

## Numerical Study on The Effect of Blade Leading Edge Rounding and Blade Number on Slurry Pump Performance

Emy Zuraida Wuryaningrum\* , Sutardi

Institut Teknologi Sepuluh Nopember, Indonesia

Email: emyzuraida30@gmail.com\* , sutardi@me.its.ac.id

---

### ABSTRACT:

Centrifugal slurry pumps are essential for transporting solid-liquid mixtures in industries such as mining and thermal power generation. However, the presence of solid particles increases hydraulic losses, reduces efficiency, and accelerates component erosion, leading to lower performance compared to clean-water operation. Impeller geometry—particularly blade number and leading-edge shape—plays a critical role in mitigating these effects. This study numerically investigates the impact of blade number ( $Z = 3, 4, \text{ and } 5$ ) and rounded leading-edge geometry on the hydraulic performance of a centrifugal slurry pump. Three-dimensional simulations were performed using ANSYS CFX with an Eulerian-Eulerian multiphase approach and the standard  $k-\epsilon$  turbulence model to represent water and limestone particles ( $100 \mu\text{m}$  diameter). The pump operated at 1250 rpm, with performance analyzed at solid concentrations of 10% and 20% and flow rates of 30–50  $\text{m}^3/\text{h}$ . Results show that increasing solid concentration reduces head and efficiency while increasing power consumption. The rounded leading edge improves inlet flow by minimizing separation and local turbulence. Among the tested configurations, the five-blade impeller ( $Z = 5$ ) demonstrated the most stable internal flow, highest head and efficiency, and best erosion resistance. These findings align with existing literature, validating the numerical approach. This research provides valuable guidance for optimizing impeller design to enhance the efficiency and durability of slurry pumps operating at high solid concentrations.

**Keywords:** Pompa slurry, jumlah sudu, leading edge rounding, kinerja hidrolis, ANSYS CFX, konsentrasi padatan.

---

### INTRODUCTION

The Centrifugal Slurry Pump (CSP) is a pipe-based transportation system used for conveying fluids mixed with solid particles (Paradkar, 2016; Zoheidi, 2018). Slurry pumps are widely utilized in various applications, such as dredging, open-pit mining, ash transportation in mineral processing plants, and ash disposal in thermal power plants (Peng et al., 2021). These pumps are preferred due to their consistent flow rate, low maintenance costs, and excellent operational stability. The design of the Centrifugal Slurry Pump requires special consideration to ensure that the flow channel does not restrict the passage of solid particles (Tarodiya & Gandhi, 2017).

The presence of suspended solid particles in the fluid is the main cause of reduced hydraulic efficiency in this type of pump (Pradhan et al., 2025). Performance decline is further exacerbated by erosion and abrasion on the impeller. Consequently, centrifugal slurry pumps exhibit lower efficiency compared to conventional pumps (Tarodiya & Gandhi, 2017). Design factors contributing to this include robust impeller construction, a typically concentric casing design, and a relatively wide working gap and throat area (Menichino et al., 2024). Erosion progressively degrades the surface of the components, alters the hydraulic profile, and ultimately

leads to a significant reduction in pump performance, including decreased head and efficiency. In addition, the internal flow phenomena become more complex, involving particle–fluid and particle–wall interactions that can cause clogging and excessive vibration (Jaiman et al., 2023; Parameshwaran et al., 2016; Qiu et al., 2024; Xie et al., 2025; Zhang et al., 2016).

To understand and overcome these challenges, many researchers have conducted experimental and numerical studies to determine the effects of solids on pump performance and internal flow fields. The use of modern computational methods, particularly Computational Fluid Dynamics (CFD), to investigate solid–liquid flow in slurry-handling components has become increasingly prevalent (Pradhan et al., 2025; Silva, 2022; Singh & Kumar, 2026; Yadav et al., 2025). For instance, Tarodiya and Gandhi (2019) conducted numerical investigations showing that an increase in solids concentration leads to a significant decrease in pump performance. Internally, particles were found to concentrate on the pressure side of the impeller blade. Similarly, Peng et al. (2021) demonstrated that CFD simulations accurately predicted the regions of severe impeller wear, which were later confirmed through physical inspection after wear testing. The main erosion zones were identified on the leading edge and pressure side of the blade, where simulations showed concentrations of high-velocity particle impacts. Furthermore, Pradhan et al. (2025) successfully investigated the effects of slurry characteristics on vortex formation, pressure distribution along the blade, and overall pump efficiency.

Although the mechanisms of these problems are well understood, most previous studies have focused on diagnostic analysis of standard designs. There remains a research gap in exploring proactive design modifications as engineering solutions. To address the erosion issue, this study investigates the influence of impeller geometry modifications, specifically the application of rounding on blade edges and variations in the number of blades (Doshi et al., 2017; Shojaeefard & Saremi, 2022; Steiros et al., 2017; Tchada et al., 2024; Yadav et al., 2025). The proposed rounding modification is hypothesized to reduce the intensity of particle impacts, while variations in blade number are expected to alter flow structure and performance characteristics.

Therefore, this study proposes a numerical analysis of impeller geometry modifications—namely, the application of a rounding radius along the blade leading edge and variations in the number of blades—to evaluate their effects on hydraulic performance, internal flow patterns, and predicted erosion rates in centrifugal slurry pumps. Using Computational Fluid Dynamics (CFD) simulations, this study aims to deepen understanding of how impeller design interacts with slurry flow characteristics to optimize pump efficiency, reliability, and service life under heavy industrial conditions.

Based on the background and problem formulation outlined above, the objectives of this study are to: (1) numerically analyze the influence of varying the rounding radius on the impeller blade leading edge on the pump’s hydraulic performance parameters, including head, efficiency, and internal pressure distribution; (2) investigate the impact of each design variation on the resulting flow structure; (3) predict and map erosion rates on the impeller surface using CFD simulations combined with specific erosion models; and (4) identify the optimal impeller design

configuration that achieves the highest hydraulic performance and maximum wear resistance, serving as the basis for design recommendations to improve efficiency and extend the service life of the slurry pump.

This research is expected to enrich understanding of the effects of impeller geometry modifications—especially rounding and blade-number variations—on hydraulic performance and erosion resistance. From an academic perspective, it provides a reference for future studies on pump design using numerical simulations based on CFD. From a practical standpoint, the results can serve as valuable input for industries, particularly in the mining and power generation sectors, to develop pumps that are more efficient, durable, and capable of reducing maintenance costs associated with erosion and performance degradation. Thus, this research is expected to contribute not only theoretically but also as a practical engineering solution for heavy industrial applications.

## METHOD

### Research Flow Diagram

This research was carried out following systematic methodological stages, starting from the conceptual design of the pump, geometry modeling, making mesh, regulating physical conditions, solving process to analysis of results and validation. The flow chart in the stages of the research process can be seen in Figure 1.

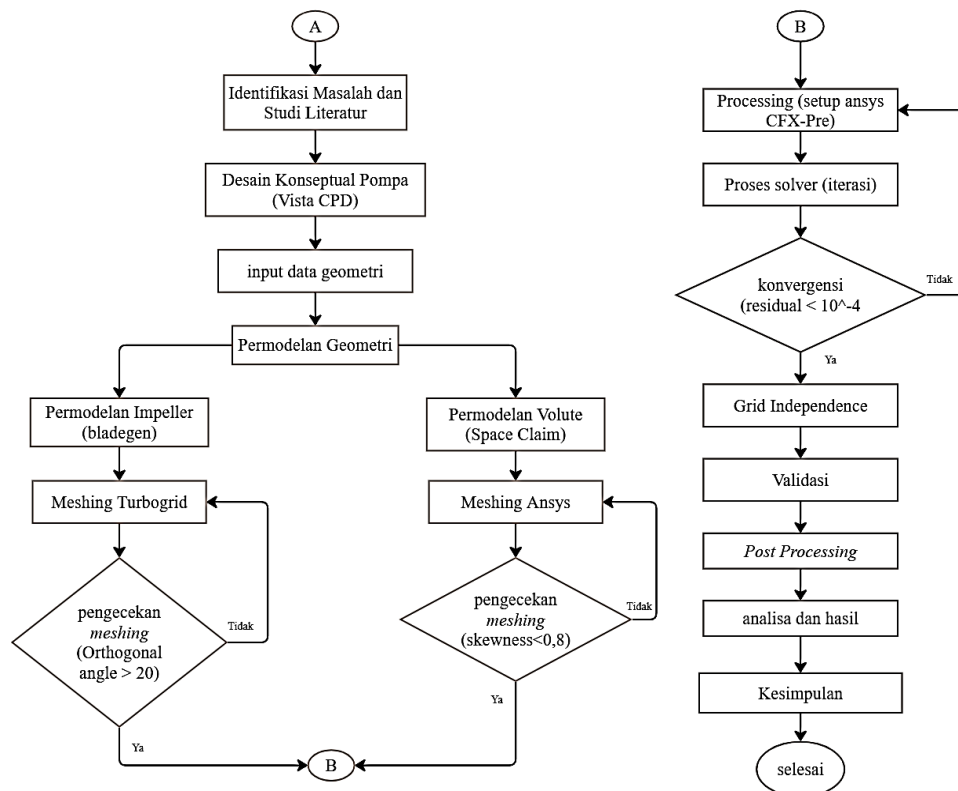


Figure 1. Research Flowchart

### Tahapan Pre-processing

Penelitian ini menerapkan metode numerik Computational Fluid Dynamics (CFD) untuk menganalisis sistem pompa. Proses simulasi diawali dengan tahap pre-processing, yang mencakup pemodelan geometri dilanjutkan dengan diskretisasi atau meshing. Setelah model terdiskretisasi, dilakukan pengaturan parameter fisika dan penentuan kondisi batas di dalam solver. Setiap langkah dalam metodologi ini dilaksanakan dengan saksama untuk menjamin bahwa model numerik yang dikembangkan mampu merepresentasikan kondisi operasi nyata secara akurat. Berikut ini tahapan pertama atau pre-processing yang dilakukan dalam proses ini.

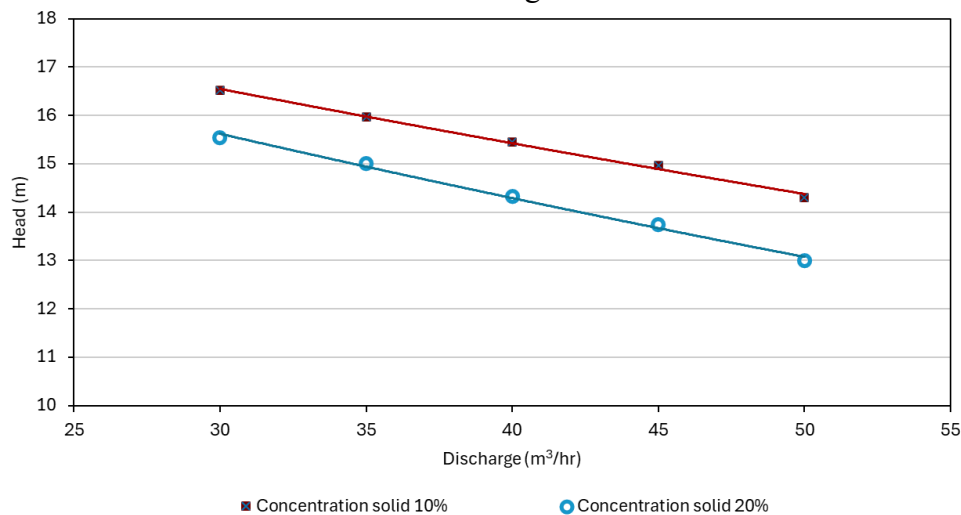
## RESULTS AND DISCUSSION

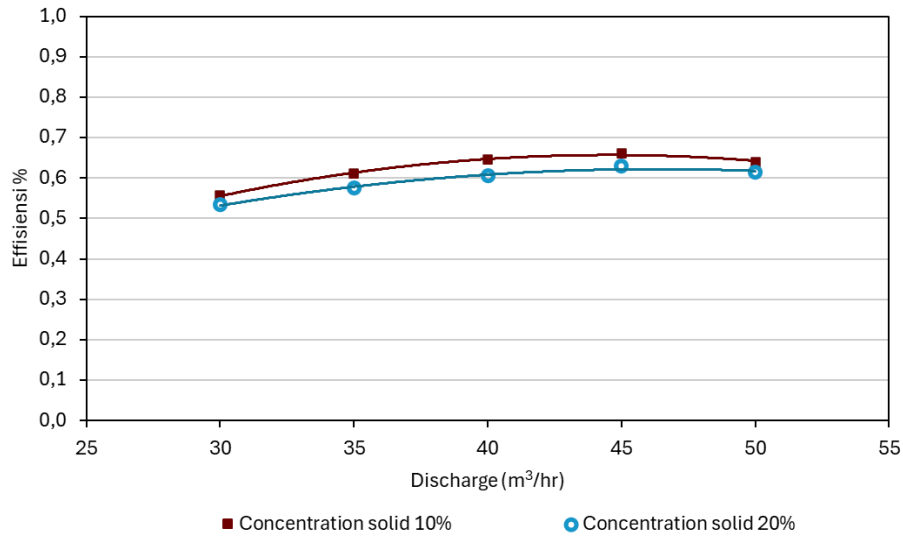
### Effect of Solids Concentration Variations on Pump Performance

This study was conducted to examine the effect of solids concentration variations on the hydraulic performance of centrifugal pumps on slurry flow. Two variations of solids concentrations, namely 10% and 20%, are used as representations of slurry fluids with different density and viscosity characteristics. This difference in solid fraction is expected to have a significant influence on flow patterns, hydraulic losses, and the ability of the pump to produce head and efficiency.

#### Comparison of Head and Efficiency

After numerical calculations, a comparison of heads and efficiency at different solids concentrations is produced. This condition is carried out in a rounding geometry condition with the number of spoons 5. After numerical calculations, the comparison of the Head and the efficiency at different concentrations is shown in Figure 2.





**Figure 2.** Comparison of (a) head (b) efficiency results at 10% and 20% concentrations

This figure shows a comparison of head values and pump efficiency under discharge variations for 10% and 20% solids concentrations. In general, the simulation results indicate that an increase in flow discharge leads to a decrease in pump head at both 10% and 20% solids concentrations. This trend is consistent with the fundamental characteristics of centrifugal pumps, where increasing discharge amplifies hydraulic losses in flow passages and impellers. This decrease in head can be observed more clearly in Figure 4.4, which presents the velocity contour; the increase in flow velocity within the impeller passage indicates a reduction in energy conversion efficiency from mechanical energy to pressure energy.

At a solids concentration of 10%, the pump produces a higher head compared to the 20% condition at the same discharge. This occurs because the lower density and viscosity of the mixture reduce flow resistance and particle interactions. The pressure contour shown in Figure 4.3 demonstrates a more stable pressure gradient at a 10% solids concentration, leading to improved head generation. In contrast, at 20% solids concentration, the higher particle density increases effective viscosity and particle collision intensity, thereby causing greater energy losses and reducing the pump's head-producing capability. This phenomenon is also depicted in Figure 4.4, which shows a broader distribution of velocity and torque at the 20% solids concentration, indicating increased energy dissipation.

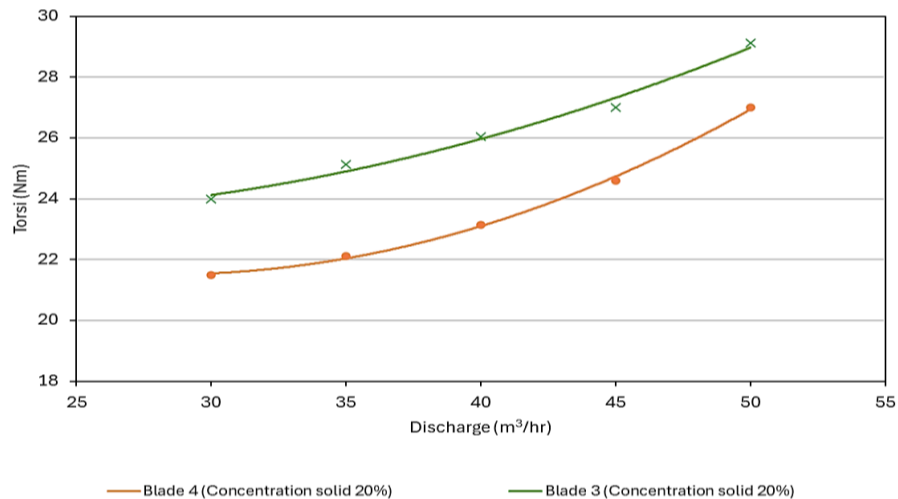
In terms of efficiency, the results show that pump efficiency increases with greater discharge until it approaches the Best Efficiency Point (BEP), after which it tends to stabilize at higher discharge rates. Efficiency at 10% solids concentration is generally higher than at 20%, indicating that a lower solids fraction provides more favorable operating conditions for the pump. As illustrated in Figure 4.3, the flow at 10% solids concentration exhibits a smoother and more uniform pressure distribution, which contributes to higher energy conversion efficiency.

The reduction in efficiency at higher solids concentrations results from increased mechanical and hydraulic losses, including fluid-particle friction, slippage between liquid and

solid phases, and flow disturbances within the impeller and volute. These factors prevent the impeller's mechanical energy from being fully converted into pressure head. As shown in Figure 4.5, the intensified flow disturbances and torque at 20% solids concentration further deteriorate the pump's efficiency.

Overall, these results demonstrate that increasing solids concentration negatively affects the hydraulic performance of the pump, both in terms of head and efficiency. Nevertheless, the observed performance trends remain consistent with centrifugal pump characteristics and slurry flow theory, indicating that the simulation results presented in Sub-Chapter 4.1.3 are physically representative.

### Power Consumption Comparison



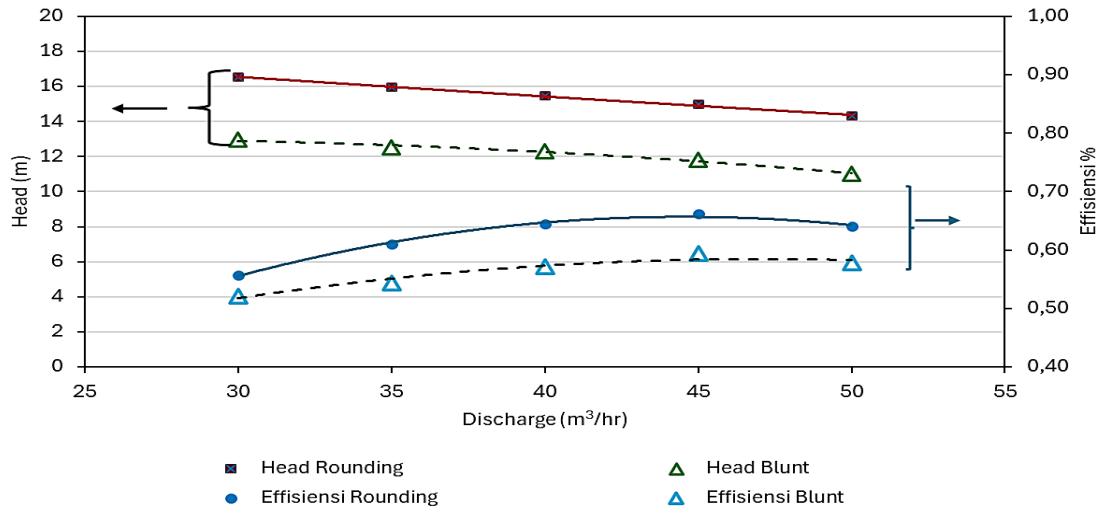
**Figure 3.** Torque ratio at solid concentrations of 10% and 20%

Based on the power–Q graph, it can be seen that power consumption increases as the flow discharge increases in both solids concentration variations. However, at a solids concentration of 20%, the increase in power consumption shows a steeper trend than the 10% concentration, which indicates that the pump requires greater energy to maintain flow discharge in slurry conditions with higher solids fractions.

Overall, these results show that increased solids concentration has a significant effect on increased pump power consumption, especially at higher discharges. These findings confirm that pump operation on slurry fluids with large solids fractions requires special consideration of the drive motor capacity and the overall efficiency of the system.

### Pengaruh Rounding (Rounding vs Non-Rounding)

The analysis of the influence of leading edge geometry was carried out by comparing the hydraulic performance between the blunt profile and the modified profile with the rounding radius. This evaluation is based on the pump characteristic curve (Head and Efficiency) as well as the visualization of the internal flow field.

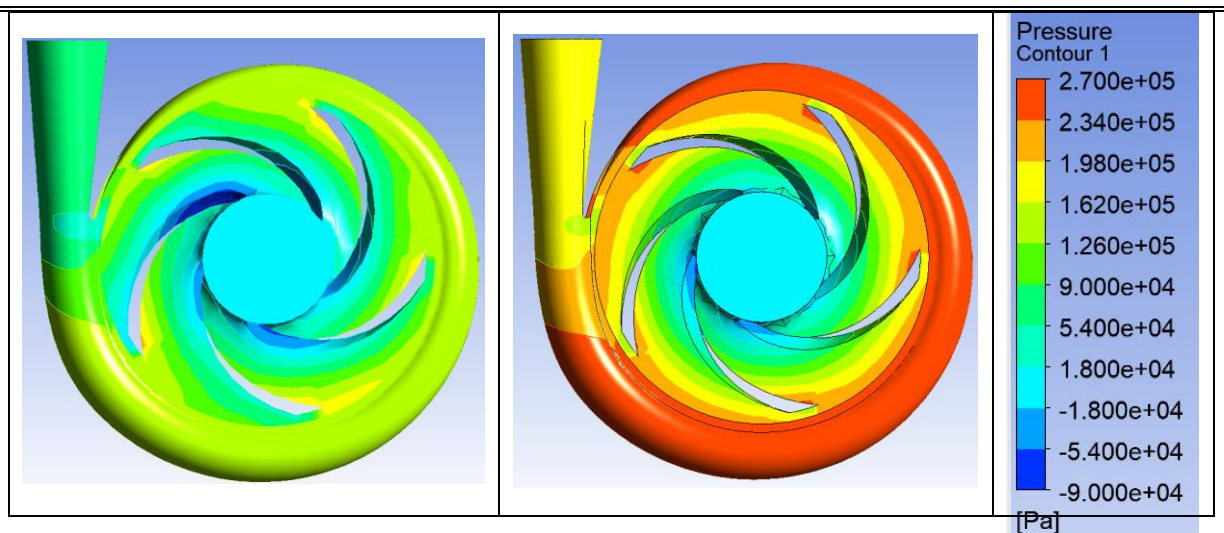


**Figure 4.** Comparison of Head and Performance Efficiency on Blunt and *Rounding Impellers* (solid concentration 20%)

Shows a comparison of centrifugal pump performance characteristics, namely head and efficiency between impeller with rounding leading edge profile and blunt profile at flow discharge variations from 30 m<sup>3</sup>/h to 50 m<sup>3</sup>/h. This comparison aims to evaluate the effect of leading edge geometry modification on the hydraulic performance of the slurry pump.

#### Internal Flow Fields in Blunt and Rounding Geometry

Flow field analysis is performed to understand the physical mechanisms that cause the difference in hydraulic performance between impellers with blunt and rounding leading edge geometry. This flow field evaluation is based on the visualization of the static pressure contour and flow velocity contour inside the impeller, which plays an important role in determining the energy transfer efficiency as well as flow stability in the slurry pump. The result of the pressure contour produced at the 20% concentration condition can be seen in the figure 5.

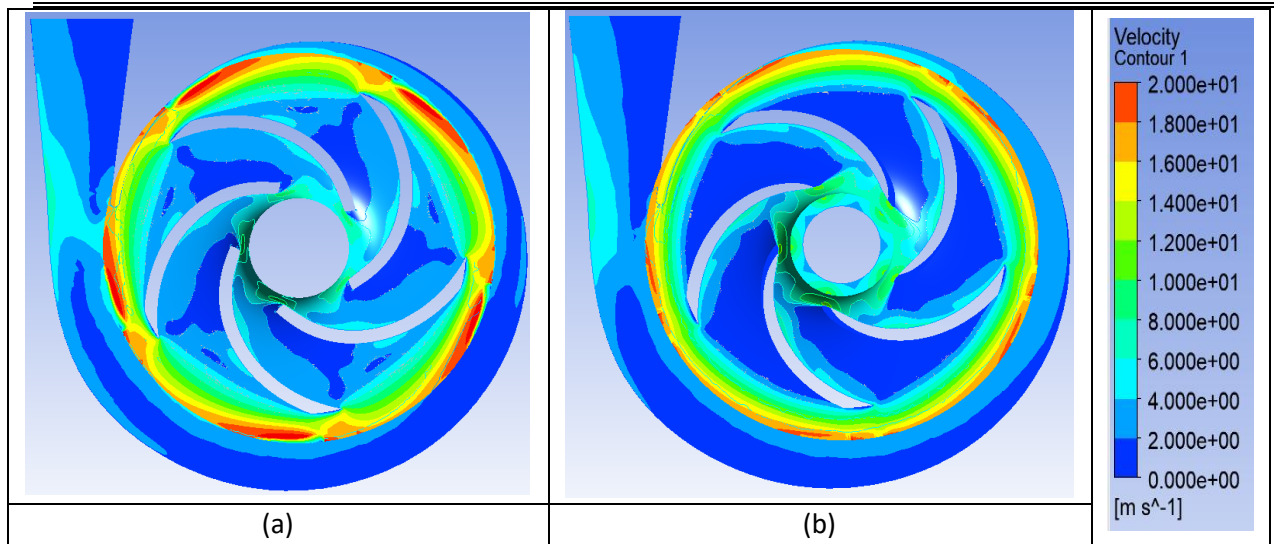




**Figure 5.** Comparison of Pressure Contours on impeller types (a) blunt and (b) rounded

This figure shows the comparison of the pressure contours on the impeller with the blunt and rounded leading edge geometry. In the blunt impeller (Figure 4.7a), a high concentration of static pressure is observed around the leading edge of the blade, which indicates the occurrence of stagnant pressure due to the inlet flow that is less aligned with the blade surface. This uneven pressure distribution and sharp pressure gradient indicate a shock loss in the inlet area, so that some of the flow energy is dissipated before the fluid flows stably in the channel between the blades.

In contrast, on impellers with rounded geometry (Figure 6.7b), the static pressure distribution appears smoother and more evenly along the surface of the blade. The stagnation pressure in the leading edge area is relatively lower, which indicates that the rounding of the leading edge is able to improve the inlet flow conditions and reduce hydraulic losses. This more stable pressure distribution supports a more effective energy transfer process, thus contributing to improved hydraulic performance of the pump compared to blunt geometry.



**Figure 6.** Comparison of Speed contours on impeller types (a) blunt and (b) rounded

The contour of the speed looks smoother and is uniformly distributed following the curvature of the blade. The rounding radius at the leading edge serves to minimize shock loss when fluid enters, so that the kinetic energy transition takes place more efficiently. This improvement in flow quality is the fundamental reason why the rounded impeller is able to produce higher Head and Efficiency than the blunt type in the previous performance graph. The phenomenon of improving flow quality can be clearly observed through the visualization of streamlining speed in Figure.

### Design Determination

The determination of the optimal impeller design was carried out by considering the results of hydraulic performance analysis, internal flow field characteristics, and erosion rate prediction across various geometric modifications and blade numbers. The main evaluation parameters included head value, efficiency, power consumption, flow stability, and resistance to erosion, which are critical factors in slurry pump operation.

Based on the hydraulic performance results, impellers with leading-edge rounding geometry demonstrated superior performance compared to blunt designs, characterized by higher head and efficiency across the entire discharge range. Internal flow field analysis supported these findings, indicating that the rounded-edge design produced a more uniform pressure and velocity distribution while reducing flow separation and turbulence at the impeller inlet. This condition shows that the rounding modification enhances the effectiveness of energy transfer from the impeller to the fluid.

In terms of blade number variation, impellers with  $Z = 5$  yielded the highest head and efficiency due to improved flow guidance and reduced fluid slip. However, regarding erosion, the  $Z = 4$  configuration exhibited the lowest relative erosion rate, reflecting a balance between flow stability and particle impingement area. Nonetheless, the erosion rate at  $Z = 5$  remained within acceptable limits and did not indicate significant deterioration compared to  $Z = 4$ .

Considering all these aspects comprehensively, the combination of leading-edge rounding geometry with five blades ( $Z = 5$ ) was identified as the most optimal impeller design for slurry pump applications under the conditions of this study. This configuration provides the best balance between enhanced hydraulic performance and erosion resistance and is therefore recommended to improve the operational efficiency and service life of slurry pumps.

### CONCLUSION

---

Based on the results of the numerical simulations and analyses, it can be concluded that the application of a leading-edge rounded geometry on the impeller of a slurry pump has a positive effect on inlet flow characteristics by minimizing flow separation and local turbulence, thereby increasing head and efficiency compared to blunt-edge geometry. Variations in blade number also had a significant influence, with the configuration containing five blades ( $Z = 5$ ) producing the most stable flow and the most consistent hydraulic performance, especially at high solids concentrations. In terms of wear resistance, the rounded geometry effectively reduces the angle and velocity of particle impacts, significantly lowering the erosion rate, with the  $Z = 5$  configuration exhibiting the best erosion resistance. Overall, the combination of the leading-edge rounded geometry with a five-blade configuration ( $Z = 5$ ) represents the most optimal design for slurry pumps, as it improves hydraulic performance while minimizing erosion rates.

For future research development, several recommendations can be made. First, conduct analyses at higher solids concentrations to capture the characteristics of non-Newtonian fluids with more complex rheological models. Second, vary the impeller rotational speed to determine the optimal operating range. Third, explore variations in angle and rounding radius on the

leading edge in greater detail. Fourth, expand the investigation to different pump types, such as horizontal and submersible centrifugal pumps. Fifth, incorporate ergonomic and operational aspects, including ease of maintenance, vibration, and noise. Finally, combine Computational Fluid Dynamics (CFD) simulations with experimental testing to enhance result validity, particularly under slurry conditions with high flow complexity.

## REFERENCES

- Doshi, A., Channiwala, S., & Singh, P. (2017). Inlet impeller rounding in pumps as turbines: An experimental study to investigate the relative effects of blade and shroud rounding. *Experimental Thermal and Fluid Science*, 82, 333–348.
- Jaiman, R., Li, G., & Chizfahm, A. (2023). *Mechanics of flow-induced vibration: Physical modeling and control strategies*. Springer Nature.
- Menichino, J. T., Frier, S., Burkardt, N., & Sorokes, J. M. (2024). Impeller manufacturing: Understanding the methods and their impact on performance.
- Paradkar, A. B. (2016). *Evaluation and development of a structurally enhanced PVC water pipe*.
- Parameshwaran, R., Dhulipalla, S. J., & Yendluri, D. R. (2016). Fluid-structure interactions and flow induced vibrations: A review. *Procedia Engineering*, 144, 1286–1293.
- Peng, G., Du, J., Chang, H., Chen, Q., Li, J., Pan, C., & Yang, Y. (2021a). Numerical and experimental analysis of influence of impeller structures on slurry pump performance. *Frontiers in Energy Research*, 9. <https://doi.org/10.3389/fenrg.2021.762159>
- Peng, G., Tian, L., Chang, H., Hong, S., Ye, D., & You, B. (2021b). Numerical and experimental study of hydraulic performance and wear characteristics of a slurry pump. *Machines*, 9. <https://doi.org/10.3390/machines9120373>
- Pradhan, A. R., Kumar, S., & Gupta, C. (2025). Impact of solids on flow patterns and performance of a centrifugal slurry pump: A numerical study. *International Journal on Interactive Design and Manufacturing*, 19(3), 2035–2048. <https://doi.org/10.1007/s12008-024-01990-0>
- Qiu, Z., Xiao, Q., Yuan, H., Han, X., & Li, C. (2024). Particle shape and clogging in fluid-driven flow: A coupled CFD-DEM study. *Powder Technology*, 437, 119566.
- Shojaeefard, M. H., & Saremian, S. (2022). Effects of impeller geometry modification on performance of pump as turbine in the urban water distribution network. *Energy*, 255, 124550.
- Silva, R. C. (2022). Experimental characterization techniques for solid-liquid slurry flows in pipelines: A review. *Processes*, 10, 597.
- Singh, J., & Kumar, S. (2026). *Slurry erosion: Flow phenomenon, complexities, and protection methods*. CRC Press.
- Steiros, K., Bruce, P. J. K., Buxton, O. R. H., & Vassilicos, J. C. (2017). Effect of blade modifications on the torque and flow field of radial impellers in stirred tanks. *Physical Review Fluids*, 2(9), 094802.
- Tarodiya, R., & Gandhi, B. K. (2017). Hydraulic performance and erosive wear of centrifugal

- slurry pumps: A review. *Powder Technology*, 305, 27–38. <https://doi.org/10.1016/j.powtec.2016.09.048>
- Tarodiya, R., & Gandhi, B. K. (2019). Numerical simulation of a centrifugal slurry pump handling solid-liquid mixture: Effect of solids on flow field and performance. *Advanced Powder Technology*, 30, 2225–2239. <https://doi.org/10.1016/j.appt.2019.07.003>
- Tchada, A. M., Tchoumboué, N., Mesquita, A. L. A., & Hendrick, P. (2024). Enhancing pump as turbine (PAT) performances: A numerical investigation into the impact of impeller leading edge rounding. *Heliyon*, 10(15).
- Xie, J., Hou, L., Yang, T., & Li, R. (2025). A comprehensive review of research on the motion behavior and liquid flow dynamics of a single particle impacting a liquid surface. *Physics of Fluids*, 37(3).
- Yadav, Y. K., Patnaik, A., Singh, A. K., & Sehgal, R. (2025). Tribological models for erosive wear in slurry flow: A review: Part I. *Johnson Matthey Technology Review*, 69(3).
- Zhang, Y., Qian, Z., Ji, B., & Wu, Y. (2016). A review of microscopic interactions between cavitation bubbles and particles in silt-laden flow. *Renewable and Sustainable Energy Reviews*, 56, 303–318.
- Zoheidi, L. (2018). *Flow characterization of milk protein foam transport in inner geometries*. Friedrich-Alexander-Universität Erlangen-Nürnberg.

---

**Copyright holder:**

**Emy Zuraida Wuryaningrum\* , Sutardi (2026)**

**First publication right:**

Asian Journal of Engineering, Social and Health (AJESH)

**This article is licensed under:**

