

Article

Multi-Criteria Response Surface Optimization of Centrifugal Pump Performance Using CFD for Wastewater Application

Edwin Pagayona * and Jaime Honra

School of Mechanical, Manufacturing and Energy Engineering, Mapúa University, Intramuros, Manila 1002, Philippines; jphonra@mapua.edu.ph

* Correspondence: enpagayona@mymail.mapua.edu.ph

Abstract: The effective transport of high-viscosity fluids in wastewater treatment systems is heavily contingent upon the operational efficiency of centrifugal pumps. However, challenges arise in operating these pumps under such conditions due to the detrimental impact of viscosity. This study is focused on enhancing the performance of centrifugal pumps by examining the influence of design and impeller configuration. By employing CFD analysis in ANSYS, this study examines the effects of varying inlet and outlet impeller diameters as well as different numbers of impeller blades on pump performance. The investigation entails three core stages: pre-processing, encompassing the creation of geometry, meshing, and study configuration; processing, which involves defining physics settings, selecting the solver type, and specifying boundary conditions; and post-processing, dedicated to the interpretation of results derived from model creation and solution. Leveraging Genetic Aggregation for response surface modelling facilitates the pinpointing of effective design configurations rooted in specific pump performance goals, thereby resulting in noteworthy performance enhancements. Notably, an optimal pump design featuring a 5-blade impeller with inlet and outlet diameters of 55.92 mm and 207.78 mm, respectively, yielded significant improvements of 26.51% in head, 2.53% in static efficiency, and 62.30% in incipient net positive suction head (NPSHi).

Keywords: centrifugal pump; CFD; response surface optimization



Citation: Pagayona, E.; Honra, J. Multi-Criteria Response Surface Optimization of Centrifugal Pump Performance Using CFD for Wastewater Application. *Modelling* **2024**, *5*, 673–693. <https://doi.org/10.3390/modelling5030036>

Academic Editor: Sergey Utyuzhnikov

Received: 2 May 2024

Revised: 29 May 2024

Accepted: 9 June 2024

Published: 27 June 2024



Copyright: © 2024 by the authors. Licensee MDPI, Basel, Switzerland. This article is an open access article distributed under the terms and conditions of the Creative Commons Attribution (CC BY) license (<https://creativecommons.org/licenses/by/4.0/>).

1. Introduction

Centrifugal pumps are commonly used to move liquid waste in wastewater applications, utilizing the energy of a rotating impeller to accelerate and propel the wastewater through a discharge pipe. These pumps are designed to handle the abrasive and corrosive nature of wastewater, making them well suited for this application. The increasing global population and urbanization have heightened the need for efficient waste management systems, especially in handling wastewater and sewerage [1]. These systems are crucial for safeguarding public health, the environment, and overall well-being.

Efficient sewage management and environmental preservation are crucial for urban infrastructure sustainability. Centrifugal pumps are vital in the transportation of high-viscosity fluids within sewage systems, but they face challenges such as reduced efficiency and increased susceptibility to cavitation due to the adverse effects of viscosity on pump performance.

Previous studies have focused on improving the performance of centrifugal pumps through various methods, such as redesigning and optimizing the impeller, conducting CFD (computational fluid dynamics) testing, and examining the effect of impeller parameters on efficiency. The impeller is a crucial part of the centrifugal pump and has a significant impact on its performance. Its design plays a key role in determining the pump's efficiency, hydraulic performance, and power consumption. Designing impellers for handling high-viscosity fluids presents unique challenges due to the increased force required for fluid movement, which affects the impeller's design and operation [2]. Research has aimed to

develop impellers with modified blade profiles and optimized geometries to overcome these challenges and maintain efficient fluid transport. The impact of viscosity, flow rate, and blade height on turbulence kinetic energy distribution has been analyzed, with results indicating a gradual reduction in pump head and efficiency as viscosity increases when handling high-viscosity fluids [3]. Additional studies have examined the relationship between flow characteristics, fluid-induced force, and impeller eccentricity, as well as the influences of impeller speed on internal flow disorder and fluid-induced force in different types of centrifugal pumps [4,5]. CFD approaches have been used for analyzing flow through the impeller, predicting head–capacity curves, flow patterns, and pressure distribution within the centrifugal pump, as well as studying cavitating flow, suspended solids’ impact, and impeller inlet widths [6–8]. Various researchers have also investigated the influence of impeller dimensions on centrifugal pump performance, with findings on impeller diameter, outlet width, and blade numbers affecting parameters such as pressure fluctuations, net head, power consumption, and overall pump efficiency [9–11].

The novelty of this research study is its emphasis on enhancing not just the head and static efficiency but also the NPSHi (incipient net positive suction head), a critical parameter for assessing the likelihood of cavitation in centrifugal pumps. Earlier investigations had not consistently integrated the NPSHi in their optimization endeavours, particularly with regard to the inlet impeller diameter, outlet impeller diameter, and the number of impeller blades. By incorporating these additional factors, this study endeavours to offer a more thorough and efficient method for enhancing the performance of centrifugal pumps used in wastewater applications.

This study focuses on the redesign of impellers and blade configurations in centrifugal pumps to enhance the efficient transport of high-viscosity fluids in sewage management. It tackles the inherent limitations of conventional pump designs optimized, addressing key aspects such as impeller redesign, blade configuration optimization, and cavitation mitigation. The primary objectives of this study include investigating the challenges faced by centrifugal pumps when handling high-viscosity fluids in sewage management, validating simulation results, studying centrifugal pump design, and developing an improved impeller configuration to enhance pump performance. This research aims to develop a new optimized impeller design and configuration that enhances the pump’s efficiency and reduces power consumption when handling high-viscosity fluids.

The significance of this study is particularly relevant to the wastewater treatment sector, as it aims to tackle the critical inefficiencies of centrifugal pumps—enabling facilities grappling with high energy costs due to handling high-viscosity fluids to reduce their power consumption substantially. The anticipated benefits include significant energy savings and the possibility for integration with renewable energy sources, enhancing sustainability and potentially lowering operational expenses. Due to the pumps’ enhanced design and performance, these improvements can lead to better profit margins for sewage treatment facilities and stimulate interest across multiple industries. This, in turn, would foster a more appealing market for pump manufacturers and advance the efficacy and environmental stewardship within the wastewater treatment landscape.

While this study proposes an improved pump design, implementing and testing the new impeller configuration may require further research and validation in real-world sewage treatment scenarios. This study’s findings might be limited to centrifugal pumps used in sewage management and may not directly apply to other pump types or industries. Within this study’s ambit, one scope involved adjusting the fluid’s viscosity to determine its effect on pump performance, a factor that is essential for ensuring the design’s efficacy under constant operating conditions. Nevertheless, this investigation was limited to computational fluid dynamics (CFD) analysis, and the hands-on testing and validation of the new design within actual sewage treatment processes are beyond the scope of this study.

2. Methodology

This study's methodology is divided into two main steps: computational fluid dynamics (CFD) study and optimization study. These steps allow us to comprehensively analyze the centrifugal pump's performance and identify optimal design parameters.

The methodological framework for this study is summarized in Figure 1.

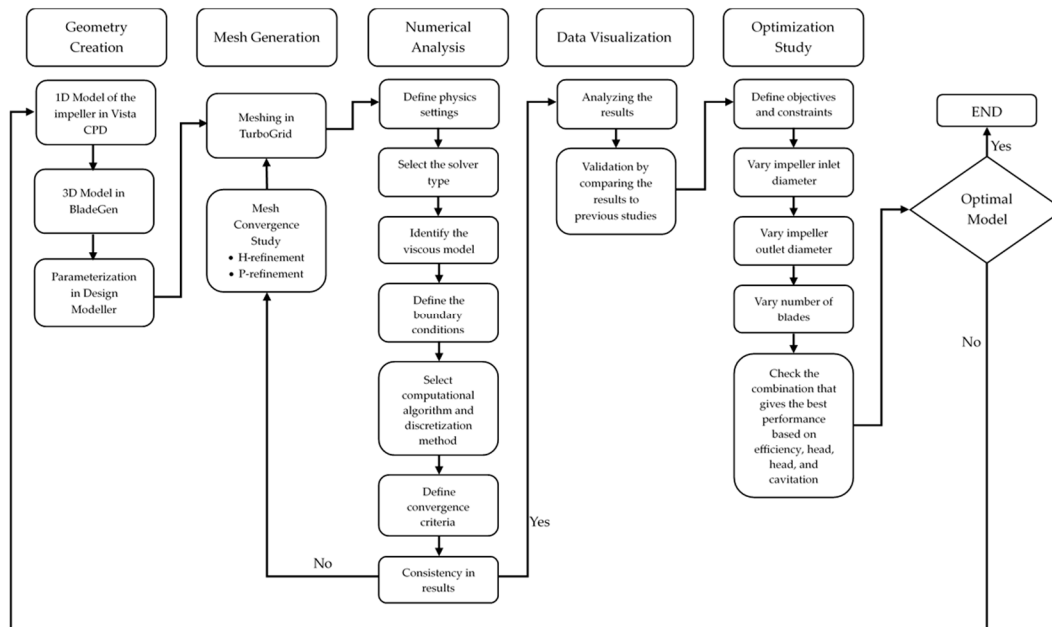


Figure 1. The methodological framework of this study.

2.1. CFD Study of Centrifugal Pump

The process will adhere to the typical CFD protocol, which involves three main stages: pre-processing, processing, and post-processing. During pre-processing, the focus is on creating the geometry, meshing, and configuring the study. The processing stage involves defining the physics settings, choosing the solver type, specifying the viscous model, establishing boundary conditions, and defining convergence criteria. The post-processing stage is the final step in the CFD study, which interprets the result relative to the assumptions made during model creation and solution.

2.1.1. Geometry Creation and Mesh Generation

To initiate the CFD study, the pre-processing stage will involve creating the pump's geometry using Ansys Vista CPD (centrifugal pump design). This encompasses the definition of the pump's performance parameters, including rotational speed, volume flow rate, and head rise. Table 1 summarizes the pump's performance parameters obtained from the pump manufacturer datasheet (KSB Pump Model: Sewatec K 065-250):

Table 1. Centrifugal pump's performance parameters.

Parameters	Unit	Value
Rotational speed	rpm	1466
Volume flow rate	m ³ /h	90
Head rise	m	10

Utilizing this information, Vista CPD was applied to generate the impeller's inlet and outlet dimensions and overall performance using one-dimensional semi-empirical calculations. While Vista CPD generates a one-dimensional geometric model without blade thickness, exporting it to ANSYS BladeGen further enhances the model. BladeGen creates

a three-dimensional (3D) model by adding blade thickness, providing a more accurate representation of the impeller's geometry. This is illustrated below in Figure 2.

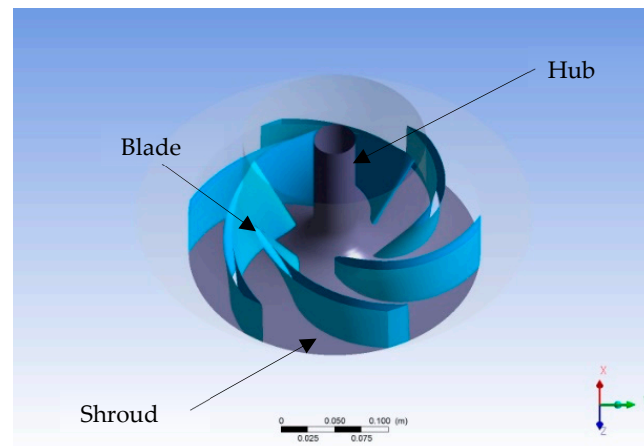


Figure 2. The isometric 3D view of the blade geometry.

Afterwards, the 3D model was transferred to Ansys DesignModeller, enabling the introduction of parametric variations in impeller features. This involved adjusting parameters such as the number of blade sets, impeller inlet diameter, impeller outlet diameter, ellipse ratio at the hub, ellipse ratio at the shroud, and the positioning of the leading and trailing edges to explore different design configurations. The parameters under consideration in this study included the number of blades (z), inlet impeller diameter ($2r_1$), and outlet impeller diameter ($2r_2$). These parameters, except for the number of blades, are depicted in Figure 3.

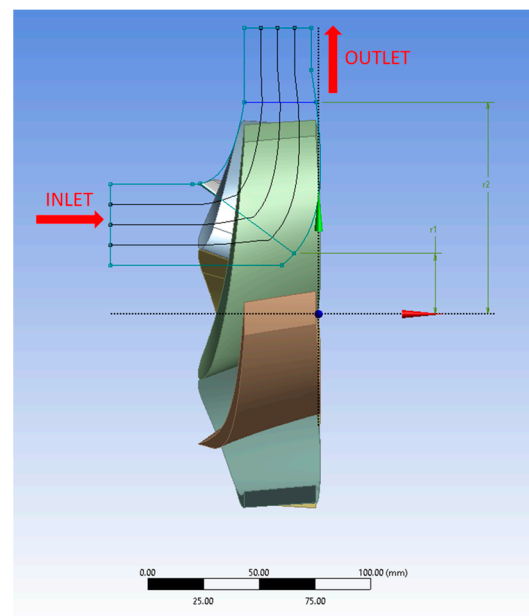


Figure 3. The meridional view of the impeller showing study parameters r_1 and r_2 .

The specification of the initial impeller geometry of the centrifugal pump is summarized in Table 2. This detailed overview serves as a valuable reference for understanding and analyzing the pump's primary components.

The 3D geometric model was exported to Ansys TurboGrid, which specializes in generating high-quality hex-mesh around turbomachinery blade rows. Proper meshing is crucial for accurate and efficient CFD simulations. The mesh quality will be checked and

refined to ensure optimal simulation accuracy and convergence. The meshing process will be conducted in a way that captures the intricate flow patterns and phenomena occurring within the pump. Figure 4 represents the generated mesh element of the pump impeller.

Table 2. Centrifugal pump impeller specification.

Parameters	Unit	Value
Impeller inlet diameter, d_1 ($2 \times r_1$)	mm	54.3
Impeller outlet diameter, d_2 ($2 \times r_2$)	mm	190.2
Impeller blade width at leading edge	mm	27.125
Impeller blade width at outlet (trailing edge)	mm	33.3
Impeller eye diameter	mm	116.4
Impeller hub diameter	mm	21.7
Impeller tip diameter	mm	280
Blade inlet angle at hub	°	27
Mean blade angle	°	19
Blade angle at exit	°	22.5
Number of blades, z	nos	6

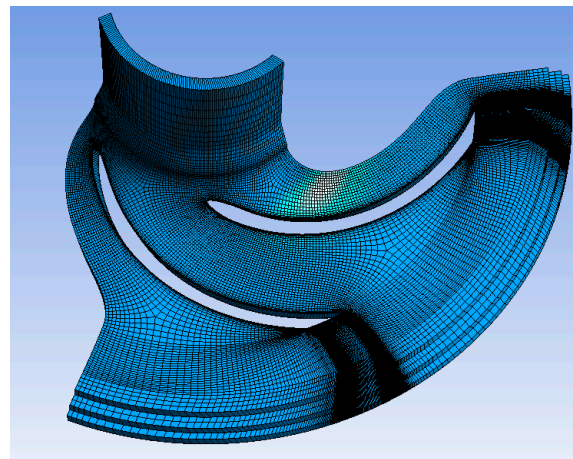


Figure 4. Mesh elements at span 50 view.

Mesh Independence Study

A mesh independence study was conducted to validate that the specific mesh used did not influence the CFD results, based on the principle that results should converge to a consistent value with successive mesh refinement. Increasing the number of nodes and mesh elements was expected to demonstrate consistency and mesh independence. The size factor and factor ratio in Turbogrid were adjusted to modify the number of elements and nodes, with higher factor ratios enabling near-wall mesh refinement but also carrying the risk of degrading mesh quality [12]. Due to resource constraints, only the head parameter was included in this study, with a minimum factor ratio of 1.5 chosen, varied at intervals of 0.2, and the size factor ranging between 1.1 and 1.4 to achieve convergence criteria below 1%, as per previous studies on mesh convergence [13]. These adjustments aimed to ensure the mesh independence and convergence of the CFD results. Table 3 summarizes this study.

Table 3. Mesh independence study result.

Size Factor	Factor Ratio	Nodes	Elements	Head (m)	% Change
1.1	1.5	275,859	248,092	12.0	0
1.2	1.7	424,080	386,640	11.924	0.6374
1.3	1.9	617,610	568,752	12.04	0.9634
1.4	2.1	798,954	741,228	12.028	0.0998

The analysis of the results reveals a minimal % change in comparison to the previous mesh setting, and the head remained consistently at approximately 12 m during the mesh variations, indicating the independence of the CFD results from the mesh. Subsequent analysis incorporated a mesh setting utilizing a size factor of 1.2 and a factor ratio of 1.7, aligning with industry best practices to capture intricate fluid behaviours. The selected mesh parameters effectively improved the simulation accuracy, demonstrating the enhanced resolution of complex flows in critical areas, emphasizing the crucial role of optimized mesh settings for robust CFD analyses while maintaining a balance between detail and computational efficiency.

2.1.2. Numerical Analysis

The meshed model will be imported into ANSYS CFX for CFD analysis, as it is well suited for turbomachinery and offers extensive capabilities for 3D models. ANSYS Fluent is an alternative, particularly for axisymmetric models and 2D simulations. CFX is preferred for its user-friendly interface, powerful algorithm, and flexibility with meshes, making it a top choice for dynamic flow analysis [14]. The flow is assumed to be incompressible and turbulent, and the $k-\omega$ Shear-Stress Transport (SST) Turbulence Model viscous model was selected, as it is well suited for turbomachinery simulations.

Steady-state simulation will be performed using a pressure-based solver suitable for analyzing centrifugal pump performance. The simulations should account for the impeller's rotation and the complex fluid dynamics occurring within the pump. The sliding mesh technique will capture the collaboration between the impeller and the volute, allowing for dynamic mesh deformation as the impeller rotates. The fluid used in the simulation will be wastewater having a temperature of 25 °C, inlet pressure of 1 atm, and 25 kg/s mass flow rate at the outlet. The dynamic viscosity will be adjusted to 100 centipoises (0.1 Pa.s) to be typical of wastewater properties [15]. All other properties were assumed to be of standard water, even though wastewater tends to have different fluid properties from standard water. This simplification allows for the simulation to be achieved easily, and it is considered that a more viscous fluid fairly simulates wastewater properties. It is assumed that at the point of pumping, the suspended particles will not be of significant quantity to require multiphase analysis, and most wastewater is dissolved. Hence, it will mainly affect viscosity. Table 4 provides a systematic rundown of the settings and conditions for the parameters utilized during the simulations conducted in this study.

Table 4. Simulation conditions used in this study.

Description	Parameter/Value Setting
Analysis Type	Steady-State Flow Analysis
Turbulence Model	Shear-Stress Transport (SST) Model
Fluid Temperature (°C)	25
Inlet Pressure (atm)	1
Mass Flow Rate (kg/s)	25
Dynamic Viscosity (centipoise)	100
Computation Algorithm and Discretization Method	Finite Volume Method with First-Order Upwind Scheme

2.1.3. Data Visualization

The simulation results were analyzed using ANSYS CFD-Post to interpret the performance of the pump under various operating circumstances. Post-processing will involve visualizing flow contours, pressure distributions, velocity profiles, and other relevant parameters to gain deeper insights into the pump's behaviour. Comparisons between simulation results and literature data or experimental measurements will be made to validate the accuracy of the numerical simulations. Sensitivity analyses will also determine the impact of various design parameters on the pump's performance.

Cavitation computation can be conducted using the Rayleigh–Plesset equation, which outlines the growth and collapse of vapour bubbles under pressure. Equation (1) presents the Rayleigh–Plesset equation that delineates the growth of gas bubbles in a liquid. Utilizing this equation enables the establishment of a correlation describing the rate of cavitation in relation to fluid properties and the local vapour pressure difference.

$$R_B \frac{d^2 R_B}{dt^2} + \frac{3}{2} \left(\frac{dR_B}{dt} \right)^2 + \frac{2\sigma}{\rho_f R_B} = \frac{p_v - p}{\rho_f} \quad (1)$$

where

R_B = bubble radius

σ = fluid surface tension coefficient

p = density

ρ_f = fluid density

p_v = vapour density

In ANSYS CFX, a cavitation model has been implemented to model both cavitation's condensation and vaporization stages. The CFX model allows for the visualization of cavitation bubbles to validate NPSH performance. However, the flow analysis involved is a two-phase one, implying that additional equations must be solved, rendering the cavitation study time-consuming and expensive. Fortunately, an alternative approach, known as cavitation inception analysis, can be performed without the need to apply the cavitation model. This method allows for the swift analysis of the pump's cavitation inception and hydraulic performance.

Visual cavitation inception within an impeller takes place when any region of the impeller registers a local minimum static pressure (P_{min}) that equals the fluid vapour pressure (P_v). The NPSH value corresponding to this phenomenon is referred to as inceptive NPSH or (NPSHi). According to Gülich [16], with a gradual reduction in the inlet pressure below the NPSHi value, the length of the cavity increases until the cavitation regions become large enough to affect the work transfer adversely. The NPSHi in question is generally higher than the required NPSH or (NPSHR). It should be noted that the NPSHR describes the given amount of cavitation needed to generate a specified variation in pump performance [17].

A steady-state, single-phase analysis can estimate cavitation inception in ANSYS CFX. To monitor a point for cavitation inception, the expressions described by Equation (2) can be used.

$$NPSHi = \frac{P_{in} - P_{min}}{\rho * g} \quad (2)$$

where

$NPSHi$ = incipient net positive suction head

P_{in} = pressure at the inlet in Pascal (Pa)

P_{min} = local minimum static pressure in Pascal (Pa)

ρ = fluid density in m^3/kg

g = gravity, $9.81 \text{ m}^2/\text{s}$

2.2. CFD Model Validation

The numerical model's accuracy was confirmed by cross-referencing a similar study conducted by Shojaeefard et al. [18]. The study used the 65–200 model manufactured by Pump Iran Co. in Tehran, Iran, which features an impeller with a 209 mm outside diameter and six backward-curved blades. An approximate representation of the pump was created in ANSYS, and CFD analysis was conducted. The fluid used in both experimental and numerical simulations was water, with a density of $998 \text{ kg}/\text{m}^3$ and a kinematic viscosity of $43 \text{ mm}^2/\text{s}$. Inlet boundary conditions entailed a total pressure of 1.4687 atm. (15.2 m) at a flow rate of 0, while outlet boundary conditions involved a mass flow rate ranging from 0

to 70 m³/h. The gathered pump performance data were compared with the experimental data from Shojaeefard et al. [18], and the findings are presented in Figure 5.

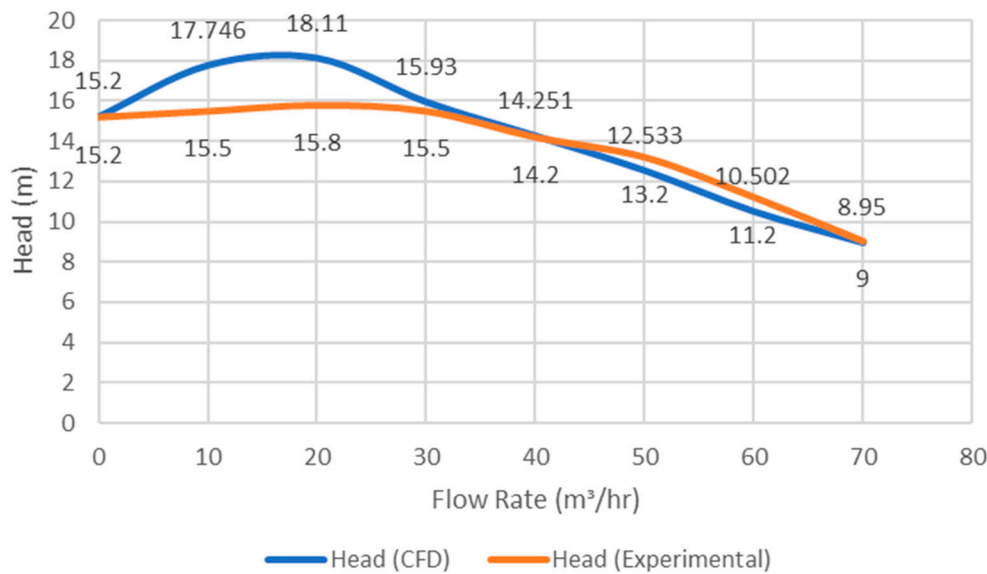


Figure 5. Comparison of pump head from experiment and CFD simulation.

Upon analyzing the experimental and numerical data for head values, it was determined that the average percentage error was 5.5%. These results suggest a satisfactory correlation between the numerical (CFD model) and experimental findings.

2.3. Optimization Study

The general workflow of response surface optimization (RSO) in ANSYS Workbench for the pump impeller geometry is illustrated in Figure 6.

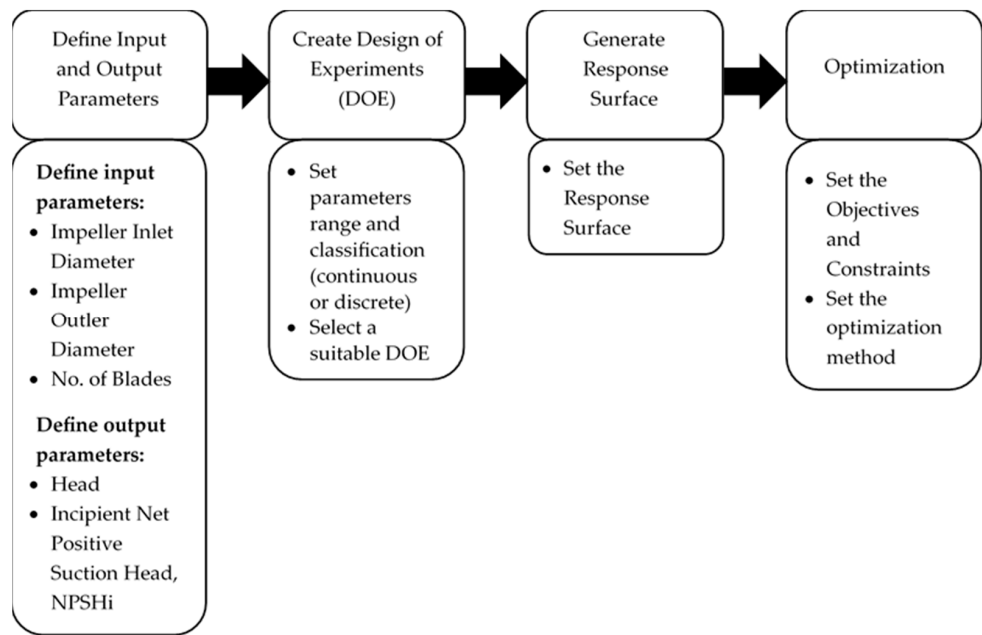


Figure 6. General workflow in ANSYS Workbench for optimization of centrifugal pump.

2.3.1. Study Variables

The set input parameters were the impeller inlet diameter ($2 \times r1$), the impeller outlet diameter ($2 \times r2$), and the number of blades (z). Thus, parameters $r1$ and $r2$ were used to

represent inlet and outlet radii for similarity and simplicity. The output parameters were set as the head, the static efficiency, and the incipient net positive suction head (NPSHi). Table 5 summarizes the study input variables and range used in this study.

Table 5. Input parameters for the optimization study.

Input Parameters	Unit	Lower Bound	Upper Bound	Classification
r1	mm	24	30	Continuous
r2	mm	85	104	Continuous
z	nos	3	8	Continuous, manufacturable

2.3.2. Type of Design of Experiments (DOE) for Study

The design of experiments (DOE) type for this study was set as CCD, Face-Centred, to establish the design matrix table initially. The Custom + Sampling design was later used to edit the design matrix table to only include design points for which the value of the number of blades specified is a whole number. This manipulation allowed us to work with the number of blades as a continuous variable for better results when working with the optimization algorithm instead of setting it as a discrete variable, which would have increased the computation time.

2.3.3. Type of Response Surface for the Study

Genetic Aggregation is more reliable and offers superior response surface quality than the other types. In addition, it is suited for multi-objective optimization, as is the case in this study. Therefore, Genetic Aggregation was selected as the response surface type for this study. The tolerance for the response surface was set as indicated in Table 6. This was decided based on the required accuracy of the output parameters. For instance, it was assumed that head and NPSHi can be within ± 0.5 m and static efficiency as $\pm 1\%$. This is important for the results' convergence. Choosing higher values would reduce accuracy but provide faster convergence. Lower values would increase accuracy but might make convergence impossible, or it would take a long time to converge.

Table 6. Response surface tolerances.

Output Parameters	Unit	Tolerance
Head	m	0.5
Static Efficiency	%	1
NPSHi	m	0.5

2.3.4. Objectives and Constraints

The optimization study aims to identify the optimal design parameters that will maximize pump efficiency, minimize cavitation, and enhance overall pump performance. The main objective function in the optimization process is to increase the performance of the centrifugal pump base model used in this study in terms of head, static efficiency, and incipient net positive suction head (NPSHi). Constraints were established to ensure that the optimized design adheres to practical considerations, such as structural limits (maintaining the volute size of the pump) and reducing the computation time of ANSYS 2024 R1 software. Additionally, cavitation and NPSHi constraints were taken into account to prevent potential damage to the pump and ensure its reliable operation. The design objectives and constraint settings are summarized in Table 7, based on the base model's performance. For instance, the base model exhibited a static efficiency of approximately 55%, a head of 11.925 m, and an NPSHi of 2.78 m. To maximize these parameters, the targets were set to exceed the performance values of the base model.

Table 7. Centrifugal pump's optimization parameter limits.

Parameter	Type	Objectives Target	Constraints
Head	Maximize	15 m	Head $\geq$ 10 m
NPSHi	Maximize	4 m	NPSHi $\geq$ 2.5 m
Static Efficiency	Maximize	60%	Static Efficiency $\geq$ 55%

The ANSYS optimization methods available for response surface optimization (RSO) include Screening, Multi-Objective Genetic Algorithm (MOGA), Nonlinear Programming by Quadratic Lagrangian (NLPQL), and Mixed-Integer Sequential Quadratic Programming (MISQP). The MOGA method was chosen due to its suitability for this study's multi-criteria nature and its ability to provide an excellent solution. The MOGA is a variant of Non-dominated Sorted Genetic Algorithm-II (NSGA-II) that operates in accordance with controlled elitism concepts. This method enables users to execute an optimization study involving multiple objectives and constraints to identify the global optimum. Through an iterative assessment of various design configurations, the optimization process aims to converge towards the optimal solution that meets the specified objectives while complying with the defined constraints.

3. Results and Discussion

This section interprets and evaluates the outcomes of the CFD simulations, shedding light on the functional aspects of centrifugal pumps as it pertains to fluid flow, pressure, velocity profiles, efficiency, and associated performance metrics. This is achieved through a systematic examination of the obtained data, comparison with theoretical formulations or experimental benchmarks, and critical interpretation of the implications of the findings on centrifugal pump design and optimization.

3.1. Base Model Results

In this study, the base model's centrifugal pump impeller performance results were generated, as summarized in Table 8.

Table 8. Centrifugal pump impeller performance results.

Parameters	Unit	Value
Rotation Speed	radians/s	153.5190
Reference Diameter	m	0.1900
Volume Flow Rate	m ³ /s	0.0251
Head	m	11.925
Flow Coefficient	-	0.0241
Head Coefficient	-	0.1406
Shaft Power	Watts	3882.83
Power Coefficient	-	0.0045
Static Efficiency	%	54.279
NPSHi	m	2.78

From the above table, the head generated by the design is 11.925 m with a static efficiency of 54.279% and NPSHi of 2.78 m.

3.1.1. Blade-to-Blade Plot

The blade-to-blade plot offers flow parameter visualization along a given rotating component span, which is the span of the centrifugal pump impeller. A contour plot of the total pressure (Pt) distribution along the impeller at a 50% radius position was generated, as shown in Figure 7. The total pressure variation along the impeller span can be seen. This plot indicates how the pump imparts energy to the fluid during the flow through the

impeller blades. A smooth and gradual increase in total pressure indicates efficient energy transfer. This can be seen to be the case, but there are pockets of sudden increases that are likely to result in reduced efficiency. From the plot, areas of higher blade loading can be seen as depicted by higher pressure gradients. Higher pressure gradients are undesirable since they may result in flow separations.

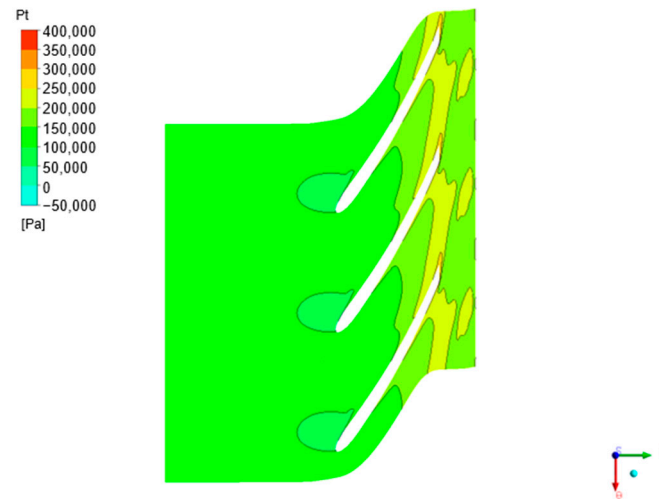


Figure 7. Contour of total pressure at 50% span.

3.1.2. Velocity Vector Plot

Velocity vector plots were also generated. Figure 8 shows a velocity vector plot at 50% radial position. From the plot, the flow path visualization is clear, and the pump seems to be guiding the fluid well through the intended path. The inlet and outlet conditions can also be seen to show uniform inlet flow and well-distributed flow across the diffuser. However, areas of stagnation points can be seen in the blue regions. This is indicative of potential flow separation and recirculation. These areas also happen to show a higher pressure gradient, as shown in Figure 7.

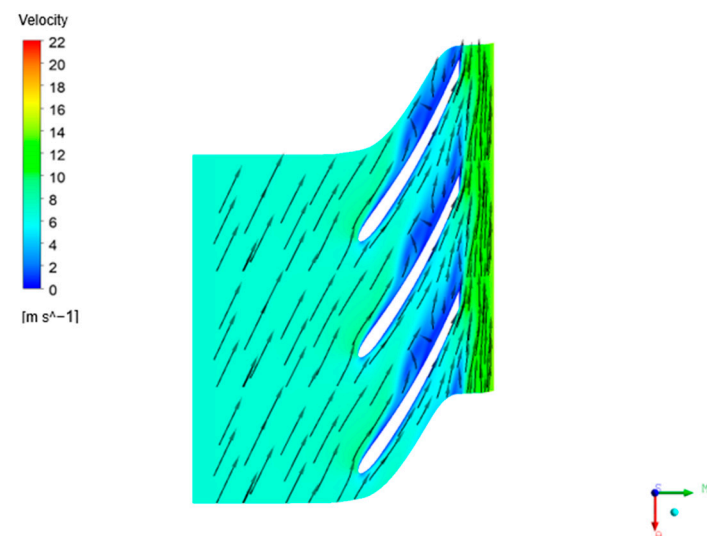


Figure 8. Velocity vectors at 50% span.

3.1.3. Meridional Plot

Insights into a pump's aerodynamic performance and characteristics can be obtained from a meridional plot indicating the vector of C_m (moment of coefficient) along a merid-

ional surface. Figure 9 presents the meridional C_m vector plot generated for impeller design.

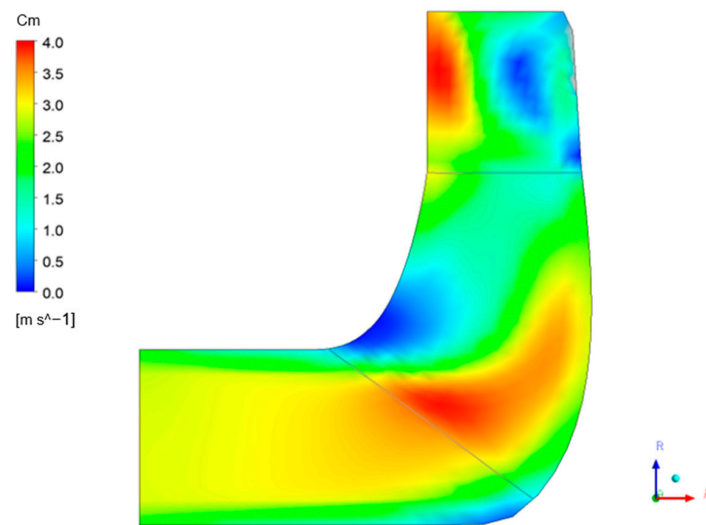


Figure 9. Vector of C_m (moment of coefficient) on meridional surface view.

Figure 9 shows noticeable deviations of specific velocity vectors from the meridional direction, which may impact the flow development. The observed deviations could lead to flow disruptions and inefficiencies within the system. It is critical to address these deviations to promote a more uniform and well-developed flow profile, ultimately contributing to improved system performance and efficiency. The blade-to-blade plot offers flow parameters.

3.1.4. Velocity Streamlines

The streamlines' plot illustrates the flow patterns inside the pump impeller. The plot in Figure 10 reveals the fluid movement as it enters the impeller, interacts with the blades, and exits towards the trailing edge. It is apparent from the plot that the flow deviates slightly from the blade profile along the impeller passage, resulting in a somewhat irregular pattern. Additionally, occurrences of streamlines' convergence are observed, suggesting the presence of flow separation.

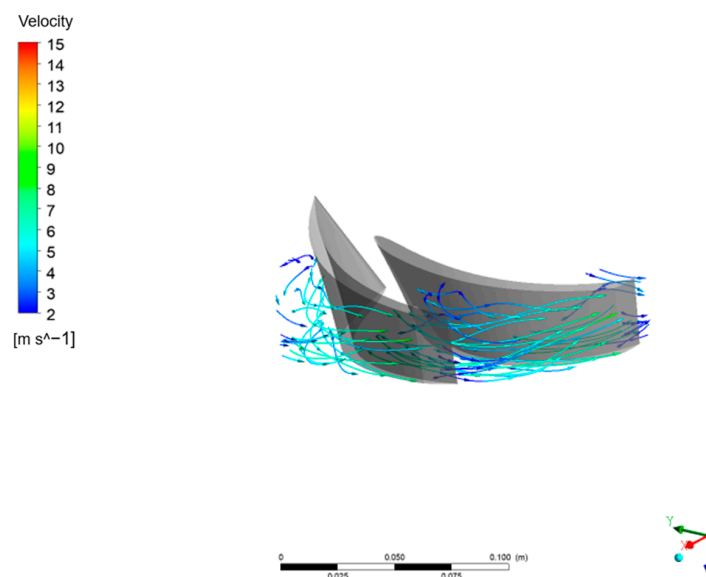


Figure 10. Velocity streamlines stream blade, trailing edge view.

3.2. Optimization Study Result

The results of an optimization study provide valuable insights into the hydrodynamic behaviour, flow patterns, and design modifications that can lead to improved pump performance. The optimization study typically involves the exploration of various design parameters, impeller geometries, and operational conditions to achieve specific engineering targets and performance objectives.

3.2.1. Design of Experiments (DOE) and Response Min–Max Results

The DOE study comprised 15 design points as the initial simulations ran, as indicated in Table 9. A mesh setting was configured with a resolution defined by a size factor of 1.2, which determined the scale of the individual mesh elements, and a factor ratio of 1.7, dictating the proportional variance between the sizes of adjacent mesh elements, to optimize the balance between simulation accuracy and computational resource requirements.

Table 9. Design of experiments of this study.

Design Point	r1 (mm)	r2 (mm)	z	Head (m)	Static Efficiency (%)	NPSHi (m)
1	27	94.5	6	11.7351	54.693	2.68927
2	24	94.5	6	11.7643	55.4117	2.38229
3	30	94.5	6	11.4983	53.7174	3.2636
4	27	85	6	7.74435	52.0232	2.29063
5	27	104	6	15.4833	55.7908	3.29158
6	27	94.5	3	10.512	50.8887	7.08343
7	27	94.5	8	11.7138	53.6082	2.20261
8	24	85	3	7.03004	51.2447	5.06267
9	30	85	3	6.45182	45.9608	6.73263
10	24	104	3	12.9427	49.574	9.61455
11	30	104	3	13.3323	50.2296	9.59621
12	24	85	8	7.50283	50.6726	3.05663
13	30	85	8	6.79925	47.4614	2.84858
14	24	104	8	16.0191	54.845	2.37666
15	30	104	8	15.5632	52.5729	2.69759

Figure 11 shows a graphical representation of design points, depicted by input parameters such as inlet and outlet impeller diameter and impeller blade number. These points are assessed for outputs like pump head, static efficiency, and incipient net positive suction head (NPSHi), identifying optimal pump configurations and critical interrelations. The graph reveals how changes in geometric and blade parameters affect output sensitivity, pinpointing areas for potential design improvements to enhance pump efficiency and operation.

The design points were first generated using the Central Composite Design (CCD) but later switched to Custom + Sampling to ensure blade numbers were whole. This change avoided fractional blade numbers and preserved the use of varied algorithm methods without categorizing blade numbers as discrete.

The response surface was created using the Genetic Aggregation method, which allows the algorithm to determine and include refinement points based on specified tolerances. However, due to the algorithm's tendency to generate blade numbers as decimals, it was crucial to closely monitor the refinement process, frequently pausing the solver to adjust the assigned number to a whole number manually. This required a time-consuming and meticulous effort.

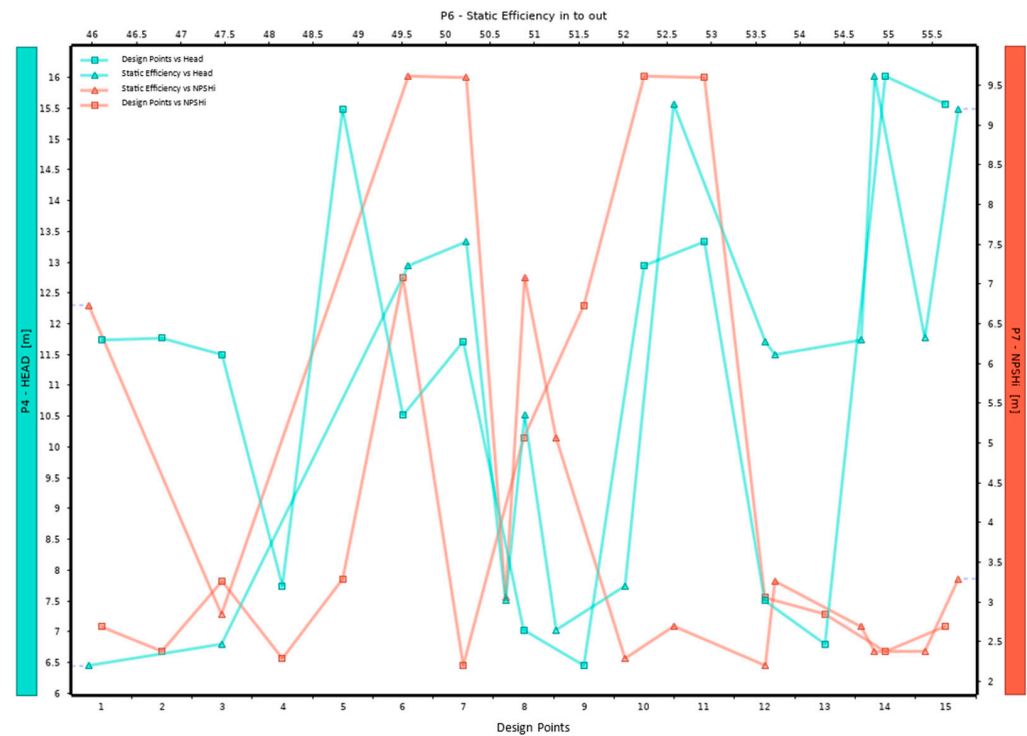


Figure 11. A chart depicting the design of experiments of this study.

The min–max search functionality was applied to determine the minimum and maximum responses obtained in the study. The min–max search results for the study are shown in Table 10.

Table 10. Study minimums and maximums.

Description.	r1 (mm)	r2 (mm)	z	Head (m)	Static Efficiency (%)	NPSHi (m)
Output Parameter Minimums						
Head	30	85	3	6.46	46.14	6.79
Static Efficiency	30	85	3	6.46	46.14	6.79
NPSHi	24	92.13	7	11.12	55.37	1.80
Output Parameter Maximums						
Head	24	104	8	16.01	54.37	2.50
Static Efficiency	24.23	95.10	4	11.51	57.31	4.98
NPSHi	26.42	104	3	13.14	48.80	9.62

The above table shows that at a maximum head of 16.01 m, the static efficiency and NPSHi are 54.37% and 2.50 m, respectively. At a maximum static efficiency of 57.31%, the head and NPSHi are 11.51 m and 4.98 m, respectively. Similarly, at the peak NPSHi of 9.62 m, the head and static efficiency are 13.14 m and 48.80%, respectively. These data reveal the rollercoaster nature of pump performance in maximizing the objectives, underscoring the need for optimization to balance the often contrasting objectives and requirements.

3.2.2. Local/Parameter Sensitivity

Local sensitivity charts show the changes in output parameters in response to each input parameter being varied independently. A positive value on the sensitivity chart implies that the output parameter is positively correlated with the input parameter. Increasing input will increase the output and contrariwise. A negative value implies that when the input increases, the output reduces and vice versa.

Figure 12 shows that the inlet impeller diameter has a marginal inverse relationship with pump head, while the outlet impeller diameter and blade number have a positive correlation with the head, with the outlet impeller diameter having the most significant impact. In terms of static efficiency, the outlet impeller diameter and blade number are positively correlated, while the inlet impeller diameter is inversely correlated. The number of blades has the most substantial impact on static efficiency. Furthermore, the NPSHi is negatively affected by the number of blades and positively correlated with the inlet and outlet impeller diameters, with the number of blades having the most significant impact. Overall, the analysis reveals that the outlet impeller diameter and blade number are the most critical variables in pump performance. These conclusions are consistent with the findings of Hlaing et al. [19].

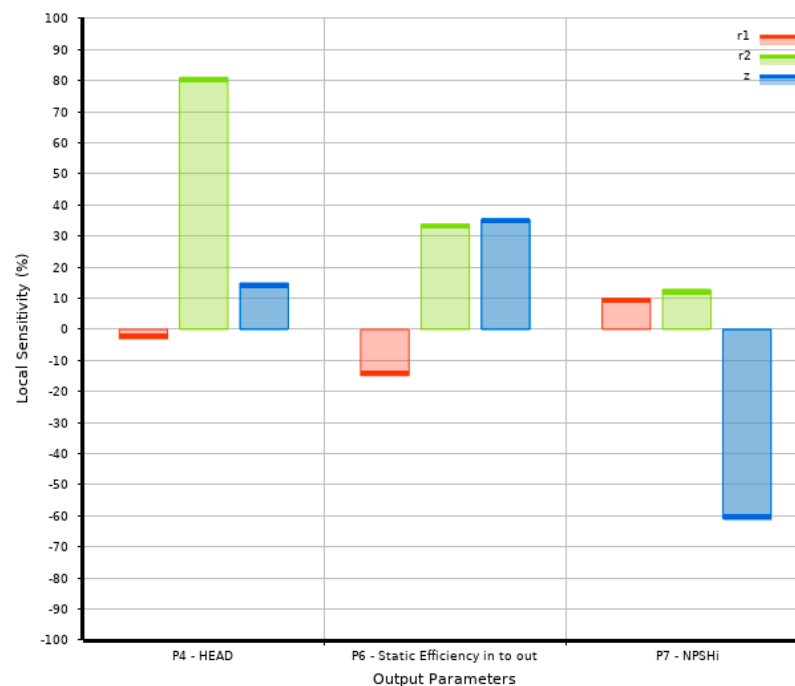


Figure 12. Parameter sensitivities chart.

3.3. Response Surface

The analysis of response surfaces for head, static efficiency, and NPSH can involve presenting 2D cross-sections. Two-dimensional cross-sections are recommended for conveying the behaviour of parameters.

3.3.1. Pump Head

The plots of the variation in the pump head with inlet and outlet impeller radii were generated for various number of blades, as shown in Figures 13 and 14.

Variation in head with inlet impeller diameter

From the Figure 13, it is evident that for all the evaluated blade numbers except for $z = 3$, an increase in the inlet impeller diameter decreases the pump head generated. For blade number 3, the head is lower at a lower inlet impeller diameter but peaks at a value around 54 (27×2) mm, beyond which any further increase results in a lower head.

The number of blades significantly influences the pump head. The plot shows that for each impeller geometry, there is an optimum number of blades for the maximum head to be attained. This observation is in line with findings by Chakraborty and Pandey [7].

Variation in head with outlet impeller diameter

As depicted in Figure 14, the graph illustrates how the pump head generated changes in response to the outlet impeller diameter variation, specifically within the range of three to eight blades. The data indicate that an increase in the outlet impeller diameter corresponds

to a notable increase in pump head. This insight suggests that the outlet impeller diameter significantly influences the pump's head performance within the specific range of the number of blades.

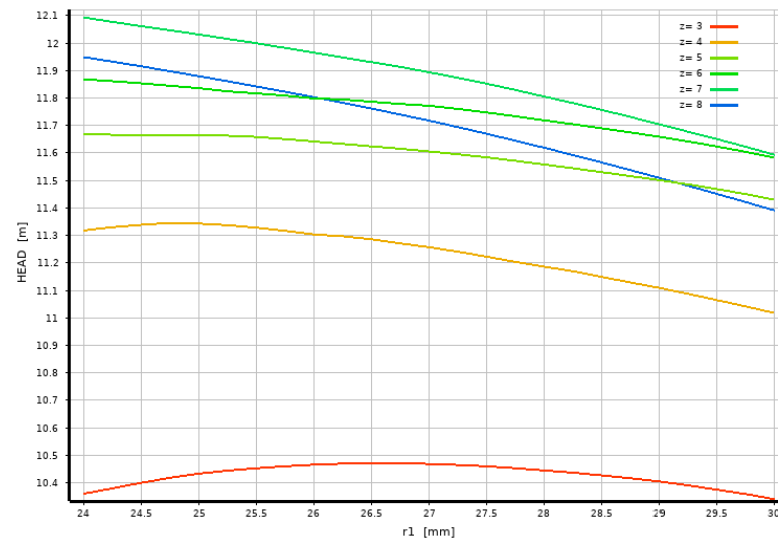


Figure 13. Variation in head with inlet impeller radius for different number of blades.

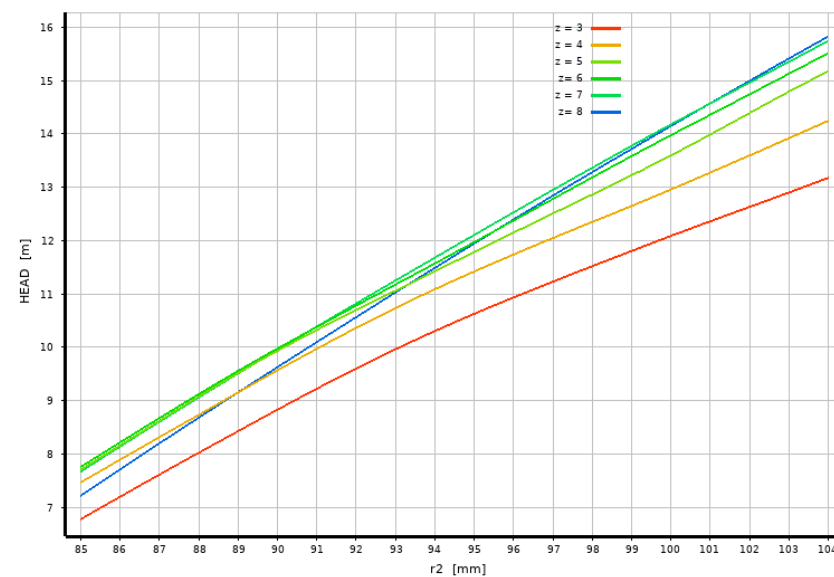


Figure 14. Variation in head with outlet impeller radius for different number of blades.

The observation depicted in Figure 14 aligns with the conclusions drawn by Chakraborty and Pandey [20] regarding the critical significance of the outlet impeller diameter in centrifugal pump performance. Adu et al. [21], in their investigation of centrifugal pump performance, also arrived at similar findings. Therefore, this study's conclusions support the existing literature, emphasizing that both the inlet and outlet impeller diameters substantially impact pump performance.

3.3.2. Static Efficiency

The variation in the static efficiency with the inlet and outlet impeller radii plots was generated and is presented in the figures below.

Variation in static efficiency with inlet impeller diameter

Figure 15 shows irregular variation in the static efficiency with the inlet impeller diameter, with the general sense being that increasing the inlet impeller diameter would

decrease static efficiency for all numbers of blades evaluated. The highest static efficiency of about 57% was recorded for $z = 4$ at an inlet radius of about 24 mm.

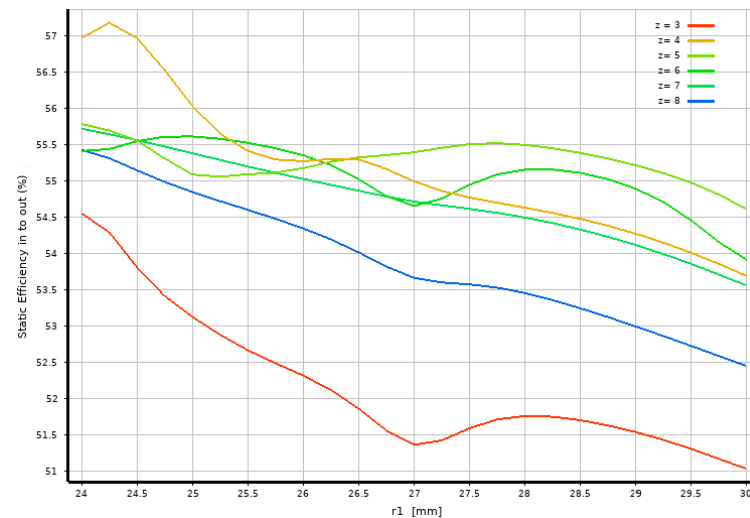


Figure 15. Variation in static efficiency with inlet impeller radius for different number of blades.

Variation in static efficiency with outlet impeller diameter

Figure 16 illustrates that the static efficiency of the centrifugal pump is impacted by changes in the outlet impeller radius, with a notable increase in static efficiency observed as the outlet radius is raised from 85 mm to its peak at 97–98 mm, followed by a subsequent decline as the radius surpasses this range. The static efficiency of a centrifugal pump is intricately linked to the outlet impeller diameter, as this crucial parameter influences the velocity and static pressure characteristics within the pump system [3]. Initially, larger outlet diameters boost static efficiency by improving fluid outflow. But continued increases in outlet size cause faster flow and more turbulence, leading to greater energy losses and reduced efficiency.

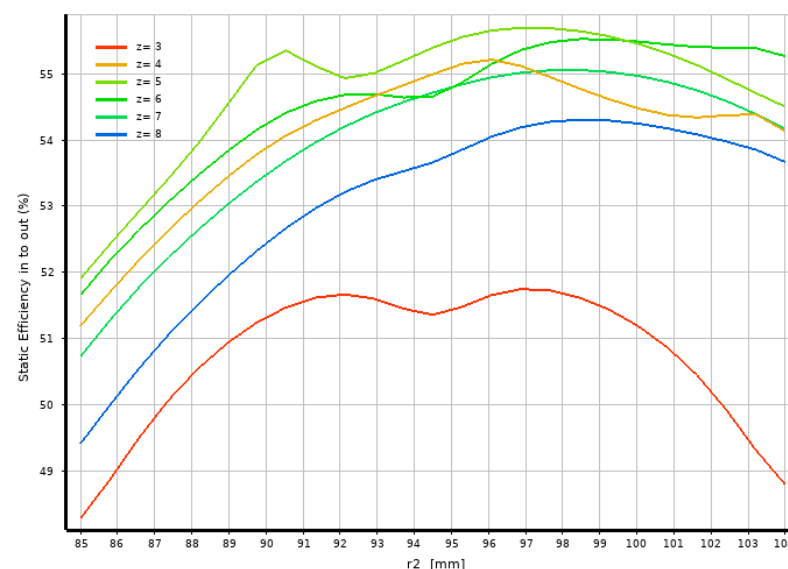


Figure 16. Variation in static efficiency with outlet impeller radius for different number of blades.

3.3.3. Incipient Net Positive Suction Head (NPSHi)

The impeller performance with respect to cavitation was studied by considering the variation in the NPSHi with the inlet and outlet impeller diameter, as shown by Figures 17 and 18.

Variation in NPSHi with inlet impeller diameter

Figure 17 shows a slight increase in the NPSHi with a larger inlet impeller radius within 24–30 mm, supporting findings from Hlaing et al. [19] that a bigger inlet can reduce cavitation. A higher NPSHi suggests better cavitation resistance, highlighting its importance for pump performance and cavitation prevention.

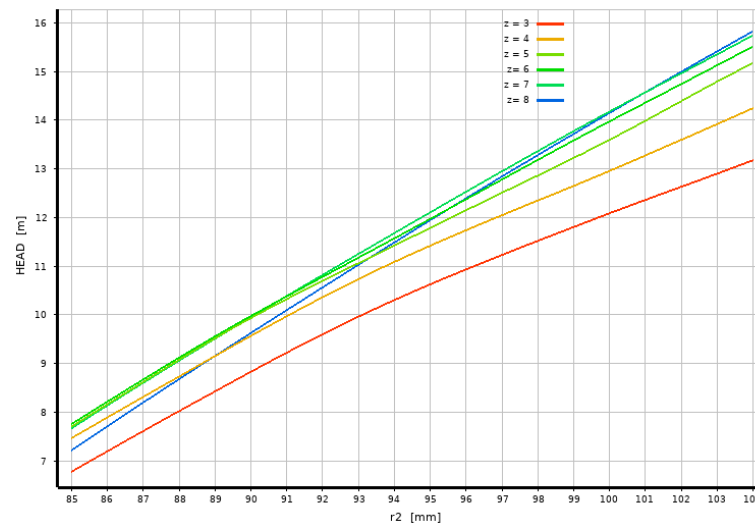


Figure 17. Variation in NPSHi with inlet impeller radius for different number of blades.

Variation in NPSHi with outlet impeller diameter

Figure 18 indicates that increasing the outlet impeller radius resulted in an increase in the NPSHi for all blade configurations evaluated except for $z = 8$, which shows a decrease within the 85–95 mm region and then an increase. As the outlet impeller diameter increases, the velocity of the fluid decreases, which can increase the static pressure and, hence, the NPSHi. This means the pump is less likely to cavitate, improving its performance. The highest NPSHi was recorded for a configuration of three blades. An increase in the number of blades appears to result in reduced NPSHi, which implies an increase in cavitation.

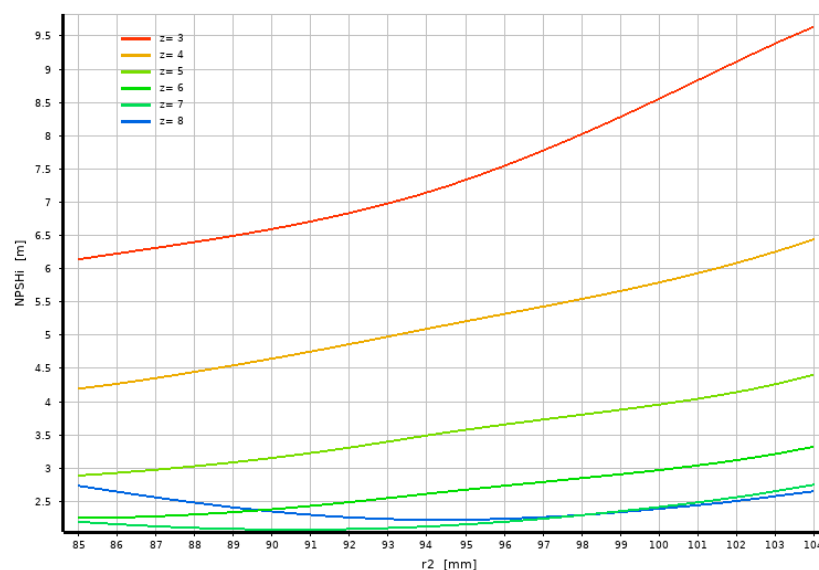


Figure 18. Variation in NPSHi with outlet impeller radius for different number of blades.

3.4. Optimization

The optimization study returned three candidates' points, as presented in Table 11.

Table 11. Optimal three candidates.

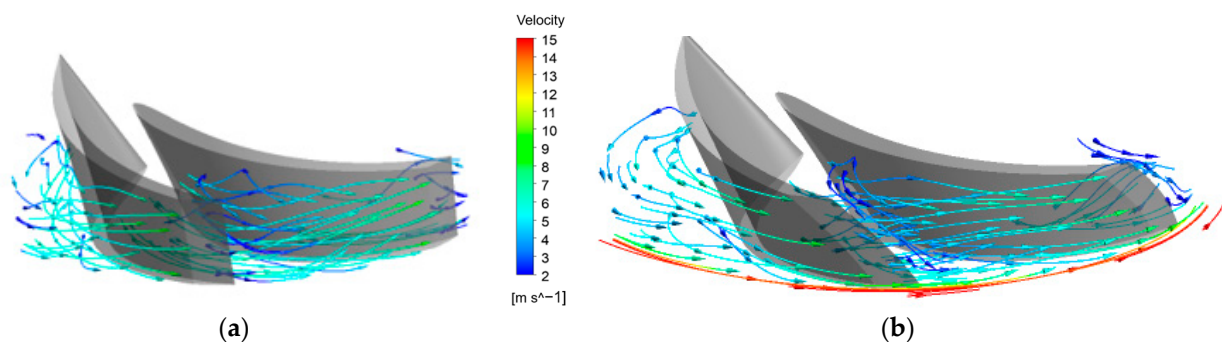
Description	r1 (mm)	r2 (mm)	z	Head (m)	NPSHi (m)	Static Efficiency (%)
Candidate Point 1	27.985	103.79	5	15.047	4.5015	55.674
Candidate Point 2	27.999	103.84	5	15.064	4.5106	55.664
Candidate Point 3	27.96	103.89	5	15.086	4.512	55.651

From the three candidates outlined in Table 11, there are minimal discrepancies among them, and any of them could be regarded as the optimal solution. Selecting Candidate 3, Table 12 details the design parameters and records the enhancements attained through the optimization process.

Table 12. Base model design vs. optimal solution design.

Parameters	Base Model Design	Optimal Solution Design	Improvement (%)
r1 (mm)	27.125	27.96	-
r2 (mm)	95.103	103.89	-
Number of Blades, z	6	5	-
HEAD (m)	11.925	15.086	26.51
NPSHi (m)	2.78	4.512	62.30
Static Efficiency (%)	54.279	55.651	2.53

From the above table, significant improvement in the head and NPSHi was achieved by 26.51% and 62.30%, respectively. Static efficiency only saw an improvement of 2.53%. Figure 19 shows a plot of velocity streamlines at the blade trailing edge that was generated for the optimal design and compared with the base model design.

**Figure 19.** A comparison between the (a) base design and (b) optimized design with respect to the velocity streamlines at the blade trailing edge.

4. Conclusions

The novelty of this study lies in its simultaneous focus on improving pump head, static efficiency, and particularly NPSHi—a critical cavitation indicator previously overlooked in the optimization of pump design variables like impeller diameters and blade quantity. This study, incorporating fluid viscosity adjustments, sought to validate design effectiveness in constant operating conditions but remained within the scope of computational fluid dynamics (CFD) without empirical field validation. This study provided key insights into the influence of impeller geometry on centrifugal pump performance, particularly in sewage management applications. It was found that a larger inlet impeller diameter reduces pump head, while a larger outlet diameter boosts pump head and efficiency. The ideal number of impeller blades is crucial for maximizing performance and significantly affects the incipient net positive suction head (NPSHi). In terms of correlations, a larger

inlet impeller diameter negatively impacts pump head but enhances the NPSHi, whereas an increased number of blades improves pump head and efficiency but may reduce the NPSHi. The optimal design for the evaluated operating conditions of the pump was a 5-blade configuration with inlet and outlet diameters of 55.92 mm and 207.78 mm, leading to performance gains of 26.51% in head, 2.53% in efficiency, and 62.30% in the NPSHi.

5. Recommendation

Considering the ongoing study into an optimized impeller design for sewage treatment and other fluid-handling applications, it is recommended that future work includes comprehensive real-world implementation and validations. This should involve collaborative efforts with wastewater treatment facilities and industries with similar fluid dynamics. Detailed attention to the specific rheological properties of sewage and industrial fluids is crucial, as these can profoundly affect impeller performance. Long-term operational studies and robust field tests will provide the empirical evidence needed to confirm the reliability, efficiency, and overall benefits of the new impeller designs under various real-world conditions. Furthermore, exploring the design's scalability and cost-effectiveness is essential to encourage its adoption across different market segments.

Author Contributions: Conceptualization, E.P.; writing—original draft preparation, E.P.; writing—review and editing, J.H.; supervision, J.H.; project administration, J.H. All authors have read and agreed to the published version of the manuscript.

Funding: This research received no external funding.

Data Availability Statement: The original data presented in the study are openly available in FigShare at [dx.doi.org/10.6084/m9.figshare.25954831](https://doi.org/10.6084/m9.figshare.25954831) (accessed on 1 June 2024).

Conflicts of Interest: The authors declare no conflicts of interest.

References

1. Debnath, N.; Mahfuz-ud_Drain, K.; Mow, N. Spatial aspect of the wastewater management system: A case study of Khalishpur, Khulna. *Khulna Univ. Stud.* **2022**, *2022*, 397–407. [\[CrossRef\]](#)
2. Wang, H.; Tsukamoto, H. Fundamental analysis on rotor-stator interaction in a diffuser pump by vortex method. *J. Fluids Eng.* **2001**, *123*, 737–747. [\[CrossRef\]](#)
3. Liu, M.; Tan, L.; Cao, S. Influence of viscosity on energy performance and flow field of a multiphase pump. *Renew. Energy* **2020**, *162*, 1151–1160. [\[CrossRef\]](#)
4. Zhou, W.; Yu, D.; Wang, Y.; Shi, J.; Gan, B. Research on the fluid-induced excitation characteristics of the centrifugal pump considering the compound whirl effect. *FU Mech. Eng.* **2023**, *21*, 223–238. [\[CrossRef\]](#)
5. Mentzos, M.; Filios, A.E.; Margaris, D.; Papanikas, D. A numerical simulation of the impeller-volute interaction in a centrifugal pump. In Proceedings of the 1st International Conference from Scientific Computing to Computational Engineering, Athens, Greece, 8–10 September 2004.
6. Caridad, J.; Asuaje, M.; Kenyery, F.; Tremante, A. Characterization of a centrifugal pump impeller under two-phase flow conditions. *J. Pet. Eng.* **2008**, *63*, 18–22. [\[CrossRef\]](#)
7. Botia, G.P.; Ochoa, G.V.; Forero, J.D. CFD analysis of hydraulic performance in small centrifugal pumps operating with slurry. *IREMOS* **2019**, *12*, 364–372. [\[CrossRef\]](#)
8. Zhou, H.; Zhang, Q.; Gao, Q.; Cui, Z. Analysis of effects of impeller inlet width on the performance of centrifugal pump. *J. Chem. Pharm. Res.* **2014**, *6*, 2078.
9. Al-Obaidi, A.R.; Qubian, A. Effect of outlet impeller diameter on performance prediction of centrifugal pump under single-phase and cavitation flow conditions. *Int. J. Nonlinear Sci. Numer. Simul.* **2022**, *23*, 1203–1229. [\[CrossRef\]](#)
10. Gobran, M.H.; Ibrahim, M.; Shaltout, R.E. Numerical simulation of the dynamic effects due to impeller-volute interaction in a centrifugal pump. *J. Fluids Eng.* **2019**, *4*, 21–29.
11. Elyamin, G.R.A.; Bassily, M.; Khalil, K.; Gomaa, M.S. Effect of impeller blades number on the performance of a centrifugal pump. *Alex. Eng. J.* **2019**, *58*, 39–48. [\[CrossRef\]](#)
12. Chitrakar, S. FSI Analysis of Francis Turbines Exposed to Sediment Erosion. Master's Thesis, Kathmandu University, Kathmandu, Nepal, July 2013.
13. Baidar, B. Selection of Optimal Number of Francis Runner Blades for a Sediment Laden Micro Hydropower Plant in Nepal. *Int. J. Fluid Mach. Syst.* **2015**, *8*, 294–303. [\[CrossRef\]](#)
14. Iacob, N.; Dragan, N. Dynamic Analysis of a Centrifugal Pump using CFD and FEM Methods. *Hidraulica* **2019**, *4*, 31.
15. Wolski, P. Analysis of rheological properties of thickened sewage sludge. *Desalination Water Treat.* **2021**, *232*, 331–338. [\[CrossRef\]](#)

16. Gülich, J.F. *Centrifugal Pumps*, 1st ed.; Springer: New York, NY, USA, 2014; pp. 7–15.
17. Karassik, I.; Messina, J.; Cooper, P.; Heald, C. *Pump Handbook*, 3rd ed.; McGraw-Hill: New York, NY, USA, 2008; pp. 18–23.
18. Shojaeefard, M.H.; Tahani, M.; Ehghaghi, M.B.; Fallahian, M.A.; Beglari, M. Numerical study of the effects of some geometric characteristics of a centrifugal pump impeller that pumps a viscous fluid. *Comput. Fluids* **2012**, *60*, 65. [[CrossRef](#)]
19. Hlaing, N.N.; Win, H.H.; Htet, Z.M. Design and significant effect of a centrifugal pump by using various blade exit angles. *Int. J. Eng. Sci. Technol.* **2014**, *3*, 3018–3024.
20. Chakraborty, S.; Pandey, K.M. Numerical studies on effects of blade number variations on performance of centrifugal pumps at 4000 rpm. *Int. J. Eng. Sci. Technol.* **2011**, *3*, 410–416. [[CrossRef](#)]
21. Adu, D.; Du, J.; Darko, R.; Antwi, E.O.; Khan, M.A.S. Numerical and experimental characterization of splitter blade impact on pump as turbine performance. *Sci. Prog.* **2021**, *104*, 1. [[CrossRef](#)] [[PubMed](#)]

Disclaimer/Publisher’s Note: The statements, opinions and data contained in all publications are solely those of the individual author(s) and contributor(s) and not of MDPI and/or the editor(s). MDPI and/or the editor(s) disclaim responsibility for any injury to people or property resulting from any ideas, methods, instructions or products referred to in the content.