Numerical Study on the Effect of Blade Angle on the Performance Characteristics of a Centrifugal Pump

Muhammad Nur Salam[1], Wan Norhayatul Husna[1], Suzairin Md Seri[1]*

[1] Faculty of Mechanical and Manufacturing Engineering
Universiti Tun Hussein Onn Malaysia 86400, Parit Raja,
Johor, MALAYSIA

*Corresponding Author: suzairin@uthm.edu.my
DOI: https://doi.org/10.30880/rpmme.2025.06.01.041

Abstract

The main focus of our research is to analyze how different blade angles affects the performance of a centrifugal pump. By using numerical analysis we hope to gain a deeper understanding of how blade angle variations can affect the pump's efficiency, pressure increase, and cavitation. Our objectives revolve around examining the connection between blade angle and pump performance, determining the optimal blade angles to achieve better efficiency, and making valuable contributions to the field of pump design and optimization. The research methodology involves utilizing computational fluid dynamics (CFD) simulations to model and analyze the behavior of a centrifugal pump under different blade angles. This numerical analysis allows for a comprehensive exploration of fluid flow patterns, pressure distribution, and performance parameters. The study will employ established CFD software and systematically vary blade angles while monitoring and recording relevant performance data to draw meaningful conclusions about the impact of blade angles on the pump's characteristics. In conclusion, the numerical study reveals that blade angle significantly affects the performance characteristics of a centrifugal pump. Optimal blade angles were identified to enhance efficiency and pressure distribution,with specific recommendations for blade angle adjustments in pump design. Further experimental validation is recommended to validate these findings and implement improvements in pump design for increased efficiency and performance.

1. Introduction

Centrifugal pumps are widely used in various industries for fluid transportation, relying on the conversion of rotational kinetic energy into hydrodynamic energy. The efficiency and performance of these pumps are influenced by several factors, including the blade angle of the impeller. This study aims to investigate the impact of blade angle on the performance characteristics of a centrifugal pump, with the objective of identifying optimal design parameters[1].

The study investigates the influence of blade angle variations on the performance characteristics of a centrifugal pump, a vital component widely employed in fluid transportation across diverse industries. Centrifugal pumps operate on the principle of converting mechanical energy into fluid kinetic energy and subsequently into potential energy within the pump casing. Central to this study is the pivotal role of the

impeller in pump performance with a specific focus on the impact of blade angle. The impeller's design, especially the blade angle, significantly affects the flow patterns within the pump, thereby influencing parameters such as head, efficiency, and power consumption. Prior research in the field of centrifugal pump performance and impeller design is reviewed to identify gaps and establish the need for further investigation. The study aims to employ numerical methods such as computational fluid dynamics (CFD) simulations to comprehensively explore the relationships between blade angle variations and the pump's performance under various operating conditions.[2] By elucidating these relationships, the research aims to contribute valuable insights to the optimization of centrifugal pump design, ultimately enhancing efficiency, reducing energy consumption, and improving overall reliability. The significance of the study lies in its potential to advance our understanding of pump dynamics, offering practical implications for the design and operation of centrifugal pumps in industrial applications.[3]

2. Methodology

This particular chapter delves into a thorough explanation of the various methods utilized for the simulations, offering a comprehensive exploration of all the related processes.

2.1 Flow Chart Process

The graphic below, "Fig. 1", provides a comprehensive representation of the entire flow process that is discussed in this paper.
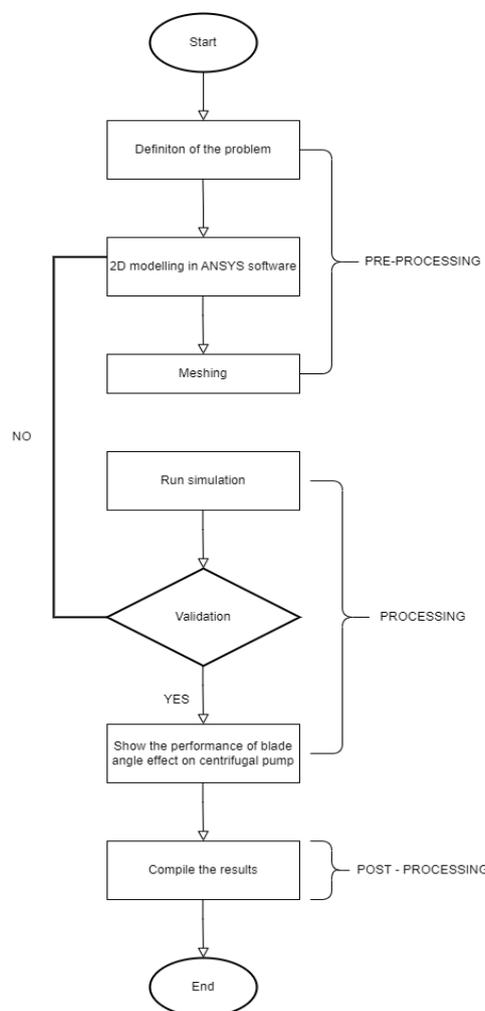

Fig. 1 *Flow Chart Process throughout this paper*

2.2 Geometry Development and Meshing

The geometry employed in the investigation will be generated using Auto Desk Inventor within ANSYS Fluent. The design Subsequently, the geometry will be discretized into a mesh that will partition the geometry into

smaller elements or cells. It is imperative to employ an appropriate meshing technique in order to ensure an accurate representation of the flow characteristics and precise calculation
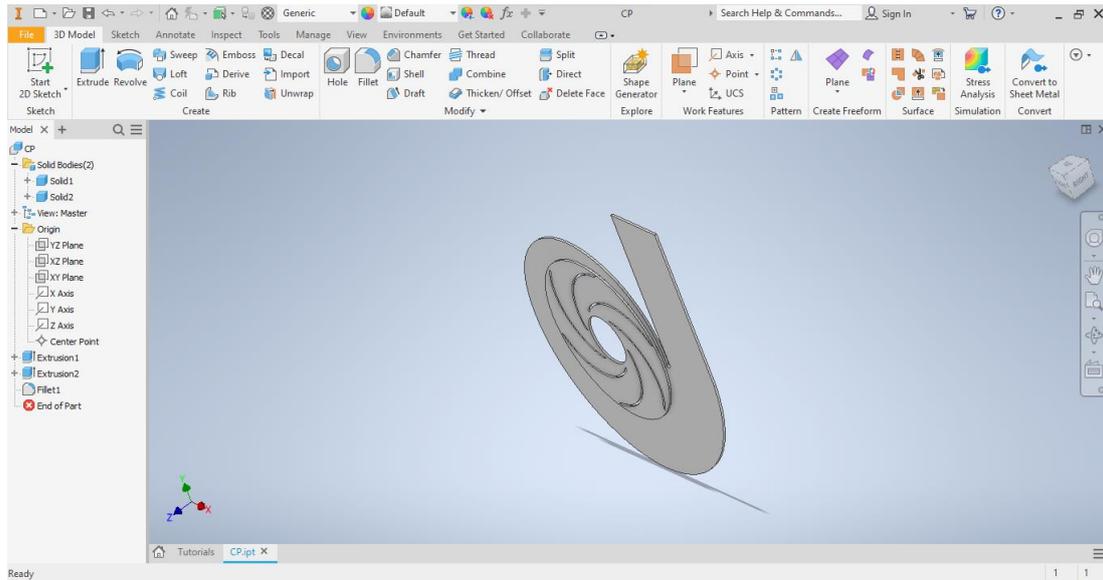

Fig. 1 Design with Auto Desk Inventor 2021

Table 1. All parameter set for the simulation

Blade Angle (°)	Angular speed (rev/min)	Volume flowrate (m^3/h)
16	1000	400
26	1200	450
36	1400	500
46	1600	550
56	2000	600
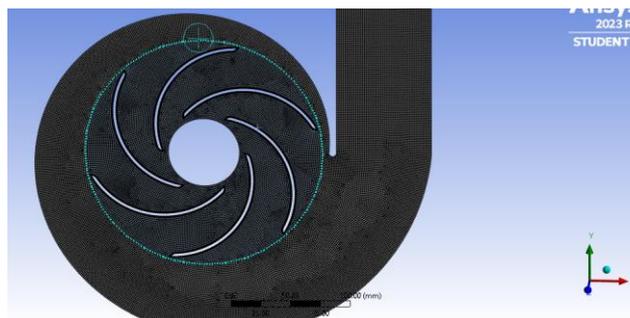
Fig 2: After Meshing the geometry

2.3 Solver Setup

In the solver setup stage, the parameters, such as the simulation's blade models, materials, cell zone conditions, and boundary conditions, are determined and set accordingly based on the desired values. Accurate specification of the setup is crucial in simulating the crossflow scenario accurately.

2.4 Model Setup

The answer will begin in a general context first and foremost. Preference requirements must be established for the scale, solver type, velocity formulation, time, and two-dimensional space. The main determinants of the outcome will be the energy and viscous models. The choice ought to make sense in light of the case study
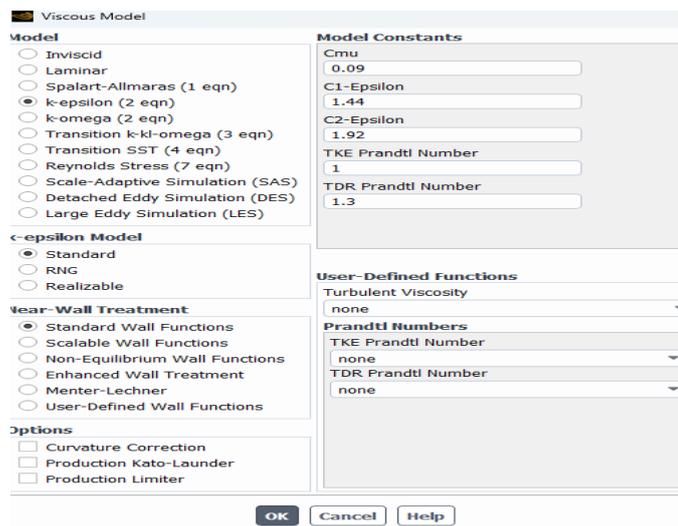

Fig 3 : Viscous Model Setup

2.5 Material Properties

Water liquid is the substance used in this model. The density of fluid in ANSYS Fluent is commonly specified as a function of temperature and pressure. Under normal circumstances, the default density value is 998.2kg/m^3. The air's viscosity in the variables of pressure and temperature also affect ANSYS Fluent. The default viscosity value under normal circumstances is around 0.001003 kg/ms.
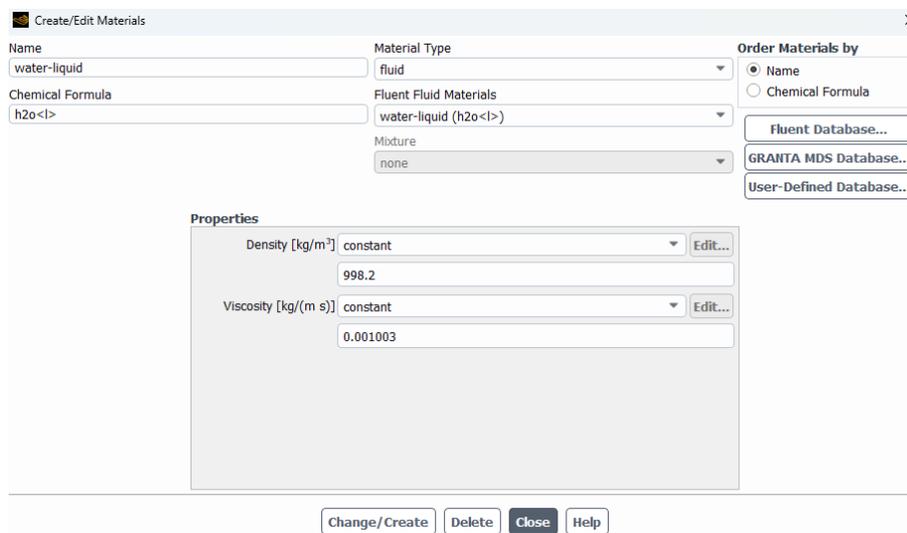

Fig 4: Material Selection

2.6 Boundary condition

In ANSYS Fluent, boundary conditions are used to specify the conditions at the boundaries of the simulation domain. The kind of flow (inlet, exit, or wall), velocity, pressure, temperature, or other properties of the fluid or solid at the boundary. To get precise and pertinent simulation results, boundary conditions must be chosen and applied with care. As an illustration, the velocity inlet selection with computed values is displayed in the picture below.
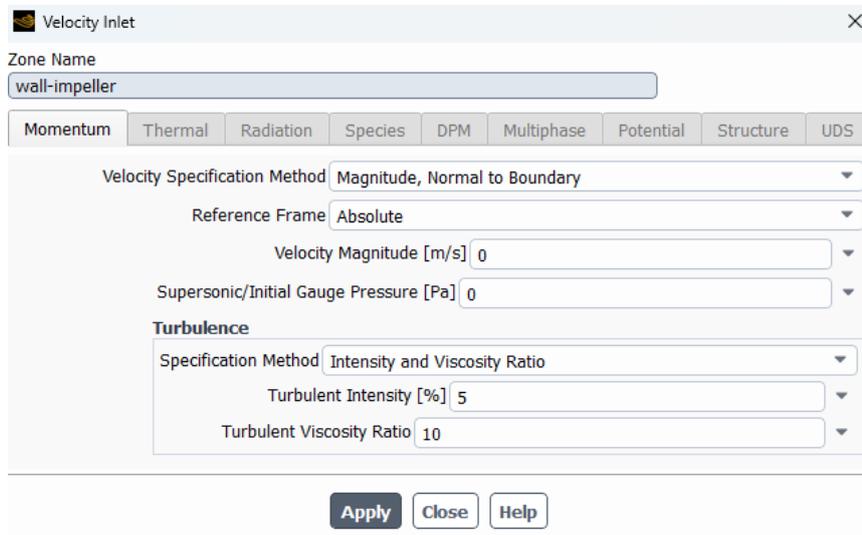

Fig. 5 *Boundary conditions in the velocity inlet*

2.7 Run Calculation

The findings and an explanation of the simulation's success or failure will be provided in the post-processing section. In order to assess the correctness of an ANSYS Fluent simulation, validation involves comparing the simulation's output to experimental or theoretical data. and accuracy. Following a simulation, the results might be contrasted with empirical data obtained from empirical measurements, theoretical forecasts derived from analytical solutions, or results from more simulations.
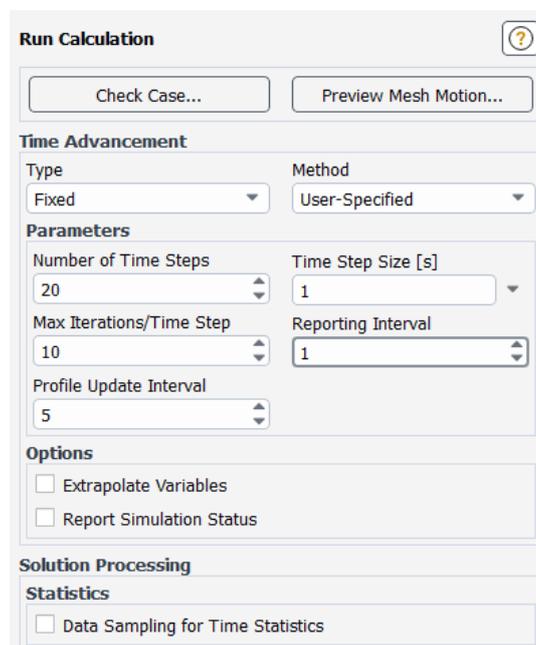

Fig. 6 *Run the Calculation*

3. Results and Discussion

Five different scenarios will be simulated using the Computational Fluid Dynamics (CFD) programme via ANSYS Fluent, based on the preceding processing phase. The study's wide scope, which covers 5 blade angle number between 16° ,26°,36°,46° and 56°. The appropriate meshing parameters have been used in every simulation to ensure exceptional validity and accuracy. The pressure increases, pressure distribution, and velocity of each model. Throughout the whole numerical test, vectors are displayed.

3.1 Pressure and Velocity Contour Visualization

When analysing and visualising the pressure distribution and flow behaviour inside a fluid domain such as the area surrounding pump blades or other aerodynamic structures contours are crucial tools.
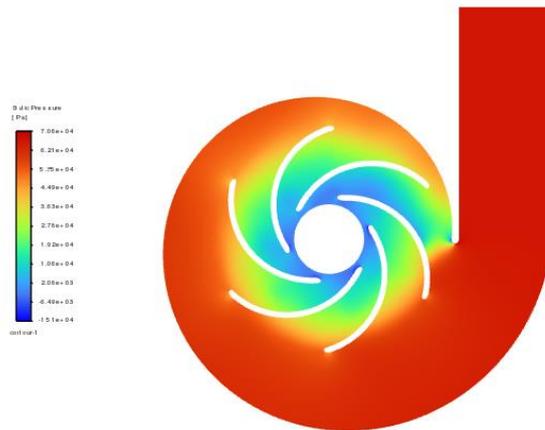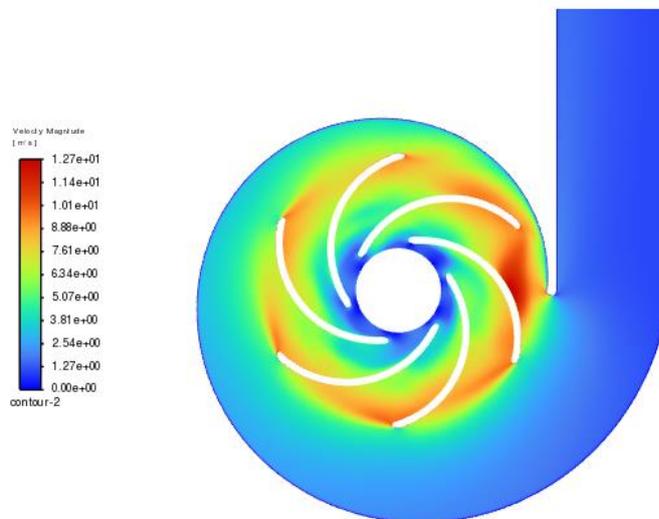

Fig. 7 *Pressure Distribution Contour*

Fig. 8 *Velocity Magnitude Contour*

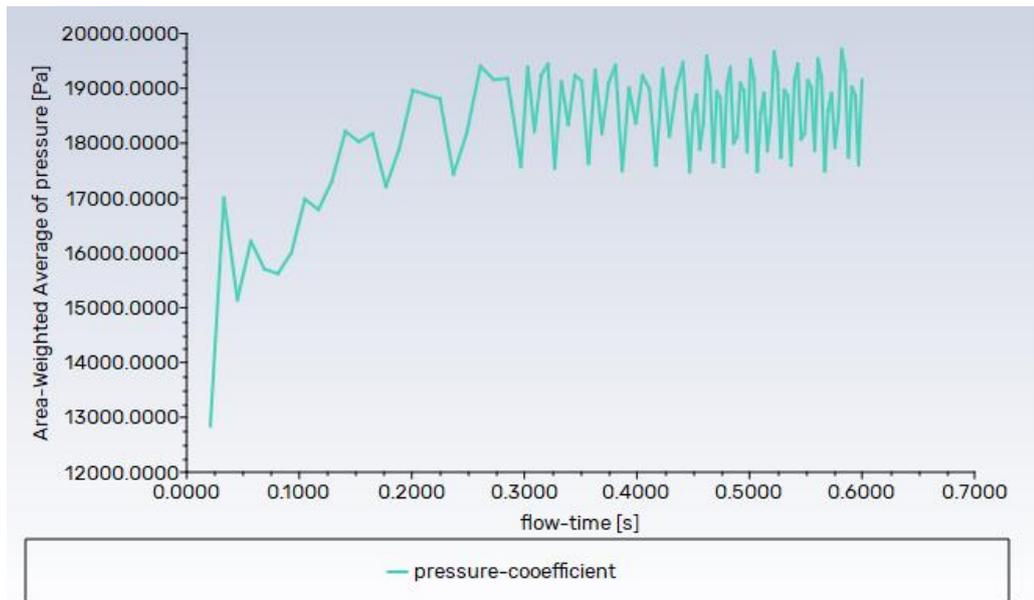## 3.2 Pressure Increase for every Blade Angle Geometry

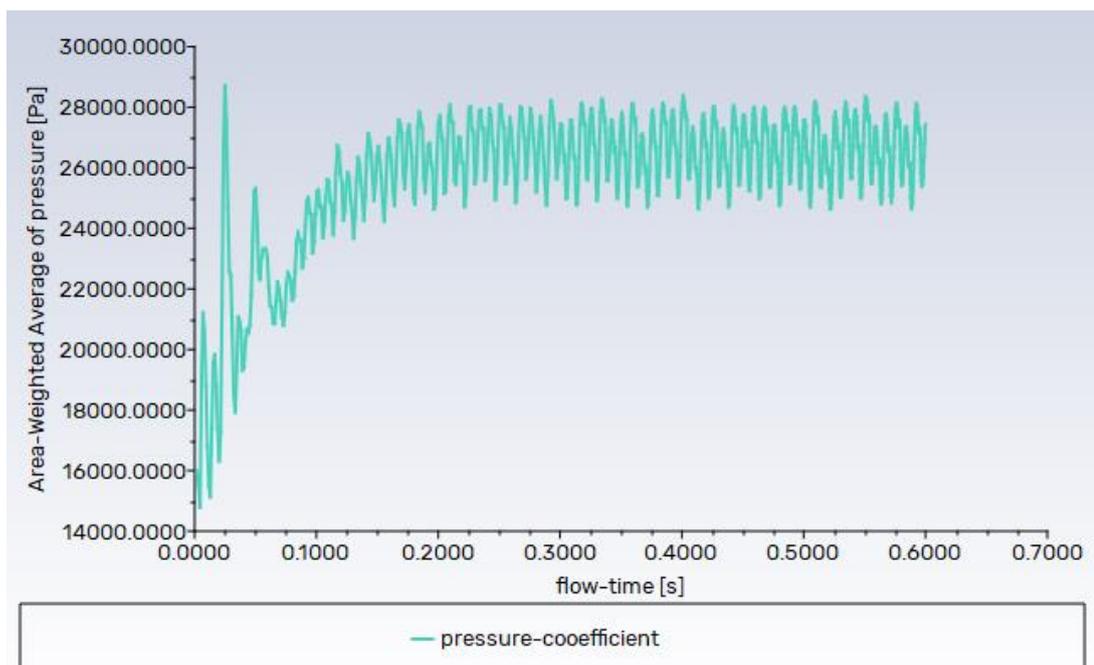**Fig. 9** *Graph pressure increase for blade angle 16°*
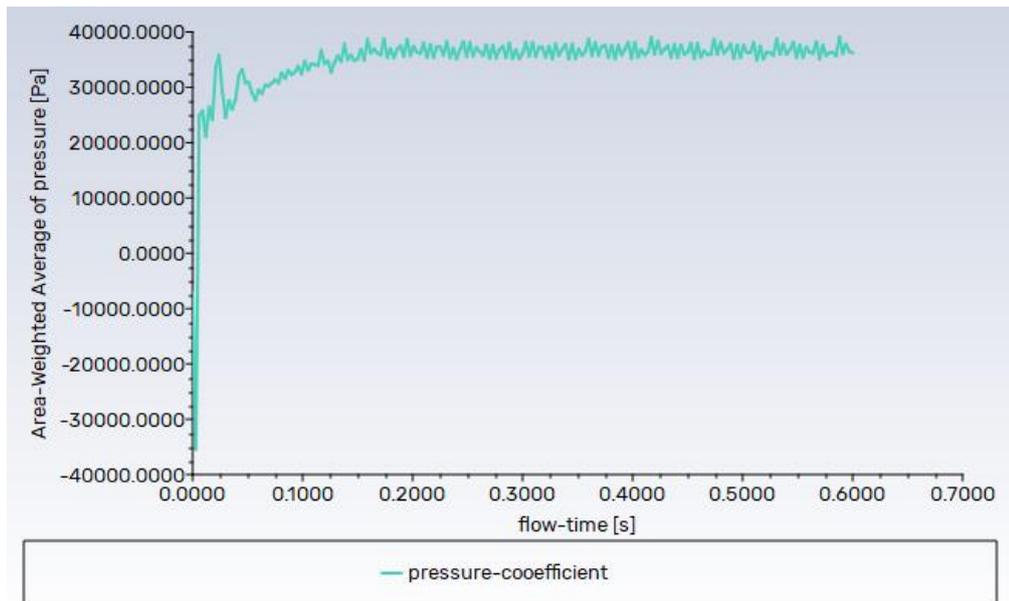

Fig. 10 *Graph pressure increase for blade angle 26°*

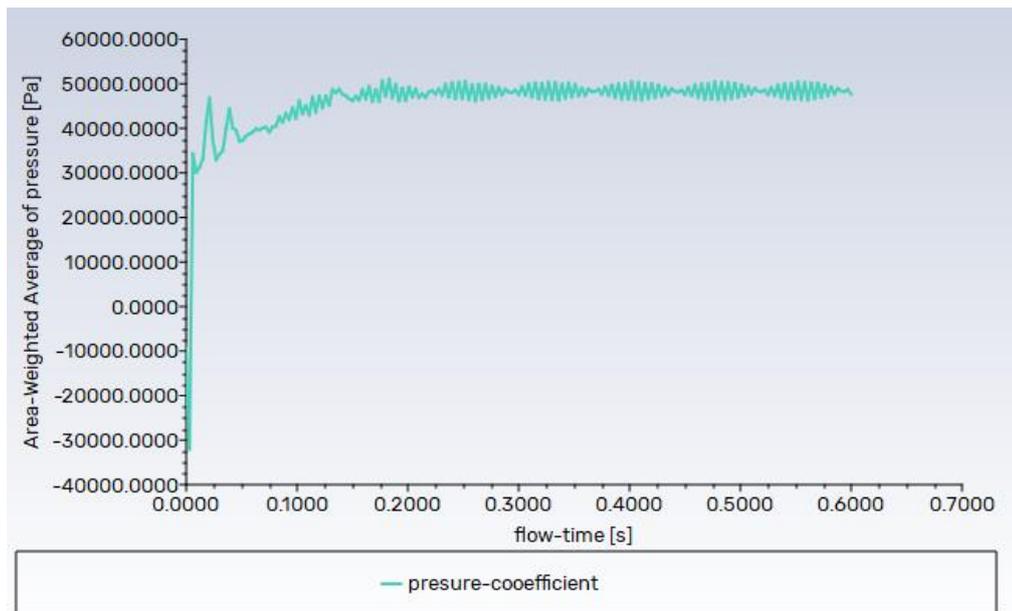**Fig. 11** *Graph pressure increase for blade angle 36°*

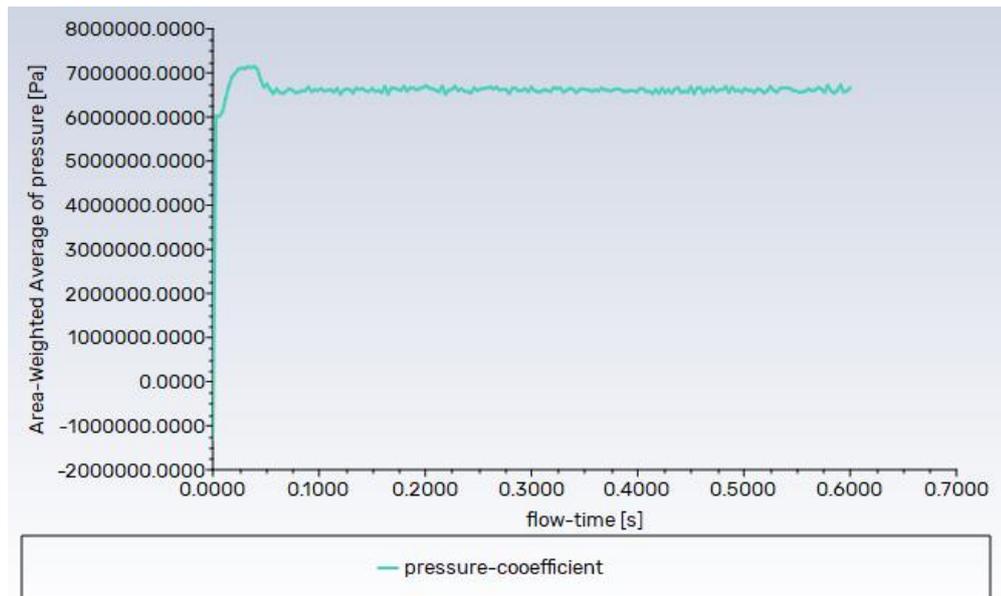**Fig. 12** *Graph pressure increase for blade angle 46°*

Fig. 13 *Graph pressure increase for blade angle 56°*

The results indicate that the pressure increase, pumping power, and efficiency of the centrifugal pump are significantly influenced by the blade angle. As the blade angle increases, the pressure increase and pumping power tend to rise, reaching optimal values at certain angles. However, beyond a specific angle, these parameters may decline, suggesting an optimal range for blade angle to maximize performance. The study's findings align with theoretical expectations and highlight the importance of precise blade angle adjustments for efficient pump operation.

Table 2 Table for all the parameters performance for every blade angle geometry

Blade Angle	Angular speed (rev/min	Volume Flowrate (m^3/h)	ΔP (Pa)	Pumping Power (W)	Efficiency (%)
16	1000	400	66377.26789	80580.2973	9992.94
26	1200	450	95285.49615	149755.9265	8725.12
36	1400	500	134536.875	296463.2648	8222.38
46	1600	550	178082.5489	482097.7928	7796.99
56	2000	600	18291676.91	504599686.7	7163.56

4. Conclusions

This study provides a comprehensive analysis of the impact of blade angle on the performance characteristics of a centrifugal pump. The results demonstrate that optimal blade angles can significantly enhance pump performance, emphasizing the need for careful design and adjustment of impeller blades. Future research should explore a wider range of blade angles and operational conditions to further refine these findings and support the development of more efficient centrifugal pumps.

Acknowledgement

References

[1]   1.Dano, M. L., & Kim, K. W. (2018). A study on the performance prediction of centrifugal pumps using CFD. *Journal of Fluids Engineering*, 140(5), 051103.2

[2]   2.Xue, Y., & Zhang, L. (2017). Numerical simulation of blade angle effects on centrifugal pump performance. In *11th International Symposium on Cavitation* (Vol. 6, pp. 237-244)..

[3]   Chen, L. (2015). *Numerical Study of Impeller Blade Shape and Angle on Centrifugal Pump Performance* (Doctoral dissertation, University of Beijing).