

CFD analysis in investigating the impact of turbine blade number on the performance of hydro turbine

Ade Putra Maulana¹, Indrayani^{1,2}, Fatahul Arifin^{1,3*}

¹Renewable Energy Engineering, Politeknik Negeri Sriwijaya, Palembang 30139, Indonesia

²Civil Engineering, Politeknik Negeri Sriwijaya, Palembang 30139, Indonesia

³Mechanical Engineering, Politeknik Negeri Sriwijaya, Palembang 30139, Indonesia

*Corresponding author: farifinus@polsri.ac.id

Abstract

The demand for electrical energy in Indonesia is growing, and therefore more effort is required to fulfill this need. Indonesia has considerable hydropower potential due to its geographical location and climate, by utilizing areas that have this potential to support the government's renewable energy program to provide electricity to the community. Impeller turbines are one option in an effort to create renewable energy generation. In this study, a comparison of the number of turbine blades was carried out using the Computational Fluid Dynamics (CFD) method, three models are 11, 13, and 15 blade impeller turbines designed at water with a runner diameter of 200 mm, blade thickness of 2 mm and an angle of 30 degrees then simulated using Comsol Multiphysics against different water flow rates. The simulation results show that the 11-bladed turbine model I has better performance because it has a lower pressure value but has a better velocity value compared to the two models simulated.

Keywords: hydropower, Impeller turbines, CFD method

1 Introduction

Energy plays a significant role in a nation's development if its availability and development coincide with that nation's development [1]. In recent years, the issue of rising energy demand has emerged as one of the most pressing [2] due to the world's increasing population. To meet energy needs, there must be a global awareness and commitment to reduce the use of nonrenewable energy and shift to renewable energy. Indonesia is renowned for its profusion of natural resources, including fossil and non-fossil energy [3], and the country's energy demand is rising in tandem with its growing population and economy [4]. The foundation for expanding the use of renewable energy in the provision, security, and independence of national energy is the Indonesian government's strategy to maximise the country's energy potential. This policy is also supported by the abundant potential for new and renewable energy in Indonesia, which differs from region to region due to the country's vast size. With a favourable geographical location and climate, hydropower has a substantial amount of potential.

Numerous rivers in Indonesia can be used as an alternative energy source, with micro-hydro power facilities being one of them [5–9]. Micro-hydro is frequently viewed as a viable option for meeting the electricity requirements of rural communities [10, 11], is more reliable and less expensive than generators powered by other raw materials, and is regarded as a renewable energy source [12]. Micro hydro depends heavily on the availability of discharge and the height of the water fall that can propel turbines [13].

The significance of design when displaying turbines Several investigations on CFD turbines have been conducted, according to Daneshkahr and Zangeneh [14]. This article explains the parametric design of a Francis turbine runner. The geometry of the runner is parameterized using a 3D inverse design method, and CFD analyses were undertaken to evaluate the hydrodynamic and suction performance of various design configurations. On the basis of this research, an optimal configuration was devised that produces a cavitation-free flow in the raceway while maintaining a high level of hydraulic efficiency. The paper focuses on design guidelines for the application of the inverse design method to Francis turbine runner components. Since they are based on flow field analyses and hydrodynamic design parameters [14], the design guidelines have general applicability and can be applied to analogous design applications.

Kaewnai & Wongwises studied to improve the runner design of the Francis turbine and analyse its performance using the Computational Fluid Dynamics (CFD) technique. The runner design procedure employs a direct method with a design conditions flow rate of 3.12 m³/sec. At the moment of design, results from calculating runner's performance were approximately 90%. Existing absolute velocity component from CFD simulation revealed undulating flow at runner exit. This method [15] can be used to simulate the runner's performance of the Francis turbine based on the comparison between simulation results and experimental data from previous work reported in the literature.

Binama et al. used a pump-as-turbines (PATs) in water distribution systems for energy regulation and hydroelectricity generation has garnered the attention of energy industry players. Due to PAT's inability to control flow in such dynamic systems, the power production efficacy of PAT is still problematic. This eventually resulted in the development of the so-called "variable operating strategy" or VOS, in which the impeller's rotational speed can be adjusted to meet the system's required flow conditions. The results of the study indicate that both the PAT's flow and pressure fields diminish as the machine's influx decreases, and that an increase in impeller rotational speed reduces PAT pressure pulsation levels under high-flow operating conditions but exacerbates them under part-load conditions. The findings of this study add to a comprehensive understanding of PAT flow dynamics, which, in the long run, contributes to the resolution of the existing technical problems. [16].

Biner et al. studied the runner fatigue damage during start-up in generating mode of a 5 MW variable speed Francis pump-turbine prototype equipped with an FSFC is numerically analysed and compared to a fixed speed solution following a BEP tracking control strategy. Simulations of 1D hydraulic transients provide boundary conditions for 3-D CFD/FEA simulations. The use and comparison of complete and reduced numerical domains are demonstrated. Using variable speed actuators for turbine start-up manoeuvres results in a significant reduction of partial damages, according to the overall findings of the present numerical study [17].

Tengs et al. studied the design optimisation of constrained to blade design dengan tujuan meningkatkan efisiensi hidrolik. As a result of numerical optimisation of the design, it is possible to acquire a small

but significant increase in torque and efficiency while drastically reducing structural stresses [18].

This study seeks to determine and obtain the relationship between the number of turbine impeller blades and computational fluid dynamics (CFD) analysis based on some of these studies.

2. Research Methods

This hydro turbine impeller research examines the effect of the number of blades chosen on the optimisation of the turbine in obtaining kinetic energy from the fluid it circulates. There are three variants of the turbine impeller, with 11 blades (model I), 13 blades (model II), and 15 blades (model III). Changing the blades of the turbine is one option for optimising energy extraction with the impeller.

This study seeks to determine and obtain the turbine impeller's efficiency by counting and configuring the number of blades. Comsol Multiphysics 6.0 is utilised to evaluate the analysis of Computational Fluid Dynamics (CFD), where CFD is the capacity to conduct experimental and analytical approaches. Water fluid flow is modelled in three dimensions. Flow Mesh and simulations generated with the CFD software Comsol Multiphysics. To obtain the quantitative characteristics of the water flow to the turbine structure, simulations were performed. The turbine impeller is made of plastic. For the purpose of determining the hydrodynamic characteristics of the impeller hydro turbine, numerical analysis is utilised. Variations in turbine angular velocity (rotating zone) are depicted in a view of turbine performance (19). Model testing on hydraulic turbines is crucial for the design and performance of turbines derived from CFD simulation results.

Three-dimensional turbulent flow is used to characterise the internal flow of a hydraulic turbine in hydrodynamics. The k-omega model is a type of Reynolds-averaged Navier-Stokes (RANS) model that is commonly used in computational fluid dynamics (CFD) simulations to model turbulent flows, which is characterised by the time-averaged N-S equation of Reynolds [20].

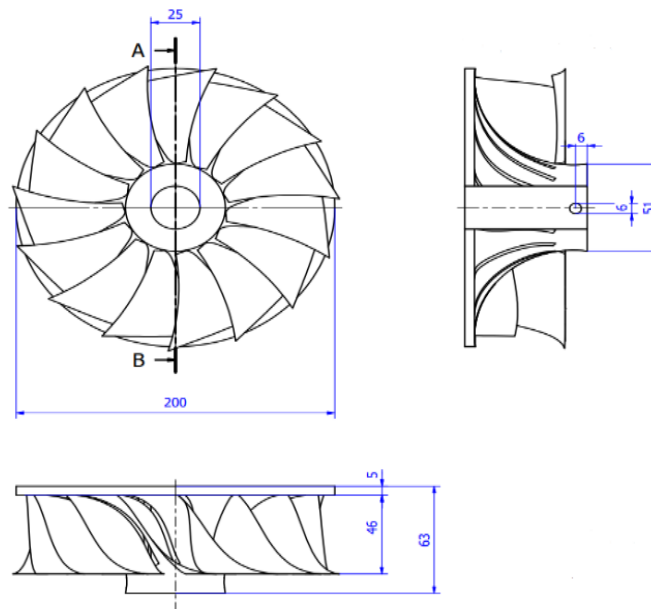


Fig. 1. Turbine dimensions

Continuity Equation (Eq. 1):

$$\frac{\partial u}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \quad (1)$$

Momentum Equation (Eq. 2):

$$\rho \frac{\partial \bar{u}_i}{\partial t} + \rho \bar{u}_j \frac{\partial \bar{u}_i}{\partial x_j} = \rho f_i \frac{\partial \bar{p}_i}{\partial x_i} + \frac{\partial}{\partial x_j} \left(\mu \frac{\partial \bar{u}_i}{\partial x_j} - \overline{\rho \dot{u}_i \dot{u}_j} \right) \quad (2)$$

K - ω Turbulence Model Equation (Eq. 3):

$$\frac{\partial(\rho k)}{\partial t} + \nabla \cdot \rho \bar{u} k = \nabla \cdot \left[\left(\mu + \frac{\mu_t}{\sigma_k} \right) \nabla k \right] + \rho_k - \hat{\beta} \rho k \omega \quad (3)$$

ρ_k is turbulent production describing as (Eq. 4-5):

$$\rho_k = \mu_t S^2 \quad (4)$$

$$arg_1 = \min \left(\max \left(\frac{\sqrt{k}}{\beta \omega y}, \frac{500 \mu}{\rho y^2 \omega} \right), \frac{4 \rho k}{CD_{k\omega} \sigma_{\omega 2} y^2} \right) \quad (5)$$

$$CD_{k\omega} = \max \left(\frac{2 \rho}{\sigma_{\omega 2} \omega} \nabla k \nabla \omega, 1.0 \times 10^{-10} \right)$$

Eddy-Viscosity describing as (Eq. 6):

$$\mu_t = \left(\frac{\rho \alpha_1 k}{\max(\sigma_2, \omega, SF_2)} \right) arg_2 = \max \left(\frac{\sqrt{k}}{\beta \omega y}, \frac{500 \mu}{\rho y^2 \omega} \right) \quad (6)$$

These six formula equations (Eq. 1-6) are used in the implementation of numerical simulation on boundary conditions in the CFD Comsol k-omega model software platform.

The steps in CFD simulation include creating geometry, meshing, defining boundary planes on the geometry, and checking the mesh [21], [22].

The first step in the numerical method is to design a hydro turbine impeller geometry model with a runner diameter of 200 mm, a blade thickness of 2 mm and an angle of 30 deg shown in Fig 1. with three variations of models consisting of 11 blades (model I), 13 blades (model II) and 15 blades (model III) shown in Fig 2.

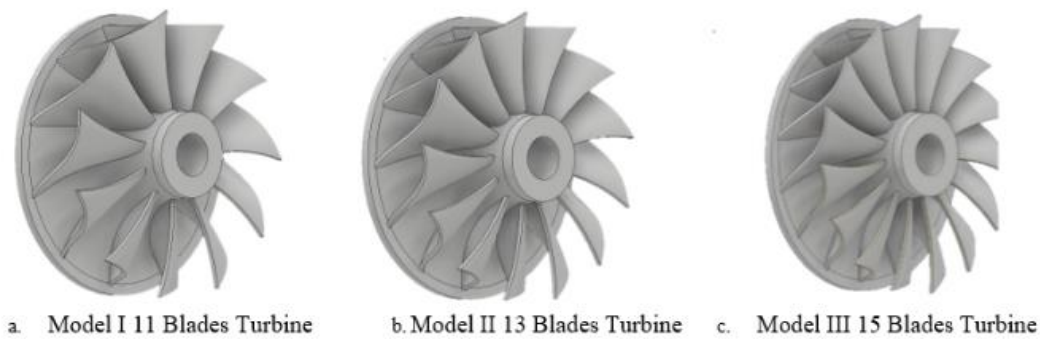


Fig. 2. Turbine design

Flow rate inlet has a constant flow rate profile for 0.2, 0.8, 1.4, and 2 m³/s (Tabel 1), and pressure outlet has a value of atmospheric pressure. For the purpose of assuring the continuity of the flow field, a rotating interface is established around the turbine's circumference. The inner zone of the flow field is referred to as the turbine zone and is designated as the interface between the domain and turbine. The turbine zone is distinguished by a mesh that rotates with the same angular velocity as the rotor [23].

Boundary Condition For fluid domain to operate, the actual operating boundary conditions must be provided. Set the inlet domain's boundary conditions to a distinct flow rate based on the working conditions, and set the outlet domain's boundary conditions to an average static pressure of 1 atm. Fig. 3 depicts the domain (3D) as a result.

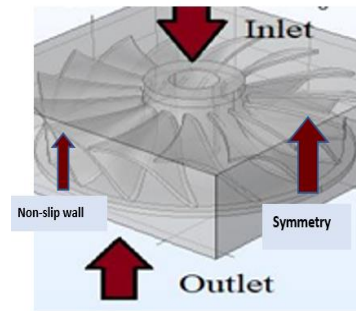


Fig. 3. Flow field

The meshing element is shown in Fig 4.

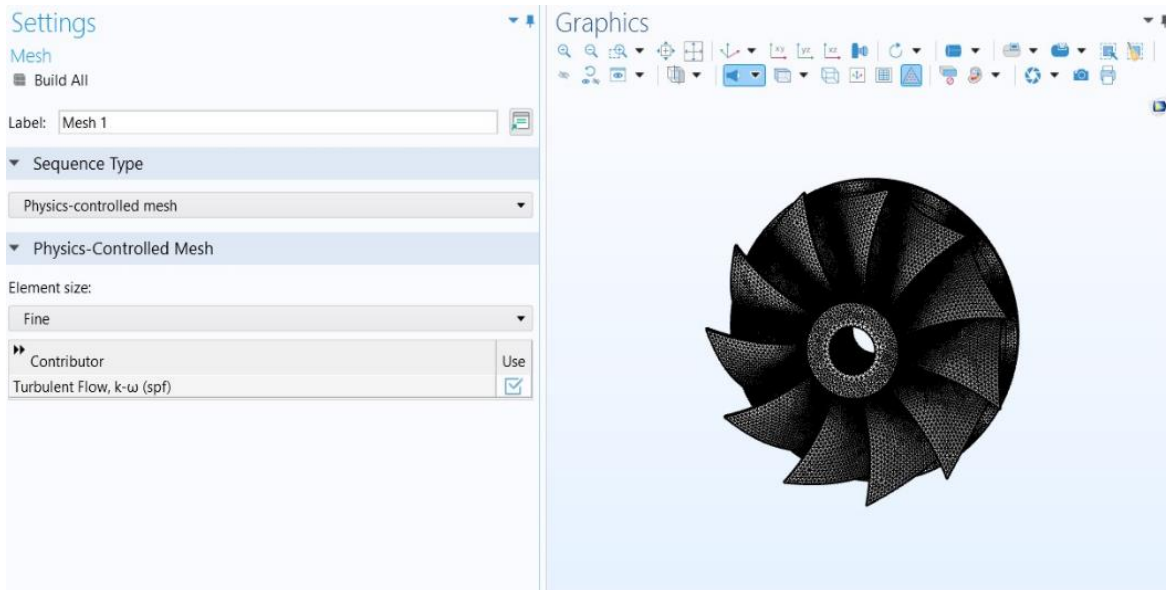


Fig. 4. Computational mesh

Tabel 1. CFD Parameters

Parameter	CFD code
Flow simulation	The k-omega model
Mesh	Structured (fine)
Fluid	H ₂ O
Material	ABS
Inlet	Flow rate 0.2, 0.8, 1.4 dan 2 m ³ /s
Outlet	1 atm

Wall	No Slip
------	---------

2 Results and Discussion

The simulation results of the three models with model I turbine 11 blades, model II turbine 13 blades and model III turbine 15 blades are shown in Fig 5-10 with 2 m³/s.

In the simulation of the 11-blade turbine model I, flow rates of 0.2, 0.8, 1.4 and 2 m³/s simultaneously obtained results in the form of maximum velocity values of 0.38, 2.02, 3.75 and 5.5 m/s while minimum velocity has values of 0.0097, 0.05, 0.1 and 0.14 m/s.

In Pressure, the maximum values are 534, 8360, 25100 and 50600 Pa and the minimum Pressure values are -17.9, -1140, -5200 and -12700 Pa.

Model II of the 13-blade turbine at simulated flow rates of 0.2, 0.8, 1.4 and 2 m³/s simultaneously obtained results in the form of maximum velocity values of 0.34, 1.86, 3.52 and 5.23 m/s respectively while minimum velocity has values of 0.0086, 0.05, 0.09 and 0.14 m/s. In Pressure the maximum values are 539, 8430, 25300 and 51000 Pa and the minimum Pressure values are -19, -957, -4560 and -11300 Pa.

The simulation results of model III of the 15-blade turbine flow rate of 0.2, 0.8, 1.4 and 2 m³/s simultaneously obtained results in the form of maximum velocity values of 0.32, 1.83, 3.5 and 5.2 m/s respectively while minimum velocity has values of 0.0082, 0.05, 0.09 and 0.13 m/s. In Pressure, the maximum values are 542, 8480, 25400 and 51200 Pa and the minimum Pressure values are -16.9, -1020, -4700 and -11700 Pa.

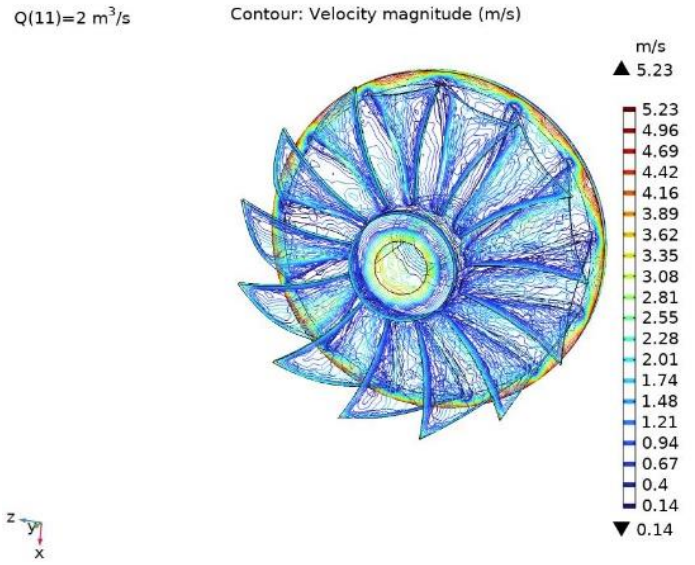


Fig. 7. Model II velocity simulation results

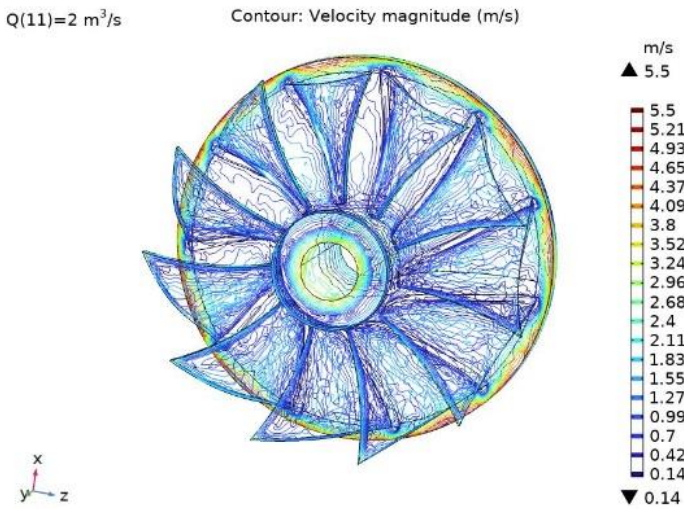


Fig. 5. Model I velocity simulation results

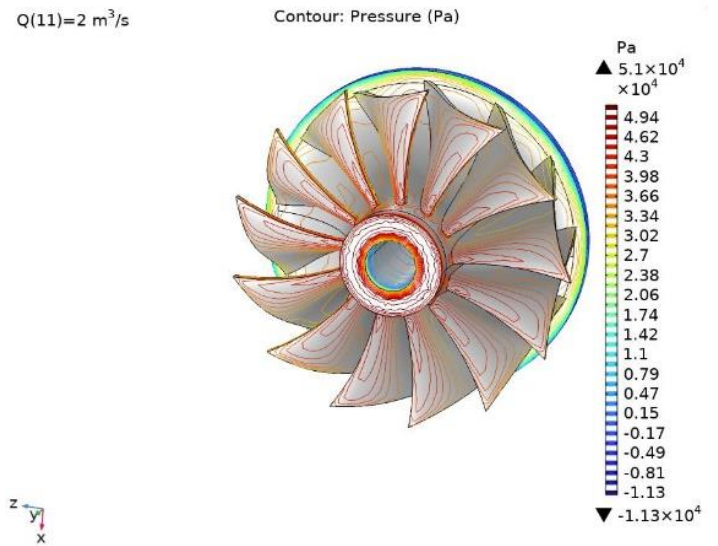


Fig. 8. Model II pressure simulation results

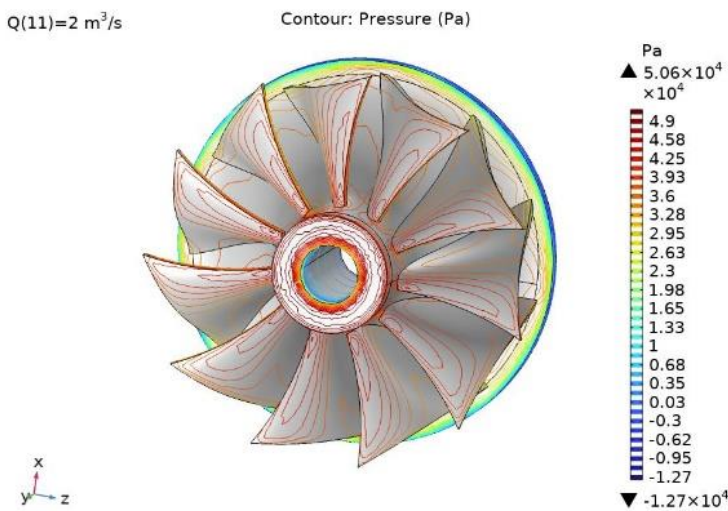


Fig. 6. Model I pressure simulation results

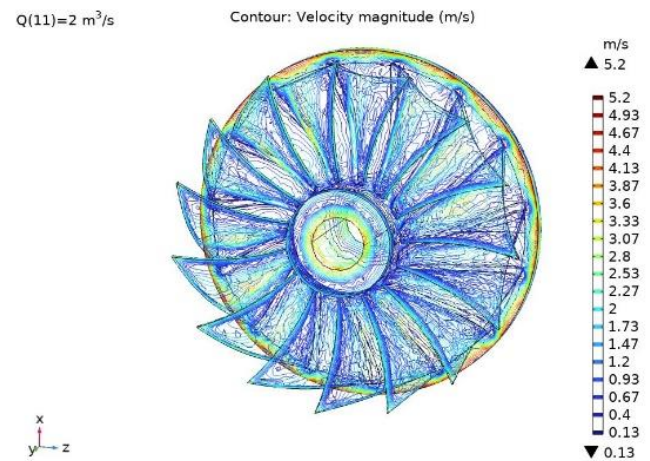


Fig. 9. Model III velocity simulation results

Q(11)=2 m³/s

Contour: Pressure (Pa)

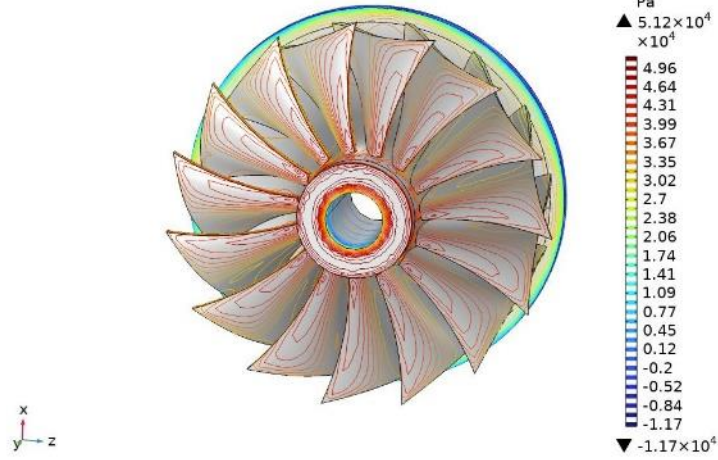


Fig. 10. Model III pressure simulation results

Fig. 5-10 depicts the runner's velocity and pressure. The results of the CFD simulation indicate that the minimum velocity occurs in the crown region and increases towards the band region, where the maximum velocity exists. It is favourable that the utmost velocity of a hydro turbine can reach up to 5 metres per second. The effects of higher velocity in a turbine are increased kinetic energy and attrition of the turbine blades, as well as a significant increase in noise.

In contrast, the CFD simulation results indicate that the pressure is lowest in the band region and increases towards the crown region, where it is highest. This is due to a different work distribution in the runner resulting from the stacking condition, which enhances blade loading towards the crown and decreases it towards the band. The pressure range for an ABS turbine will be several hundred pounds per square inch [24, 25].

ABS is therefore a material with high mechanical strength and tensile strength, and it is frequently used in applications requiring resistance to impact and attrition. No cavitation occurs. The CFD simulation outcomes indicate.

From the simulation results that have been carried out, a graph is obtained to further clarify the description, shown in Fig 11-14.

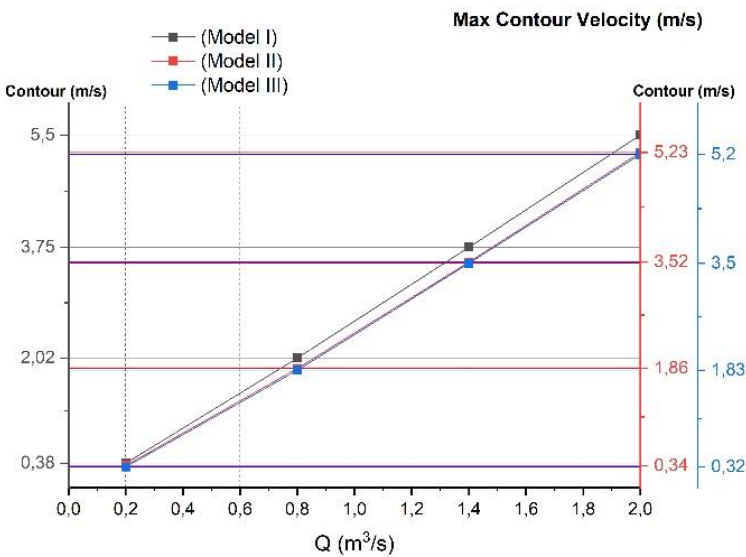


Fig. 11. Max contour velocity

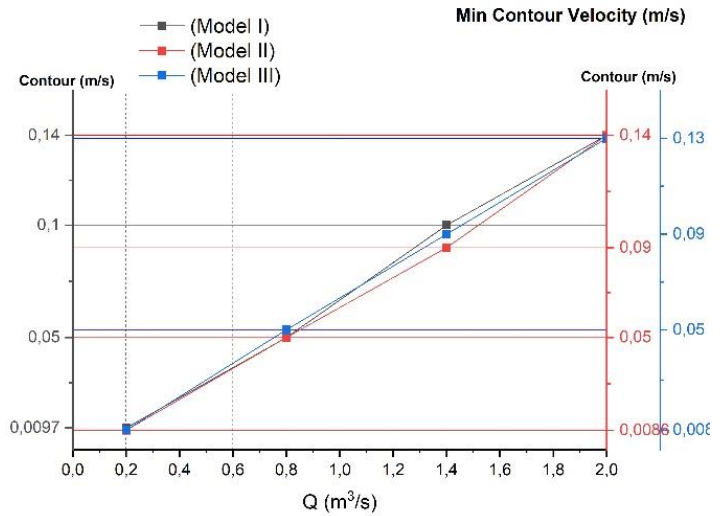


Fig. 12. Min contour velocity

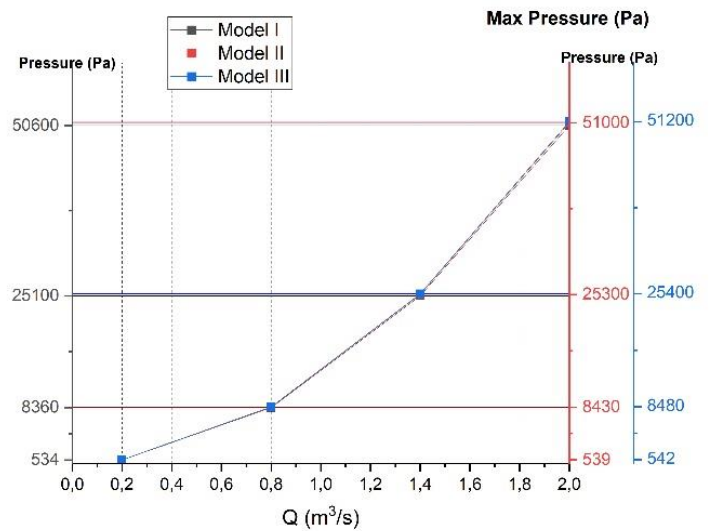


Fig. 13. Max pressure

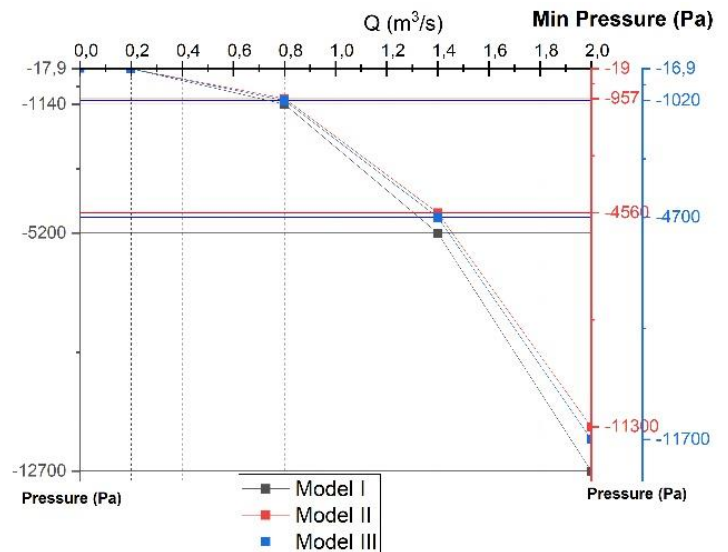


Fig. 14. Min pressure

CFD simulation results on turbine performance can be consistent with the increase in flow. Model I 11-bladed turbine has a higher maximum value compared to the other two models because it is lighter.

Pressure results show a linear relationship to flow rate with an increase in value influenced by flow. Model III 15 blades has the highest pressures value because it has more area than the two models.

The CFD simulation results on turbine performance can be consistent with the increase in flow. This is shown by the linear relationship between the water flow parameters and the velocity and pressure results [15]. The velocity parameter shows how much the water flow velocity varies through the water turbine configuration with various flow velocities. The velocity variation increases consistently with increasing flow rate. The velocity of the 11-bladed turbine configuration in Fig 3. is higher than that of the 13-bladed and 15-bladed turbines at the maximum velocity contour with a value of 0.3 m/s. The increase in pressure accompanies the increase in flow rate and field area. The 15-bladed turbine configuration has a higher maximum pressure than the 11 and 13-bladed turbines in the flow rate parameter. The maximum pressure at 2 m³/s is 51200 Pa

3 Conclusions

In a water turbine configuration, the CFD simulation results demonstrate a linear relationship between water flow parameters, velocity, and pressure. As the flow rate increases, so does the velocity variation, and as the flow rate and field area increase, so does the pressure. At the maximum velocity contour, the 11-bladed turbine configuration has a higher velocity than the 13 and 15-bladed turbines, while the 15-bladed turbine configuration has a higher maximum pressure than the 11 and 13-bladed turbine configurations at the given flow rate of 2 m³/s. These results indicate that CFD simulation can provide valuable insights into turbine performance and assist in optimising turbine design for optimum efficiency. Model I of the 11-bladed turbine performs better than the other two simulated models due to its lower pressure value and higher velocity value.

Acknowledgement

The authors would like to express sincere gratitude to PT. Bukit Asam Tbk, Plakat Village and, Politeknik Negeri Sriwijaya providing the research location and supporting this research.

References

- [1] R. Ploetz, Rusdianasari, and Eviliana, "RENEWABLE ENERGY: ADVANTAGES AND DISADVANTAGES," *Proceeding Forum in Research, Science, and Technology (FIRST)*, 2016.
- [2] A. T. Wardhana, A. Taqwa, and T. Dewi, "Design of Mini Horizontal Wind Turbine for Low Wind Speed Area," in *Journal of Physics: Conference Series*, Mar. 2019, vol. 1167, no. 1. doi: 10.1088/1742-6596/1167/1/012022.
- [3] E. Okdinata, A. Hasan, and C. Sitompul, "Performance Test of Pelton Micro-Hydro Turbine with the Variations of Parameter to Produce the Maximum Output Power," in *Journal of Physics: Conference Series*, Mar. 2019, vol. 1167, no. 1. doi: 10.1088/1742-6596/1167/1/012025.
- [4] Y. Dinata, Indrayani, and T. Dewi, "Analysis of Reservoir Water Discharge at Solar Power Plant Tanjung Raja Village as a Basis for Pico Hydro Power Plant Planning in Paddy-Field Area," 2022.
- [5] I. Indrayani, A. Syarif, S. Yusi, M. N. Nugraha, and R. C. Ramadhani, "Utilization of the Kelekar River Flow as Micro-Hydro Power Plant," in *Atlantis Press*, 2022. doi: 10.2991/ahe.k.220205.008.
- [6] R. C. Ramadhani, M. Yerizam, and I. Indrayani, "Analysis of Ogan Ilir Regency's Kelakar River Runoff Discharge in Micro Hydro Power Plant (PLMTH) Planning," *Science and Technology Indonesia*, vol. 5, no. 2, p. 41, Apr. 2020, doi: 10.26554/sti.2020.5.2.41-44.
- [7] Indrayani and R. Renny Citra, "Design of Microhydro Power Plant Prototype Based on Kelekar River Flow Discharge," in *IOP Conference Series: Earth and Environmental Science*, Aug. 2021, vol. 832, no. 1. doi: 10.1088/1755-1315/832/1/012065.
- [8] M. N. Nugraha, R. D. Kusumanto, and Indrayani, "Preliminary Analysis of Mini Portable Hydro Power Plant Using Archimedes Screw Turbine," in *2021 International Conference on Computer Science and Engineering (IC2SE)*, Nov. 2021, vol. 1, pp. 1–5. doi: 10.1109/IC2SE52832.2021.9791966.
- [9] R. Citra Ramadhani and M. Yerizam, "Preliminary Design of Micro Hydro Power Plant in Kelekar River, Ogan Ilir District," 2020.
- [10] Firmansyah, A. Syarif, Z. Muchtar, and Rusdianasari, "Study of the Supply Water Discharge at the Micro Hydro Power Installation," in *IOP Conference Series: Earth and Environmental Science*, Mar. 2021, vol. 709, no. 1. doi: 10.1088/1755-1315/709/1/012002.
- [11] J. Jamal and L. Lewi, "Utilization of Irrigation Flow for the Construction of Micro-Hydro Power Plant," in *AIP Conference Proceedings*, Jun. 2018, vol. 1977. doi: 10.1063/1.5043030.
- [12] K. Kananda, D. Corio, H. K. Restu, H. Aziz, and T. B. Wira, "Potential Analysis of Hydro Power Plants in Pesisir Barat District, Lampung Province," in *ICOSITER 2018 Proceeding Journal of Science and Applicative Technology*, 2018, vol. 100.
- [13] F. Arifin, H. Sutanto, I. Iskandar, and R. Sukwadi, "The Design and Fabrication of Waterwheels with System Floating Pontoon," *International Journal of Research in Vocational Studies (IJRVOCAS)*, vol. 2, no. 3, p. 1, 2022, doi: 10.53893/ijrvocas.v2i3.143.
- [14] K. Daneshkah and M. Zangeneh, "Parametric design of a Francis turbine runner by means of a three-dimensional inverse design method," *IOP Conf Ser Earth Environ Sci*, vol. 12, p. 012058, Aug. 2010, doi: 10.1088/1755-1315/12/1/012058.
- [15] S. Kaewnai and S. Wongwises, "under a Creative Commons Attribution (CC-BY) 3.0 license Improvement of the Runner Design of Francis Turbine using Computational Fluid Dynamics," *American J. of Engineering and Applied Sciences*, vol. 4, no. 4, pp. 540–547, 2011.
- [16] M. Binama, H. X. Chen, Y. Zheng, D. Zhou, and W. T. Su, "A numerical investigation into the pat hydrodynamic response to impeller rotational speed variation," *Sustainability (Switzerland)*, vol. 13, no. 14, Jul. 2021, doi: 10.3390/su13147998.
- [17] D. Biner, S. Alligné, C. Nicolet, D. Dujic, and C. Münch-Alligné, "Numerical fatigue damage analysis of a variable speed Francis pump-turbine during start-up in generating mode," *IOP Conf Ser Earth Environ Sci*, vol. 1079, no. 1, p. 012079, Sep. 2022, doi: 10.1088/1755-1315/1079/1/012079.
- [18] E. Tengs, F. Charrassier, M. Jordal, and I. Iliev, "Multidisciplinary optimization of a Francis turbine runner," *IOP Conf Ser Earth Environ Sci*, vol. 1079, no. 1, p. 012077, Sep. 2022, doi: 10.1088/1755-1315/1079/1/012077.
- [19] A. Susandi, F. Arifin, and R. D. Kusumanto, "Simulation of Diffuser Parameters in the Performance of Horizontal Axis Wind Turbine using Computational Fluid Dynamics," 2021.
- [20] B. M. Umar, J. Cao, and Z. Wang, "EXPERIMENTAL and CFD SIMULATION VALIDATION PERFORMANCE ANALYSIS OF FRANCIS TURBINE," in *IOP Conference Series: Earth and Environmental Science*, Institute of Physics, 2022. doi: 10.1088/1755-1315/1037/1/012003.

- [21] A. Rosidi, D. Haryanto, N. Adi Wahanani, Y. Dwi Setyo Pambudi, and M. Hadi Kusuma, "The simulation of heat transfers and flow characterization on wickless loop heat pipe," *Jurnal Polimesin*, vol. 20, no. 1, 2022, doi: 10.30811/jpl.v20i1.2497.
- [22] F. Arifin, D. Arnoldi, E. Sundari, F. Putri, F. Agasa, Y. Ramadhan, G. Susetyo, and Y.D. Herlambang "Studi analisis simulasi kekuatan beban pada alat bantu pembuatan lubang dengan sudut kemiringan 45 derajat," *Jurnal Polimesin*, vol. 18, no. 2, 2020, doi: 10.30811/jpl.v18i2.1837.
- [23] A. Garmana, F. Arifin, and Rusdianasari, "CFD Analysis for Combination Savonius and Darrieus Turbine with Differences in the Number of Savonius Turbine Blades," in *AIMS 2021 - International Conference on Artificial Intelligence and Mechatronics Systems*, Apr. 2021. doi: 10.1109/AIMS52415.2021.9466009.
- [24] N. W. Khun and E. Liu, "Thermal, mechanical and tribological properties of polycarbonate/ acrylonitrile-butadiene-styrene blends," *Journal of Polymer Engineering*, vol. 33, no. 6, pp. 535–543, Sep. 2013, doi: 10.1515/polyeng-2013-0039.
- [25] M. Rakibuzzaman, K. Kim, and S. H. Suh, "Numerical and experimental investigation of cavitation flows in a multistage centrifugal pump," *Journal of Mechanical Science and Technology*, vol. 32, no. 3, pp. 1071–1078, Mar. 2018, doi: 10.1007/s12206-018-0209-6.