphasized that proper turbulence model selection is crucial for achieving reliable CFD simulations in centrifugal pump analysis. They concluded that the SST k–ω model is most suitable for capturing complex flow interactions and should be preferred for design optimization and performance evaluation of pump components..

Rao and Joshi (2020) performed an extensive parametric analysis to examine how impeller outlet width and vane thickness influence the hydraulic efficiency and performance characteristics of a centrifugal pump. The study utilized CFD simulations conducted in ANSYS CFX to visualize internal flow patterns and pressure distributions under different design configurations. By systematically varying the outlet width and vane thickness, the authors evaluated their effects on head generation, flow velocity, and turbulence behavior. The results revealed that reducing the impeller outlet width leads to a more uniform velocity distribution and smoother flow through the volute casing. However, this reduction also caused a slight decrease in head due to the limited flow area available for energy transfer. Conversely, increasing the vane thickness enhanced structural strength but introduced additional frictional losses and reduced efficiency. The study highlighted that an optimal combination of outlet width and vane thickness is essential to balance flow smoothness with energy conversion efficiency. The pressure contour plots and velocity vectors confirmed that smaller outlet widths minimize flow recirculation and secondary vortices. Rao and Joshi concluded that careful geometric tuning of impeller parameters can substantially improve hydraulic efficiency while maintaining mechanical stability and operational reliability of centrifugal pumps..

Zhao et al. (2019) carried out a detailed numerical investigation using ANSYS CFX to study the effect of impeller blade outlet angle on the hydraulic efficiency of a centrifugal pump. The main objective of their research was to determine the optimal blade angle that maximizes energy transfer while minimizing flow separation and recirculation losses. The study considered several outlet angles ranging from 20° to 35° and evaluated their influence on head, discharge, and overall efficiency. CFD simulations were performed under steady-state conditions to obtain velocity vectors, pressure contours, and turbulence intensity distributions across the impeller and volute. The results revealed that smaller outlet angles (below 22°) led to flow separation near the blade trailing edge, reducing efficiency, whereas larger angles (above 30°) caused increased frictional losses and secondary vortices. The analysis showed that an outlet angle between 23° and 27° provided the best performance, ensuring uniform flow at the impeller exit and efficient energy conversion. The authors validated their simulation data with experimental measurements, finding close agreement in predicted head and efficiency values. Zhao et al. concluded that optimizing the impeller blade outlet angle within the identified range can significantly improve hydraulic performance, reduce recirculation losses, and enhance overall pump efficiency..

Mehta and Banerjee (2018) conducted a combined computational and experimental study to investigate the influence of volute casing design on the flow uniformity and efficiency of a centrifugal pump. Their research aimed to compare different volute geometries, including circular and logarithmic spiral shapes, to determine their effect on pressure recovery and hydraulic performance. Using CFD simulations in ANSYS FLUENT, they analyzed velocity distributions, pressure contours, and turbulence intensity within the volute for multiple flow rates. The results demonstrated that the conventional circular volute produced uneven pressure distribution and higher flow separation near the tongue region, leading to energy losses and vibration. In contrast, the logarithmic spiral volute casing facilitated a smoother flow path and more uniform velocity distribution around the impeller periphery. This geometric configuration enhanced pressure recovery and minimized secondary flow formation. The CFD findings were validated with experimental tests conducted on a prototype pump, showing close agreement in performance curves. Mehta and Banerjee reported that the logarithmic spiral design improved overall efficiency by approximately 5–7% compared to the traditional design. They concluded that optimizing volute geometry plays a crucial role in enhancing hydraulic efficiency, reducing flow-induced losses, and improving operational stability in centrifugal pumps..

Ali and Rahman (2017) conducted a detailed experimental and numerical analysis to evaluate the performance of a centrifugal pump operating under part-load conditions. The main objective of their study was to identify the causes of reduced efficiency and unstable flow patterns that occur when the pump operates below its design flow rate. Using a combination of laboratory experiments and CFD simulations in ANSYS FLUENT, they examined pressure variations, velocity fields, and turbulence distribution within the impeller and volute regions. The results indicated that part-load operation leads to significant energy losses caused by backflow at the impeller inlet and secondary circulation zones near the volute tongue. These flow instabilities were found to generate pressure pulsations and increased vibration, contributing to mechanical wear and noise. The CFD analysis provided detailed visualization of recirculation regions and velocity reversals that were difficult to capture experimentally. The authors observed that improving impeller inlet geometry and optimizing blade curvature could mitigate backflow effects and stabilize flow behavior. Validation of CFD results with experimental data showed good correlation, confirming the reliability of the numerical model. Ali and Rahman concluded that CFD-based design optimization is essential for minimizing hydraulic losses under part-load conditions and improving the overall efficiency and operational stability of centrifugal pumps..

Rao et al. (2016) carried out a comparative study to evaluate the performance differences between conventional and optimized impeller designs of a centrifugal pump using Computational Fluid Dynamics (CFD) simulations. The primary goal of their research was to analyze how variations in blade curvature and geometric profile affect internal flow patterns, turbulence behavior, and overall hydraulic efficiency. The study employed ANSYS CFX to simulate steady-state flow conditions and assess parameters such as velocity distribution, pressure gradient, and turbulence intensity within the impeller passages. Results from the simulations revealed that the optimized impeller with modified blade curvature produced smoother flow trajectories and reduced regions of flow separation near the blade suction surface. This design improvement significantly minimized turbulence intensity and hydraulic losses compared to the conventional impeller. Furthermore, the optimized model achieved approximately a 10% increase in overall pump efficiency without compromising head generation. The authors validated the CFD predictions through experimental testing, which showed close agreement with the simulated performance curves. Rao et al. concluded that the incorporation of optimized blade curvature not only enhances energy conversion but also improves the operational stability and durability of centrifugal pumps. Their study emphasized the importance of CFD-driven design refinement in achieving high-performance and energy-efficient pump systems..

5. CONCLUSION

The review of recent studies on centrifugal pump performance using CFD tools reveals a significant evolution in design optimization techniques and computational modeling accuracy. Researchers consistently emphasize the importance of impeller geometry, volute design, and flow interaction in achieving higher hydraulic efficiency and stable operation. Advanced turbulence models such as SST k–ω and transient cavitation simulations have enabled better prediction of internal flow behavior and performance characteristics. Studies also demonstrate that optimized parameters like blade outlet angle, number of blades, and volute curvature can collectively enhance efficiency by 6–10%. CFD-based optimization has proven to be a powerful and reliable approach for analyzing complex flow phenomena within pumps, minimizing the need for costly experimental trials. Moreover, coupling CFD with design of experiments (DOE) and response surface methodology (RSM) has facilitated multi-parameter optimization with greater precision. From impeller–volute interactions to cavitation control, computational studies provide deep insights into loss mechanisms and help in formulating effective design strategies. In summary, the literature indicates that continuous advancements in CFD tools and modeling techniques have revolutionized the design process of centrifugal pumps. The integration of numerical simulations with experimental validation ensures improved accuracy, energy efficiency, and reliability in pump design. Future research can focus on multiphase flow modeling, unsteady load analysis, and AI-driven optimization for developing next-generation high-efficiency centrifugal pumps.

REFERENCES

[1] S. Kumar and R. Sharma, "CFD-based optimization of single-stage centrifugal pump using ANSYS CFX," *International Journal of Fluid Machinery and Systems*, vol. 17, no. 2, pp. 145–156, 2024.

[2] Y. Li and Q. Chen, "Optimization of impeller geometry parameters using response surface methodology and CFD simulation," *Energy Conversion and Management*, vol. 295, pp. 118–134, 2023.

[3] D. Patel, M. Shah, and A. Desai, "Numerical analysis of impeller–volute interaction using k–ω SST turbulence model in ANSYS FLUENT," *Engineering Applications of Computational Fluid Mechanics*, vol. 17, no. 3, pp. 425–440, 2023.

[4] R. Singh and V. Verma, "Transient CFD analysis of cavitation inception and suppression in centrifugal pumps," *Journal of Mechanical Science and Technology*, vol. 36, no. 8, pp. 3921–3934, 2022.

[5] H. Zhang, W. Liu, and T. Gao, "Influence of impeller blade number on centrifugal pump performance: A three-dimensional CFD study," *Applied Thermal Engineering*, vol. 213, pp. 118563, 2022.

[6] N. Pandey and H. Patel, "Comparison of turbulence models for internal flow simulation in centrifugal pumps using ANSYS CFX," *International Journal of Applied Mechanics and Engineering*, vol. 26, no. 4, pp. 987–1002, 2021.

[7] K. Rao and S. Joshi, "Parametric study of impeller outlet width and vane thickness on centrifugal pump efficiency," *Journal of Fluids Engineering*, vol. 142, no. 12, pp. 121301, 2020.

[8] L. Zhao, F. Xu, and Y. Wang, "Numerical investigation on impeller blade outlet angle and its effect on pump hydraulic performance," *Alexandria Engineering Journal*, vol. 58, no. 4, pp. 1239–1248, 2019.

[9] R. Mehta and S. Banerjee, "CFD and experimental evaluation of volute casing design on flow uniformity and efficiency," *Mechanical Systems and Signal Processing*, vol. 110, pp. 158–170, 2018.

[10] M. Ali and M. Rahman, "Experimental and numerical study on part-load performance of centrifugal pumps," *Flow Measurement and Instrumentation*, vol. 54, pp. 45–53, 2017.

[11] P. Rao, T. Sharma, and K. Nair, "Comparative CFD analysis of conventional and optimized impeller designs for performance enhancement," *Journal of Computational Design and Engineering*, vol. 3, no. 4, pp. 315–326, 2016.

[12] D. Wang, J. Lee, and B. Kim, "Effect of blade curvature and volute tongue configuration on flow behavior in centrifugal pumps," *Procedia Engineering*, vol. 127, pp. 1121–1130, 2015.

[13] A. Sahu and M. Chattopadhyay, "Numerical prediction of performance and flow instability in centrifugal pumps using CFD," *Journal of Thermal Science and Engineering Applications*, vol. 6, no. 2, pp. 021007, 2014.

[14] G. Patel and R. Gupta, "Design optimization and performance evaluation of centrifugal pump impeller using CFD techniques," *International Journal of Rotating Machinery*, vol. 2013, Article ID 584731, pp. 1–8, 2013.