Design and Simulation of Control Flow Vane Nozzle at Cross Flow Turbine

Isnaini Aliman[1,*], Josi Ayu Wulandari Pratama Putri[2], and Priyono Sutikno[3]

[1]Center of New and Renewable Energy Development, Faculty of Mechanical and Aerospace Engineering, Institut Teknologi Bandung

[2]Department of Mechanical Engineering, Faculty of Mechanical and Aerospace Engineering, Institut Teknologi Bandung

[3]Fluid Machinery Laboratory, Faculty of Mechanical and Aerospace Engineering, Institut Teknologi Bandung

*Correspondence email: isnainaliman@gmail.com

Abstract. Huge potential of hydro power sources in Indonesia leads renewable energy innovations into micro hydro power plants as the promising solution. Cross Flow Water Turbine is one of micro hydro power plant turbine which is easy to manufacture and maintenance. As an impulse turbine, nozzle is the key of technology for increasing turbine efficiency performance. Nozzle design of cross flow water turbine by using control flow vane is developed in this journal. Simulation is conducted with the variation heads from 6 meters until 30 meters and 0 - 30 degrees control flow vane angles using numerical analysis in CFD Fluent ANSYS 16.0. The simulation results shows the maximum efficiency at fully open vane condition is reached 87.3%, at 30 meters head, at 650 rpm rotational speed, and power 18.7 kW. At 5 degree control flow vane position, the maximum efficiency is 85.9% at 12 meters head in rotational speed at 350 rpm and power 15.1 kW. Control flow vane at more 5 degree affects the increasing of head losses but the advantageous from this control flow vane is mass flow can be adjusted directly. Changing in incidental angle and increasing of head loss coefficient in nozzle are the main reason for decreasing efficiency of turbine at high head potential.

Keywords: control flow vane, cross flow turbine, efficiency, CFD, head

© 2017. BKSTM-Indonesia. All rights reserved

Intoduction

Indonesia is a country with large electricity needs, in 2016 it was noted that Indonesia's electricity demand reached 232 TWh. This is also accompanied by an upward trend in energy demand that reaches 6.6% annually. Even in 2050, Indonesia's electricity demand is estimated to reach 2,008 TWh [1]. This huge electricity requirement is also followed by an increase in the fossil fuels consumption in electric generating generators which will produce many CO_2 and other wastes that are harmful to the environment.

The environmental issues and the increasing demand for electricity make the government, especially the ministry of energy and mineral resources, targeting by 2050 the role of new and renewable energy will be at least 31% compared to less than 20% of petroleum, 25% of coal, and natural gas by 24% [1]. Renewable energy is dominated by large-scale geothermal energy, hydroelectric power generation, wind generation and new and renewable energy sources through a small scale power plant.

On the other hand, hydroelectricity power is one of the most potential energy generation considering the available hydro energy source of 75,000 MW in Indonesia [1], but its utilization (installed capacity) has only reached 8,671 MW [1]. If this potential is compared with electrical problems in Indonesia in the form of electrification ratio that reached 88.3% (still many villages that have not received electricity), then Micro Hydro Power Generation (PLTMH) is the right solution. PLTMH is an electricity generation that has a capacity below 100 kW. PLTMH usually uses several types of generator turbines such as Axial Turbines, French Turbines, Pelton Turbines, and the others type of Turbine.

Cross-turbine is the most easily manufactured turbine, and easy to maintain. So it is possible Indonesia can be independent in this technology. Therefore, the continuous research on this turbine is required. Basically, research is done by experiment or simulation. However, due to the high cost required to conduct the experiment, this research used simulation technology through computational fluid dynamic method.

Methods

The stages along the construction of this journal can be seen in the flowchart from the **Error! Reference source not found.**. The first stage is to conduct a literature study. The literature study was conducted by studying cross flow turbine characteristics, basic principles of fluid mechanics, and basics on CFD simulations. Literatures that containing information on existing cross flow turbine designs is required as a design reference for this research. Turbine design based on calculations from literature is drawn using CAD applications. Turbine design in the form of CAD will be used in CFD simulation. The CFD process is repeated until the design and performance of the nozzle and turbine housing fulfill the criteria of parameter configuration.

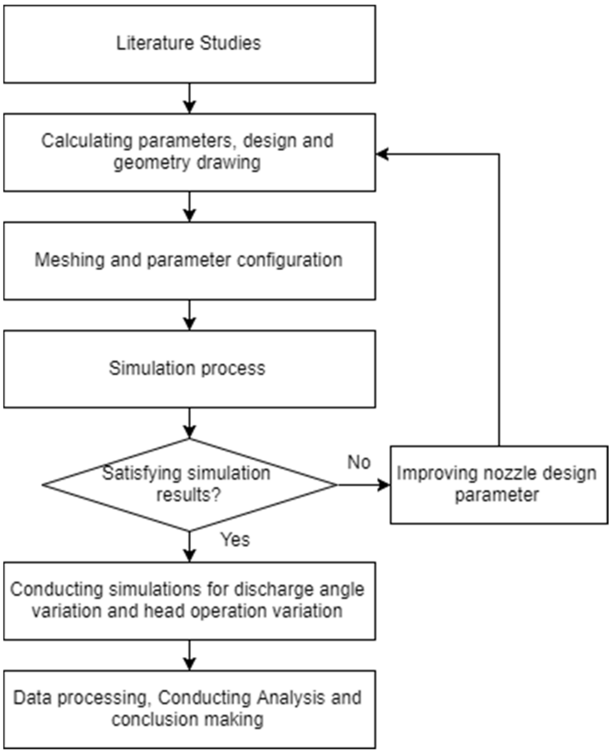


Figure 1. Flowchart of the research

Once the turbine nozzle design has been completed and meets the parameter configuration criteria, the simulation is performed by varying the nozzle angle openings and the variations of the head flow. The simulation results are analyzed by calculating and finding the design optimization point and evaluating the design based on the conformity of the generated parameters.

1. Geometry Design

The design of Cross Flow Turbine used in this research was designed by Dara Seyhak [2]. There are some fixed turbine design parameters that are implemented from Seyhak's turbine design, i.e. the outer diameter of the turbine, the diameter ratio, as well as the inner diameter of the turbine.

The selected outer diameter of the turbine is 175 mm, with a diameter ratio of 0.66, obtained from the equation (1) and (2) [2]. Thus, the inner diameter of the turbine is 116 mm.

$$x^2 - \left[1 - \left(\frac{W_1}{U_1}\right)^2\right].x - \left(\frac{W_1}{U_1}\right)^2 .\sin^2(\beta_1) = 0 \tag{1}$$

$$\left(\frac{r_2}{r_1}\right) = x^{1/2} = 0.66 \tag{2}$$

Turbine efficiency is the ratio of power generated to the power potential of fluid. Based on equation (3) and (4), if the (η) turbine efficiency, to $\frac{U_1}{C_1}$ graph is made through the calculation, it will produce an efficiency curve shows on **Error! Reference source not found.**.

$$\eta_i = \frac{W}{W_{th}} = \frac{W}{{C_{1th}}^2/2} = 2.\varphi^2.\frac{C}{{C_1}^2} \tag{3}$$

$$\eta_i = 2.\varphi^2.\frac{U_1}{C_1}.(1+\psi).(\cos(\alpha_1) - \frac{U_1}{C_1}) \tag{4}$$

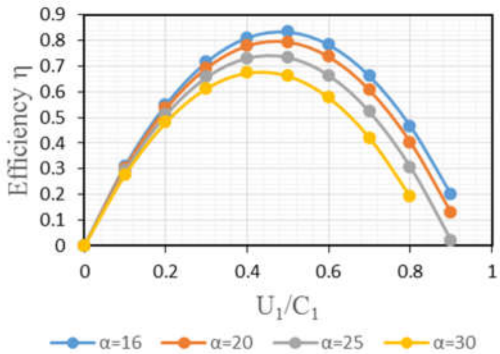


Figure 2. Cross Flow Turbine efficiency for each Angle of Attack

The angle of attack $\alpha_1 = 16°$ is determined by the highest efficiency shown in **Error! Reference source not found.**. It produced turbine efficiency around 85%.

Based on the simulation and experiment result of the turbine by Dara Seyhak, through the width of the inlet blade (λ) variation (λ=60º, λ=70º, λ=80º) and the number of blade (Z) variation (24, 28 and 32), the highest efficiency of the turbine is showed in the turbine by the number of λ=60º and Z=28 [2].

The inlet blade angle (β_1) is determined through equation (5) and (6) [2].

$$\frac{U_1}{C_1} = \frac{\cos(\alpha_1)}{2} \tag{5}$$

$$W_1 \cos(\beta_1) = C_1 \cos(\alpha_1) \tag{6}$$

$$\tan(\beta_1) = 2.\tan(\alpha_1) \tag{7}$$

By the number of $\alpha_1 = 16°$, so the number of $\beta_1 = 29{,}83°$.

Based on some practical limitations described on the basic theory of cross flow turbine, the tangent angle of the turbine runner is worth 43.2 º, the outlet angle I is 90 º, the selected blade radius is 27.8 mm, and the arc length is 33.8 mm. The turbine blade parameter and final design are shown in Table 1 and **Error! Reference source not found.**.

Table 1. Turbine blade parameter

Design Parameters	Units	Number

Outer diameter of turbine	mm	175
Inner diameter of turbine	mm	116
Angle of Attack	°	16
Blade radius	mm	27.8
Turbine and Nozzle Width	mm	110
Number of Blade	-	28
Inlet nozzle width	°	60

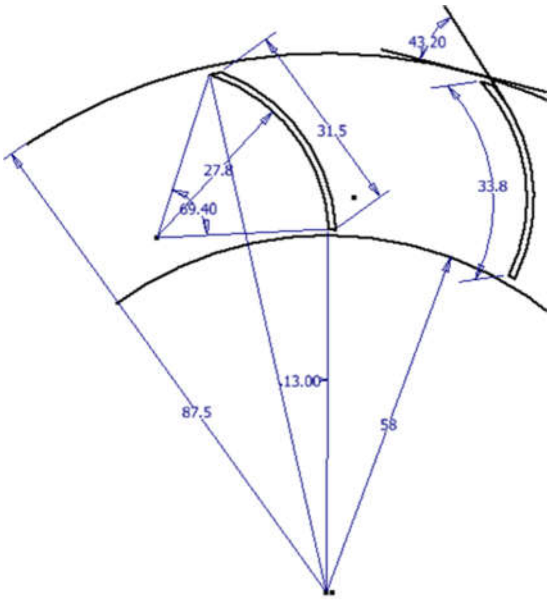


Figure 3. Geometry details configuration of the designed CFWT Turbine

2. Nozzle, Control Flow Vane, and Turbine Housing Design

Control flow vane design is designed considering as an initial parameters turbine design and following the details of the nozzle inspired from several papers or patents manufacture that has a flow vane control (Ossberger, Karl.F, 1986) [3] shown on **Error! Reference source not found.**. The parameters are shown in Table 2.

Table 2. Parameters used in turbine house design

Parameters	Value
The angle of attack	α = 16 °
The admittance angle of flow vane control that directs the flow of fluid into the turbine wheel	in the range of 30 ° to 50 ° to the horizontal plane, 40 ° is the most recommended value.
The nasal control flow vane on the lower side	increased by about 45°
The fully open valve position	Angle of the turbine attack is at 16°
Decreasing and rising angles on the sides of the nozzle housing	in the range of 40 ° to 50°

The nozzle designed based on a trial and error method and designed through a CFD simulations to allow the flow of water into the turbine have a laminar flow and avoid stagnation point on the nozzle surface. The Control Flow Vane initial design adapted from L.A.M.HI, Italy [4] shown on **Error! Reference source not found.**. However, the nozzle wall design used is very different in order to achieve a large input of the blade in the range of 60°.

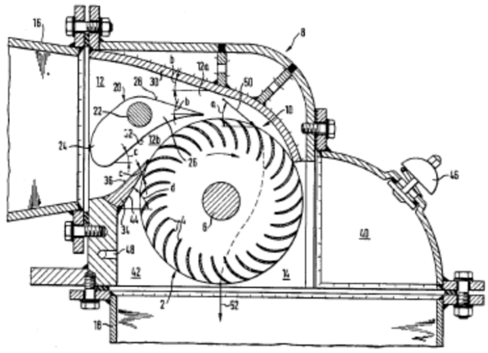


Figure 4. Control flow vane patent by Ossberger [3]

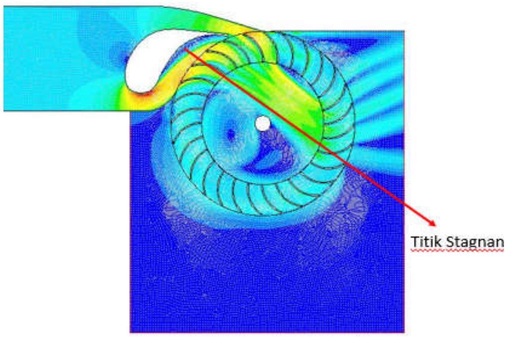


Figure 1. Control flow vane nozzle design initial design with unwanted stagnation point

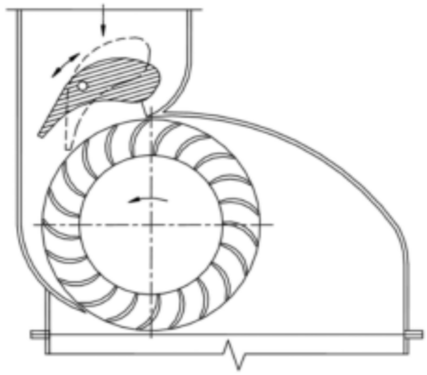


Figure 6. Control flow vane design by L.A.M.HI [4]

The problem seen on **Error! Reference source not found.**, the lower wall of the flow vane control occurs separated flow, where the fluid flowing there is in stagnation condition. Therefore, design improvements are made by permeating the wall of the flow vane control and extending the nozzle. So that, the fluid can adjust the flow and there was no stagnation flow.

After the nozzle design was repaired, there is no separation flow under the control flow vane, but there is no uniformity of flow on the nozzle input side. This is not desirable because the flow velocity when entering the face of the nozzle becomes lower. Therefore, an improvement is made at the base of the nozzle. The base of the nozzle is made more curved and the wall with a large angle is avoided.

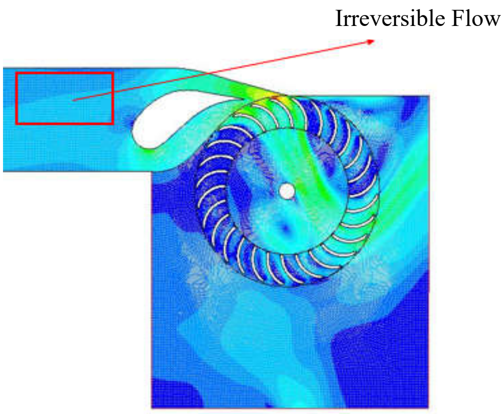


Figure 7. Control Flow Vane Nozzle secondary design with irregular flow

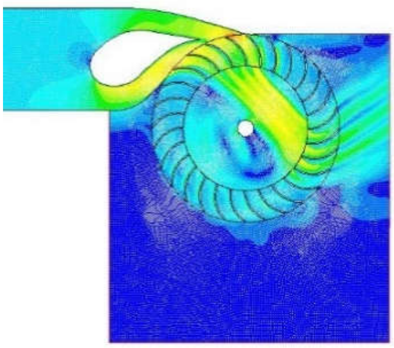


Figure 8. Control Flow Vane Nozzle final design with laminar flow

After the simulation is done on the high head and rotation speed, the working fluid requires a larger space at the upper of turbine housing. Additional space is proposed based on the simulation results in order to avoid the counter flow that could hit the turbine housing and disrupt the turbine flow. This simulation result can be observed on contour velocity, so that the turbine house is repaired as in the **Error! Reference source not found.**.

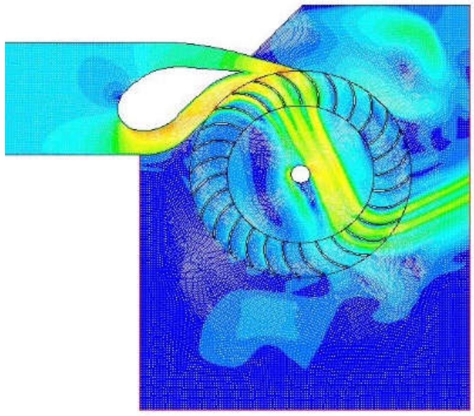


Figure 9. Final Design of Turbine Housing

The flow on the top side of the turbine house can be seen in **Error! Reference source not found.**. This turbine house change results in higher turbine efficiency than previous turbine home designs on the high head.

3. Modeling and Simulation

The result of 2-D turbine component geometry from **Error! Reference source not found.**, **Error! Reference source not found.** and **Table 1** can be seen in **Error! Reference source not found.**.

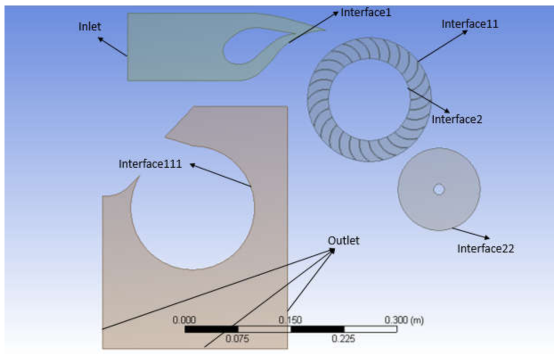


Figure 10. The Interface naming for each selected interface of the geometry parts

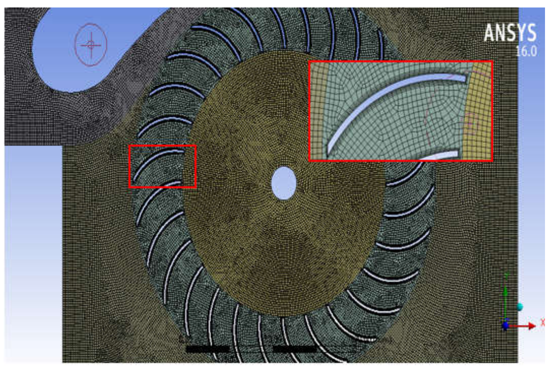


Figure 11. The Boundary Layers and mesh on each blades of the turbine runner

The simulations are performed in ANSYS Fluent under 2-D CFD simulations with its simulation parameters and variables can be seen on **Table** and **Table** . **Error! Reference source not found.** and **Error! Reference source not found.** are generated to describe the interface naming and meshing configuration of the model

Table 3. Variables used in control flow design

Variables	Value
Head variation (m)	6, 9, 12, 15, 18, 21, 24, 27, 30
Flow vane nozzle discharge angle (deg)	0, 5, 10, 15, 20, 25, 30

Table 4. Simulation parameter

Viscous Model	k-omega SST
Cell Zones Condition	Frame Motion (runner)
Boundary Condition	Moving Wall – Rotational (wall runner)
Solution Initialization	Standard initialization

Results and Discussion

The relation of the efficiency, power produced, and angular velocity of the runner to the head shown on **Error! Reference source not found.** and **Error! Reference source not found.**.

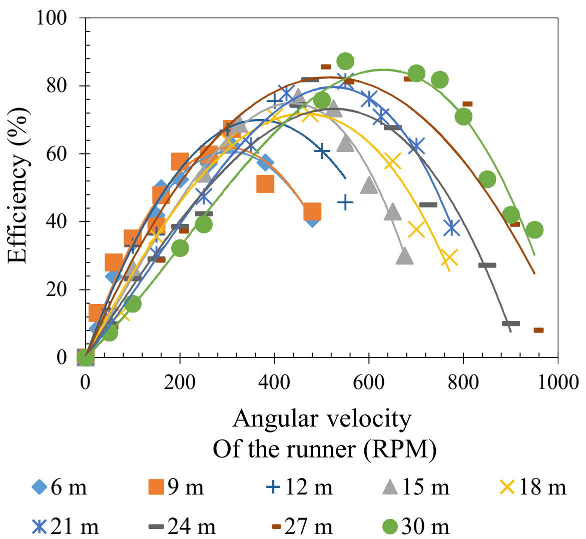


Figure 12. Graphic Efficiency vs Angular Velocity of the Runner (RPM) with full opening Control Flow Vane Nozzle

The **Error! Reference source not found.** and **Error! Reference source not found.** are generated through a series of CFD simulations using the Ansys Fluent program. It was found that the highest efficiency lays on 87.50 % value with full opening angle control vane, 650 RPM and 30 meters head operation condition. The maximum power produced through this simulation is 18,710 W value at 550 RPM, 30 m head, and with full opening angle control vane.

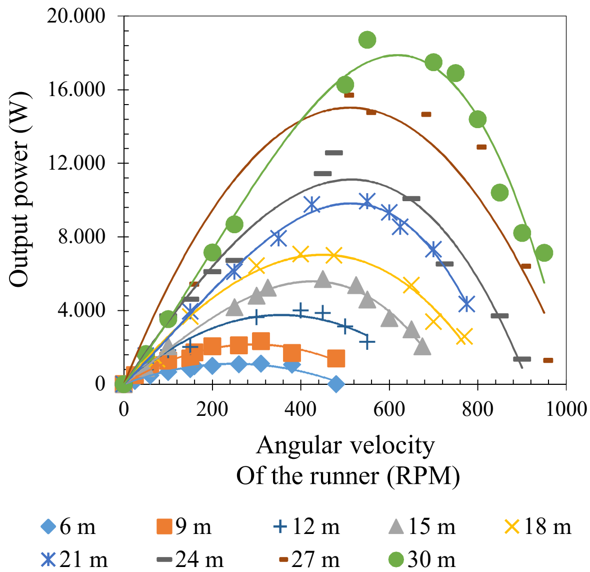
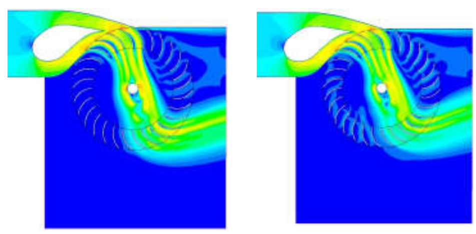


Figure 13. Graphic Output Power vs Angular Velocity of the Runner (RPM) with full opening Control Flow Vane Nozzle

The output power and efficiency graphics pattern shows a parabolic curve appeared to be bigger and larger as the head value increasing due to the higher value of absolute velocity that enters the runner are produced on the higher head value operation. The greater parabolic curve also shows the change of peripheral velocity value to a higher value which made the peak efficiency and power of the Cross Flow Turbine shifts to the right side with the same size of diameter runner.

The tendency of increasing output power and efficiency value as the increasing angular velocity value operated will stop at one highest point and both of the output power and efficiency will soon decrease with the continued angular velocity value. This characteristic value applied due to the incomplete fluid that crashes into blades as it will disrupt the fluid flow existed inside the runner shown on **Error! Reference source not found.**.



0 RPM

200 RPM

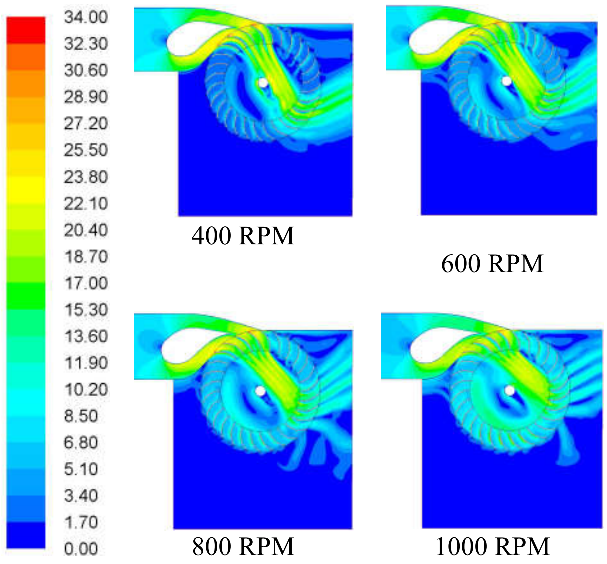


Figure 14. Velocity magnitude contour of Control Flow Vane Nozzle with 30 meters head

At the 5-degree angle opening simulation result shown on **Error! Reference source not found.**, the highest efficiency is at 12 meter head with efficiency reach 85.9 % at 350 rpm rotational speed and maximum turbine output reach 15.1 kW. The graph at 5-degree valve openings on Fig. 14 formed an irregular pattern compared to full valve openings. This occurs because the velocity triangle changes in the fluid flow through the turbine that is affected by the change of angle of attack the nozzle. Changes in the angle of flow vane control also greatly affect the head loss factor in the nozzle as shown on **Error! Reference source not found.**.

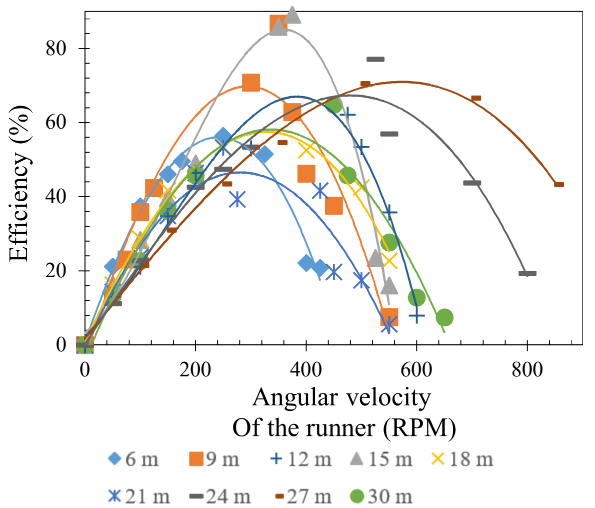


Figure 15. Graphic Output Power vs Angular Velocity of the Runner (RPM) with 5 degrees opening Control Flow Vane Nozzle

Error! Reference source not found. shows that the head loss coefficient curves produced irregularly. The lowest head loss occurs only at full opening Control Flow Vane Nozzle. The value of head loss will increase with the opening of the nozzle valve with unpredicted value of head loss coefficient pattern.

The velocity magnitude contour of opening Control Flow Vane Nozzle variation is shown on **Error! Reference source not found.**. As the increasing number of opening control flow valve, the amount of water flowrate flowing through the nozzle is decreasing. This can be seen from the decrease of the fluid flow velocity affecting the turbine rotor on **Error! Reference source not found.**.

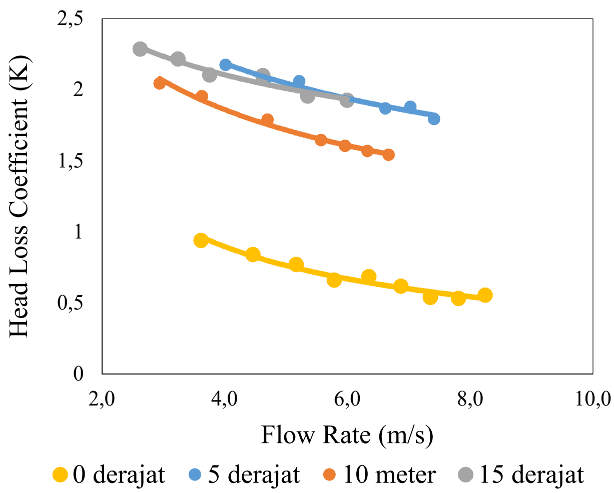
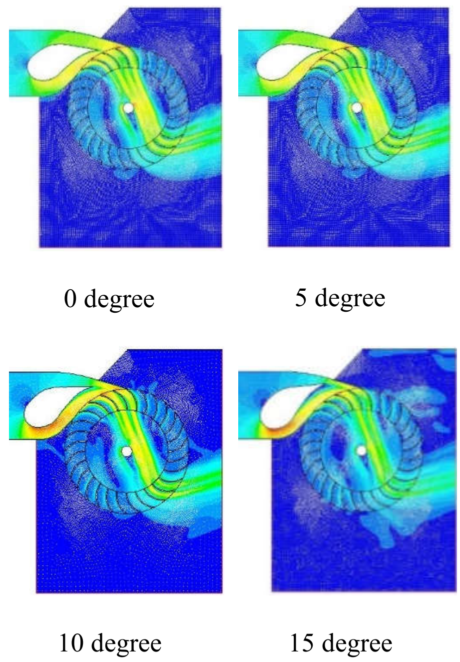


Figure 16. Graphic of head loss coefficient on opening Control Flow Vane Nozzle variation



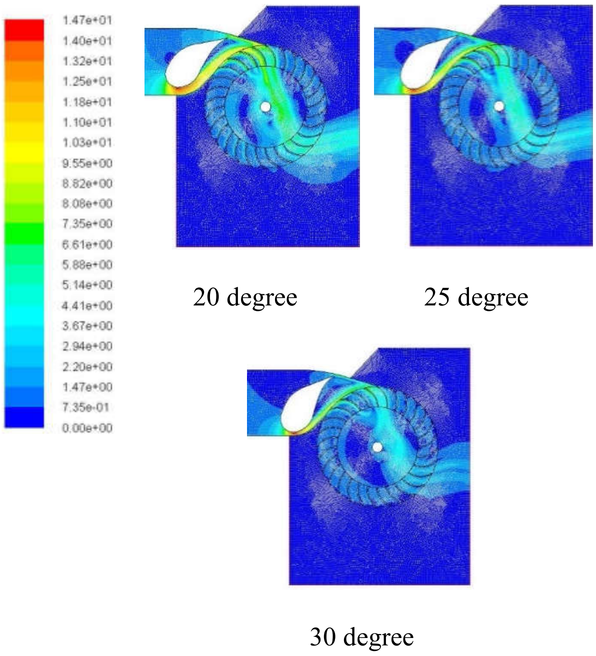


Figure 17. Velocity magnitude contour of opening Control Flow Vane Nozzle variation

The relationship between mass flows with opening Control Flow Vane Nozzle variation shown on **Error! Reference source not found.**. This linear relationship is advantageous later on in real turbine operation. Mass flow can be adjusted directly by the operator by controlling the nozzle angle outside the turbine house. The control flow vane will fully closed if the nozzles angle set to 40 degrees and produces no water mass flow into the turbine.

The 3D Model of the designed turbine is displayed on **Error! Reference source not found.** included the configuration of several turbine parts such as nozzle, body, bearings, additional duct and the runner itself. This 3D model is made based on calculations to recover power from a water turbine from a flow with head 30 m. This design will be made for experiments in the next research.

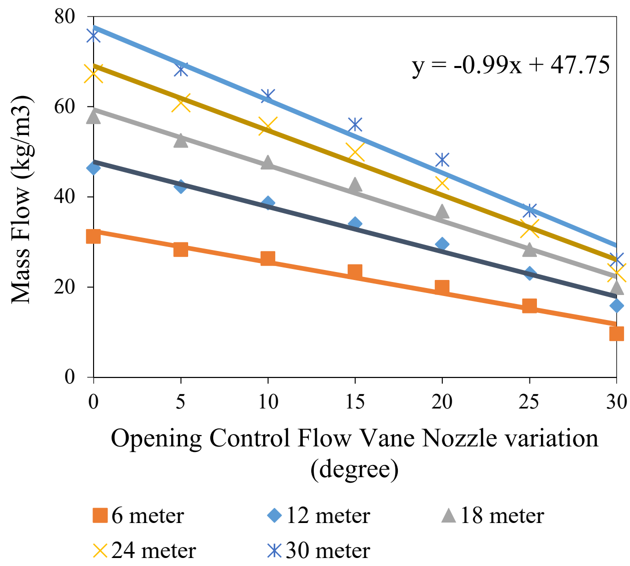


Figure 18. Graphic of mass flow to the opening Control Flow Vane Nozzle variation

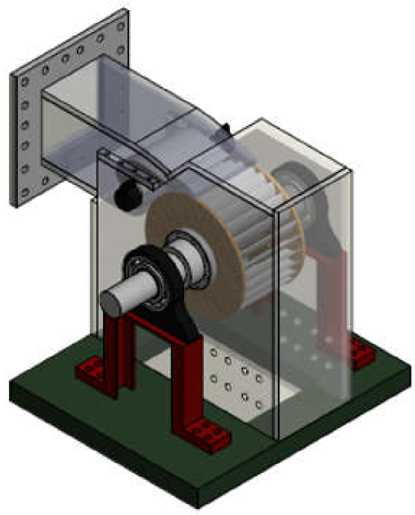


Figure 19. 3-D Model of the designed CFWT Turbine for experiment

Conclusions

The designed Cross Flow Water Turbine with Control Flow Vane Nozzle for various heads are constructed with the geometries list as follows:

- D1 = 175 mm and inner diameter
- D2 = 116 mm
- Number of blade = 28 number of blades
- Angle of attack (α_1) = 16º
- Blade's arc radius (ς) = 27.8 mm

Maximum efficiency generated by the design with full open control flow vane discharge angle is of 87.50 % for the angular velocity operation on 650 RPM and 30 meters head condition. The simulations for various heads had produced the biggest power output on head 30 meters with a number of 18,710

W 30 m head through a full opening angle control vane.

The simulation result for various head values shows the larger and wider parabolic curve as the increase of head values. This occur due to the increasing absolute velocity of fluid entering the runner and affect the peripheral and angular velocity.

The efficiency graphic through a valve openings variation formed an irregular pattern compared to full valve openings. This occurs because the velocity triangle changes in the fluid flow through the turbine that is affected by the change of angle of attack the nozzle. Changes in the angle of flow vane control also greatly affect the head loss factor in the nozzle.

The advantageous from this design is mass flow can be adjusted directly by the operator by controlling the nozzle angle from the outside of turbine house.

References

[1] Keputusan Menteri Energi dan Sumber Daya Mineral Republik Indonesia, RUPTL PLN 2016-2025, 10 Juni 2016.

[2] Dara, S., 2016. A Numerical Study and Experimental Measurement on The Design Of A Cross Flow Water Turbine Model.

[3] Ossberger, K., O. Reider, and P. Partzsch,, 1986. Horizontal-Inflow, Vertical-Outflow cross-flow turbine. United States Patent.

[4] Pequenas Centrales Hidraulicas, Facultad De Ingenieria, Laoratorio De Maquinas Hidraulicas (LA.M.HI.), Italia.