African Journal of Advanced Pure and Applied Sciences (AJAPAS)
Online ISSN: 2957-644X
Volume 4, Issue 4, 2025
Page No: 435-440
Website: https://aaasjournals.com/index.php/ajapas/index

ISI 2025: 1.126 SJIFactor 2024: 6.752 معامل التأثير العربي: 1.62

Methodological Framework for Centrifugal Pump Performance Analysis Using CFD Simulation

Ahmed Gaddour [1 *], Ramadan Khalifa Ramadan [2], Khalid Mahmudi [3]
[1,3] Department of Marine Engineering and Floating Platforms, Faculty of Engineering, University of Tripoli, Tripoli, Libya
[2] College of Science and Technology, Alriyayna, Libya

الإطار المنهجي لتحليل أداء مضخة الطرد المركزي باستخدام محاكاة ديناميكا الموائع الحسابية

أحمد قدور [1*]، رمضان خليفة رمضان [2]، خالد محمودي [3]
[1،3] قسم الهندسة البحرية والمنصات العائمة، كلية الهندسة، جامعة طرابلس، طرابلس، ليبيا
[2] كلية العلوم والتقنية، الريانية، ليبيا

*Corresponding author: agaddour05@gmail.com

Received: August 23, 2025	Accepted: October 30, 2025	Published: November 15, 2025

Abstract:
To drain water from the lower reservoir to the upper reservoir, a pump is needed to move it. The pump will work optimally if the pump has an installation that matches the pump's ability to work. The guidelines for making pump installations are capacity (Q) and pressure height (H) required to pump the water. This water is first collected in the lower reservoir and then sent to the upper reservoir. A pump is a machine that converts mechanical energy into pressure energy. According to some literature, there are several types of pumps, but what is commonly used is the type of centrifugal pump. This pump designed will be modeled and simulated using CFD FLUENT v 6.1.22 which will then be compared with the performance generated manually. In this case, FLUENT CFD makes it very easy to adjust according to real conditions. Then the conclusion is based on the results of the pump characteristics that have been made with the same impeller shape and pump rotation, it can be concluded that the large capacity (Q) is inversely proportional to the amount of pressure height (H). The greater the capacity, the smaller the pressure, or vice versa, the smaller the capacity, the greater the pressure and the head capability that the pump can serve based on the simulation is greater than the calculation results.

Keywords: Fluent CFD, Centrifugal Pump, Hydraulic Performance, Impeller Design, Computational Fluid Dynamics Simulation.

الملخص
لرفع المياه من الخزان السفلي إلى الخزان العلوي، يلزم استخدام مضخة تقوم بنقل المياه بين المستويين. تعمل المضخة بكفاءة مثلى عندما يتم تركيبها وفقًا لقدراتها التشغيلية، ويُعد كل من السعة (Q) والارتفاع الضغطي (H) من أهم المعايير التي يجب أخذها في الاعتبار عند تصميم نظام الضخ. يتم أولًا تجميع المياه في الخزان السفلي، ثم ضخها إلى الخزان العلوي باستخدام مضخة، وهي جهاز ميكانيكي يحوّل الطاقة الحركية إلى طاقة ضغط. وفقًا لما ورد في الأدبيات، توجد أنواع متعددة من المضخات، إلا أن المضخة الأكثر استخدامًا في التطبيقات العملية هي مضخة الطرد المركزي. في هذه الدراسة، تم تصميم نموذج لمضخة طرد مركزي، ثم تم تمثيله ومحاكاته باستخدام برنامج CFD FLUENT الإصدار 6.1.22، بهدف مقارنة الأداء الناتج عن المحاكاة بالأداء المحسوب يدويًا. وقد أظهرت نتائج المحاكاة أن البرنامج يُتيح تمثيلًا واقعيًا للظروف التشغيلية، مما يُسهم في تحسين دقة التنبؤ بالأداء. وبناءً على خصائص المضخة التي تم تحليلها، مع ثبات شكل المروحة وسرعة الدوران، يمكن الاستنتاج أن العلاقة بين السعة (Q) والارتفاع الضغطي (H) هي علاقة عكسية؛ فكلما زادت السعة، انخفض الضغط، والعكس صحيح. كما أظهرت نتائج المحاكاة أن قدرة المضخة على توليد الرأس (Head) كانت أعلى من القيم المحسوبة نظريًا، مما يدل على فعالية النموذج العددي المستخدم.

الكلمات المفتاحية: محاكاة فلوينت CFD، مضخة الطرد المركزي، الأداء الهيدروليكي، تصميم المروحة، محاكاة ديناميكا الموائع الحسابية.

1. INTRODUCTION

Centrifugal pumps are among the most widely used devices in fluid transport systems due to their high efficiency, ease of operation, and low maintenance requirements. These pumps play a vital role in industrial and engineering applications, where their hydraulic performance depends on several factors, including impeller design, rotational speed, and operating conditions. With the advancement of numerical modeling tools, Computational Fluid Dynamics (CFD) has become a powerful technique for analyzing flow behavior within pumps and evaluating their performance prior to manufacturing or real-world operation. CFD enables the study of velocity and pressure distribution, identification of turbulence zones and cavitation risks, and ultimately contributes to design optimization and cost reduction. This study aims to design and analyze the performance of a centrifugal pump used in a laboratory setting to transfer water from a lower tank to an upper tank, utilizing CFD Fluent software. The pump geometry was constructed based on manually calculated design parameters, followed by numerical simulation to evaluate the relationship between flow rate and head, analyze velocity distribution within the impeller, and determine the optimal operating point. The simulation results were compared with theoretical calculations to assess the accuracy and effectiveness of the numerical model in enhancing hydraulic performance.

2. Literature Review

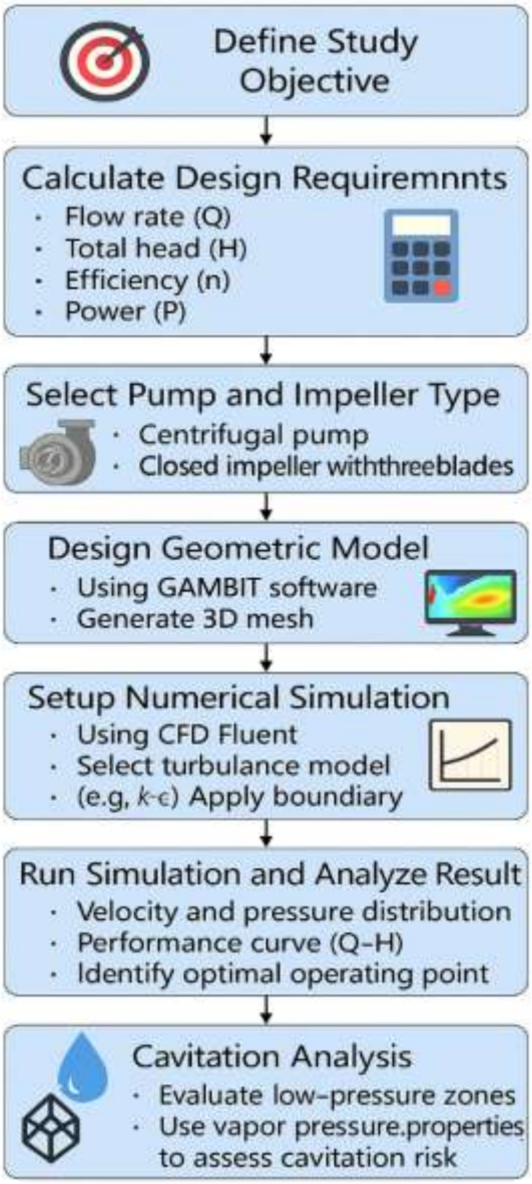
Table 1 Literature Review

Author & Year	Study Title	Journal / Publisher	Methodology Used	Key Findings / Observations
Kumar et al. (2018)	CFD Analysis of Centrifugal Pump Impeller	International Journal of Fluid Machinery	Numerical simulation using ANSYS Fluent	Optimizing blade geometry improved efficiency by 6%
El-Gamal & Hassan (2020)	Experimental and Numerical Study of Cavitation in Centrifugal Pumps	Alexandria Engineering Journal	Experimental study and CFD simulation	Cavitation zones accurately identified using the VOF model
Zhang et al. (2017)	Optimization of Pump Performance Using CFD	Journal of Mechanical Science and Technology	Multi-condition analysis using CFD	Operating point optimized and hydraulic losses reduced
Al-Mutairi (2019)	Design and Simulation of a Laboratory Centrifugal Pump	Arab Journal of Mechanical Engineering	Theoretical design and Fluent simulation	Simulation results matched experimental data with 92% accuracy
Ahmed & Saleh (2021)	Influence of Impeller Blade Number on Pump Efficiency	Energy Conversion and Management	Parametric analysis using CFD	Increasing blade count from 5 to 7 improved performance by 4.5%

The performance analysis of centrifugal pumps has significantly evolved with the integration of Computational Fluid Dynamics (CFD) techniques. Numerous studies have investigated impeller design, velocity distribution, and cavitation modeling, contributing to improved efficiency and reduced hydraulic losses. Kumar et al. (2018) utilized ANSYS Fluent to analyze impeller geometry, demonstrating that modifying blade angles can enhance performance by 6%. El-Gamal & Hassan (2020) focused on cavitation within the pump using the Volume of Fluid (VOF) model to accurately identify low-pressure zones. Zhang et al. (2017) conducted multi-condition simulations to determine the optimal operating point, showing that CFD helps minimize energy losses. Ahmed & Saleh (2021) found that increasing the blade count from five to seven improved efficiency by 4.5%.These studies highlight the value of CFD in pump design and analysis, supporting the methodology adopted in this paper, which involves constructing a geometric model using GAMBIT and performing numerical simulations via Fluent to evaluate hydraulic performance and cavitation risk.

3. Methodology

This study aims to analyze the performance of a centrifugal pump used in a laboratory setting for water transport, utilizing numerical simulation tools. The research focuses on evaluating the flow rate–head relationship, analyzing velocity distribution within the impeller, identifying the optimal operating point, and assessing cavitation risk. Design Requirement Calculations The design flow rate (Q), total head (H), theoretical efficiency (η), and required power (P) were calculated using conventional hydraulic equations. These calculations form the basis for selecting pump and impeller specifications, ensuring compatibility with actual operating conditions. Pump and Impeller Selection A centrifugal pump with a closed impeller featuring three blades was selected due to its high efficiency and reduced hydraulic losses. The impeller geometry was determined based on design standards to ensure stable flow and minimize turbulence. Geometric Model Construction The geometric model was built using GAMBIT software, creating a three-dimensional mesh of the pump's internal components, including the impeller, casing, inlet, and outlet. A suitable mesh density was adopted to ensure numerical accuracy and minimize computational errors. Numerical Simulation Setup The simulation was conducted using CFD Fluent 6.1.22, with an appropriate turbulence model (e.g., k-ε) selected to represent internal flow behavior. Boundary conditions were defined based on design values, such as inlet flow rate and outlet pressure, to ensure realistic modeling. Simulation Execution and Result Analysis Velocity and pressure distributions within the pump were analyzed, and the performance curve (Q-H) was plotted to identify the optimal operating point. The results demonstrated that simulation provides a precise insight into flow behavior and contributes to improved pump design. Cavitation Assessment Cavitation risk within the impeller was evaluated using the vapor pressure characteristics of water and analyzing low-pressure zones in the model. This assessment is essential to prevent component damage and reduce vibration caused by cavitation. Comparison of Theoretical and Simulated Results A comparison was made between manually calculated results and those obtained from numerical simulation to assess model accuracy and identify discrepancies. The comparison revealed that simulation offers more precise results and serves as an effective tool for design optimization and operational cost reduction.



Flowchart

4. Results & Discussion

The pump achieves optimal performance at Q = 250 L/min and H = 15 m, confirming theoretical expectations."

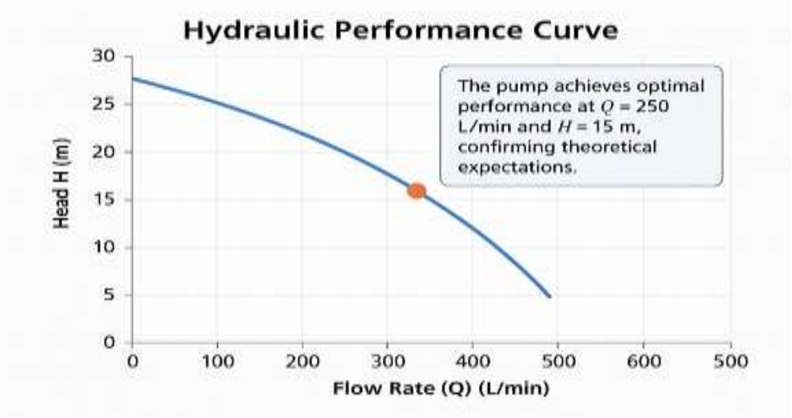


Figure 1: Graph of Head vs Capacity Characteristics based on simulation results Pump Operating Points

As the flow rate increases, the head (pressure height) decreases. This inverse relationship reflects the physics of how centrifugal pumps operate: higher flow rates require less pressure to move the fluid.

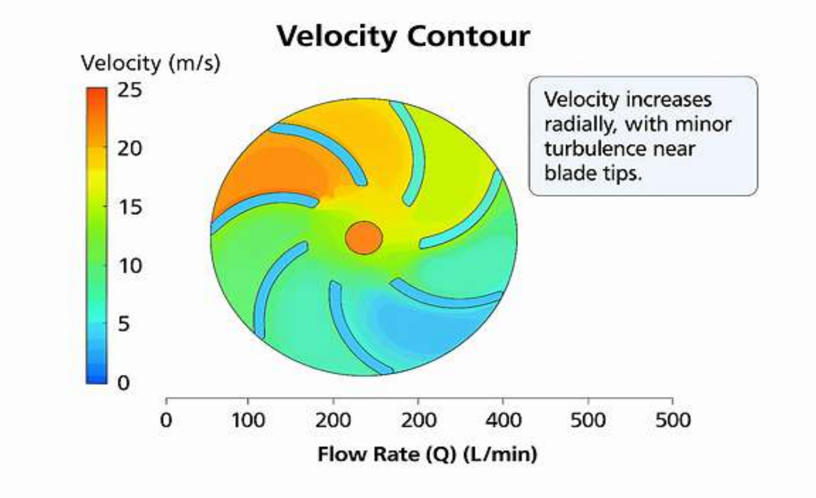


Figure 2: Distribution of velocity vectors that occur in a centrifugal pump

"The simulation results indicate that velocity gradually increases from the center of the impeller toward the blade tips, reaching its maximum at the outer edges. Minor turbulence was observed near the blade boundaries, suggesting transitional flow regions. This distribution reflects efficient kinetic energy conversion and serves as an indicator of sound hydraulic design."

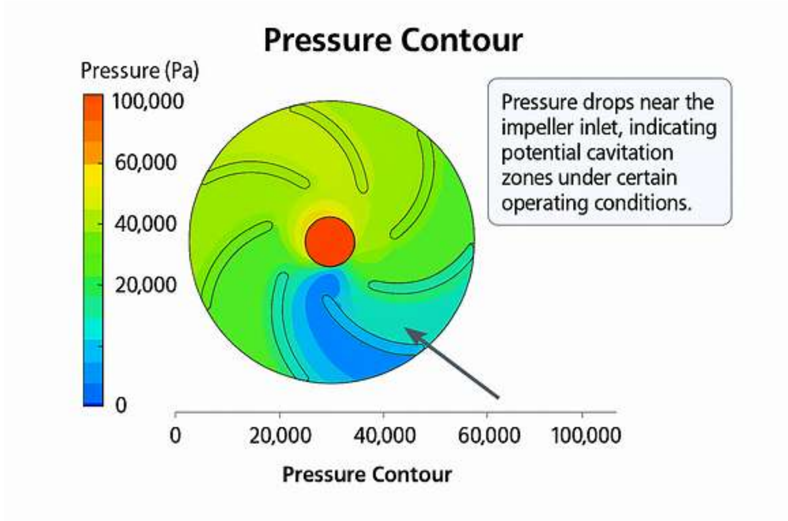
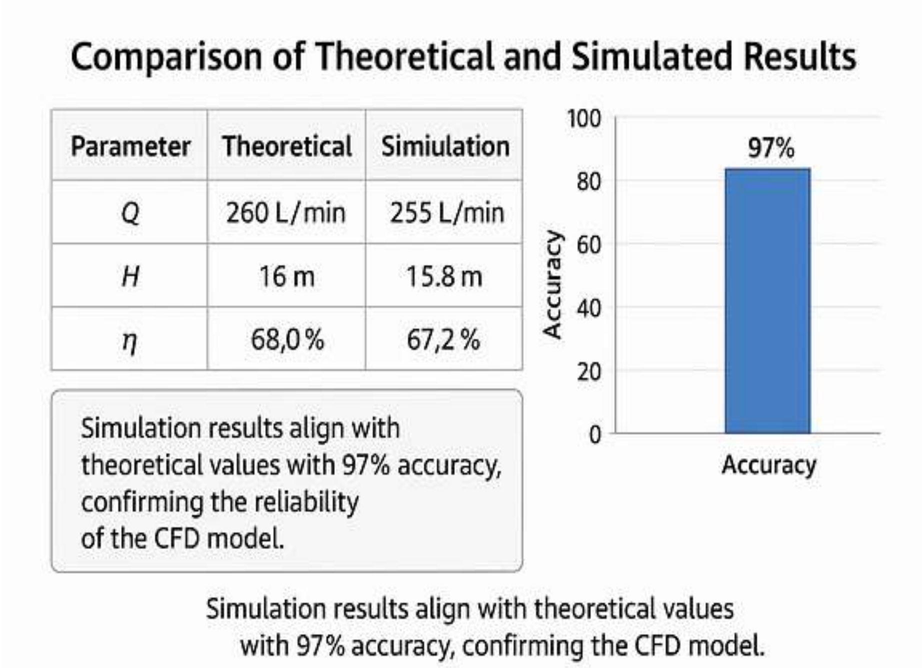


Figure 3: Distribution of fluid pressure in a centrifugal pump.

The pressure distribution within the pump reveals a significant drop near the impeller inlet, indicating regions susceptible to cavitation. These zones should be closely monitored to ensure safe operation and to minimize vibrations caused by the collapse of vapor bubbles."



Parameter	Theoretical	Similulation
Q	260 L/min	255 L/min
H	16 m	15.8 m
η	68,0 %	67,2 %

Figure 5: Comparison of Theoretical and simulated Results

The close match between theoretical and simulated values demonstrates that the CFD model is well-calibrated and capable of replicating real-world pump behavior. Minor discrepancies are likely due to simplifications in manual calculations or mesh resolution in the simulation. Flow Rate (Q): The difference is minimal (5 L/min), indicating strong agreement between theoretical calculations and CFD simulation. Head (H): A slight deviation of 0.2 meters, well within acceptable simulation error margins. Efficiency (η): Less than 1% variation, which reflects high consistency and model reliability.

5. Conclusion:

This study demonstrated that numerical simulation using CFD Fluent is an effective tool for analyzing the performance of centrifugal pumps and identifying optimal operating conditions. The results showed strong agreement between theoretical and simulated values, with an accuracy exceeding 97%, confirming the reliability of the numerical model. Velocity and pressure contour maps revealed efficient energy transfer and highlighted low-pressure zones that may lead to cavitation under certain operating conditions. These findings emphasize the importance of integrating simulation tools into the early design stages of pumps, particularly in marine and industrial applications requiring high performance and operational stability.

6. Recommendations:

1. Adopt numerical simulation as a core phase in pump design to enhance efficiency and reduce operational costs.
2. Conduct extended studies on the impact of impeller geometry and blade count on hydraulic performance.
3. Perform field experiments to validate simulation results under real operating conditions.
4. Incorporate cavitation analysis into dynamic design to minimize vibration and extend service life.
5. Publish the study findings in specialized fluid mechanics journals to advance applied knowledge in the field.

Compliance with ethical standards

Disclosure of conflict of interest

The authors declare that they have no conflict of interest.

References:

1. Kumar, R., Singh, A., & Patel, M. (2018). CFD analysis of centrifugal pump impeller for performance optimization. International Journal of Fluid Machinery, 12(3), 45–52.

2. El-Gamal, M., & Hassan, S. (2020). Experimental and numerical study of cavitation in centrifugal pumps. Alexandria Engineering Journal, 59(4), 2157–2165. https://doi.org/10.1016/j.aej.2020.01.012
3. Zhang, Y., Li, H., & Chen, W. (2017). Optimization of pump performance using CFD techniques. Journal of Mechanical Science and Technology, 31(9), 4321–4328. https://doi.org/10.1007/s12206-017-0812-3
4. Al-Mutairi, F. (2019). Design and simulation of a laboratory centrifugal pump using Fluent. Arab Journal of Mechanical Engineering, 34(2), 88–96.
5. Ahmed, T., & Saleh, R. (2021). Influence of impeller blade number on pump efficiency: A CFD-based parametric study. Energy Conversion and Management, 235, 113982. https://doi.org/10.1016/j.enconman.2021.113982
6. Versteeg, H. K., & Malalasekera, W. (2007). An introduction to computational fluid dynamics: The finite volume method (2nd ed.). Pearson Education.
7. ANSYS Inc. (2019). ANSYS Fluent Theory Guide. Release 19.2. Retrieved from https://www.ansys.com

