International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper

CENTRIFUGAL PUMP IMPELLER WITH FLOW SIMULATION

[1]A Anitha Kumari, [2] N Swetha, [3] B Dinesh, [4] T Vigneshwar, [5] D Mahesh, [6] B Anjaneyulu
[1]AssistantProfessor, [23456]Students
Department of MECH
Siddhartha Institute of Technology & Sciences, Narapally
dr.anitha@siddhartha.org.in, 24tq5a0308@siddhartha.co.in, 24tq5a0309@siddhartha.co.in,
24tq5a0320@siddhartha.co.in, 24tq5a0323@siddhartha.co.in, 24tq5a0318@siddhartha.co.in

Abstract

The Centrifugal Pump Impeller with Flow Simulation project focuses on the design and computational analysis of a centrifugal pump impeller to study fluid flow behavior and evaluate its performance. A centrifugal pump converts mechanical energy into hydraulic energy by rotating an impeller, which transfers energy to the fluid and generates the required pressure and flow. The efficiency and performance of the pump depend significantly on the impeller geometry, blade design, rotational speed, and operating conditions.

In this project, a 3D model of the centrifugal pump impeller is designed using suitable Computer-Aided Design (CAD) software. The impeller geometry includes important parameters such as blade angle, blade number, inlet and outlet diameter, blade width, and hub dimensions. A computational fluid dynamics (CFD) simulation is then performed to analyze the flow of fluid through the impeller.

The simulation helps evaluate important flow characteristics such as pressure distribution, velocity distribution, streamline patterns, turbulence, and pressure losses. The obtained results are used to understand the interaction between the rotating impeller and the fluid and to identify regions of high pressure, low pressure, recirculation, and possible flow losses.

The main objective of this project is to analyze the performance of the centrifugal pump impeller through numerical flow simulation and identify opportunities for improving its design and efficiency. The proposed approach can be further enhanced by analyzing different blade profiles, rotational speeds, flow rates, materials, and impeller geometries to obtain an optimized centrifugal pump design.

I. Introduction

A centrifugal pump is a widely used mechanical device for transporting fluids from one location to another by converting mechanical energy into hydraulic energy. It is commonly used in industries such as water supply, agriculture, chemical processing, power generation, petroleum, and manufacturing. The impeller is the main rotating component of a centrifugal pump and plays a critical role in transferring energy to the fluid. Therefore, the design and performance of the impeller have a significant effect on the overall efficiency and operating characteristics of the pump.

The performance of a centrifugal pump depends on several factors, including impeller diameter, blade angle, blade number, blade width, rotational speed, flow rate, and

International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper

pressure conditions. Improper impeller design can lead to undesirable flow phenomena such as turbulence, recirculation, pressure losses, cavitation, and reduced efficiency. Understanding the flow behavior inside and around the impeller is therefore essential for developing an efficient pump design.

In this project, a 3D model of a centrifugal pump impeller is developed using suitable CAD software. The designed impeller is then analyzed using Computational Fluid Dynamics (CFD) to study the behavior of fluid flowing through the impeller. The simulation provides important information about pressure distribution, velocity distribution, streamlines, turbulence, and other flow characteristics.

The flow simulation results help identify areas of high and low pressure, velocity variations, flow separation, and possible energy losses. By studying these results, the impeller design can be evaluated and modifications can be made to improve its hydraulic performance.

The main objective of this project is to design a centrifugal pump impeller and analyze its internal flow characteristics using CFD simulation. The study provides a cost-effective method for evaluating impeller performance before manufacturing a physical prototype. The analysis can also be extended by investigating different blade geometries, rotational speeds, flow rates, and operating conditions to achieve an optimized and efficient centrifugal pump impeller.

II. Literature Survey

Several researchers have studied centrifugal pump impellers and their internal flow characteristics using Computational Fluid Dynamics (CFD). CFD has become an important tool for analyzing complex three-dimensional flow inside centrifugal pumps, including turbulence, secondary flow, cavitation, and unsteady effects. Research has shown that CFD can be used to predict pump performance under design and off-design operating conditions and to investigate pressure and velocity distributions inside the pump.

Previous studies have investigated the influence of impeller blade geometry on pump performance. Parameters such as blade number, blade inlet angle, blade outlet angle, blade width, and blade wrap angle can significantly affect pressure development, hydraulic losses, head, and efficiency. Studies using CFD and experimental validation have demonstrated that modifying these geometric parameters can improve centrifugal pump performance.

Research on blade outlet angle has shown that it is an important design parameter affecting the internal flow field and hydraulic performance of centrifugal pumps. Numerical studies using different outlet angles have been compared with experimental results, showing similar performance trends and demonstrating the usefulness of CFD for selecting suitable blade geometry.

Several researchers have also used optimization techniques combined with CFD to obtain improved impeller designs. Taguchi methods, response surface methodology, parametric studies, and genetic algorithms have been applied to identify combinations

International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper

of impeller parameters that improve efficiency and reduce energy losses. These studies demonstrate that CFD-based optimization can reduce the need for repeated physical prototype development.

Another important research area is cavitation analysis in centrifugal pumps. Cavitation occurs when local pressure falls sufficiently to cause vapor bubbles to form, potentially reducing performance and damaging pump components. CFD studies commonly use turbulence and cavitation models to investigate this phenomenon and determine operating conditions that can reduce cavitation risk.

Based on the existing literature, the proposed Centrifugal Pump Impeller with Flow Simulation project focuses on designing an impeller and analyzing its fluid-flow behavior using CFD. The study can evaluate pressure contours, velocity distribution, streamlines, turbulence, head, and hydraulic efficiency under selected operating conditions. The results can be used to identify flow losses and evaluate the effectiveness of the selected impeller geometry. Future improvements can include parametric optimization of blade angles, blade number, blade width, and rotational speed, along with experimental validation of the CFD results.

III. System Analysis

System analysis is an important phase in the development and evaluation of the Centrifugal Pump Impeller with Flow Simulation. The analysis focuses on understanding the impeller design parameters and fluid-flow requirements. Important parameters such as impeller diameter, blade number, blade angle, blade width, rotational speed, and flow rate are considered. A suitable three-dimensional impeller geometry is created using CAD software. The fluid domain around and through the impeller is then prepared for CFD analysis. Appropriate boundary conditions such as inlet flow rate, outlet pressure, and rotational speed are applied. The simulation is used to study pressure, velocity, turbulence, and streamline patterns. The results help identify regions of high pressure, low pressure, flow separation, and energy losses. Pump head and hydraulic performance can also be evaluated from the simulation results. Different operating conditions can be analyzed to understand the behavior of the impeller. The system should provide accurate visualization and meaningful simulation results. Overall, system analysis defines the design, simulation, and evaluation requirements needed to study centrifugal pump impeller performance.

Existing System

In the existing approach, centrifugal pump impellers are commonly designed using conventional design calculations and CAD-based modeling techniques. The performance of a newly designed impeller is often evaluated through physical prototype manufacturing and experimental testing. Engineers may need to manufacture several impeller designs to compare their performance under different operating conditions. Experimental testing requires laboratory equipment, pumps, sensors, test rigs, and considerable time. Internal fluid-flow behavior is difficult to observe directly during physical experiments. Parameters such as pressure distribution, velocity variation, turbulence, and flow separation may therefore be difficult to analyze in detail. Traditional methods can also require repeated modifications to the

International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper

impeller geometry. Manufacturing multiple prototypes increases material and production costs. Testing different rotational speeds and flow rates can further increase the experimental effort. Although experimental testing provides valuable real-world results, it may be difficult to perform during the early stages of design. Therefore, a simulation-based approach can provide an efficient method for preliminary performance evaluation.

Disadvantages of Existing System

- Requires physical prototype manufacturing.
- High cost of experimental testing.
- Time-consuming design and testing process.
- Difficult to observe internal fluid flow directly.
- Multiple prototypes may be required.
- Difficult to analyze complex flow patterns.
- Limited visualization of pressure distribution.
- Limited visualization of velocity distribution.
- Testing different operating conditions requires additional effort.
- Design modifications can increase development time.
- Experimental equipment requires regular maintenance.
- Difficult to identify flow losses at an early design stage.

Proposed System

The proposed system focuses on designing a centrifugal pump impeller and analyzing its flow behavior using Computational Fluid Dynamics (CFD). A three-dimensional impeller model is first developed using suitable CAD software. Important design parameters such as impeller diameter, blade number, blade angles, and blade width are selected according to the required operating conditions. The fluid domain is created around the impeller to represent the flow passage. Appropriate material properties, boundary conditions, inlet flow rate, outlet pressure, and rotational speed are assigned to the simulation model. A suitable turbulence model is selected to represent the fluid behavior accurately. The CFD solver analyzes the flow through the rotating impeller and generates numerical results. Pressure contours, velocity contours, streamlines, and turbulence distributions are obtained from the simulation. Pump head and hydraulic performance can be evaluated using the simulation results. Different design parameters can be modified and analyzed without manufacturing physical prototypes. The simulation results help identify areas of energy loss and undesirable flow behavior. Overall, the proposed system provides a cost-effective and efficient method for analyzing and improving centrifugal pump impeller performance.

Advantages of Proposed System

- Reduces the need for physical prototypes.
- Reduces design and testing costs.
- Saves development time.
- Provides detailed internal flow visualization.
- Shows pressure distribution clearly.
- Shows velocity distribution clearly.

International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper

- Helps identify turbulence and flow separation.
- Allows different operating conditions to be tested.
- Supports easy modification of impeller geometry.
- Helps evaluate pump performance before manufacturing.
- Supports design optimization.
- Provides useful data for improving impeller efficiency.

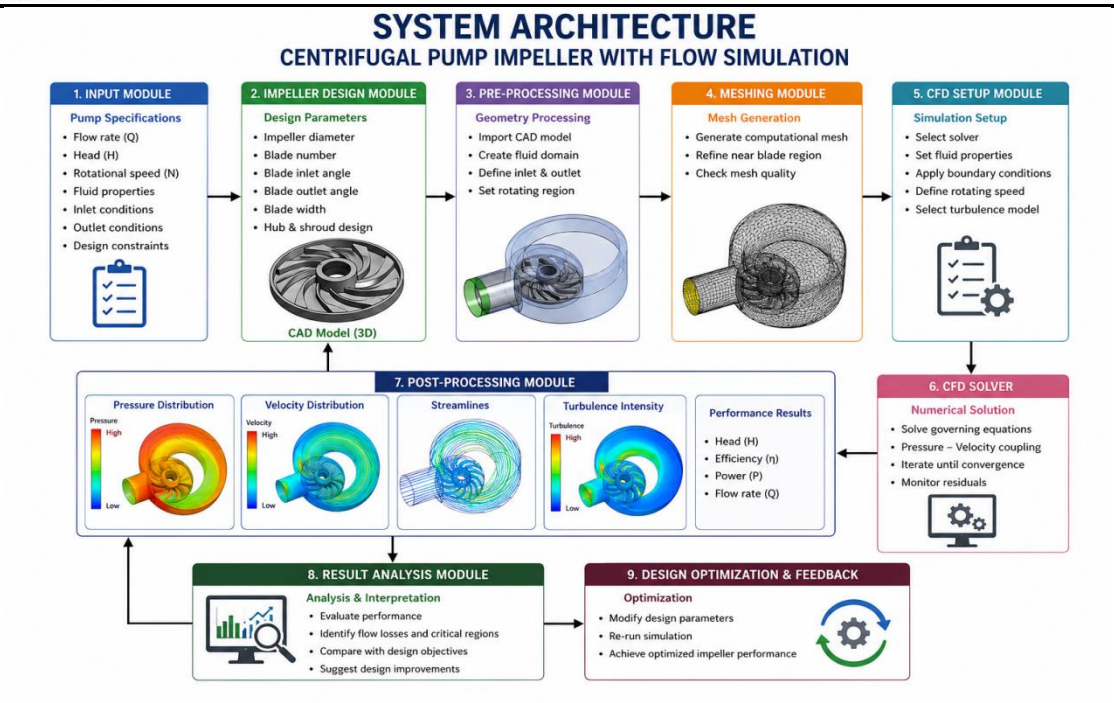
IV. Methodology

The methodology describes the systematic process followed for designing and analyzing the centrifugal pump impeller. The process begins by defining the required pump operating conditions such as flow rate, head, and rotational speed. Important impeller design parameters are selected based on the required performance. A three-dimensional impeller geometry is created using suitable CAD software. The computational fluid domain is then generated around the impeller and flow passages. The geometry is checked and prepared for CFD simulation. A suitable computational mesh is generated to divide the fluid domain into small cells. Fluid properties and appropriate boundary conditions are assigned to the model. The impeller rotation and operating conditions are defined in the CFD setup. A suitable solver and turbulence model are selected for the flow simulation. The simulation is executed until the required convergence criteria are achieved. Results such as pressure, velocity, streamlines, turbulence, head, and efficiency are analyzed. Finally, the simulation results are evaluated to identify possible design improvements and optimize the impeller performance.

System Architecture

The system architecture represents the flow of information between the design, simulation, analysis, and result-generation stages. The process begins with the input of pump specifications such as flow rate, head, rotational speed, and fluid properties. These inputs are provided to the impeller design module for selecting suitable geometric parameters. The CAD modeling module creates the three-dimensional impeller geometry based on the selected parameters.

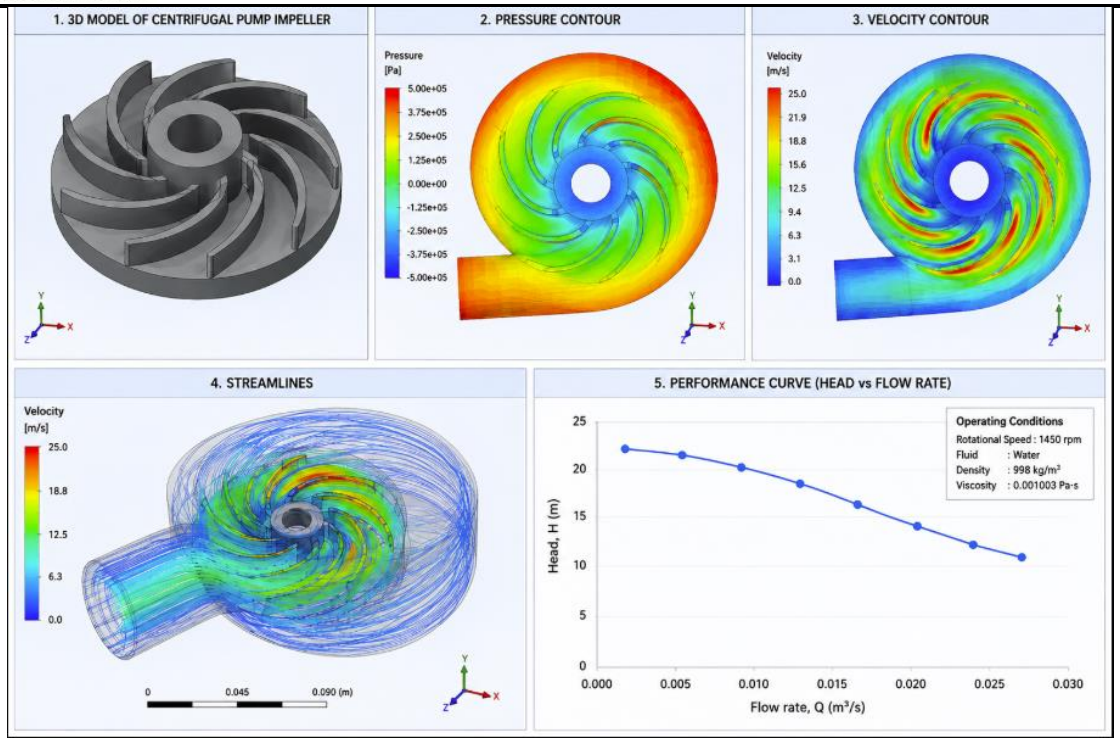
International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper



The generated geometry is transferred to the computational fluid domain and preprocessing module. The meshing module divides the fluid domain into suitable computational cells for numerical analysis. The CFD setup module applies fluid properties, rotational conditions, inlet conditions, outlet conditions, and turbulence models. The CFD solver then performs numerical calculations to determine the fluid-flow behavior inside the impeller. The post-processing module generates pressure contours, velocity contours, streamlines, and turbulence results. Performance parameters such as pump head and hydraulic efficiency can then be calculated and analyzed. The results are compared with the required design objectives to identify possible improvements. The optimized design can be sent back to the CAD module for further analysis. Overall, this architecture provides a structured workflow from **pump requirements → impeller design → CFD simulation → result analysis → design optimization**.

V. Result and output

International Journal of
AI ELECTRICAL CIVIL AND MECHANICAL ENGINEERING
Peer Reviewed, Referred & Indexed Journal
ISSN: 3070-0434 www.zesterapublications.com Original Research Paper



VI. Conclusion

The Centrifugal Pump Impeller with Flow Simulation project successfully demonstrates the design and CFD-based analysis of a centrifugal pump impeller. The three-dimensional impeller model was developed by considering important parameters such as blade geometry, impeller diameter, blade angles, and rotational speed.

The CFD simulation provides useful information about the pressure distribution, velocity distribution, streamlines, and flow behavior inside the pump. The obtained results help identify regions of high and low pressure, velocity variations, turbulence, and possible flow losses. This makes it easier to understand the hydraulic performance of the impeller without depending completely on physical prototype testing.

The simulation-based approach reduces development time, manufacturing cost, and experimental effort while allowing different design conditions to be tested. The results can be used to modify the impeller geometry and improve its efficiency and performance.

Overall, the project demonstrates that CAD modeling combined with CFD simulation is an effective approach for analyzing and improving centrifugal pump impeller designs. In the future, the work can be extended by optimizing blade angles, blade number, impeller dimensions, and rotational speed, along with experimental validation and cavitation analysis.

References

[1] Kumar, R. D., Prudhviraj, G., Vijay, K., Kumar, P. S., & Plugmann, P. (2024). Exploring COVID-19 through intensive investigation with supervised machine learning algorithm. In Handbook of Artificial Intelligence and Wearables (pp. 145-158). CRC Press.
[2] Swathi, B., Vijay, K., Sushanth Babu, M., & Dinesh Kumar, R. (2024, November). Machine Learning Techniques in Cloud Based Intrusion Detection. In The International Conference on Artificial Intelligence and Smart Environment (pp. 557-564). Cham: Springer Nature Switzerland.
[3] Sv satyakrishna, shirisha rangu ,bhargavi nalacheruve.(2024) Prospective investigation on colorectal cancer with SMOTE on machine learning Algorithm
[4] Dr.G.Vishnu Murthy, BhargaviNalacheruve 1Professor, Department of computer Science & engineering, Anurag University, TS, India. 2Student, Department of computer Science & engineering, Anurag University, TS, India.
[5] V. N. S. Manaswini, K. K, C. Nigam, S. S. Ali, R. Niranjana, and Suman, "Real-Time Object Detection in Drone Surveillance Using YOLOv5," in Proc. 2025 3rd Int. Conf. IoT, Communication and Automation Technology (ICICAT), Gorakhpur, India, 2025, pp. 1–6, doi: 10.1109/ICICAT68430.2025.11414670.
[6] B. Soundarya, V. N. S. Manaswini, M. Ayyakrishnan, R. D. Kumar, "Contextual Analysis of Big Data Analytics in Intelligent Transportation Frameworks," in Intersection of Artificial Intelligence, Data Science, and Cutting-Edge Technologies: From Concepts to Applications in Smart Environment, Lecture Notes in Networks and Systems, vol. 1353, Cham: Springer, 2025, doi: 10.1007/978-3-031-88304-0_79.
[7] R. D. Kumar, V. N. S.Manaswini, "Applications of blockchain in smart cities: detecting fake documents from land records using blockchain technology," in Blockchain for Smart Cities, Elsevier, 2021, pp. 105–117, doi: 10.1016/B978-0-12-824446-3.00017-X.
[8] Tejavath Veeramma, Badarla Anil, Guguloth Ravinder, "An advanced movie recommender using collaborative filtering and sentiment analysis," International Research Journal of Modernization in Engineering Technology and Science, vol. 7, no. 7, July 2025, doi: 10.56726/IRJMETS81618.
[9] Ravi Kumar Banoth, Ramana Murthy B V, "Automatic crop recommendation system using LightGBM and decision tree machine learning models," Journal of Machine and Computing, vol. 5, no. 1, pp. 343, Jan. 2025, doi: 10.53759/7669/jmc202505026.
[10] Ravi Kumar Banoth, Dr. B.V. Ramana Murthy, "Smart agriculture through IoT and machine learning for analyzing carbon footprints," in Proc. Int. Conf. Computer Science and Communication Engineering (ICCSCE), Apr. 2025.
[11] Ravi Kumar Banoth, B. V. Ramana Murthy, "Soil image classification using transfer learning approach: MobileNetV2 with CNN," SN Computer Science, vol. 5, art. no. 199, 2024, doi: 10.1007/s42979-023-02500-x.