International Journal of Research Publication and Reviews

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

Review of Design and CFD Analysis of Centrifugal Pump

Lokesh chouhan[1], Prof. kamlesh Gangrade[2]

[1]*PGStudent, Department of Mechanical Engineering, Sagar Institute of Research & Technology Indore, India*
[2]*Professor, Department of Mechanical Engineering, Sagar Institute of Research & Technology Indore, India*

ABSTRACT:-

This paper presents a comprehensive overview of recent developments in the design and Computational Fluid Dynamics (CFD) analysis of centrifugal pumps. The focus is on crucial components like impellers and casings, exploring emerging trends in materials and manufacturing methodologies. The study examines the application of CFD to optimize pump performance, addressing numerical techniques, associated challenges, and illustrating insights through practical case studies. By integrating traditional design principles with CFD advancements, the objective is to improve the efficiency, reliability, and sustainability of centrifugal pump systems across a variety of industrial applications.

Keywords: Centrifugal Pump, Design methodology, impeller design, design methodology, fluid dynamics, volute, turbulence, flow rate, head.

1. Introduction:-

Centrifugal pumps, indispensable in diverse industrial processes, are experiencing noteworthy progress in both design and analytical methodologies. This paper investigates the dynamic intersection of conventional design principles and the incorporation of Computational Fluid Dynamics (CFD) to optimize the performance of centrifugal pumps. Central to this exploration is the meticulous design consideration of crucial components such as impellers, casings, and volutes, where advancements in materials and manufacturing processes are fostering improved pump reliability and efficiency.

Simultaneously, the ascent of Computational Fluid Dynamics has transformed the comprehension and enhancement of centrifugal pump dynamics. Through the utilization of CFD simulations, engineers gain unparalleled insights into fluid flow within these pumps, enabling them to refine designs for heightened efficiency and reduced energy consumption. Throughout this paper, we will scrutinize these advancements, highlighting trends in pump design, material progress, and featuring real-world applications through case studies. The ultimate objective is to provide a comprehensive perspective on the synergy between conventional design principles and CFD, delivering practical guidance for professionals in the field.

2. Objectives of the Study:

The primary objective of this study is to comprehensively investigate the design principles and Computational Fluid Dynamics (CFD) analysis applied to centrifugal pumps. Specifically, the study aims to:

- **Examine Design Principles:** Delve into the fundamental design considerations of centrifugal pumps, focusing on key components such as impellers, casings, and volutes. Explore recent trends in materials and manufacturing processes to enhance pump efficiency and reliability.
- **Explore Computational Fluid Dynamics (CFD) Applications:** Investigate the utilization of CFD in analyzing and optimizing centrifugal pump performance. Evaluate various numerical techniques and models employed in CFD simulations, addressing their applicability and limitations.
- **Address Challenges in CFD Analysis:** Highlight challenges associated with CFD analysis of centrifugal pumps, including turbulence modeling, mesh generation, and boundary condition selection. Discuss how advancements in computing power and simulation software contribute to more accurate predictions.
- **Showcase Practical Applications:** Explore and present case studies where CFD has been instrumental in optimizing centrifugal pump performance. Illustrate real-world examples of how CFD simulations contribute to design improvements, energy efficiency, and maintenance reduction.
- **Integrate Traditional Design with CFD Insights:** Emphasize the symbiotic relationship between traditional design principles and CFD analysis. Provide insights into how combining these approaches can lead to informed decisions, ultimately improving the reliability, efficiency, and sustainability of centrifugal pump systems.

3. Literature Survey:-

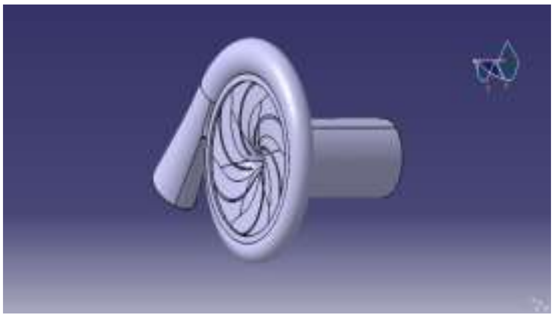
- **Wang and Tsukamoto [2003]** developed an experimental and numerical study of instability occurrences under non-ideal operating conditions for a diffuser pump. Pressure variations in a diffuser pump caused by rotor-stator contact and the spinning stall were computed using the vortex approach and compared to experimental data. They found that for any given flow rate, the impeller's downstream pressure is always dominated by the frequency at which the blades pass the rotor, as well as its higher harmonics.
- **Shojaee and Boyaghchi [2007]** studied through both experiment and simulation how a centrifugal pump with sharply angled output blades performs. The pump could handle water and thick oils since they are Newtonian fluids. The internal flows of the centrifugal pump were simulated in three dimensions using a numerical model under various operating circumstances and blade outlet angles. Characterising the turbulent flow process using the k- turbulence model. The findings show that as the outlet angle rises, the centrifugal pump becomes more effective at handling viscous fluids.
- **Vasant Godbole, Rajashri Patil, and Gavade S S S (2012)** conducted an experimental evaluation that confirmed the radial tension created within the volute is supported by the pump casing design and that axial thrust appears on a centrifugal pump. A double volute case, designed to reduce radial force, is created when the fluid's velocity is not uniform across the volute's circumference.

4. Methodology:

The methodology for this study involves the following key steps:

- **Literature Review:** Conduct an extensive review of existing literature to gather insights into traditional design principles and recent advancements in the field of centrifugal pump design. Explore peer-reviewed articles, research papers, and industry publications to establish a solid foundation for the study.
- **Design Principles Analysis:** Investigate the fundamental design principles of centrifugal pumps, emphasizing key components such as impellers, casings, and volutes. Analyze trends in materials and manufacturing processes, considering their impact on pump performance and reliability. Consult established engineering standards and guidelines for benchmarking.
- **Computational Fluid Dynamics (CFD) Application:** Explore the application of CFD in simulating and analyzing fluid flow within centrifugal pumps. Select appropriate CFD software and numerical techniques for accurate simulations. Develop computational models representing the pump geometry and fluid properties. Conduct simulations to analyze performance parameters, including pressure distribution, velocity profiles, and efficiency.
- **Challenges in CFD Analysis:** Identify and address challenges associated with CFD analysis, including turbulence modeling, mesh generation, and boundary condition selection. Evaluate the accuracy of simulation results by comparing them with experimental data or established theoretical models. Discuss limitations and propose potential improvements.
- **Case Studies:** Investigate and present real-world case studies where CFD has been applied to optimize centrifugal pump performance. Highlight specific challenges faced in each case and demonstrate how CFD insights contributed to design improvements, energy efficiency enhancements, or reduced maintenance requirements.
- **Integration of Traditional Design with CFD:** Explore the integration of traditional design principles with CFD insights. Discuss how knowledge gained from CFD simulations informs and refines the traditional design process. Highlight successful examples where this integration has led to improved pump systems in terms of efficiency, reliability, and sustainability.
- **Data Analysis and Conclusion:** Analyze the collected data from the literature review, design principles analysis, CFD simulations, and case studies. Summarize key findings and draw conclusions regarding the effectiveness of combining traditional design approaches with CFD analysis in optimizing centrifugal pump performance.

5. Design of Centrifugal pump:-



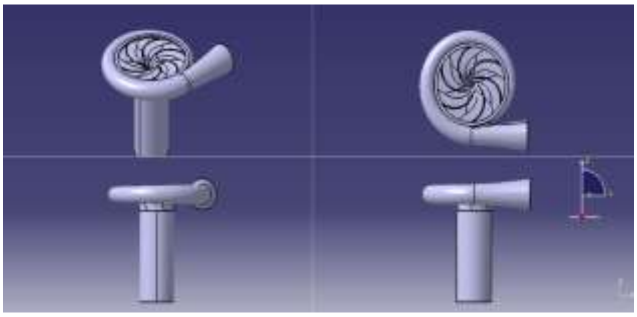


Figure 1: Designing of type-1 centrifugal pump **Figure 2:** Multi- section view of type-1 design

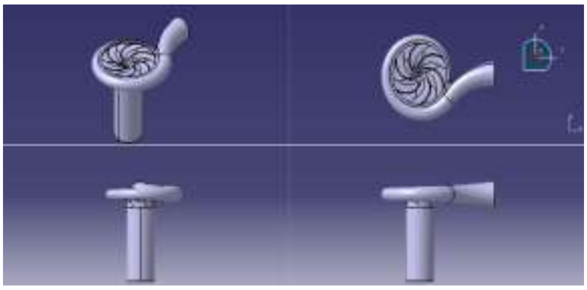


Figure 3: Isometric view of type -2 centrifugal pump **Figure 4:** Multi-view of type-2 Design of centrifugal pump

6. Analysis of Centrifugal Pump Type-1:-

Meshing and boundary condition of Press Tool

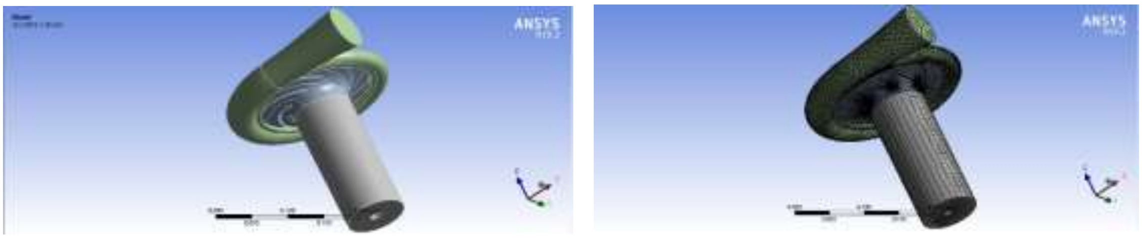


Figure 5: Importing and Meshing of Centrifugal pump type-1

Statistics	Value
Node	148868
Element	749447

Analysis of Centrifugal Pump Design type – 2:-

Meshing and boundary condition of Press Tool

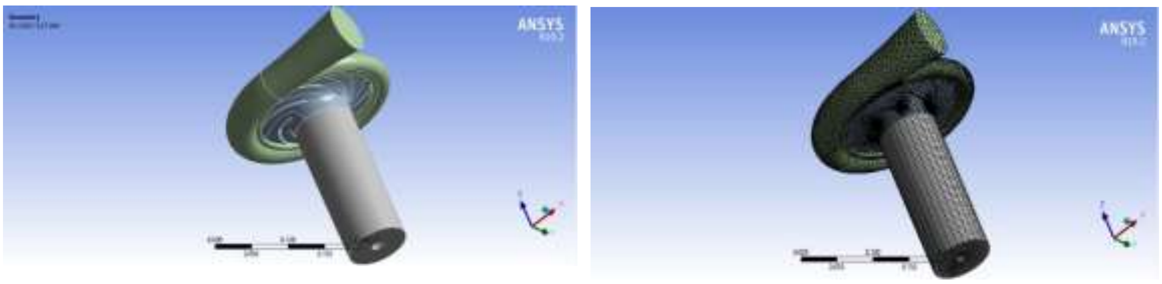


Figure 6: Importing and Meshing of Centrifugal pump type-2

Statistics	Value
Node	166430
Element	848913

Result of Centrifugal Pump type -1

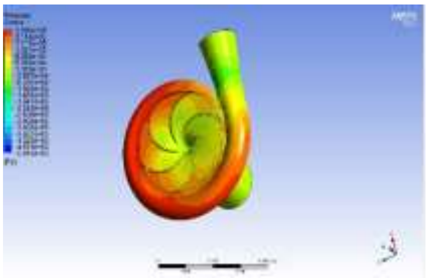


Figure 7: Pressure Contour

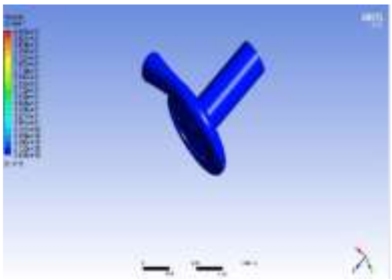


Figure 8: Velocity Counter

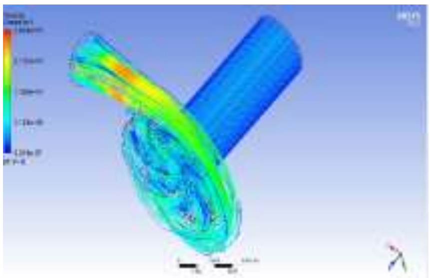


Figure 9: Velocity Streamline

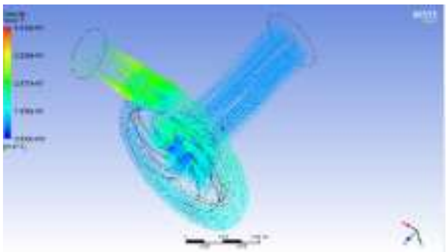


Figure 10: Velocity vector

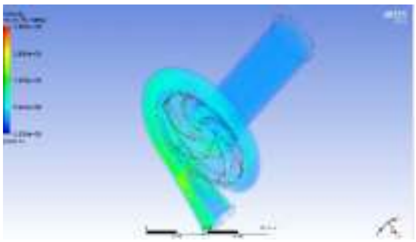


Figure 11: Velocity Volume Rendering

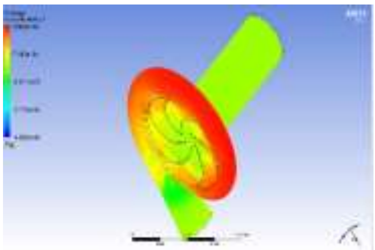


Figure 12: Pressure volume rendering

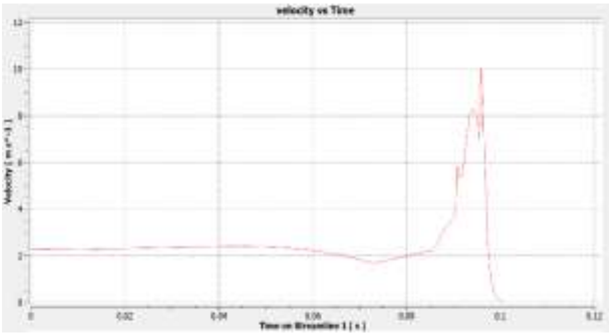


Figure 13: Velocity vs Time Graph

Result of centrifugal pump type-2

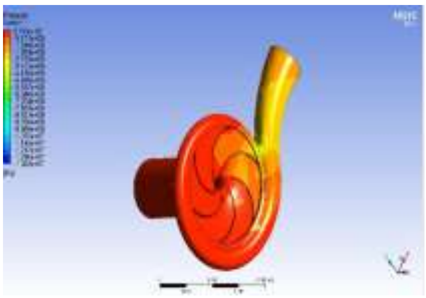


Figure 14: Pressure Contour

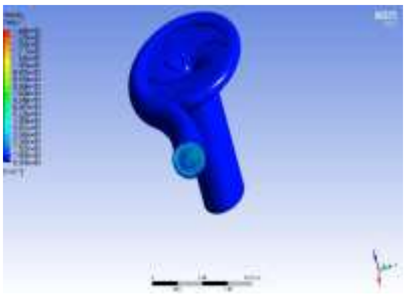


Figure 15: Velocity Counter

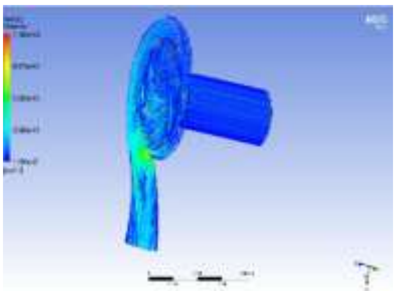


Figure 16: Velocity Streamline

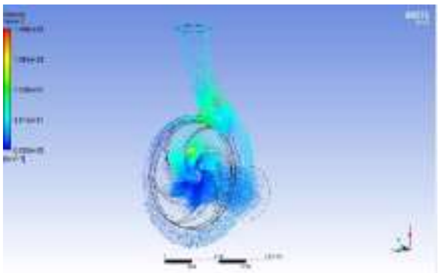


Figure 17: Velocity vector

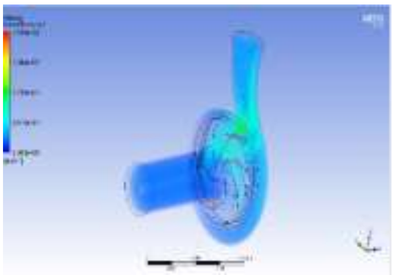


Figure 18: Velocity Volume Rendering

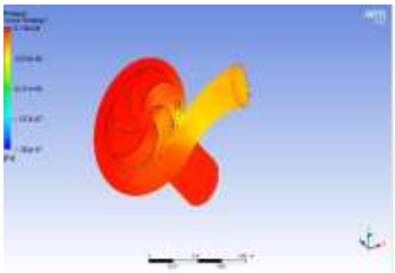


Figure 19: Pressure volume rendering

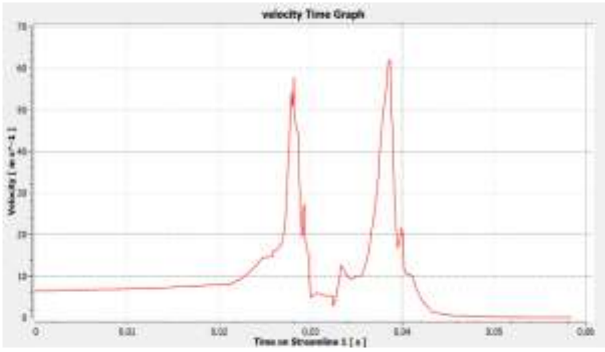


Figure 20: Velocity vs Time Graph

Conclusion:-

This study has revealed key insights into centrifugal pump design by integrating traditional principles and Computational Fluid Dynamics (CFD) analysis. Examining components like impellers and casings highlighted evolving trends in materials, impacting pump efficiency. CFD simulations proved instrumental in understanding fluid dynamics, overcoming challenges, and optimizing performance. Real-world case studies underscored CFD's practical applications, showcasing design enhancements, improved energy efficiency, and reduced maintenance needs. The study emphasizes the symbiotic relationship between traditional design and CFD insights, offering valuable guidance for enhancing the reliability and efficiency of centrifugal pump systems across diverse industrial applications.

References:-

[1] Rakibuzzaman, “A study on multistage centrifugal pump performance characteristics for variable speed drive system”, Procedia Engineering 105 (2015) 270 – 275.

[2] S.Shaaban, “Design optimization of a centrifugal compressor vaneless diffuser”, S0140-7007(2015)00188-7.

[3] Janusz Skrzypacz, “Numerical modelling of flow phenomena in a pump with a multi-piped impeller”, Chemical Engineering and Processing 75 (2014) 58-66,

[4] H.Stel , “Numerical investigation of the flow in a multistage electric submersible pump”, Journal of Petroleum Science and Engineering, 136 (2015) 41–54

[5] Wenjie Wang, “Application of different surrogate models on the optimization of centrifugal pump”, Journal of Mechanical Science and Technology, 30 (2) (2016) 567-574

[6] Shuai yang, “Dynamic analysis of the pump system based on MOC–CFD coupled method”, Journal on Annals of Nuclear Energy, 78 (2015) 60–69

[7] Richard B. Medvitz, “Performance Analysis of Cavitating Flow in Centrifugal Pumps Using Multiphase CFD”, Transactions of the ASME, Vol. 124, JUNE 2002

[8] S.Rehman , “Pumped hydro energy storage system of Renewable and Sustainable Energy”, Reviews 44 (2015) 586–598.

[9] F.Pugliese, “Characterization of two pumps as turbine for hydro power generation”, Renewable Energy 99 (2016) 180- 187.

[10] Bikram Singh Solanki , “Centrifugal Pump in Turbine Mode for Small Hydro Power Generation Mechanical Engineering Department”, UIT-RGPV, Bhopal, India 25 May 2016, Vol.6, No.3 (June 2016).