COMPARATIVE ANALYSIS OF NOZZLE BLADE PRESSURE IMPACT ON PELTON TURBINE BUCKETS

[*1]Fikra Maulana,[2]Desmarita Leni,[3]Muchlisinalahuddin

[1,2,3]Mechanical Engineering, Faculty of Engineering
Universitas Muhammadiyah Sumatera Barat, Indonesia

Author's email:
[1]fikramaulana93@gmail.com; [2]desmaritaleni@gmail.com; [3]muchlisinalahuddin.umsumbar@gmail.com

**Corresponding author:desmaritaleni@yahoo.com*

Abstract. *The Pelton turbine is a hydroelectric power generation technology whose performance is greatly influenced by the nozzle design, particularly the angle of the water spray to the blades. This study aims to analyze the effect of varying nozzle angles (15°, 30°, and 35°) on the pressure and force distribution on the Pelton turbine blades. Fluid flow simulations were performed using the Computational Fluid Dynamics (CFD) method with SolidWorks Flow Simulation to model the fluid interaction with the blades in detail. The simulation results show that an angle of 15° produces maximum impact force in the FY direction with a very sensitive response to increasing pressure, while an angle of 35° dominates the force in the FZ direction at high pressure. An angle of 30° provides a more stable force distribution. The conclusion of this study confirms that selecting the right nozzle angle is crucial for optimizing energy transfer and maintaining the structural durability of the Pelton turbine.*

Keywords: *CFD; Nozzle Angle; Pelton Turbine; Pressure Distribution*

1. INTRODUCTION

The global need for sustainable and efficient renewable energy sources is driving the development of reliable energy conversion technologies. Hydropower, as a renewable energy source, offers advantages in terms of supply continuity, high efficiency, and relatively low environmental impact. In hydroelectric power systems with large head falls, Pelton turbines are the primary choice due to their ability to convert fluid potential energy into mechanical energy through an impulse mechanism. (Ma'ruf et al., 2023).

A Pelton turbine works by directing a high-pressure jet of water from a nozzle onto the surface of a bucket. The interaction between the jet and the bucket produces an impulse force that causes the turbine shaft to rotate. (Karudin et al., 2025) Fluid pressure at the nozzle is a crucial parameter because it directly affects the velocity and kinetic energy of the flow transmitted to the bucket. Variations in nozzle pressure can cause significant changes in the pressure distribution on the bucket surface, which ultimately affects the energy conversion efficiency and structural reliability of turbine components (Setyawan & Tetuko, 2023).

While numerous studies have examined the general performance of Pelton turbines, comprehensive studies specifically comparing the effect of nozzle pressure on the pressure distribution in the bucket are relatively limited. A numerical approach based on Computational Fluid Dynamics (CFD) allows for detailed visualization and analysis of pressure on the bucket surface, allowing for more accurate evaluation of potential design optimizations and efficiency improvements. (Darmawan et al., 2023).

This research focuses on a comparative analysis of the effect of nozzle pressure variations on pressure distribution in a Pelton turbine bucket. The results of this analysis are expected to enrich technical understanding of fluid-structure interactions in impulse turbine systems and provide more efficient and reliable design recommendations for actual hydropower plant implementations. (Tonadi, 2021).

2. LITERATURE REVIEW

2.1 Water Turbine

A water turbine is a key component in a hydroelectric power plant (PLTA) system, converting the potential and kinetic energy of water into mechanical energy in the form of shaft rotation, which is then converted into electrical energy by a generator. One of the advantages of a water turbine is its use as an environmentally friendly and efficient technology. (Hidayat, 2019), by utilizing the continuously available water flow.

Based on the working principle and type of flow, water turbines can be classified into reaction turbines and impulse turbines. a. Reaction turbines, such as Francis and Kaplan turbines, utilize changes in pressure and fluid flow velocity in the turbine blades, making them suitable for areas with low to medium water head but high-water discharge. b. Impulse turbines, such as Pelton turbines, utilize changes in momentum from high-speed water jets hitting the turbine blades, and are ideal for use in areas with high head values and low to medium water discharge.

2.2 Pelton Turbine

Education serves as a key driver of change in society and contributes directly to almost all SDGs. Quality education (SDG 4) not only provides basic skills such as literacy and numeracy, but also builds the capacity of individuals to actively participate in social, economic, and political life. Sustainable development-based education encourages students to understand the relationship between their actions and their impact on the world globally.

The Education for Sustainable Development (ESD) program initiated by UNESCO aims to integrate sustainability principles into all aspects of education. This includes curriculum development that emphasizes global problem-solving, social innovation, as well as green entrepreneurship. This kind of education is needed to equip the younger generation with the critical thinking skills and transformative skills needed to implement solutions to the challenges of the SDGs.

In addition, education plays an important role in narrowing social and economic disparities by increasing access to knowledge and technology. Equitable education paves the way for social justice and poverty reduction (SDG 1). Education also serves as a mechanism to build community resilience to natural disasters and climate change, by increasing risk awareness and adaptation skills at the local level.

2.3 Techniques in Achieving the SDGs

The Pelton turbine is a type of impulse turbine designed to convert the kinetic energy of a high-pressure water jet into mechanical energy from the rotation of the runner and turbine shaft. Invented by Lester Allan Pelton, this turbine is widely used in areas with high head values. (Susanto et al., 2019).

The working principle of the Pelton turbine Relying on the energy of a water jet that exits a nozzle at high speed and is directed toward a double-bowl-shaped blade (bucket). This water jet splits to fit the bucket shape and deflects the flow by 160–170°, enabling almost all of the water's kinetic energy to be converted into mechanical energy with a high degree of efficiency. (Pratama et al., 2021).

Main components of a Pelton turbine consist of:

1. The nozzle and needle valve function to control the water flow and jet pressure to obtain an efficient flow pattern and minimize turbulence.
2. The runner and bucket function to receive the energy of the water jet and convert it into mechanical energy. The symmetrical design of the bucket allows the water flow to be divided and the energy to be utilized with high efficiency.
3. The casing functions as a protector and channel for residual water after hitting the bucket, ensuring the work area remains dry and safe.
4. The shaft functions to transmit the runner's rotation to the transmission system or directly to the generator to be converted into electrical energy.

Characteristics and Advantages of the Pelton turbine include:
1) High efficiency, can reach values of more than 90%.
2) Optimal working capability in areas with high head values.
3) Fast operational response with the help of the nozzle and needle valve control system.
4) Relatively simple and durable structure, easy maintenance, and operation.

2.3 Nozzles and Spray Angles

The nozzle plays a key role in the operation of the Pelton turbine, converting the potential energy of water into kinetic energy by creating a high-speed jet of water that strikes the turbine blades. One critical parameter of the nozzle is the spray angle, which affects the efficiency of the water impact with the bucket. An inappropriate spray angle can result in energy loss due to splashing, turbulence, and inefficient water impact. The selection of the ideal spray angle must take into account the bucket design, number of nozzles, runner configuration, and the working head value of the turbine. (Riani et al., 2022a).

Studies using CFD simulation methods are also used to optimize the spray angle to reduce the effects of turbulence and prevent cavitation phenomena, thereby ensuring a higher level of efficiency and a longer turbine life.

2.4 Flow Pressure and Turbine Performance

The pressure of the water flow from the reservoir to the nozzle determines the value of the kinetic energy of the water jet used to drive the turbine runner. (Leni. D, 2023) Bernoulli's equation is used to calculate the relationship between pressure and water jet velocity. High initial pressure does not guarantee optimal performance unless supported by an appropriate spray angle to optimize the impact power of the water on the turbine blades (Saputra et al., 2020).

Furthermore, excessively high flow pressure can trigger cavitation, the formation of steam bubbles that can cause significant damage to the blade surface. Therefore, the pressure and nozzle angle control system play a vital role in ensuring the stability and durability of the Pelton turbine (Riani et al., 2022b).

2.5 Temperature in Turbine System

Temperature also plays a small but significant role in Pelton turbine performance. Friction and turbulence in the water flow can generate heat, impacting turbine efficiency. These effects can be further analyzed using thermal simulations to identify areas with inefficient flow patterns and energy dissipation, minimizing the risk of wear and long-term efficiency decline (Sonjaya et al., 2023).

2.6 Computational Fluid Dynamics (CFD) in Pelton Turbine Analysis

CFD is used as a simulation method to study water flow patterns, pressure distribution, jet velocity, nozzle angle effects, water-bucket impact patterns, and the effects of turbulence and temperature on Pelton turbines. CFD allows for non-invasive analysis of various turbine design parameters, including multi-nozzle flow patterns, impulse force distribution, and the effects of changes in spray angle. The simulation also provides numerical data and graphical visualizations that can be used to evaluate and optimize turbine designs before physical implementation (Sandmaier et al., 2023).

3. RESEARCH METHODS

This research is an experimental study with a quantitative approach that aims to analyze the effect of pressure variations and nozzle angles on the temperature values produced by the Pelton turbine. This research was conducted at the Energy and Energy Conversion Laboratory, Mechanical Engineering Study Program, Muhammadiyah University of West Sumatra, from May to July 2025. During this period, various stages were carried out starting from design, manufacture and testing of equipment, data collection, to data processing and analysis. The stages of this research can be seen in Figure 1 Flowchart of the research which presents the sequence of research to be carried out.

Figure 1. Flow chart

3.1 Problem Identification

The main problem in this study is how variations in input pressure (1, 2, and 3 bar) and nozzle angles (15°, 30°, and 35°) affect the pressure distribution, flow velocity, impact force, and torque generated in the impulse turbine system. This problem identification serves as the foundation for determining simulation parameters and analysis objectives.

3.2 Literature Review

A comprehensive literature review was conducted to strengthen the theoretical and methodological basis of this study. It focused on:

a. Fluid flow in nozzles and impulse turbines (particularly Pelton-type systems).
b. Jet impact phenomena and their influence on turbine buckets.
c. Applications of Computational Fluid Dynamics (CFD) in flow simulation studies.

Sources included fluid mechanics textbooks, indexed journals, and prior experimental and numerical research reports.

3.3 Determination of Pressure

In this study, input pressures of 1, 2, and 3 bar were selected as the main variables to analyze their effect on turbine performance. The inlet pressure was set at these levels, while the outlet boundary condition was atmospheric pressure. Water at room temperature was used as the working fluid, assuming steady, incompressible, and isothermal flow conditions.

3.4 Design

Three nozzle models with angles of 15°, 30°, and 35° were designed using CAD software to represent different flow directions relative to the turbine blades. The nozzle length and diameter were kept constant to ensure consistent comparative analysis.

Meshing was structured, with finer elements in critical areas such as around the nozzle exit and turbine bucket, to capture detailed gradients of pressure and velocity. A k–omega SST turbulence model was applied for higher accuracy in predicting flow behavior near walls and impact regions. Mesh independence validation was performed to ensure that results were not affected by mesh size.

3.5 Analysis

CFD simulations were carried out using commercial CFD software. The governing equations included the conservation of mass and momentum (Navier-Stokes). Key parameters analyzed were:

a. Static, total, and dynamic pressures (minimum & maximum values).
b. Maximum fluid velocity at critical regions.
c. Impact forces along the X, Y, and Z axes.
d. Torques acting on the turbine bucket along the X, Y, and Z axes.

Each scenario (combination of angle and pressure) generated a dataset extracted after achieving simulation convergence (residuals approaching 0 and progress at 100%).

3.6 Evaluation of Results and Discussion

Visualization of simulation results included pressure contours and velocity vectors for

each configuration. The evaluation highlighted:
a. Shifts in high-pressure zones.
b. Patterns of flow impact on the bucket surface.
c. Distribution of forces and torques resulting from different nozzle angles and pressures.

Qualitative comparisons were also made with previous studies to assess the reliability and consistency of trends in pressure and force distribution.

3.7 Conclusion

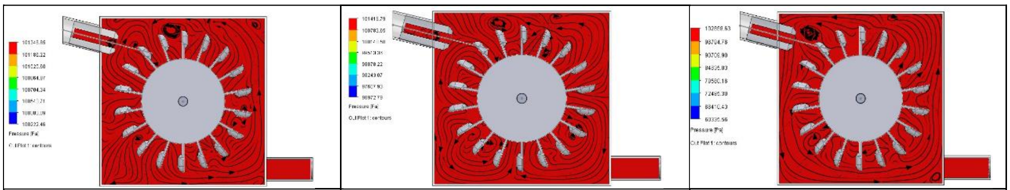
The study concludes that variations in nozzle angles and input pressures significantly influence the hydrodynamic performance of impulse turbine systems. The findings suggest that specific combinations of nozzle angle and pressure can minimize drag losses and optimize impact forces, improving overall turbine efficiency. Technical recommendations are provided regarding optimal nozzle design and working pressure for enhanced performance in impulse turbine applications.

4. RESULTS AND DISCUSSION

4.1 Results

This research begins with the design stage of the nozzle geometry and blades of the Pelton turbine. The design was made by considering three variations of nozzle angles, namely 15°, 30°, and 35°, to examine the effect of each configuration on the pressure distribution on the blade surface. Fluid flow simulations were carried out using the Computational Fluid Dynamics (CFD) method based on SolidWorks Flow Simulation, which allows for in-depth analysis of flow patterns, pressure distribution, fluid velocity, and forces and moments acting on the system. The simulation results show differences in flow characteristics at each nozzle angle, which are then evaluated to determine the most optimal configuration to support the performance of the Pelton turbine, both in terms of energy efficiency and structural resistance to mechanical loads.

1. 15° angle



(a) (b) (c)

Figure 2. Testing with a slope of 15°(a) Pressure 1 Bar (b) Pressure 2 Bar (c) Pressure 3 Bar

From the simulation carried out above by applying a nozzle tilt angle of 15° with 3 variations of pressure given, a comparison of the results is obtained which is shown in Table 1.

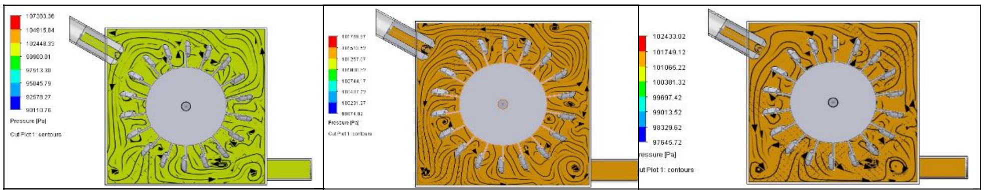
Table 1. Comparison results of 15° angle simulation

Parameter	Unit	Pressure 1 bar	Pressure 2 bar	Pressure 3 bar
Minimum Static Pressure	[Pa]	101306.4	101253.7	100743.57
Maximum Static Pressure	[Pa]	101346.9	101418.8	102155.17
Minimum Total Pressure	[Pa]	101306.4	101253.7	100743.57
Maximum Total Pressure	[Pa]	101358.1	101514.9	102971.48

Maximum Dynamic Pressure	[Pa]	34.78	142.46	1077.07
Maximum Velocity	[m/s]	0.61	1.24	3.75
Force (X)	[N]	-0.00012	-0.00047	-0.00443
Force (Y)	[N]	0.0001	0.00034	0.00514
Force (Z)	[N]	-8.93E-05	-0.00055	-0.00251
Torque (X)	[N·m]	-0.00019	-0.00083	-0.00798
Torque (Y)	[N·m]	-0.00011	-0.00027	-0.00426
Torque (Z)	[N·m]	0.00017	0.00062	0.00703

The comparison of these three pressure variations confirms that increasing inlet pressure not only improves fluid flow performance but also results in greater structural loads on the nozzle components. Therefore, design optimization requires consideration of both flow efficiency and mechanical durability to ensure the system operates safely at high pressures.

2. 30° angle



(a) (b) (c)

Figure 3. Testing with a 30° slope (a) Pressure 1 Bar (b) Pressure 2 Bar (c) Pressure 3 Bar

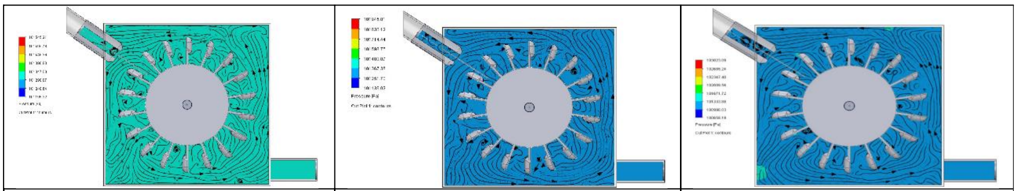
In the CFD simulation carried out on a nozzle with a 30° inclination, the comparative results of the 3 pressure variations tested on the platoon turbine were obtained, which are presented in Table 2.

Table 2. Comparison results of 30° angle simulation

Parameter	Unit	Pressure 1 bar	Pressure 2 bar	3 bar pressure
Minimum Static Pressure	[Pa]	98,687.65	101,164.02	100,523.36
Maximum Static Pressure	[Pa]	102,526.23	101,392.27	101,561.92
Minimum Total Pressure	[Pa]	98,687.65	101,164.02	100,523.36
Maximum Total Pressure	[Pa]	102,757.77	101,401.95	101,610.04
MaximumDynamic Pressure	[Pa]	517.17	77.68	147.27
Maximum Velocity	[m/s]	2.05	0.9	1.17
Force (X)	[N]	0.00123	0.00023	0.00063
Force (Y)	[N]	0.00341	0.00032	0.00105
Force (Z)	[N]	0.00073	7.86E-05	0.00046
Torque (X)	[N·m]	-0.0037	-0.00034	-0.00092
Torque (Y)	[N·m]	0.00098	0.00023	0.00045
Torque (Z)	[N·m]	0.0012	1.07E-05	0.00015

The comparison of input pressure variations of 1 bar, 2 bar, and 3 bar on a 30° nozzle angle shows that increasing input pressure significantly affects the pressure distribution, flow velocity, and forces and moments on the nozzle wall. The maximum total pressure and dynamic pressure increase with increasing input pressure, indicating a greater accumulation of fluid energy. The flow velocity shows fluctuations, with the highest value at 1 bar and increasing again at 3 bar, indicating a redistribution of fluid energy under high pressure conditions. In addition, forces and torques on all axes increase, indicating a greater mechanical load on the nozzle structure. Overall, these results confirm that increasing input pressure improves fluid performance but also increases the risk of structural loads, so nozzle design must consider aspects of flow efficiency and material durability.

3. 35° angle



(a) (b) (c)

Figure 4. Testing with a 30° slope (a) Pressure 1 Bar (b) Pressure 2 Bar (c) Pressure 3 Bar

Based on CFD simulation at nozzle tilt angle of 35°with 3 variations of pressure applied as at an angle of 15°and 30°the comparative results are obtained as shown in Table 3.

Table 3. Comparison results of 35° angle simulation

Parameter	Unit	Pressure 1 bar	Pressure 2 bar	Pressure 3 bar
Minimum Static Pressure	[Pa]	101227.5	101141.4	100658.2
Maximum Static Pressure	[Pa]	101545.2	101945.8	102924.5
Minimum Total Pressure	[Pa]	101227.5	101148.4	100658.2
Maximum Total Pressure	[Pa]	101568.1	102010.9	103066.4
Maximum Dynamic Pressure	[Pa]	124,434	270,657	962,616
Maximum Velocity	[m/s]	0.499	0.737	1,389
Force (X)	[N]	-2.19E-05	-0.00013	-0.00266
Force (Y)	[N]	-0.00014	-0.00053	-0.00145
Force (Z)	[N]	0.000333	0.000707	0.004252
Torque (X)	[N·m]	0.000493	0.00136	0.005828
Torque (Y)	[N·m]	-0.00022	-0.00058	-0.0058
Torque (Z)	[N·m]	-6.09E-05	-0.0002	0.00135

A comparison of the three input pressure variations shows that increasing pressure has a direct effect on flow performance. There is an increase in total pressure, dynamic pressure, flow velocity, and surface forces and moments. On the other hand, a decrease in minimum static pressure indicates the presence of a flow narrowing phenomenon or venturi effect in certain areas. These findings are consistent with the basic principles of fluid mechanics, namely the law of continuity and the Bernoulli equation, and provide a strong basis for nozzle geometry optimization and structural load prediction in the system.

Based on the results of the nozzle angle comparison analysis, it shows that the angle variation has a significant effect on the pressure distribution, flow velocity, and the forces and moments acting on the system. At an angle of 15°, an increase in input pressure from 1 bar to 3 bar results in a very sharp increase in the impact force, as shown by the blue line in the pressure and impact force relationship graph. Figure 6. This indicates that nozzles with smaller angles tend to produce a larger reactive force in the FY direction, as the pressure increases. Conversely, at an angle of 30°, the impact force tends to be more stable with a relatively small increase even though the pressure increases, as seen in the relatively flat orange line. Meanwhile, an angle of 35° shows a different phenomenon, where the impact force in the FZ direction increases significantly at high pressure (3 bar), as depicted in the yellow line which has a steep increasing trend after 2 bar. This pattern indicates that a larger nozzle angle directs the fluid momentum more to the Z axis, so that the dominant force shifts to the FZ direction. Thus, the graph reinforces the results of the previous table comparison that the nozzle angle not only affects the pressure distribution but also modifies the dominant direction of the impact force according to its geometric configuration.

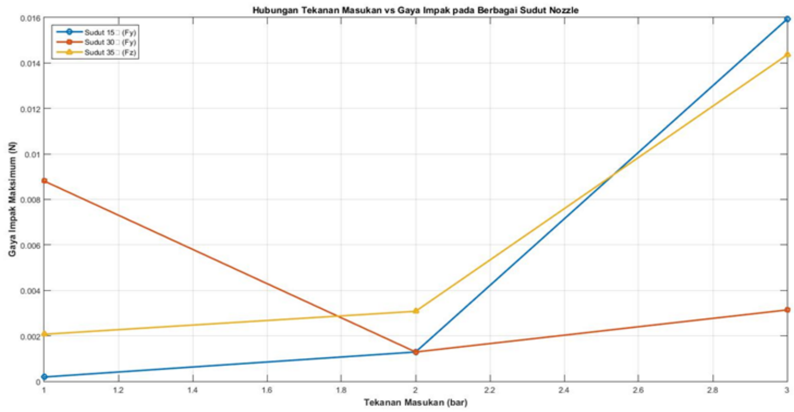


Figure 6. Relationship Between Pressure and Impact Force

1. Blue Line

The blue line in the graph depicts the relationship between input pressure (1 bar, 2 bar, 3 bar) and the maximum impact force in the FY direction for a 15° nozzle angle. The observed trend shows a very significant increase, especially when the pressure reaches 3 bar. This indicates that nozzles with smaller angles tend to direct the fluid flow linearly parallel to the Y-axis, so that the reactive force in that direction increases sharply with increasing pressure. This phenomenon is relevant in the context of applications requiring high thrust, but also requires greater attention to structural strength due to the load spike at higher operating pressures.

2. Orange Line

The orange line represents the same relationship for a nozzle angled at 30° in the FY direction. In contrast to the previous configuration, this graph shows a relatively stable trend, with only a slight increase in impact force at 3 bar. This stability indicates that the 30° angle produces a more balanced flow distribution, so that changes in input pressure do not have an extreme effect on the reactive force in the FY direction. This configuration is ideal for systems that prioritize force stability and more controlled structural loads across a wide range of pressure variations.

3. Yellow Line

The yellow line in the graph depicts the relationship between the input pressure and the maximum impact force in the FZ direction for a nozzle with a 35° angle. The increasing trend is more pronounced after the pressure exceeds 2 bar, indicating that the larger nozzle angle significantly directs the fluid momentum toward the Z axis. Consequently, the reactive force in the FZ direction increases dramatically at high pressures. This configuration requires special consideration in structural design, especially to handle the more complex lateral and torsional loads due to the force distribution shifting to the vertical direction.

The results of this study have been validated against various recent studies, including those by Wang & Yang (2023), Gao et al. (2023), and Van de Sanden et al. (2025), which independently demonstrated that variations in nozzle angle and geometry parameters significantly affect the pressure distribution, flow velocity, and reactive force in fluid and cold spray systems. Wang & Yang (2023) using RSM and CFD simulations concluded that changes in nozzle cone angle can increase jet efficiency by up to 8.38%, in line with our findings regarding the sensitivity of FY force at an angle of 15°. Gao et al. (2023) validated the critical role of nozzle throat angle and length on pressure distribution in hydraulic perforating applications, consistent with the observations of pressure distribution on the bucket surface in this study. Meanwhile, Van de Sanden et al. (2025) through CFD cold spray analysis demonstrated that changes in nozzle geometry, especially the throat and divergent section, move fluid momentum toward the FZ, resulting in increased impact force at high pressures. The consistency between the results of this study and the current literature strengthens the reliability of the analysis, as well as substantially supports the recommendation of optimizing nozzle design at angles of 15°, 30°, and 35° for various high-pressure industrial applications.

CONCLUSION

The results of this study indicate that variations in nozzle angles (15°, 30°, and 35°) significantly affect the pressure distribution, flow velocity, and forces and moments acting on the pressurized fluid system. An angle of 15° results in an increase in the maximum impact force in the FY direction which is very sensitive to the increase in input pressure, especially at a pressure of 3 bar, while an angle of 30° shows a more stable and moderate force distribution, and an angle of 35° indicates the dominance of forces in the FZ direction at high pressures due to changes in the flow pattern. The resulting pressure distribution shows the conversion of pressure energy into kinetic energy according to Bernoulli's principle, reflected in the increasing values of dynamic pressure, fluid velocity, and reactive forces on the nozzle surface.

REFERENCES

Ardika Tommy Saputra, IM, Jasa, L., & Arta Wijaya, IW (2020). THE EFFECT OF WATER PRESSURE AND NOZZLE ANGLE ON OUTPUT CHARACTERISTICS OF A PROTOTYPE OF A MHP WITH A PELTON TURBINE. SPEKTRUM Journal, 7(4), 17. https://doi.org/10.24843/SPEKTRUM.2020.v07.i04.p3

Darmawan, S., Sutanto, H., De Fretes, A., Octaviani, S., Tanujaya, H., Rosehan, R., & Marcellio, R. (2023). FLOW ANALYSIS IN A 1 KW PELTON TURBINE BUCKET WITH SPEAR NOZZLE VARIATIONS USING CFD METHOD. POROS, 19(1), 27–38. https://doi.org/10.24912/poros.v19i1.25627

Hidayat, W. (2019). Working Principles and Components of Hydroelectric Power Plants (PLTA). INA-Rxiv. https://doi.org/10.31227/osf.io/drv58

Karudin, A., Leni, D., Usmeldi, U., Purnama, A., & Akbar, Y. (2025). Implementation of a Steam Turbine Trainer Prototype as a Learning Medium at SMKN 1 West Sumatra. Jurnal Vokasi, 9(1), 43. https://doi.org/10.30811/vokasi.v9i1.6231

Leni. D, K. (2023). Performance Analysis of Hydrokinetic Turbines with Diffuser Casings Under Yaw Misalignment Conditions Based on Water Flow Velocity Variations. Journal of Materials, Manufacturing, and Energy Engineering, 6(2). https://doi.org/10.30596/rmme.v6i2.16190

Ma'ruf, K., Surono, Darmono, Agung Fadlullah, Y., & Putra Setiyawan, B. (2023). DESIGN AND CONSTRUCTION OF MINI HYDRO TURBINE POWER PLANT AS AN ENVIRONMENTALLY FRIENDLY ENERGY SOURCE. Multidisciplinary Journal of West Science, 2(05), 314–323. https://doi.org/10.58812/jmws.v2i5.314

Pratama, PA, Malyadi, M., & Wicaksono, YA (2021). EXPERIMENTAL STUDY OF VARIATIONS OF BLADE SHAPE AND ANGLE ON PELTON TURBINE PERFORMANCE. AutoMech: Journal of Mechanical Engineering, 1(01). https://doi.org/10.24269/jtm.v1i01.4254

Riani, NI, Masykur, M., & Abidin, Z. (2022a). PERFORMANCE ANALYSIS OF A TRIANGULAR BLADE PELTON TURBINE WITH NOZZLE SPRAY ANGLE VARIATIONS. Jurnal Mekanova: Mekanikal, Inovasi Dan Teknologi, 8(1), 113. https://doi.org/10.35308/jmkn.v8i1.5512

Riani, NI, Masykur, M., & Abidin, Z. (2022b). PERFORMANCE ANALYSIS OF TRIANGLE-BLADE PELTON TURBINE WITH NOZZLE SPRAY ANGLE VARIATIONS. Jurnal Mekanova: Mekanikal, Inovasi Dan Teknologi, 8(1), 113. https://doi.org/10.35308/jmkn.v8i1.5512

Sandmaier, L., Meusburger, P., & Benigni, H. (2023). Transient 3D CFD Simulation of a Pelton Turbine—A State-of-the-Art Approach for Pelton Development and Optimization. International Journal of Turbomachinery, Propulsion and Power, 8(1), 10. https://doi.org/10.3390/ijtpp8010010

Sonjaya, AN, Nugroho, FS, & Vindiawan, MV (2023). Pelton Turbine Design for a 100-watt Microhydro Power Plant. Journal of Technology, 11(1), 100–109. https://doi.org/10.31479/jtek.v11i1.266

Susanto, L., Priangkoso, T., & Darmanto, D. (2019). DESIGN OF A 1 KW CAPACITY PICO HYDRO-SCALE PELTON TURBINE. MOMENTUM SCIENTIFIC JOURNAL, 15(2). https://doi.org/10.36499/jim.v15i2.3076

Tonadi, E. (2021). ANALYSIS OF THE EFFECT OF THE NUMBER OF BLADES ON THE EFFICIENCY OF A PELTON TURBINE WITH CONSTANT PRESSURE. Teknosia, 15(1), 36–42. https://doi.org/10.33369/teknosia.v1i1.15390

Yohanes Setyawan, E., & Tetuko, B. (2023). Observation of Nozzle Performance in a Pelton Turbine-Based Hydroelectric Power Plant. Proceedings of SENIATI, 7(1), 156–160. https://doi.org/10.36040/seniati.v7i1.8139