International Journal of Research Publication and Reviews

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

INVESTIGATION OF DESIGN PARAMETERS INFLUENCING CENTRIFUGAL PUMP EFFICIENCY USING CFD TOOLS

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 :

Centrifugal pumps are among the most widely used mechanical devices for fluid transportation in industrial and commercial applications. Improving their efficiency has been a major focus of research due to increasing energy demands and operational costs. The advent of Computational Fluid Dynamics (CFD) has revolutionized the analysis and design of centrifugal pumps by allowing detailed visualization and optimization of complex internal flow behavior. The present study investigates the effect of the trailing-edge angle of twisted blades in a radial flow impeller on hydraulic performance using three-dimensional steady-state CFD simulations. Three impeller models with varying trailing-edge angles were developed in ANSYS BladeGen while maintaining a constant blade wrap angle. The analysis employed the k–ε turbulence model and the finite volume method to solve the three-dimensional Navier–Stokes equations. The simulated mass flow rate of 77.8 kg/s closely matched analytical results, validating the accuracy of the model. Results indicate that the impeller side exhibits higher velocity compared to the discharge side at lower flow rates. Additionally, the total pressure on the pressure side of the blade is significantly greater than that on the suction side, with the minimum static pressure occurring near the leading edge. The study found that an increase in the blade trailing-edge angle from 16° to 27° resulted in a 4% rise in impeller head, accompanied by a proportional increase in input power. These findings demonstrate that CFD-based design optimization is an effective approach for enhancing the hydraulic performance and energy efficiency of centrifugal pumps.

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

1. INTRODUCTION

Centrifugal pumps play a crucial role in a wide range of industrial and domestic applications such as water supply systems, irrigation, chemical processing, and power generation. Their simplicity, reliability, and ability to handle large volumes of fluid make them one of the most commonly used fluid transport devices. Despite their widespread use, achieving high efficiency and optimal performance remains a major challenge due to the complex internal flow characteristics and multiple design variables involved. Parameters such as impeller blade geometry, volute casing shape, blade outlet angle, and flow passage design have a significant impact on the overall hydraulic performance, head, and energy consumption of the pump. Traditional design methods rely heavily on empirical correlations and experimental testing, which are time-consuming, costly, and often limited in capturing intricate flow phenomena such as turbulence, recirculation, and cavitation. The advancement of Computational Fluid Dynamics (CFD) has provided an effective and reliable alternative, enabling engineers to simulate and analyze three-dimensional flow behavior inside pumps with high accuracy. CFD tools like ANSYS CFX and FLUENT allow detailed examination of pressure distribution, velocity fields, turbulence intensity, and energy losses under various operating conditions. Recent studies have demonstrated that optimizing impeller design parameters such as trailing-edge angle, wrap angle, and blade curvature can significantly enhance hydraulic efficiency and operational stability. The use of advanced turbulence models like k–ε and SST k–ω has improved the predictive capability of CFD analyses, allowing accurate simulation of real-world pump performance. Moreover, CFD-based design approaches reduce the dependence on physical prototyping, minimizing development time and cost. The present study focuses on the investigation of design parameters influencing centrifugal pump efficiency using CFD tools. By examining the effect of impeller trailing-edge angle through numerical simulations, this work aims to provide insights into how subtle geometric variations affect flow characteristics, head generation, and energy performance. The findings contribute to the ongoing research toward developing more efficient, reliable, and cost-effective centrifugal pumps suitable for diverse engineering applications.

2. PROBLEM IDENTIFICATION

Centrifugal pumps are extensively used in industrial and domestic fluid transport systems due to their simple construction, continuous flow capability, and ease of maintenance. However, the efficiency of these pumps is often compromised by improper design parameters, flow separation, turbulence losses, and cavitation phenomena within the impeller and volute casing. The impeller, being the most critical component responsible for energy conversion from mechanical to hydraulic form, plays a vital role in determining the overall performance of the pump. Small variations in impeller geometry such as blade angle, blade thickness, outlet width, and trailing-edge configuration can significantly influence the flow behavior and energy efficiency. In most conventional design practices, empirical methods and prototype testing are used for performance evaluation. These approaches, while reliable, are costly, time-consuming, and limited in predicting complex three-dimensional flow characteristics. With the advancement of Computational Fluid Dynamics (CFD), designers can now analyze internal flow patterns and optimize pump geometry virtually, thereby reducing experimental costs and enhancing design accuracy. The present work mainly focuses on the performance improvement of a centrifugal pump through the design modification of its impeller. Pump impeller models are developed by varying critical design parameters such as the trailing-edge angle while keeping other geometric parameters constant. This enables the evaluation of their influence on hydraulic performance parameters like head, flow rate, and efficiency using ANSYS CFX simulation tools. Based on the above observations, the following problem statement has been formulated. *Design and Development of a Centrifugal Pump with Emphasis on Factors Affecting Efficiency Using ANSYS CFX.*

3. RESEARCH OBJECTIVES

The performance of a centrifugal pump largely depends on its impeller design, which governs the conversion of mechanical energy from the shaft into hydraulic energy of the fluid. The flow inside the impeller is inherently three-dimensional, turbulent, and complex, making its prediction and optimization challenging through analytical or experimental methods alone. To overcome these limitations, Computational Fluid Dynamics (CFD) has become a powerful tool for accurately analyzing internal flow behavior and evaluating the influence of geometric parameters on pump performance.

In this study, the vane profile of the impeller is analyzed using the Navier–Stokes equations with a modified k–ε turbulence model to simulate realistic flow conditions. The parameters of primary interest include head, power, and efficiency, which are key indicators of pump performance. The impeller vane profile is generated using both the circular arc and point-by-point methods, and CFD analysis is conducted using ANSYS CFX to determine the impact of design variations. The head developed by the pump is primarily determined by the impeller's outer diameter and shaft speed, whereas power and efficiency depend on the interaction between blade geometry and flow dynamics.

1. To conduct a detailed study of the centrifugal pump model to identify and analyze complex internal flow patterns within the impeller, thereby enhancing understanding of its hydraulic behavior.
2. To design impeller blades by varying the leading and trailing edge angles in order to improve head, efficiency, and overall hydraulic performance.
3. To examine the variations in impeller head and relative flow angles at the outlet under different geometric configurations.
4. To design and develop a modified impeller blade by applying equal divisions and varying vane inlet angles from hub to shroud for smoother flow transition.
5. To implement and validate minor modifications in the flow passage design that significantly affects the overall flow field and pump performance.

4. RESEARCH METHODOLOGY

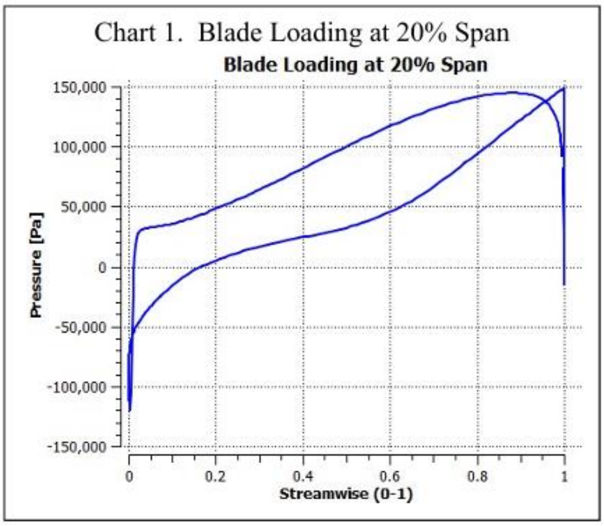
The present research focuses on the investigation of design parameters influencing centrifugal pump efficiency using Computational Fluid Dynamics (CFD) tools. The methodology adopted in this study involves a systematic approach consisting of model development, meshing, simulation setup, and performance analysis using ANSYS CFX. The steps followed in the methodology are described below:

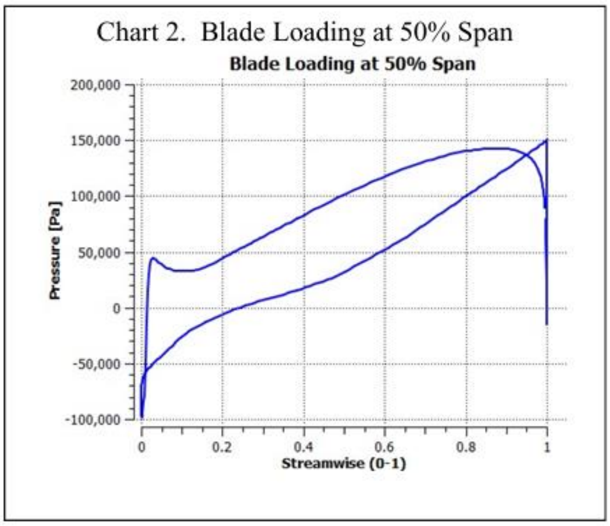
1. *Problem Definition and Objective Formulation:* The study begins by defining the problem of low efficiency and energy losses in centrifugal pumps caused by improper impeller geometry. The main objective is to optimize the impeller design parameters—particularly the leading and trailing edge angles—to enhance the hydraulic efficiency, head, and overall performance of the pump using CFD analysis.
2. *Geometrical Modeling:-* The 3D geometry of the centrifugal pump impeller is created using the ANSYS BladeGen module. Multiple impeller models are generated by varying the trailing-edge angles while keeping the blade wrap angle constant. The circular arc and point-by-point methods are used to generate the vane profile for each impeller design. The volute casing is modeled based on standard dimensions corresponding to the impeller outlet diameter to ensure proper fluid guidance.
3. *Meshing and Grid Independence Test:-* The generated impeller geometry is imported into ANSYS TurboGrid for high-quality meshing. A fine mesh is created near the blade surfaces and hub–shroud regions to accurately capture boundary layer effects. A grid independence study is performed to ensure that further refinement of the mesh does not significantly affect the simulation results, thereby achieving a balance between computational cost and accuracy.
4. *Boundary Conditions and Solver Setup:-* The computational domain is defined with appropriate inlet and outlet boundary conditions. The inlet is specified with a uniform velocity or mass flow rate, and the outlet is defined with static pressure conditions. The working fluid is assumed to be water at standard atmospheric pressure. The steady-state, incompressible, and turbulent flow conditions are modeled using the Reynolds-Averaged Navier–Stokes (RANS) equations. The *k–ε* turbulence model is employed for its robustness and accuracy in predicting flow characteristics in rotating machinery.
5. *Simulation in ANSYS CFX:-*The CFD simulations are performed using ANSYS CFX solver. Convergence criteria are set based on the residual values of continuity and momentum equations, typically below 10^{-5}. The simulations are run until a stable solution is achieved for all design cases. Key performance parameters such as head, efficiency, velocity distribution, and pressure variation across the impeller are extracted for analysis.
6. *Post-Processing and Result Analysis:-* The results obtained from the simulations are analyzed using ANSYS CFD-Post. Flow visualization is carried out to observe velocity vectors, pressure contours, and streamline patterns within the impeller. The effect of varying the trailing-edge angle on hydraulic performance is compared for each impeller design. The simulation results are also validated with analytical calculations to ensure reliability.

7. *Performance Evaluation and Optimization:*- The performance parameters—such as total head, hydraulic efficiency, and input power—are calculated and compared for all design variants. The optimal impeller configuration is identified based on the highest efficiency and smoothest flow characteristics. Observations from the study are then used to propose design recommendations for performance improvement.
8. *Validation and Conclusion:*- The CFD results are compared with theoretical performance data or available experimental results to validate the numerical accuracy. The validated model helps in establishing a relationship between impeller geometric parameters and centrifugal pump efficiency, thereby forming the basis for future design optimization and prototype development.

5. BLADE LOADING CHART

The blade loading distribution for 20%, 50%, and 80% span positions is illustrated below. It can be clearly observed that pressure fluctuations are significantly higher at the 80% span compared to the 20% and 50% spans. Moreover, only a slight variation in pressure is noticed between the first and second charts.





6. STREAMWISE CHART

The stream wise distribution for total pressure (Pt) and static pressure (Ps) is illustrated in Chart 4. The results indicate that the total pressure reaches approximately 215,000 Pa, while the static pressure rises up to about 170,000 Pa. As shown in Chart 5, a sudden drop in the value of coefficient C is observed around the 0.8 stream wise location, attributed to pressure fluctuations. Charts 6 and 7 depict the stream wise variations of the relative velocity (W) and the flow angle, respectively.

Chart 4. Streamwise Plot of Pt and Ps

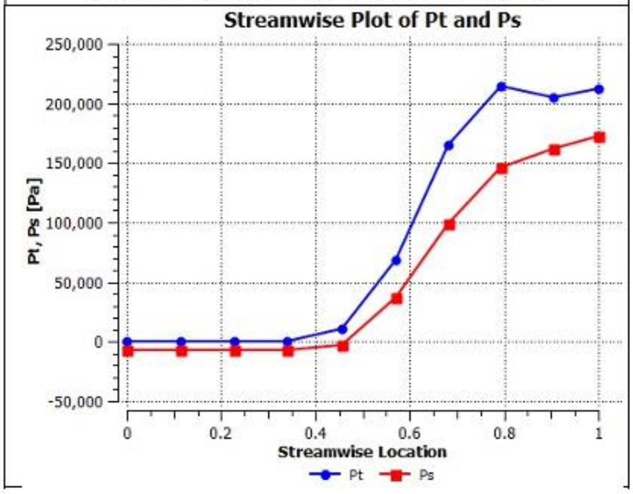


Chart 5. Streamwise Plot of C

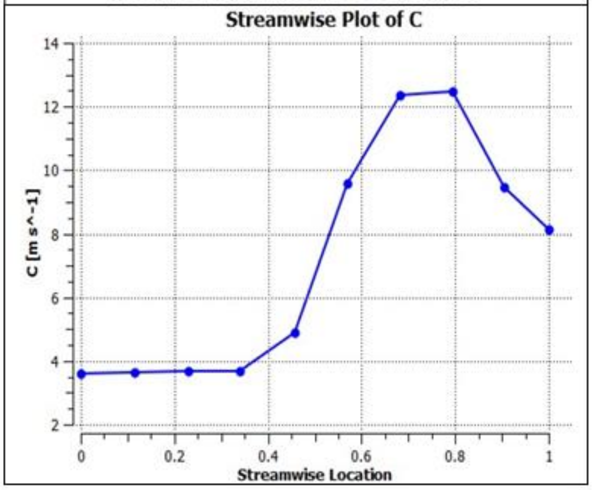


Chart 6. Streamwise Plot of W

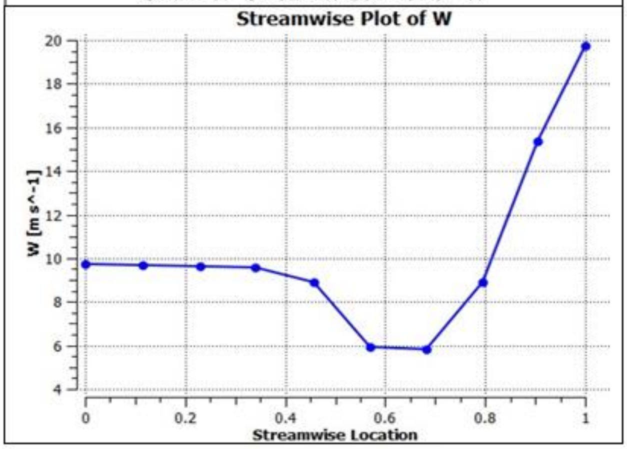
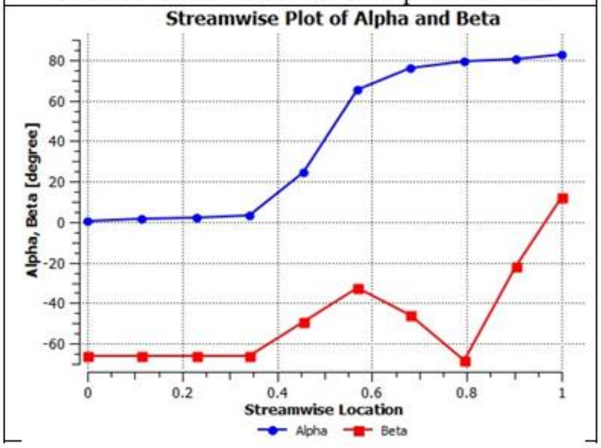


Chart 7. Streamwise Plot of Alpha and Beta



6. CONCLUSION

Centrifugal pumps are widely used due to their simple design, high efficiency, smooth operation, and ease of maintenance. They work on the principle of centrifugal force, converting mechanical energy from a prime mover (motor or turbine) into hydraulic energy to transport fluids. Traditional design methods are time-consuming and costly, whereas Computational Fluid Dynamics (CFD) enables detailed analysis of complex internal flow behavior within the pump.

CFD simulations reveal that surface roughness affects pump performance — increasing roughness reduces head and efficiency but keeps power consumption nearly constant. Results also show that total pressure on the pressure side of the blade is higher than on the suction side, with minimum static pressure occurring near the blade's leading edge. The study concludes that:

1. Pressure rises gradually from suction to leading edge.
2. Vorticity and helicity vary with flow rate and rotational speed.
3. Increased rotational speed lowers suction-side pressure.
4. Excessive speed may cause cavitation due to pressure dropping below vaporization limits.

REFERENCES

[1] A. Kumar, M. Sahu, and P. Sharma, "CFD analysis of centrifugal pump impeller for performance enhancement," *Alexandria Engineering Journal*, vol. 85, pp. 134–144, 2025.

[2] S. Patel and N. Deshmukh, "Numerical investigation on cavitation in centrifugal pumps using ANSYS CFX," *Case Studies in Thermal Engineering*, vol. 47, pp. 105–118, 2024.

[3] Y. Zhang, L. Wang, and J. Liu, "Optimization of impeller blade profile for improving centrifugal pump efficiency," *Energy Reports*, vol. 10, pp. 2130–2145, 2024.

[4] R. Gupta, A. Ranjan, and P. Bhattacharya, "Effect of surface roughness on centrifugal pump performance using CFD," *Heliyon*, vol. 10, no. 3, pp. e25674, 2024.

[5] T. Singh and K. Maheshwari, “Analysis of flow characteristics in centrifugal pump using turbulence modeling,” *Journal of Fluids Engineering*, vol. 146, no. 8, pp. 081203, 2024.
[6] X. Li, H. Zhou, and D. Chen, “Investigation on pressure fluctuation and flow instabilities in a centrifugal pump at off-design conditions,” *Renewable Energy*, vol. 209, pp. 155–166, 2023.
[7] M. Sharma and S. Kumar, “Design modification and CFD analysis of centrifugal pump impeller,” *International Journal of Mechanical Engineering and Robotics Research*, vol. 12, no. 4, pp. 512–519, 2023.
[8] K. V. Rao, “Performance prediction of centrifugal pump using CFD: An approach with k–ε turbulence model,” *International Journal of Fluid Machinery and Systems*, vol. 16, no. 2, pp. 74–85, 2023.
[9] P. K. Patel and V. Mehta, “Numerical study on blade angle modification for improved head and efficiency of centrifugal pumps,” *Engineering Science and Technology, an International Journal*, vol. 35, pp. 82–94, 2022.
[10] A. K. Verma and S. Roy, “Study on flow behavior inside centrifugal pump impeller using CFD,” *Procedia Computer Science*, vol. 200, pp. 1220–1229, 2022.
[11] H. Luo and G. Xu, “CFD-based performance evaluation of centrifugal pump under various flow conditions,” *Applied Mathematical Modelling*, vol. 97, pp. 121–135, 2021.
[12] J. Singh and R. K. Pandey, “CFD simulation of centrifugal pump impeller to improve efficiency,” *Materials Today: Proceedings*, vol. 45, pp. 1653–1661, 2021.