ISSN 2963-7511
Journal of Engineering Researcher and Lecturer, Vol. 4, No. 1, pp. 22-30, 2025

A concave impeller: A new modified semi-open impeller for higher performance of centrifugal pumps

Rahmat Azis Nabawi, Egi Fadillah*, Haris Shiddiq Mulyadi, Muhammad Shadiq Fahrezi, and Firza Fernanda Putra

Department of Mechanical Engineering, Faculty of Engineering, Universitas Negeri Padang, **Indonesia**

*Corresponding Author: eginesta0@gmail.com

Received: 07th January 2025; 1st Revised: 27th March 2025; 2nd Revised: 17th April 2025; Accepted: 22th April 2025

https://doi.org/10.58712/jerel.v4i1.178

Abstract: The performance enhancement of centrifugal pumps is continuously pursued to meet increasingly complex operational demands. Ongoing improvements in pump component design, particularly the impeller, are essential to achieving optimal configurations. The impeller is a critical component that significantly influences the overall performance of centrifugal pumps. This study aims to develop an impeller design that offers higher performance compared to the original one. The methodology utilized was Computational Fluid Dynamics (CFD). The primary focus was on modifying the semi-open impeller type. Two modified impellers were developed with concave and convex design. They were simulated and compared to the original impeller configuration. The simulation results indicate that pressure distribution at the inlet remains similar for all designs. However, pressure variations at the outlet and differences in volumetric flow rate between inlet and outlet were found. The contour visualization of pressure and flow velocity for each impeller configuration shown from the CFD simulation results is further discussed, in terms of pressure distribution and flow trajectory pressure.

Keywords: centrifugal pump; impeller design; CFD simulation; pressure distribution; volume flow rate; semi-open impeller

1. Introduction

Centrifugal pumps have been widely used in various fields, both in the industrial and household sectors. Their popularity is due to their economical characteristics, relatively simple construction (Kosseva, 2017), and high level of operational reliability (Si et al., 2018). Efforts to improve the efficiency of these pumps need to continue as a strategic step to reduce operational costs and improve overall system performance (Shi et al., 2023). Centrifugal pumps are widely used in various applications, so pump systems are often required to operate over a wide range of flow rates, depending on the operational needs of each application. The diversity of fluid types used in various applications is also one of the main drivers in the research and development of centrifugal pump design, especially in order to increase the efficiency and adaptability of the pump to varying operational conditions. In addition, innovations in pump design are also directed at reducing the time and cost of the production process, resulting in a more efficient, economical, and sustainable system.

Published studies on centrifugal pumps generally focus more on the effect of the degree of inclination and the number of blades on pump performance. Peng et al. (2020) conducted a study on the effect of outlet angle variation on the impeller blades of a centrifugal pump, with an angle range of 16°, 20°, 24°, 28°, and 32°, using ANSYS-CFX simulation software. The results show that the blade outlet angle has a significant effect on the external flow characteristics, as well as the resulting power curve. Appropriate adjustment of the outlet angle can be used as a strategy to control pump power and prevent overloading (Li et al., 2020; Nadaraja et al., 2023; Peng et al., 2020; Wang et al., 2023). The number of impellers has an influence on centrifugal pump performance (Abo Elyamin et al., 2019; Aldio et al., 2023; Sakran et al., 2022; Subroto & Effendy, 2019; Susilo & Setiawan, 2021).

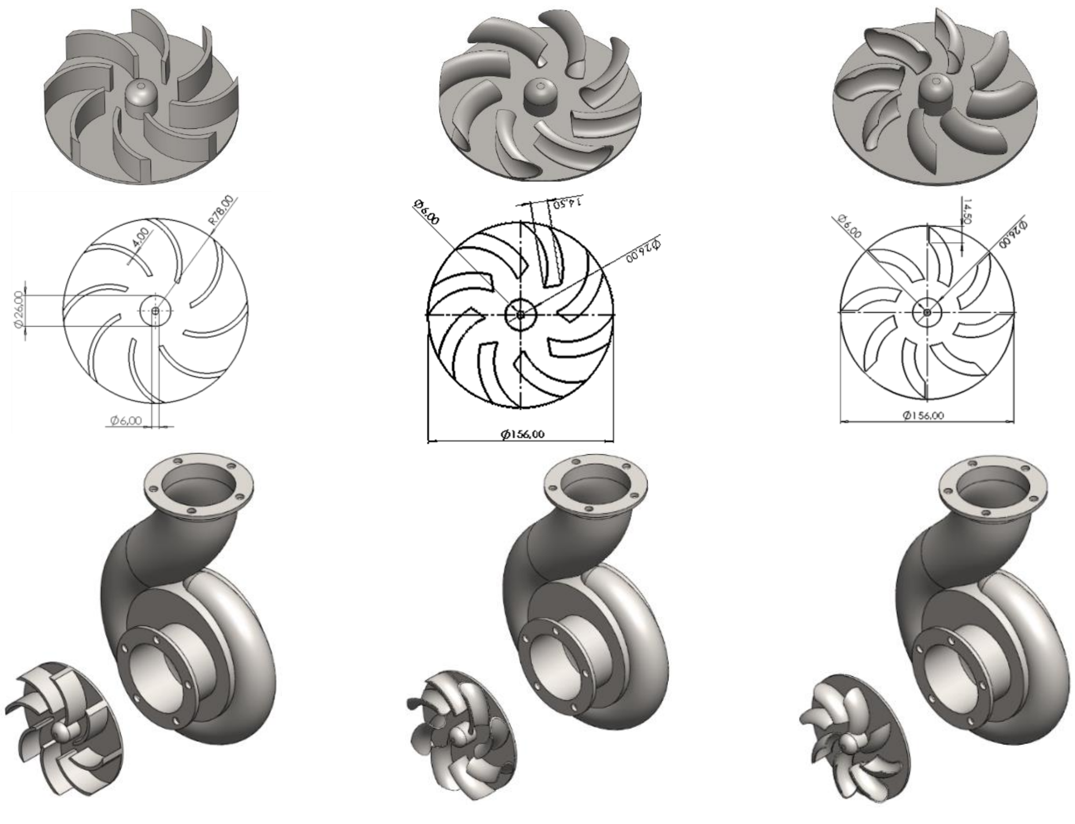
This study aims to develop an impeller design that offers higher performance compared to the original one. This study is conducted in order to improve the performance of centrifugal pumps. The problem behind this research is the limitation of previous studies that generally only evaluate the performance of centrifugal pumps through the study of the number of blades and angle changes. Therefore, this study explores impeller designs with different shapes, in the form of two geometric variations: concave, and convex. For comparison, an impeller with the original shape commonly used in centrifugal pumps was also simulated. This research is expected to gain a deeper understanding of the effect of changes in impeller design on the overall performance of centrifugal pumps.

2. Material and methods

This study was conducted using the Computational Fluid Dynamics (CFD) method to evaluate the performance of each design based on a number of key parameters, comprising pressure, volume flow rate, and efficiency. The software used in this simulation is SolidWorks (Research License 2021). Through the CFD approach, the complex internal flow behavior inside various pump components can be analyzed in detail under various operating conditions. Various studies on pump simulation have been conducted using CFD methods (Sakran et al., 2022; Zhou et al., 2003).

2.1 Centrifugal pump impellers

In this study, three types of centrifugal pump impeller designs were compared, including original, concave, and convex impellers (Figure 1).



(a) (b) (c)

Figure 1. Centrifugal pump impellers. (a) original impeller, (b) concave impeller, and (c) convex impeller

2.2 Simulation parameters

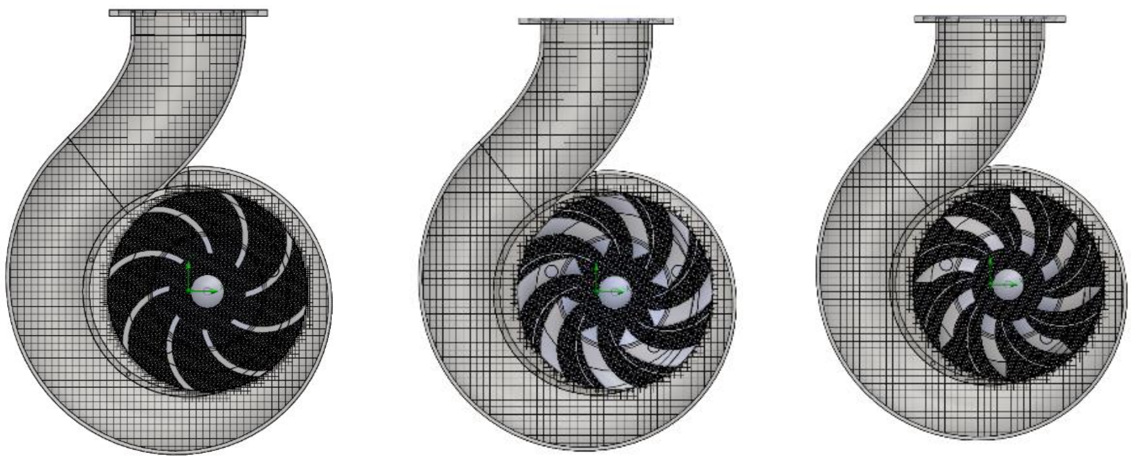
The simulation parameters carried out using SolidWorks Flow Simulation in this study are presented in Table 1.

Table 1. Simulation parameters

Unit system	Pressure Volume flow rate Torque	Mpa m^3/s S Nm
Analisis type	Internal analysis Total analysis time Out put time step Rotating local region	 0 0 Sliding
Rotational speed of the impeller	5000 RPM(523,6 rad/s)	
Fluid	Water	
Wall condition	Default wall condition	
Initial condition	Default initial condition	

2.3 Meshing

The results of research conducted by Salmat et al. (2023) on the application of centrifugal pumps for air pumping, and by Aldio et al. (2023) on crude oil pumping using SolidWorks Flow Simulation recommended the use of mesh level 5. This is based on the finding that simulation results at this level show pressure and volume flow rate values at the inlet and outlet channels that are not significantly different compared to mesh level 7, which is the highest mesh level. In this study there are two types of mesh settings used, global mesh and local mesh. Global mesh uses level 5 and on the impeller in contact with the fluid using local mesh with a smaller mesh size. The total cells in each model sell as many as 2 million cells.



(a) (b) (c)

Figure 2. Meshing, (a) original impeller, (b) concave impeller, and (c) concex impeller

2.4 Simulation setup

In the impeller part, a 'fluid region' was made with the 'Local Region Sliding' type. This 'fluid region' was rotated at a speed of 5000 RPM (Figure 2).

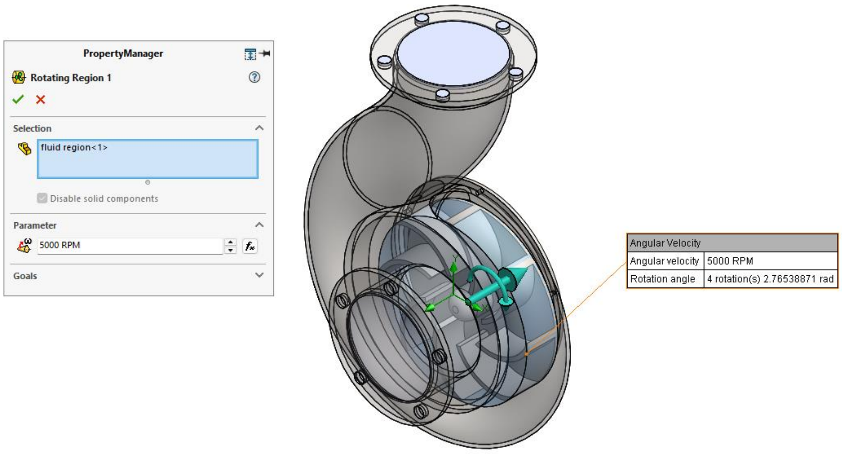


Figure 2. Fluid region

In setting boundary conditions, the inlet and outlet were set as ‘environment pressure’, this is in accordance with the actual condition of the pump when used. The setting of boundary conditions in this study is presented in Figure 3.

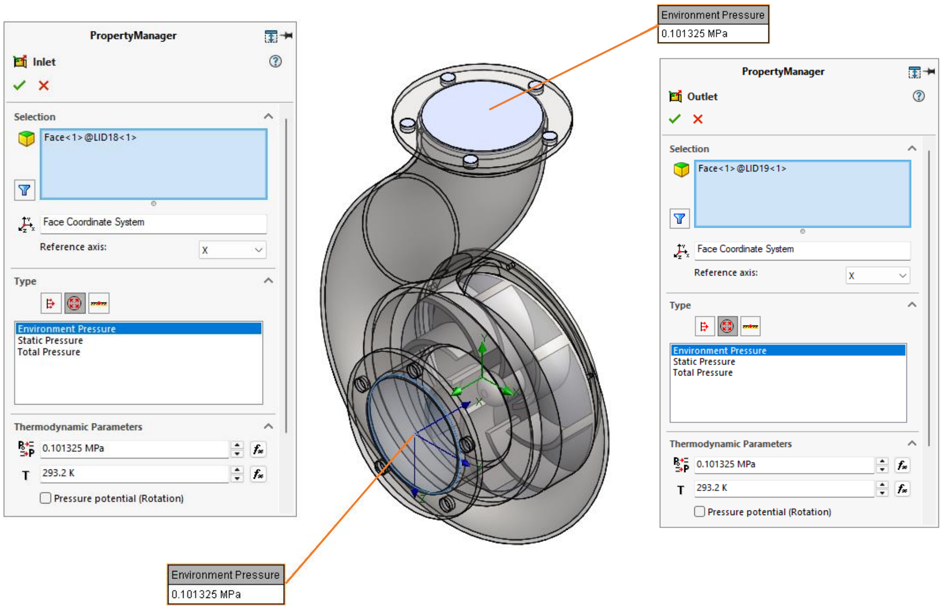


Figure 3. Boundary conditions

In the Goal section, the selected parameters include pressure and volume flow rate at the inlet and outlet sides, with the aim of knowing the pressure characteristics that occur at both points. The pressure drop was calculated by subtracting the pressure at the inlet from the pressure at the outlet (Eq. 1). To determine the value of torque acting on the impeller, the pump impeller area was selected and the torque parameter was determined through the surface goal (Figure 4). The pump efficiency was obtained based on the multiplication of the pressure drop and volume flow rate at the inlet side, then divided by the amount of mechanical power calculated based on the impeller rotational speed (5000 RPM = 532.6 rad/s), as shown in Eq. 2.

$$Pressure\ drop = Pressure\ inlet - pressure\ outlet \quad (1)$$

$$Efficiency = \frac{Pressure\ drop\ \times Volume\ Flow\ Rate\ Inlet}{523.6\ \times Torque} \quad (2)$$

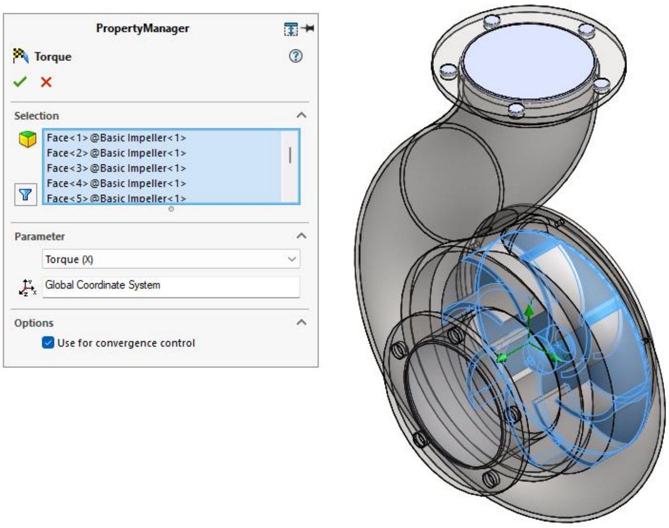


Figure 4. Torque

3. Results and discussion

Based on the simulation results that have been carried out, it is found the total pressure value on the inlet and outlet sides of each centrifugal pump with different impeller type variations, as shown in Figure 5. The simulation results show that the pressure at the inlet side for each pump has the same value, which is 0.1 MPa. The pump using original and convex impeller shows the same pressure value on the outlet side, which is 0.3 MPa. Meanwhile, the centrifugal pump with concave blades shows a higher pressure value on the outlet side, which is 0.32 MPa.

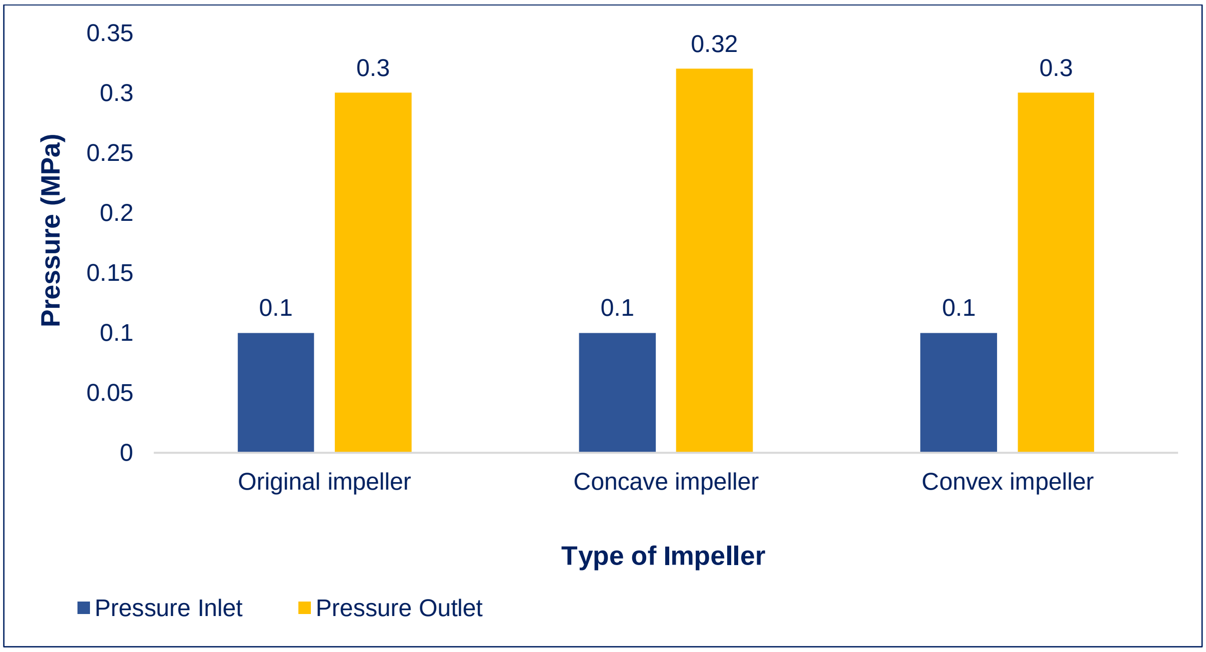


Figure 5. Pressure at inlet and outlet

The volume flow rate of each centrifugal pump with impeller variation is presented in Figure 6. On the inlet side, the highest volume flow rate is obtained in the centrifugal pump using a convex impeller. Meanwhile, on the outlet side, the highest volume flow rate occurs in centrifugal pumps with concave impellers. In centrifugal pumps with convex impellers, the volumetric flow rates at the inlet and outlet show the same value. In contrast, in centrifugal pumps using original and concave impellers, there is an increase in flow rate from inlet to outlet.

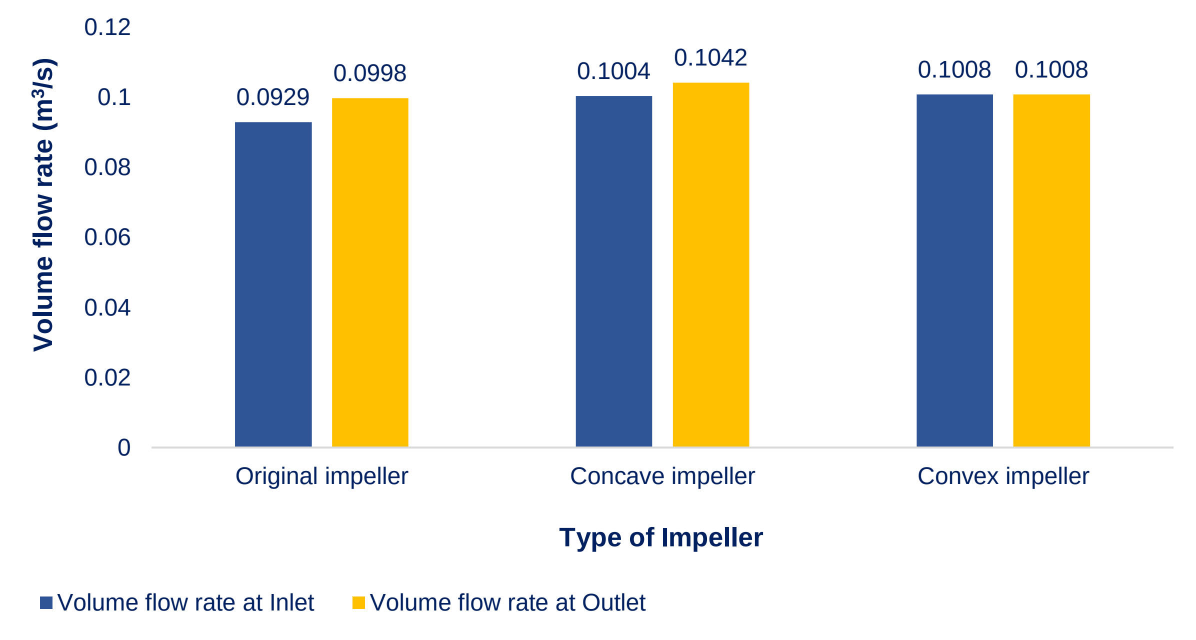


Figure 6. Volume flow rate at inlet and outlet

The impeller design with the highest efficiency was the concave impeller, which achieved an efficiency of 32%. Meanwhile, the impeller design with the lowest efficiency is the centrifugal pump using convex impellers, with an efficiency of 26%. The centrifugal pump using the original impeller has an efficiency of 27%.

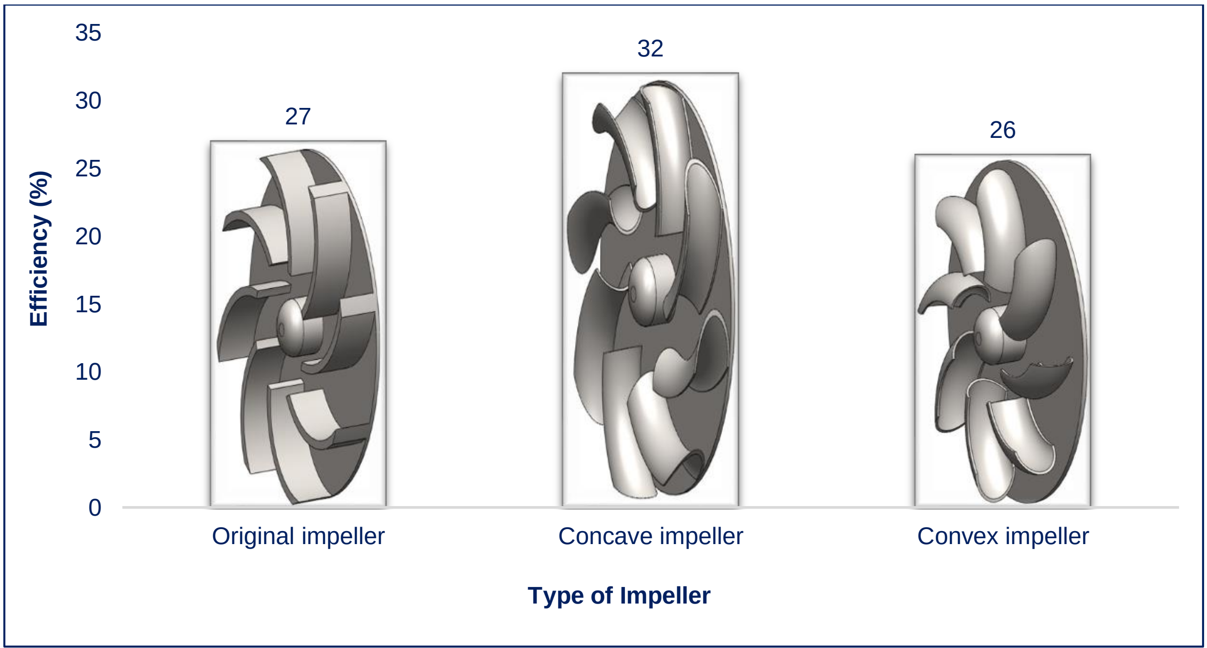
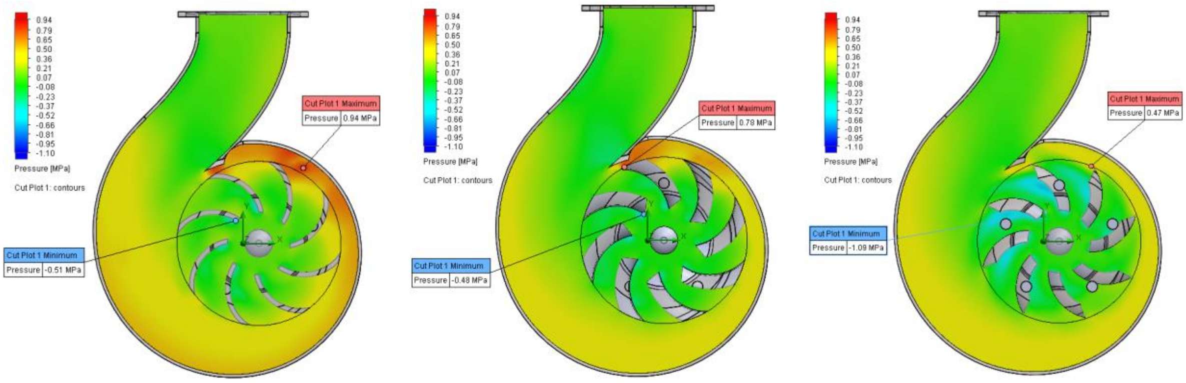


Figure 7. Efficiency

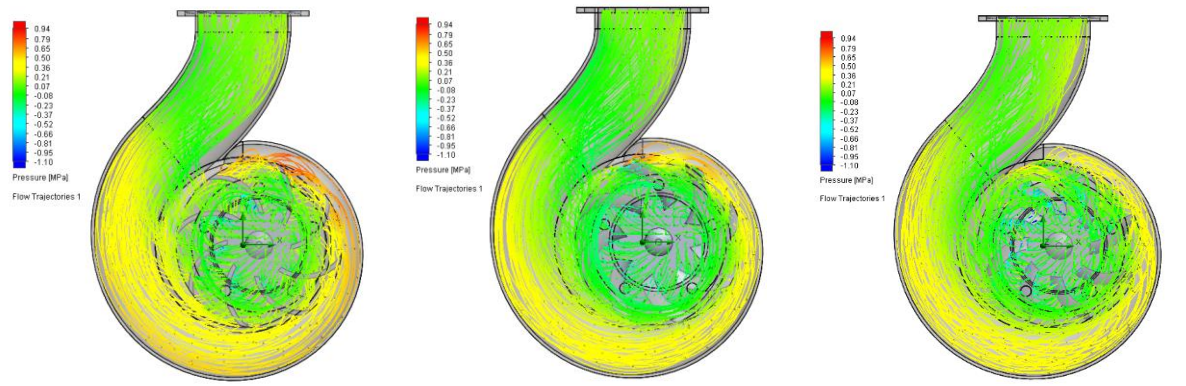
The pressure distribution of the working fluid in each centrifugal pump with different impeller designs is shown in Figure 8. Based on the simulation results, there are differences in the location of the maximum pressure in the three types of impellers. In the centrifugal pump with concave impeller, the maximum pressure is located near the channel leading to the outlet. Meanwhile, in pumps with original and convex impellers, the maximum pressure occurs in different areas.



(a) (b) (c)

Figure 8. Pressure distrubution on: (a) Original impeller, (b) concave impeller, and (c) convex impeller

In the working principle of a pump with an impeller, the fluid pressure increases when the fluid leaves the impeller due to the conversion of kinetic energy into pressure energy. In a concave impeller, the pressure increase occurs from the centre of the impeller towards the edge. Meanwhile, in the original impeller, high pressure is concentrated around the pump body wall, which causes a decrease in fluid pressure as it flows towards the outlet. The interaction between the fluid flow and the pump body wall affects the pressure distribution and flow velocity. Figure 9 shows the flow trajectory pressure, where in the original impeller, the fluid appears to impinge on the pump wall, causing a significant pressure drop.



(a) (b) (c)

Figure 9. Flow Trajectory Pressure on: (a) Original impeller, (b) concave impeller, and (c) convex impeller

In Figure 9, it can be seen that the fluid flow in the pump is concentrated in the centre of the pump body and leads directly to the outlet channel, thus reducing the interaction and friction between the fluid and the pump wall. In the pump configuration with the original impeller, there is a phenomenon of flow separation that leads to the outlet channel, resulting in collisions between flows that result in a pressure drop. On the other hand, in a pump with a convex impeller, the fluid being pushed has a relatively lower pressure. This is due to the geometry of the convex impeller which is not able to produce high pressure as the concave impeller shape.

4. Conclusion

This study has designed and evaluated two centrifugal pump impeller models with new blade geometries, including concave and convex shapes, as alternatives to the original impeller design which has a flat plane-shaped blade. The hydraulic performance evaluation of the three impeller designs was conducted through Computational Fluid Dynamics (CFD) simulations. The simulation results showed that the impeller with concave blade geometry performed better than the original and convex design, in terms of efficiency and fluid flow pressure distribution in the pump. This finding indicates the potential for improving pump performance through modification of the impeller blade shape. It is recommended for further research to study on velocity diagram analysis, the number of blades, as well as performance testing on different fluid.

Author's Declaration

Author contribution

Rahmat Azis Nabawi: conceptualization, methodology, validation, and Writing - Review & Editing. **Egi Fadillah:** resources, software, writing - Original Draft, and visualization. **Haris Shiddiq Mulyadi:** software, writing - Original Draft, and visualization. **Muhammad Shadiq Fahrezi:** software, writing - original draft, data curation, and visualization. **Firza Fernanda Putra:** resource, software, writing - original draft, and visualization.

Funding statement

This research received no specific grant from any funding agency in the public, commercial, or not-for-profit sectors.

Acknowledgement

The authors would like to thank those who have helped in the implementation of this research.

Competing interest

The authors declare no conflict of interest from the conduct of research and publication of this article.

Ethical Clearance

This research does not involve humans or animals as subjects.

AI Statement

This article is an original work of the authors without any AI generated sentences, tables, and images.

Publisher's and Journal's Note

Researcher and Lecturer Society as the publisher, and the Editor of Journal of Engineering Researcher and Lecturer state that there is no conflict of interest towards this article publication.

References

Abo Elyamin, G. R. H., Bassily, M. A., Khalil, K. Y., & Gomaa, M. Sh. (2019). Effect of impeller blades number on the performance of a centrifugal pump. *Alexandria Engineering Journal*, *58*(1), 39–48. https://doi.org/10.1016/j.aej.2019.02.004

Aldio, M. F., Waskito, W., Purwantono, P., & Lapisa, R. (2023). Optimization of impeller blade number in centrifugal pump for crude oil using Solidworks Flow Simulation. *Journal of Engineering Researcher and Lecturer*, *2*(3), 80–93. https://doi.org/10.58712/jerel.v2i3.116

Kosseva, M. R. (2017). Chemical Engineering Aspects of Fruit Wine Production. In *Science and Technology of Fruit Wine Production* (pp. 253–293). Elsevier. https://doi.org/10.1016/B978-0-12-800850-8.00006-5

Li, J., Tang, L., & Zhang, Y. (2020). The influence of blade angle on the performance of plastic centrifugal pump. *Advances in Materials Science and Engineering*, *2020*. https://doi.org/10.1155/2020/7205717

Nadaraja, D., Taib, I., Darlis, N., Kadir, R., Osman, K., & Khudzari, Z. (2023). Analysis of Flow Characteristics for Different Blade Outlet Angle in LVAD. *CFD Letters*, *15*(11), 79–91. https://doi.org/10.37934/cfdl.15.11.7991

Peng, G., Chen, Q., Zhou, L., Pan, B., & Zhu, Y. (2020). Effect of Blade Outlet Angle on the Flow Field and Preventing Overload in a Centrifugal Pump. *Micromachines*, *11*(9), 811. https://doi.org/10.3390/mi11090811

Sakran, H. K., Abdul Aziz, M. S., Abdullah, M. Z., & Khor, C. Y. (2022). Effects of Blade Number on the Centrifugal Pump Performance: A Review. *Arabian Journal for Science and Engineering*, *47*(7), 7945–7961. https://doi.org/10.1007/s13369-021-06545-z

Salmat, S., Yanti Sari, D., Fernanda, Y., & Prasetya, F. (2023). SolidWorks Flow Simulation: Selecting the optimal mesh for conducting CFD analysis on a centrifugal fan. *Journal of Engineering Researcher and Lecturer*, *2*(3), 94–103. https://doi.org/10.58712/jerel.v2i3.104

Shi, Y., Sheng, B., Zhu, J., Chen, G., Zhang, T., & Luo, R. (2023). A Human-Centric Design Method for Industrial Centrifugal Pump Based on Digital Twin. *Processes*, *12*(1), 42. https://doi.org/10.3390/pr12010042

Si, Q., Bois, G., Jiang, Q., He, W., Ali, A., & Yuan, S. (2018). Investigation on the Handling Ability of Centrifugal Pumps under Air–Water Two-Phase Inflow: Model and Experimental Validation. *Energies*, *11*(11), 3048. https://doi.org/10.3390/en11113048

Subroto, & Effendy, M. (2019). *Optimization of centrifugal pump performance with various blade number*. 020016. https://doi.org/10.1063/1.5112400

Susilo, S. H., & Setiawan, A. (2021). Analysis of the number and angle of the impeller blade to the performance of centrifugal pump. *EUREKA: Physics and Engineering*, *2021*(5), 62–68. https://doi.org/10.21303/2461-4262.2021.002001

Wang, Y. Y., Zhao, W. G., Han, X. D., Fan, P. J., Liu, Z. L., & Hu, J. Q. (2023). Effects of the Centrifugal Pump Outlet Blade Angle on Its Internal Flow Field Characteristics under Cavitation Condition. *Journal of Applied Fluid Mechanics*, *16*(2), 389–399. https://doi.org/10.47176/jafm.16.02.1241

Zhou, W., Zhao, Z., Lee, T. S., & Winoto, S. H. (2003). Investigation of Flow Through Centrifugal Pump Impellers Using Computational Fluid Dynamics. *International Journal of Rotating Machinery*, *9*(1), 49–61. https://doi.org/10.1155/S1023621X0300006X