Spiral Casing Design Analysis for Francis Turbine Water Flow Distribution Using CFD Method

Oktavian Provalindo Putra[1], Luchyto Chandra Permadi[2*], Mojibur Rohman[3], Bella Cornelia Tjiptady[4], Ratna Fajarwati Meditama[5], Kiki Darmawan[6]

[1,2,3,4,5,6] *Universitas Islam Raden Rahmat, Malang Jawa Timur, Jl. Raya Mojosari No.2, Dawuhan, Jatirejoyoso, Kec. Kepanjen, Kabupaten Malang, Jawa Timur 65163, Indonesia*

Keywords

Computational Fluid Dynamics; Francis Turbin; Spiral Casing.

****Correspondence Email:***
luchy.chandra@gmail.com

Abstract

The most important component affecting the efficiency of a Francis turbine in converting water energy into electricity is the spiral casing, which functions to distribute water flow evenly to the turbine blades. The decrease in water discharge during the dry season causes suboptimal flow distribution and reduces turbine performance. This study aims to analyze the effect of the spiral casing design on the distribution of pressure, velocity, and flow patterns using the Computational Fluid Dynamics numerical simulation method with ANSYS Fluent software. The method used is the Research and Development approach with two variations of the spiral casing design, namely the 2538 mm inlet design and the 2600 mm inlet design. Simulations were conducted to compare the distribution of pressure, velocity, and streamline patterns in each design. The results showed that the modified design was able to improve flow distribution, reduce turbulent zones, and produce more even pressure and velocity.

1. Introduction

Indonesia's electricity demand continues to increase in line with population growth and industrial development. Reports indicate that global electricity consumption will increase by 4.3% by 2024 (International Energy Agency, 2025). For the past few decades, fossil fuel-based power plants, such as coal and natural gas, have dominated electricity systems in various countries (Haegel & Kurtz, 2025; Igbeghe et al., 2023; Logan et al., 2020). This situation has encouraged many countries, including Indonesia, to switch to renewable energy as a more environmentally friendly and sustainable solution. Indonesia has enormous potential for renewable energy development, with a total capacity of 419 GW, of which 75 GW comes from hydropower (Pambudi et al., 2023). In the National Energy General Plan (RUEN), the government targets the use of new and renewable energy (EBT) of 23% by 2025 (Pambudi et al., 2023 & Junihartomo et al., 2025). One form of implementation of this target is the development of small-scale hydroelectric power plants (PLTA), such as the Wonorejo PLTA in Tulungagung, East Java, which has a capacity of 6.5 MW and uses a Francis water turbine(Mujiburrakhman et al., 2024).

1.1 Literature Review

The Francis turbine is a type of reaction turbine widely used in Indonesia because it is suitable for medium head conditions, namely around 40-400 meters (Dahal et al., 2019; Tomczyk et al., 2023; Umar et al., 2024).

One of the important components of this turbine is the spiral casing, a circular channel that is responsible for distributing water flow evenly to the guide vanes. The efficiency of the turbine system is greatly influenced by the performance of the spiral casing in maintaining stable pressure and flow velocity before reaching the runner (Nakkina et al., 2016; Lazarevikj & Markov, 2022; Hariadi et al., 2021; Shrestha & Choi, 2020).

Its complex design requires highly precise hydraulic calculations, as even small errors can lead to uneven flow distribution (Luo et al., 2024 & Dahal et al., 2019). Uneven flow distribution can cause turbulence, vibration, and reduced system efficiency (B. Umar & Huang, 2024 & Wafo et al., 2024). Furthermore, spiral casings are generally designed for stable flow conditions (Shrestha & Choi, 2020a). In reality, flow fluctuations, particularly during the dry season, can lead to decreased flow distribution performance. This phenomenon was observed at the Wonorejo Hydroelectric Power Plant, where the flow decreased drastically to an average of 3.48 m^3/s during the dry season, compared to a maximum flow of 10.54 m^3/s. This forces the guide vanes to operate in the fully open position to maintain a stable turbine speed of 500 rpm, which impacts efficiency and equipment life. Previous studies were generally conducted under constant flow conditions in the laboratory. Designed a spiral casing for microhydro applications and concluded that the casing geometry significantly influences the pressure distribution pattern. Research by Wafo et al. (2024) on a hydroelectric power plant in Cameroon showed that low-pressure zones due to uneven distribution can trigger cavitation. Meanwhile, (Umar & Huang, 2024)) proved that the flow in the casing becomes unstable at partial loads, especially under low flow conditions.

Few studies have specifically analyzed the performance of spiral casings in Francis turbines under low flow conditions in the field using numerical methods. However, Computational Fluid Dynamics (CFD)-based simulations allow for more detailed and efficient flow visualization than physical experiments. Several studies, such as those by Dumitrache & Deleanu (2022) and Tiwari et al. (2022), have shown that CFD is effective in mapping fluid pressure and velocity distributions in turbine designs. This study aims to analyze the effect of changes in the spiral casing design on pressure distribution, flow velocity, and streamline patterns using CFD simulations with ANSYS Fluent software. Two models are compared, namely the original spiral casing design with an inlet diameter of 2538 mm and a modified design with a diameter of 2600 mm. Through this study, it is expected to obtain a design solution that is more responsive to seasonal discharge fluctuations and is able to increase the efficiency of the water turbine system as a whole.

2. Research Methods

In this section, each researcher is expected to be able to make the most recent contribution related to the solution to the existing problems. Researchers can also use images, diagrams and flowcharts to explain the solutions to these problems.

This research uses a Research and Development (R&D) approach, utilizing Computational Fluid Dynamics (CFD)-based numerical simulation methods to evaluate the performance of two spiral casing designs in a Francis turbine (Li et al., 2025). The simulations were conducted using ANSYS Fluent software, which allows for three-dimensional modeling of fluid flow phenomena.

Geometry Design and Modeling two spiral casing models were analyzed

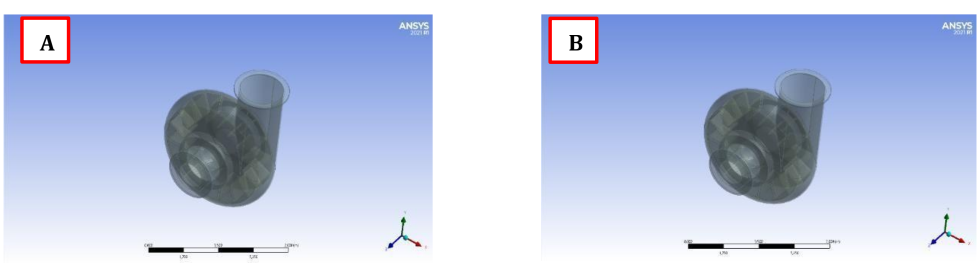

Fig 1. Design Geometry a) Modified Inlet diameter: 2600 mm b) Geometry Inlet diameter: 2538 mm

The spiral casing geometry was created using Autodesk Inventor Pro 2019 based on the technical parameters of the Wonorejo hydropower plant. The 3D modeling focused on the spiral casing, omitting the runner and draft tube to simplify the simulation domain.

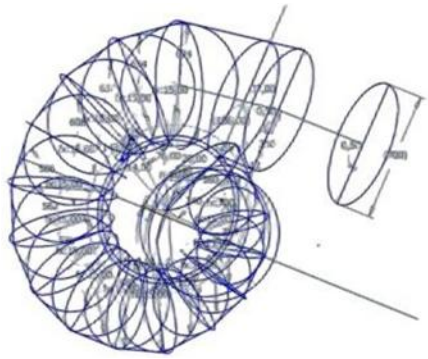

Fig 2. Spiral casing geometry

Mesh and Simulation Domain Creation

The meshing process was performed in the ANSYS Meshing module with a 50 mm element size and tetrahedral element type, resulting in approximately 1.2 million elements. Refinement was performed in the inlet area and near the guide vane to capture high flow gradients.

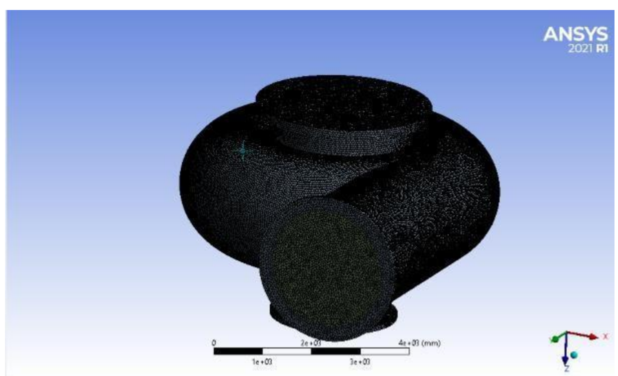

Fig 3. Meshing 50 mm

Boundary Conditions and Simulation Parameters

The numerical simulations in this study were conducted using ANSYS Fluent software to analyze the water flow behavior within the spiral casing of the Francis turbine. The flow was modeled as turbulent to accurately represent the actual operating conditions of the hydropower plant. The realizable k–ε turbulence model was employed due to its proven numerical stability and reliability in simulating internal flow characteristics, as recommended by Shrestha and Choi (2020). The inlet boundary condition was defined based on measured discharge data from the Wonorejo Hydroelectric Power Plant, representing dry-season operating conditions with flow rates ranging from 3.48 m^3/s to 10.54 m^3/s. These values were selected to capture the effects of seasonal discharge variations on the spiral casing performance.

Observed Variables

The analysis focused on three primary flow parameters to evaluate the hydraulic performance of the spiral casing. These parameters included pressure distribution, velocity distribution, and flow patterns within the casing. Pressure distribution was examined to assess flow uniformity and potential low-pressure zones that could affect turbine efficiency and cavitation risk. Velocity distribution was analyzed to understand the acceleration and deceleration of the flow before entering the guide vanes. Additionally, streamline patterns were investigated to visualize flow behavior and identify regions of turbulence, recirculation, or flow separation.

Data Analysis

The simulation results were analyzed qualitatively and quantitatively using contour plots and streamline visualizations generated in ANSYS Fluent. Pressure and velocity contours were used to evaluate spatial variations throughout the spiral casing, while streamline plots provided insight into the overall flow structure and stability under different discharge conditions. This visualization-based approach enabled a clear interpretation of how changes in the spiral casing design influence flow distribution and hydraulic performance.

3. Result and Discussion

In this study, all modeling and simulation parameters were kept uniform, except for the variation in the spiral casing design, namely the difference in inlet size. Before analyzing the pressure distribution, flow velocity, and streamline patterns between designs, the flow distribution visualization results for each spiral casing model were first reviewed. Fig. 4 displays the results of numerical simulations of several design variations analyzed.

Analysis Pressure Distribution

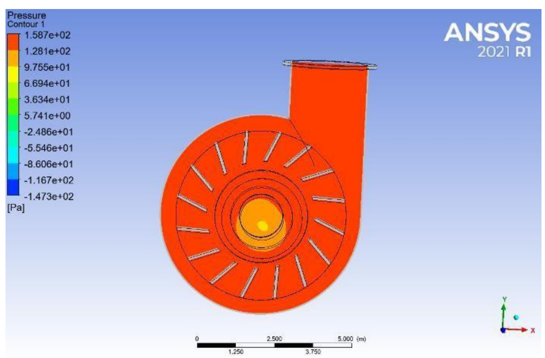

Fig 4. 2D Geometry Pressure Contour 1

The pressure contour visualization in Fig. 4 the 2D pressure contour of geometry 1 shows a fairly even pressure distribution along the spiral casing. The highest distributed pressure value was recorded at 158.7 Pa stably on the inlet side and decreased gradually towards the guide vane. This indicates that the design of geometry 1 is able to direct water flow with controlled and efficient pressure. This can be seen in Fig. 1 the pressure contour graph of geometry 1 below.

Fig 5. Geometric Pressure Contour 1

The following are the average stress values at several observation points in Geometry design 1:

Table 1. Pressure Estimates at Three Observation Points

Observation Point	Pressure (Pa)
Inlet	158,7
Center Spiral	130,2
Near Guide Vane	110,6

Table 1 presents the average pressure values at three observation points in the spiral casing of Geometry 1 based on the CFD simulation results. The highest pressure was detected at the inlet with a value of 158.7 Pa, indicating the concentration of fluid energy when it first enters the casing. Subsequently, the pressure gradually decreases to 130.2 Pa at the center of the spiral, and reaches 110.6 Pa near the guide vane. This pressure drop pattern is gradual and relatively stable, indicating that the Geometry 1 design is able to distribute pressure evenly along the casing. This distribution supports flow stability and minimizes the potential for sharp pressure gradients that can cause flow disturbances such as turbulence or cavitation.

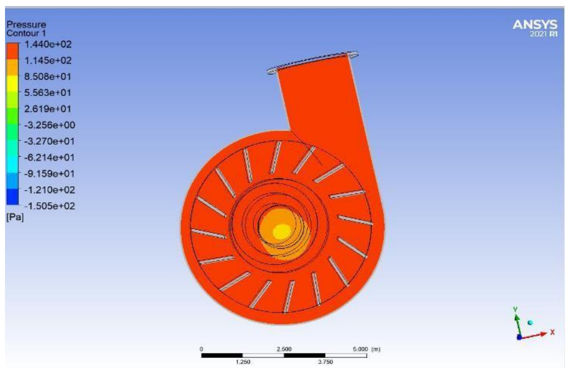

Fig 6. 2D Geometry Pressure Contour 2

Meanwhile, the pressure contour visualization in Fig. 6 for Geometry 2 shows an uneven pressure distribution along the spiral casing. The highest pressure value recorded was 147 Pa and tended to accumulate on the inlet side, then decreased sharply towards the guide vane. This pattern indicates that the Geometry 2 design is not yet able to direct water flow efficiently with controlled pressure. This can be seen in Fig. 7. The pressure contour graph for Geometry 2 is shown in the following figure.

Fig 7. Geometric Pressure Contour 1

The following are the average stress values at several observation points in Geometry design 2:

Table 2. Pressure Estimates at Three Observation Points

Observation Point	Pressure (Pa)
Inlet	147,4
Center Spiral	118,9
Near Guide Vane	95,2

Table 2 presents the average pressure values at three strategic points in the spiral casing of Geometry 2, the original spiral casing design. The highest pressure was recorded at 147.4 Pa at the inlet point, then decreased to 118.9 Pa at the center of the spiral, and reached 95.2 Pa near the guide vane. The sharp pressure drop from the inlet to the guide vane indicates that the pressure distribution in Geometry 2 is not as optimal as in Geometry 1. A large pressure gradient can cause flow imbalance, increase the potential for turbulence formation, and reduce the efficiency of fluid flow to the turbine runner. This uneven pressure distribution pattern indicates that the Geometry 2 design still has limitations in directing water flow stably and efficiently along the spiral casing (Altimemy et al., 2019; Krzemianowski & Steller, 2021; Yang et al., 2023).

Based on the CFD simulation results, a significant performance difference is seen between Geometry 1 (modified) and Geometry 2 (original) in terms of pressure distribution, velocity, and flow pattern (streamline). In terms of pressure, Geometry 1 shows a more stable and even distribution, with a maximum pressure of 158.7 Pa at the inlet and a gradual decrease up to the guide vane. In contrast, Geometry 2 recorded a maximum pressure of 147.4 Pa with a sharper decrease, indicating a less controlled high pressure gradient. This even pressure distribution is in accordance with the findings of Qin et al. (2025) who stated that an optimally designed spiral casing is able to produce a more even pressure distribution and high flow efficiency.

Analysis Speed Distribution

The velocity contour visualization in Fig. 8 shows a fairly even velocity distribution pattern along the spiral casing for the Geometry 1 design. The highest velocity value recorded was 19.6 m/s and was distributed stably on the inlet side, then decreased gradually towards the guide vane. This pattern indicates that Geometry 1 is able to direct the water flow at a controlled and efficient speed. Details of the speed at several observation points can be seen in Fig. 9. The velocity contour graph of Geometry 1 is in Fig. 8.

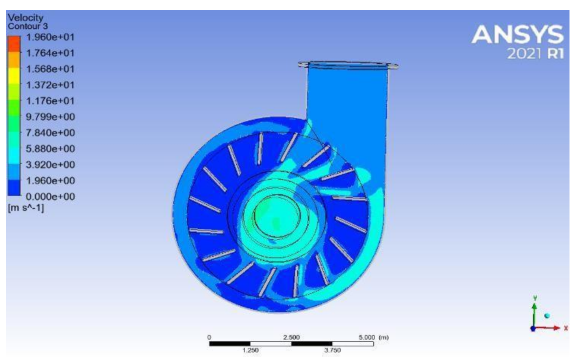

Fig 8. 2D Geometry Velocity Contour 1

Fig 9. Geometric speed range 1

The average speed values at several observation points in Geometry Design 1 are shown in the following table:

Table 3. Estimated Speed at Three Observation Points 1

Observation Point	Speed (m/s)
Inlet	1.96
Center Spiral	7.84
Near Guide Vane	13.7

Table 3. above shows the average fluid velocity values at three observation points within the spiral casing. At the inlet point, the fluid velocity starts at 1.96 m/s, reflecting the initial flow into the casing. Interestingly, the velocity actually increases as the flow moves toward the guide vanes, with a value of 7.84 m/s at the center of the spiral, and reaching 13.7 m/s near the guide vanes. This pattern of increasing velocity indicates that the spiral casing design successfully accelerates the flow gradually and directionally toward the guide vanes. This is very advantageous in hydropower systems because the fluid's kinetic energy increases in the critical area, namely before entering the turbine runner. This uniformly accelerated flow contributes to hydraulic efficiency and reduces the potential for vortex formation along the casing.

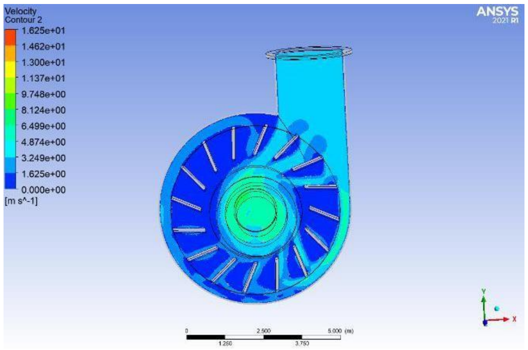

Fig 10. 2D Geometry Velocity Contour 2

Meanwhile, the velocity contours of the simulation results for Geometry 2 show an uneven flow distribution pattern from the inlet to the guide vane. The maximum velocity recorded was 17.8 m/s at the inlet area, but there was a sharp and unstable decrease along the spiral casing. In the center of the casing, the velocity distribution appears asymmetrical and indicates flow disturbances in the form of stagnation zones or low-velocity areas. This irregularity in the velocity pattern indicates that Geometry 2 is not yet able to efficiently direct the water flow to the guide vane. This uneven distribution of kinetic energy can cause flow imbalance, increase local turbulence, and reduce the overall hydraulic efficiency of the turbine system. This can be seen in Fig. 10. The velocity contour graph for Geometry 2 is shown in the following figure.

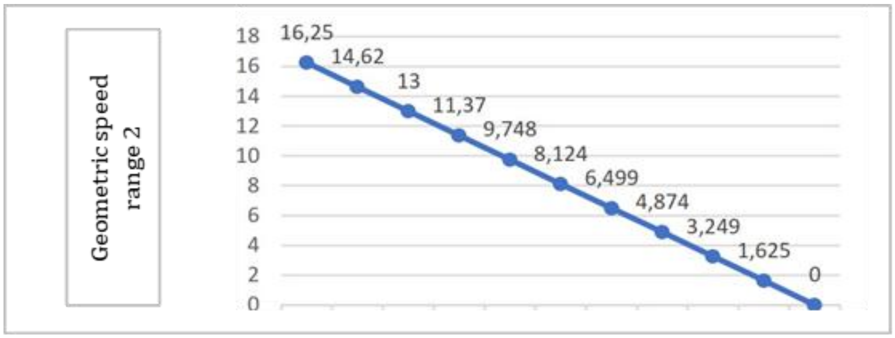

Fig 11. Geometric speed range 2

The average speed values at several observation points in Geometry Design 2 are shown in the following table:

Table 3. Estimated Speed at Three Observation Points 2

Observation Point	**Speed (m/s)**
Inlet	1.6
Center Spiral	6.5
Near Guide Vane	11.4

The average fluid velocity values at three observation points in the spiral casing of Geometry 2 are shown in Table 3. At the inlet, the velocity was recorded at 1.6 m/s, increasing to 6.5 m/s in the center of the spiral, and reaching 11.4 m/s near the guide vane. Despite this increase, the velocity distribution pattern indicates a tendency for uneven flow. The relatively low initial velocity value at the inlet indicates the potential for the formation of a stagnation zone or slow flow that could cause disturbances upstream of the spiral casing. Furthermore, the significant velocity jump from the inlet to the downstream indicates a sharp velocity gradient, which could lead to an imbalance in the distribution of kinetic energy (Abu & Kim, 2024; Showkat et al., 2025). This pattern supports the previous visual findings of the velocity contours that Geometry 2 has less efficient flow performance and tends to produce local turbulence that is detrimental to turbine efficiency.

In terms of velocity, Geometry 1 recorded a gradual and consistent flow, from 19.6 m/s at the inlet to 12.2 m/s near the guide vane, indicating better flow stability and kinetic energy control. Meanwhile, Geometry 2 showed an uneven and less symmetrical velocity distribution pattern, from 17.8 m/s to 9.7 m/s. This indicates the potential for flow imbalance and the formation of local turbulence. Wang et al. (2025) stated that small changes in the spiral casing design can significantly affect the direction and stability of the flow, even if the changes are not statistically significant.

Analysis Polar Streamline

The simulation results in Fig 12. show that the streamline pattern in Geometry 1 has stable and symmetrical flow characteristics from the inlet to the guide vane. The streamlines follow the spiral shape of the casing tangentially without any major disturbances, such as eddies or stagnation zones. This pattern distributes fluid flow efficiently throughout the circumference of the casing, ensuring that each guide vane receives an even fluid supply. This good flow distribution reflects the design capability of Geometry 1 in maintaining flow stability, as well as reducing the potential for vortex formation or turbulence that can reduce system efficiency. This stability also supports the process of converting energy from pressure to kinetic energy more effectively. This can be seen in Table 8. The pressure contour graph of Geometry 2 is shown in Fig 1.

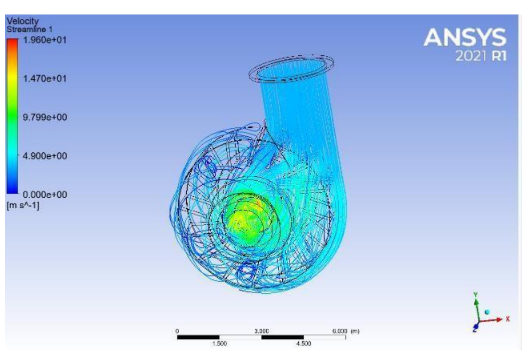

Fig 12. Streamline Geometry 1

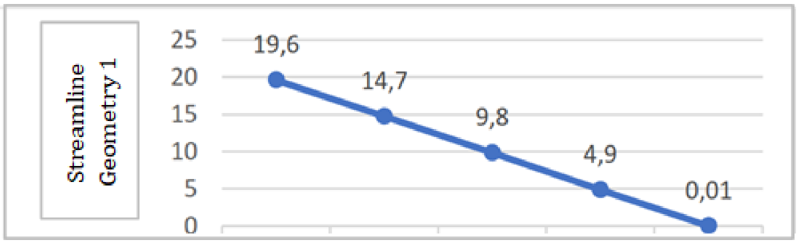

Fig 13. Streamline Geometry 1

Fig. 13 Geometry 1's streamline graph shows a smooth and consistent decrease in value from 19.6 to 0.01, indicating stable and efficient fluid flow along the casing spiral. The streamlines follow the casing contour without significant disruption, indicating symmetrical flow distribution and minimal turbulence. This reflects good hydraulic performance and design efficiency in optimally directing flow toward the guide vane.

Meanwhile, the streamline pattern simulated for Geometry 2 shows uneven flow distribution from the inlet to the guide vanes. Although the flow appears quite controlled at the inlet, irregularities begin to appear along the spiral casing, particularly in the center. The streamlines do not completely follow the tangential contour of the spiral, and there is even a tendency for small eddies to form and deviations in flow direction. This pattern indicates that Geometry 2 is not yet able to efficiently direct the water flow to the guide vanes. This uneven flow distribution can cause pressure imbalances, local turbulence, and reduce the overall hydraulic efficiency of the turbine system. This is shown in Fig. 15. A visualization of the streamline pattern for Geometry 2 is shown in the following fig. 14.

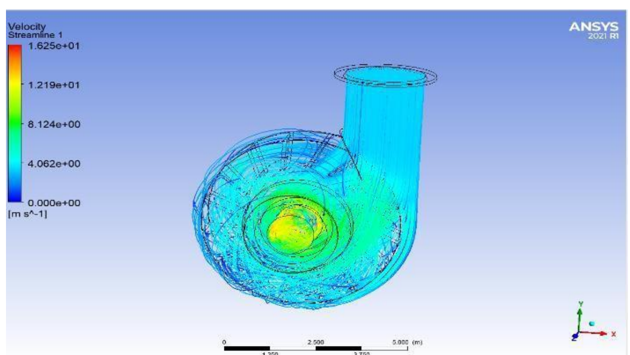

Fig 14. Streamline Geometry 2

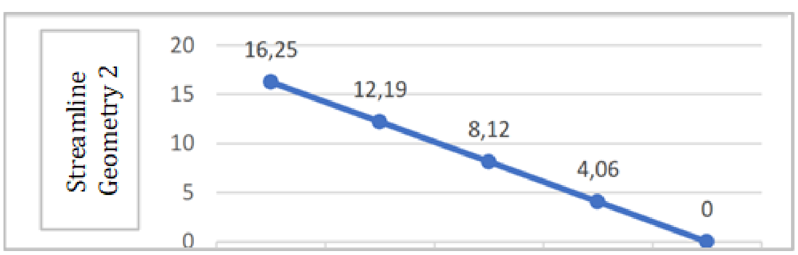

Fig 15. Streamline Geometry 2

Visualization of the streamline pattern also reinforces the advantages of Geometry 1. The fluid flow lines follow the spiral contour of the casing tangentially and are well-distributed without any eddies or stagnant zones. In contrast, Geometry 2 exhibits a divergent flow pattern and forms areas with potential vortices. This is in line with the view of Sudsuansee et al. (2025) and Lee et al. (2025) who stated that suboptimal casing design can cause flow deviations and increase potential energy losses. Technically, Geometry 1 showed better performance, with stable flow, uniform pressure, and efficient flow direction. This statistical insignificance can be attributed to the relatively small size of the geometric differences, especially in the inlet dimensions. Although the statistical results were not significant, Geometry 1 was visually and technically proven to be superior in terms of pressure distribution, flow velocity, and streamline stability, which contributed to the hydraulic efficiency of the Francis turbine system.

There is no fixed recipe for presenting the findings of a study. We will, therefore, first consider general guidelines and then turn our attention to options for reporting descriptive statistics and the results of the hypothesis test.

4. Conclusions

Based on the results of research that has been conducted through modeling and numerical simulation using the Computational Fluid Dynamics (CFD) method on two Francis turbine spiral casing designs. The spiral casing design with an inlet size enlarged to 2.6 cm in Geometry 1 produces a more even pressure distribution along the casing path. The maximum pressure achieved is 150.7 Pa higher than Geometry 2, the maximum pressure achieved is 144.0 Pa, and the decrease is more stable, without any extreme pressure spikes. Changes in the spiral casing design affect the water flow pattern (streamline). Geometry 1 displays a more regular, symmetrical, and non-vortex flow pattern, while Geometry 2 shows flow irregularities and recirculation in several areas. Showing that the Geometrys 1 design is more optimal in directing the flow towards the turbine runner. 3. Geometry 1 provides a more even flow distribution than Geometry 2. It can be seen from the more stable flow, more balanced pressure and speed, and the flow pattern that follows the spiral casing shape well. Therefore, Geometry 1 is considered more efficient and suitable for use in low water discharge conditions.

5. References

Abu, M., & Kim, J. (2024). Investigation of role of fins in a Francis turbine model ' s cavitation-induced instabilities under design and off-design conditions. Energy, 292(January), 130514. https://doi.org/10.1016/j.energy.2024.130514

Altimemy, M., Attiya, B., Daskiran, C., Liu, I., & Oztekin, A. (2019). International Journal of Heat and Fluid Flow Mitigation of flow-induced pressure fluctuations in a Francis turbine operating at the design and partial load regimes — LES simulations. International Journal of Heat and Fluid Flow, 79(March), 108444. https://doi.org/10.1016/j.ijheatfluidflow.2019.108444

Dahal, D., Chitrakar, S., Kapali, A., Thapa, B., & Neopane, H. (2019). Design of Spiral Casing of Francis Turbine for Micro Hydro Applications. Journal of Physics Conference Series, 1266, 12013. https://doi.org/10.1088/1742-6596/1266/1/012013

Dahal, D. R., Chitrakar, S., Kapali, A., Thapa, B. S., & Neopane, H. P. (2019). Design of Spiral Casing of Francis Turbine for Micro Hydro Applications Design of Spiral Casing of Francis Turbine for Micro Hydro Applications. Journal of Physics: Conference Series, 1266(012013). https://doi.org/10.1088/1742-6596/1266/1/012013

Dumitrache, C., & Deleanu, D. (2022). SPIRAL TURBINE CASING -IMPELLER ASSEMBLY DESIGN AND FLUID

FLOW ANALYSIS. In International Journal of Modern Manufacturing Technologies (Vol. 14). https://doi.org/10.54684/ijmmt.2022.14.2.69

Haegel, N. M., & Kurtz, S. R. (2025). Global Progress Toward Renewable Electricity : Tracking the Role of Solar (Version 4). IEEE Journal Of Photovoltaics, 15(2), 206–214.

Hariadi, Muchlisinalahuddin, & Meilisa, M. (2021). Analisis Perbandingan Unjuk Kerja Turbin PLTA Batang Agam Terhadap Kondisi Pada Saat Komisioning. Rang Teknik Journal, 4(2), 398–405. https://doi.org/https://doi.org/10.31869/rtj.v4i2.2667

Igbeghe, C. B., Mizik, T., Gabnai, Z., & Bai, A. (2023). Trends and Characterization of Primary Energy Sources by Energy and Food Prices. Energies, 16, 3066. https://doi.org/https:// doi.org/10.3390/en16073066 Academic

International Energy Agency. (2025). Electricity 2025 Analysis and forecast to 2027. IEA Publications. www.iea.org

Junihartomo, M. T. C., Thamrin, S., & Boedoyo, S. (2025). Potential Analysis and Regulations of Solar Power Plant Development in Indonesia. International Journal of Innovative Science and Research Technology, 7(4), 518–522. https://doi.org/https://doi.org/10.5281/zenodo.6519378

Krzemianowski, Z., & Steller, J. (2021). High speci fi c speed Francis turbine for small hydro purposes - Design methodology based on solving the inverse problem in fl uid mechanics and the cavitation test experience. Renewable Energy, 169, 1210–1228. https://doi.org/10.1016/j.renene.2021.01.095

Lazarevikj, M., & Markov, Z. (2022). Automated hydraulic design procedure for a Francis turbine spiral casing.

Lee, J., Sadia, S., Shrestha, U., Adeeb, E., Ham, S., Kim, S., Park, J., Ko, K., Choi, Y., & Ha, H. (2025). Energy Conversion and Management : X Francis turbine instabilities during off-design and transient operations : A comprehensive review of the state-of-the-art and recent research and development. Energy Conversion and Management: X, 28(November), 101384. https://doi.org/10.1016/j.ecmx.2025.101384

Li, L., Zhao, D., Sun, W., & Huang, Q. (2025). A micro Francis turbine for smart water meter self-generation : Study on the effects of structural parameters on turbine performance based on CFD simulations and experiments $. Energy, 328(52179089), 135865. https://doi.org/10.1016/j.energy.2025.135865

Logan, B. E., Rossi, R., Baek, G., Shi, L., Oconnor, J., & Peng, W. (2020). Energy Use for Electricity Generation Requires an Assessment More Directly Relevant to Climate Change. ACS Energy Letters, 351(11), 3514–3517. https://doi.org/10.1021/acsenergylett.0c02093

Luo, Y., Li, Z., Zhang, S., Ren, Q., & Wang, Z. (2024). Hydrodynamic Characteristics of Preloading Spiral Case and Concrete in Turbine Mode with Emphasis on Preloading Clearance. Processes, 12, 2056. https://doi.org/10.3390/pr12092056

Mujiburrakhman, M., Jayadi, R., Istiarto, & Nurdianyoto, I. (2024). An optimization model for integrated water allocation in the parallel system of Segawe regulating weir and Wonorejo multipurpose dam, Tulungagung, East Java. IOP Conference Series: Earth and Environmental Science, 1343, 012021. https://doi.org/10.1088/1755-1315/1343/1/012021

Nakkina, P., Prakash, K., & Gurunathan, S. K. (2016). Numerical studies on fluid flow characteristics through different configurations of spiral casing. Engineering Applications of Computational Fluid Mechanics, 10, 297–311. https://doi.org/10.1080/19942060.2016.1149103

Pambudi, N. A., Firdaus, R. A., Rizkiana, R., Ulfa, D. K., Salsabila, M. S., Suharno, & Sukatiman. (2023). Renewable Energy in Indonesia : Current Status , Potential , and Future Development. Sustainability, 15, 2342. https://doi.org/https://doi.org/10.3390/ su15032342 Academic

Qin, Y., Li, D., Shi, Q., Wei, X., Wang, X., & Zhu, B. (2025). Transient instability elimination during turbine runaway process of an. Energy, 336(April), 138552. https://doi.org/10.1016/j.energy.2025.138552

Showkat, S., Harmain, G. A., & Masoodi, J. H. (2025). Cavitation control in Francis turbines by design optimization across variable operating conditions. Next Energy, 9(April), 100391. https://doi.org/10.1016/j.nxener.2025.100391

Shrestha, U., & Choi, Y. Do. (2020a). Improvement of flow behavior in the spiral casing of Francis hydro turbine model by shape optimization. Journal of Mechanical Science and Technology, 34, 1–10. https://doi.org/10.1007/s12206-020-0817-9

Shrestha, U., & Choi, Y. Do. (2020b). A CFD-Based Shape Design Optimization Process of Fixed Flow Passages in a Francis Hydro Turbine. Processes, 8, 1392. https://doi.org/10.3390/pr8111392

Sudsuansee, T., Phitaksurachai, S., Pan-aram, R., & Sritrakul, N. (2025). Design and hydrodynamic performance of low head propeller hydro turbine for wide range high efficiency operation. International Journal of Thermofluids, 27(April), 101228. https://doi.org/10.1016/j.ijft.2025.101228

Tiwari, G., Kumar, J., Prasad, V., & Patel, V. kumar. (2022). CFD Investigation for Surface Roughness Effects on the Hydrodynamics of Cavitating Turbulent Flow through a Low Head Prototype Francis Turbine. Journal of Applied Fluid Mechanics, 15, 1593–1607. https://doi.org/10.47176/jafm.15.05.33301

Tomczyk, P., Mastalerek, K., Wiatkowski, M., Kuriqi, A., & Jurasz, J. (2023). Assessment of a Francis Micro Hydro Turbine Performance Installed in a Wastewater Treatment Plant. Energies, 16, 7214. https://doi.org/https://doi.org/ 10.3390/en16207214

Umar, B., & Huang, X. (2024). Numerical Investigation of Performance and Flow Characteristics of Francis Turbine in Part Load Condition. In Journal of Physics: Conference Series (Vol. 2752). https://doi.org/10.1088/1742-6596/2752/1/012006

Umar, B. M., Huang, X., & Wang, Z. (2024). Experimental Flow Performance Investigation of Francis Turbines from Model to Prototype. Applied Sciences, 14, 7461. https://doi.org/https:// doi.org/10.3390/app14177461

Wafo, R. D., Tientcheu-Nsiewe, M., Tcheukam-Toko, D., Kenmeugne, B., & Kuitche, A. (2024). Flow Modeling in the Wake of a Francis Turbine with Hydraulic Charge : Case of Songloulou-Cameroon. Journal of Applied Fluid Mechanics, 17(4), 926–938. https://doi.org/https://doi.org/10.47176/jafm.17.4.1900 Flow

Wang, D., Zhou, R., Li, M., Zhang, K., Wang, Z., & Li, K. (2025). International Journal of Heat and Fluid Flow Study on the influence of tube type and fluid flow channel on cooling heat transfer characteristics of supercritical CO 2 in spirally grooved casing tubes. International Journal of Heat and Fluid Flow, 114(March), 109804. https://doi.org/10.1016/j.ijheatfluidflow.2025.109804

Yang, J., Peng, C., Li, C., Liu, X., Liu, J., & Wang, Z. (2023). Design and verification of Francis turbine working in sand laden. Renewable Energy, 207(51876099), 40–46. https://doi.org/10.1016/j.renene.2023.02.088