Design and Simulation of Three-Way Nozzle on Cross Flow Water Turbine for Various Heads

Ika Kurniawati[1,*], Isnain Aliman[2], and Priyono Sutikno[3]

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

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

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

*Correspondence email: ikakurniawati_x6@yahoo.com

Abstract. The influence of Three-Way Nozzle and the various head operations for a Cross Flow Water Turbine (CFWT) under a CFD approach is proposed on this current work. Many advantages performed by A CFWT as an impulse turbine such as its simple, economical, and easy-to-manufacture type of turbine. The advantage characteristic of its various head operations is also performed in a CFWT turbine. The usage of CFWT as Micro Hydro Power Plant Turbine in rural areas based on those advantages is understood as an importance background of this work. The variation of discharge angle for The Three-Way Nozzle, the design of Nozzle Roof and the various head value operation are included as design parameters find its best performance. Efficiency level as performance result on the level of 81.73% is generated by the 90º discharge angle with 5 meters head operation under the 2-D steady state flow CFD simulation. Stable characteristic of efficiency levels with the operations of various heads are performed by The Three Way Nozzle CFWT under an error value between efficiencies below 6%.

Keywords: Cross Flow Water Turbine, three-way nozzle, CFD, head, pressure

Introduction

The constant development of Indonesia's energy sector occurred along with its increasing energy consumption rate making the needs of reliable, simple and remote for rural area power plant become higher. One of the promising power plant which can answer those characteristics is the Micro Hydro Power Plant (PLTMH). The value of peak load that increases from the number 35,828 MW at 2016 until the level of 74,373 MW at 2025 shows the continued growth that still occur [1]. This is also followed by the increasing number of PT. PLN (Persero) electricity consumer that reaches the level of 85.27 million at 2025[1]

Indonesia particularly serve potentials of alternative energy resources aside from coals and fossil fuels in energy generation power source, one of them is Hydro Power. According to *Master Plan Study of Hydropower Development in Indonesia* by PT. PLN (Persero), there are 116 Hydro Power potential sites spread throughout Indonesia that already completed the screening process [2].

According to the mentioned information, in order to find solutions for electricity demands especially in rural areas, Micro Hydro Power Plant could be the promising option answering the existing energy demand. Thus, the development of Micro Hydro Power Plant should be improved, expanded and increased on its efficiency sector for the improvement on project investment.

Cross Flow Water Turbine is one of the preferred turbine type for the Micro Hydro Power Plant based on its easy-to-manufacture and inexpensive characteristic. Most researches have not perform any numerical CFD simulations for the three-way Cross Flow Turbine that can be one of option for turbine design to improve the plant's efficiency. This bachelor thesis is constructed to find the performance characteristic of the three-way Cross Flow Water Turbine as the option of Micro Hydro Power Plant Water Turbine.

Methods

The stages along the construction of this journal can be seen in the flowchart from the Fig. 1. The design parameters for the research along with equations used for design and calculation are included on the second stage of calculating parameters, design, and geometry drawing. Based on Euler equation applied for the turbine runner, the efficiency as a function from angle of attack are set up as follows [3]:

$$W = (U_1 C_{1U} - U_2 C_{2U}) + (U_3 C_{3U} - U_4 C_{4U}) \quad (1)$$

$$W = U_1 (W_1 \cos(\beta_1) + \psi W_1 \cos(\beta_1)) = U_1 (1+\psi) W_1 \cos(\beta_1) \quad (2)$$

$$= U_1\,(1+\psi)(C_{1U} - U_1)$$
$$= (1+\psi)(C_1\cos(\alpha_1) - U_1)$$
$$\eta = \frac{W}{Wmax} = 2\varphi^2\frac{W}{C_1{}^2} \qquad (3)$$
$$= 2\varphi^2\frac{U_1}{C_1}(1+\psi) * (\cos(\alpha_1) - \frac{U_1}{C_1})$$

Under assumption where the value of β_2 (angle between relative velocity and absolute velocity in section 2) = 90°, four design angle of attack options are set up for the range 16°, 20°, 25°, and 30°. Fig. 2 are generated showing the best angle of attack using for the design under analytical calculation [3].

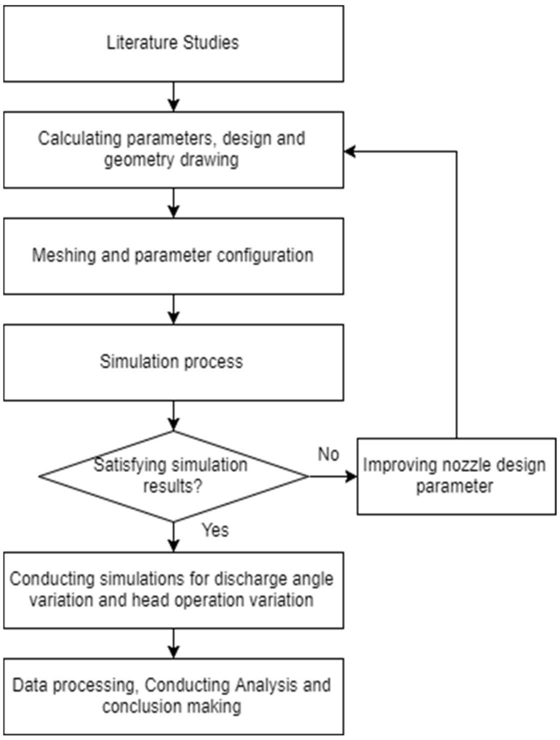


Figure 1. Flowchart of the research construction

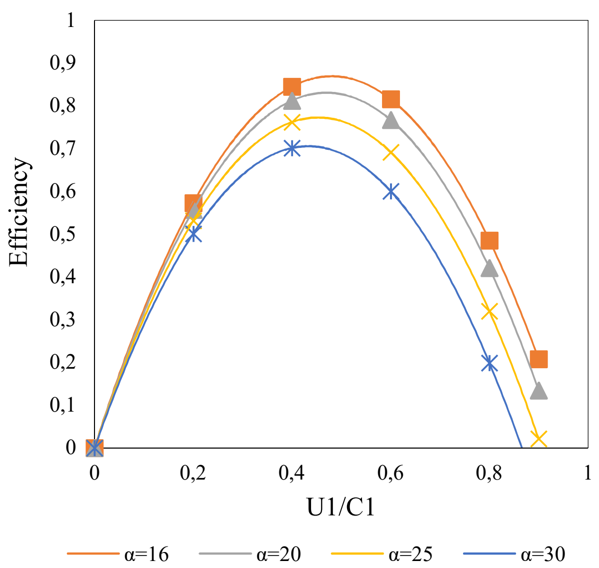


Figure 2. Cross Flow Turbine efficiency for each angle of attack

The absolute velocity value of fluid entering the runner can be determined in order to find the size of runner outer diameter where it is set up as follows:

$$C_1 = \varphi\sqrt{2.g.H} \qquad (4)$$
$$U_1 = 0.5.C_1.\cos(\alpha_1) \qquad (5)$$
$$U1 = \frac{\pi.Do.500}{60} = 4.5867 \text{ m/s}$$
$$Do = 0.1752\ m = 175.2\ mm$$

Those calculation are performed under several assumptions such as 500 RPM runner angular velocity, 0.96 nozzle efficiency (φ), 9.81 m/s^2 gravity acceleration and head value 5 m. The ratio of diameter can be found using the derivation of several calculation which set up as follows:

$$x^2 - \left[1 - \left(\frac{W_1}{U_1}\right)^2\right]x - sin^2(\beta_1) = 0 \qquad (6)$$
$$\text{Where } \tan(\beta_1) = 2\tan(\alpha_1)$$
$$x^2 - [1-(1.1527)^2]x - (1.1527)^2 sin^2(29.83) = 0 \qquad (7)$$
$$x^2 + 0.328x - 0.661 = 0$$
$$x = 0.453$$
$$\left(\frac{r2}{r1}\right) = \sqrt{x} = \sqrt{0.453} = 0.66 \qquad (8)$$

A quadratic equation [3] as shown below is used to derive the ratio value of diameter and radius of the runner. The value of blade radius arc are set up as follows regarding equation below [3]:

$$\varsigma = \frac{(r_1{}^2 - r_2{}^2)}{2.r_1.\varsigma.\cos\beta_1} \qquad (9)$$

where r_1 equals outer radius of runner, r_2 equals outer radius of runner. The value of arc opening angle of the blade can be found using this equation below [3]:

$$\tan(\frac{1}{2}\delta) = \frac{\cos(\beta_1)}{\frac{r2}{r1} + \sin(\beta_1)} \qquad (10)$$

where r_2/r_1 ratio = 0.66 and β_1 = 29.83° are used.

Fig. 3 are generated to describe the turbine designed geometry to be imported into 2-D Workbench Fluent for simulation process. The significant variables involved in Nozzle's Geometry design are the S_o (the magnitude of the highest point from the nozzle's roof), the discharge angle of nozzle (λ), nozzle's angle of attack (α_1), W (the width of runner) and the value of angle formed by the highest point of the nozzle room with the left side of nozzle's discharge area (γ) as shown in Fig. 4. Using the continuity equation at the inlet area of the nozzle for derivations as shown below, the equations to calculate the value of S_o can be found. Thus, we also can compute the coordinates to find the shape of nozzle's roof arc ($R(\Theta)$).

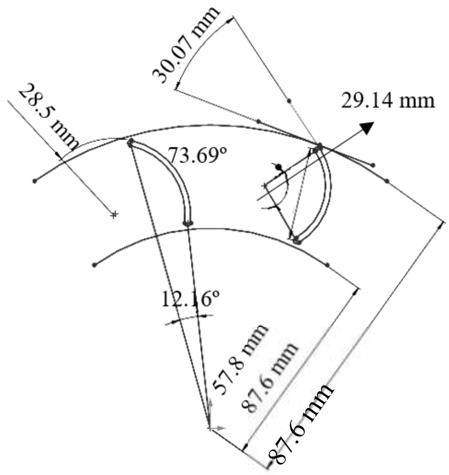


Figure 1. Geometry details configuration of the designed CFWT Turbine

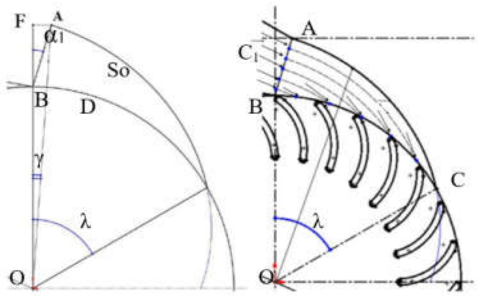


Figure 2. The configuration of nozzle roof and its shape with the involved fluid flow [3]

The following equations are set up to found the nozzle's geometries value:

$$So = \sin(\alpha_1).\lambda.(\frac{D1}{2}) \qquad (11)$$

For the value of discharge angles 100º, 90º, 80º, 75º, 60º, 45º, 30º, the value of S_o would be calculated.

The value of γ can be found using the triangle geometry concept for the value of *discharge angle* 100º as explained in the next passages

$$AF = So \sin(\alpha_1) \qquad (12)$$

$$FO = \frac{D1}{2} + So \cos \alpha_1 \qquad (13)$$

$$AO^2 = FO^2 + AF^2 \qquad (14)$$

$$\gamma = cos^{-1}(\frac{FO}{AO}) \qquad (15)$$

The relationship between So and the value of discharge angle can be shown into a graphic below as a fine linear line as explained by equation 18.

$$So = 24.145\,\lambda \qquad (16)$$

The coordinates of nozzle's roof shape (R(Θ)) are set up using the following equations and its value are described on Table 1.

$$AD = So \cos \angle BAD \qquad (17)$$

$$\angle BAD = 180 - \angle ABO - \gamma$$

$$\angle BAD = 180º - (180º - 16º) - 5.1536º = 10.8464º$$

Where $0 < \Theta < 89.1536º$

$$R(\Theta) = So \cos \angle BAD + (D_1/2) \qquad (18)$$

$$R(\Theta) = (24.145\lambda) \cos \angle BAD + (\frac{D1}{2})$$

Table 1. The radius of nozzle roof's highest point for each degree value (Θ)

Θ (º)	Θ (rad)	R	x	Y
0	0	87.61	0.00	87.61
5	0.087266	89.67	89.33	7.81
10	0.174533	91.74	90.34	15.93
15	0.261799	93.81	90.61	24.28
20	0.349066	95.88	90.09	32.79
25	0.436332	97.95	88.77	41.39
30	0.523599	100.01	86.62	50.01
35	0.610865	102.08	83.63	58.56
40	0.698132	104.15	79.79	66.95
45	0.785398	106.22	75.11	75.11
50	0.872665	108.29	69.61	82.95
55	0.959931	110.36	63.29	90.40
60	1.047198	112.43	56.21	97.37
65	1.134464	114.50	48.39	103.77
70	1.22173	116.57	39.87	109.54
75	1.308997	118.64	30.70	114.59
80	1.396263	120.71	20.96	118.88
85	1.48353	122.78	10.70	122.31
89.15	1.556024	128.63	0.00	124.50

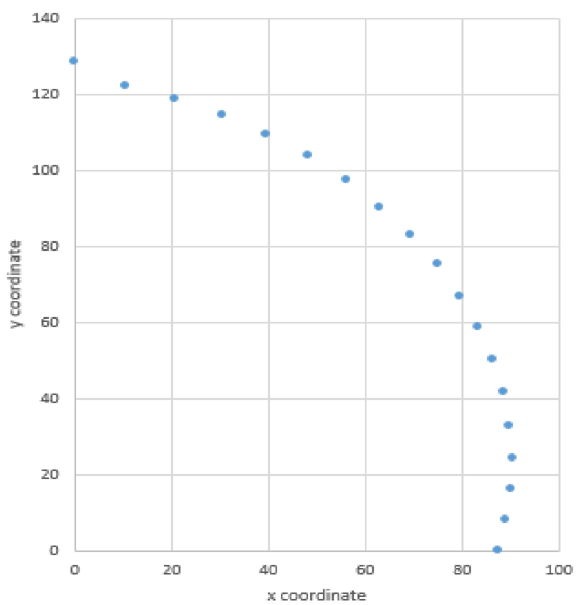


Figure 3. Nozzle roof shape coordinates

The result of 2-D assembly geometry can be seen in Fig. 6.

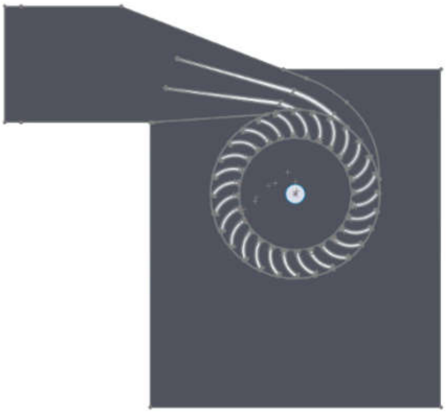


Figure 4. 2-D Assembly view of the designed Cross Flow Turbine

Several change variables are implemented to see the characteristic of the designed turbine as shown in Table 2. The simulations are performed in ANSYS Workbench Fluent 15.0 under 2-D CFD simulations with its simulation parameters can be seen on table 4 in appendix page. Figs. 7 and 8 are generated to describe the interface naming and meshing configuration of the model.

Table 2. The list of Change variables to be implemented on each 2-D modelling and simulations

Variables	Value
Discharge angle (deg)	30, 45, 60, 75, 80, 90, 100
Head variation (m)	5, 10, 15, 20, 25, 30

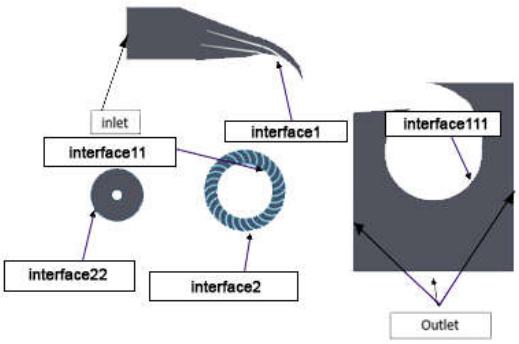


Figure 5. The Interface naming for each selected interface of the geometry parts

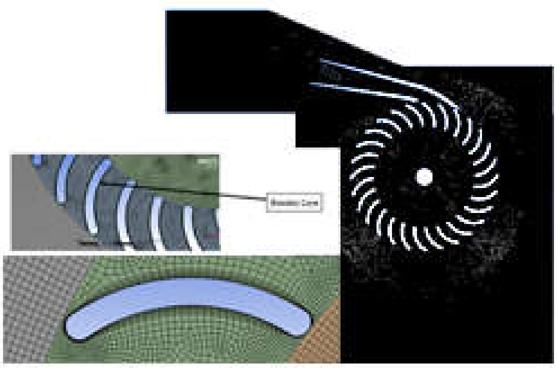


Figure 6. The Boundary Layers and mesh on each blades of the turbine runner

Result and Discussion

Analysis were made upon the simulation results of the discharge angle variation operation and it was found that the highest efficiency of the designed CFWT lays on the design with discharge angle 90º on 81.73% value at 400 RPM and 5 meters head operation condition. The highest result of efficiency with 81.73% has a higher value compared to the previous thesis result by Mr. Dara Seyhak with a value 81.18% [3] on the same angular velocity value 400 RPM for the 5 meters operation head as shown in Fig. 10.

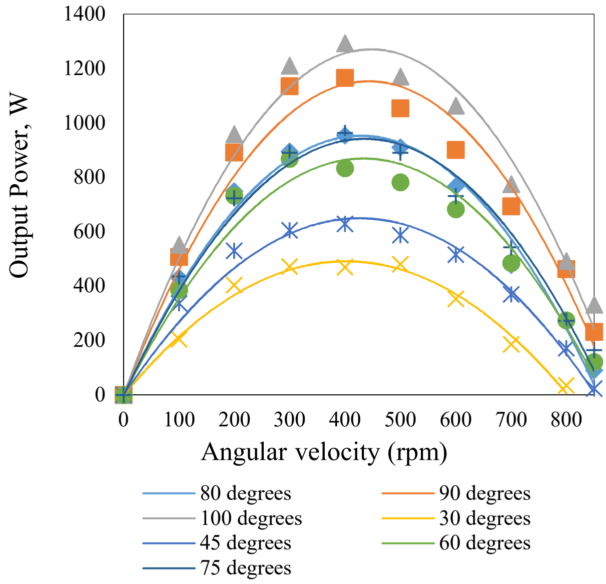


Figure 7. Graphic Output Power vs Angular Velocity of the Runner (RPM) for each design variation of the designed Cross Flow Turbine

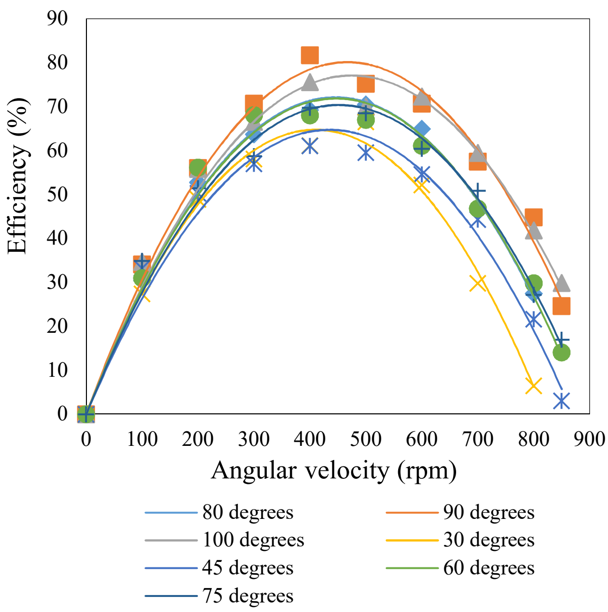


Figure 8. Graphic Efficiency vs Angular Velocity of the Runner (RPM) for each design variation of the designed Cross Flow Turbine

The design with 100º discharge angle generated a decreasing efficiency value despite the increasing output power value due to its larger discharge area that lead the fluid flows crashing into the shaft. Thus, some losses are produced in this condition.

The significant different of this designed Cross Flow Turbine with three-way nozzle compared to the previous thesis [3] is the highest efficiency reached on 90º *discharge angle* design where the previous thesis has the highest efficiency on 60º *discharge angle* design. This resulted in conclusion that larger area is needed for the partition of three way space itself on three-way nozzle CFWT.

Another section of simulation is performed to see the performance upon the head variation value operation (5 m, 10 m, 15 m, 20 m, 25 m, and 30 m). Result was generated that the torque value improved way higher with the increasing value of head used. The characteristic of increasing torque described the tendency of higher power generated with the higher value of head used that is shown in graphic from Fig. 9.

Both of the output power and efficiency graphic on Fig. 11. and Fig.12., the pattern of 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. This parabolic form of efficiency curve where its peak point slightly shifts to the right which indicates the need for greater value of angular velocity is the characteristic that has the similarity form with another Cross Flow Turbine efficiency-speed curve tested for several operation head with range of 5 to 60 meters by Mr. Chandran [4].

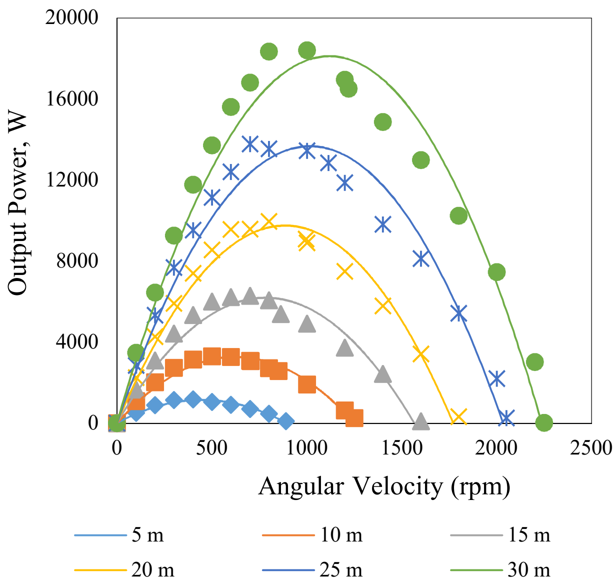


Figure 9. The graphic shows the output power in respect to Angular Velocity (RPM) for each various head value

The various head simulations had shown stable efficiency level of the turbine when it operates for various heads with its error for each peak efficiency points are all below 6% as shown in Table 13. This turbine design will be very promising for flexible head operation and high head value condition with considering for further material selections analysis.

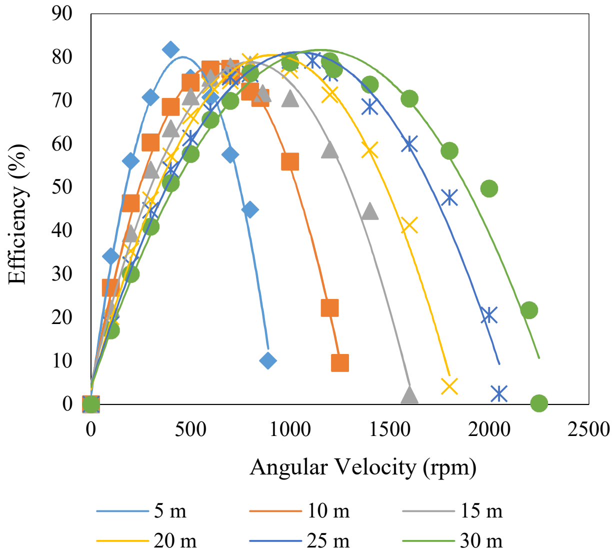


Figure 10. The graphic shows the efficiency value in respect to Angular Velocity (RPM) for each various head value

Table 3. The error range between efficiencies of the various head operation

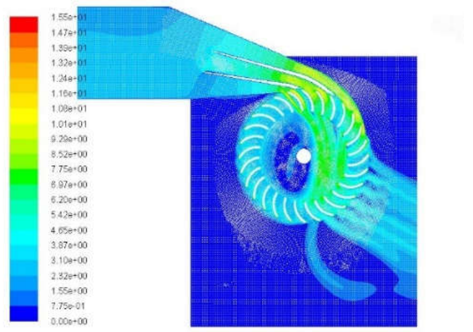
Head (m)	Highest Efficiency (%)	Error (%)
5	81.73	0
10	77.27	5.46
15	78.24	1.25
20	78.96	0.92
25	79.24	0.35
30	79.04	0.25

On the 30 meters head operation condition for the designed turbine, it results around 18.41 kW for highest output power and 79.04% for highest efficiency level.

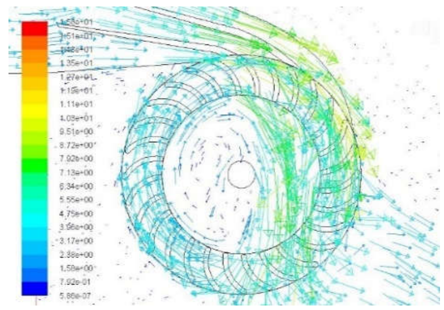
An experiment was conducted by previous thesis [3] used as a reference for CFWT improvement on this paper. Result was generated on experimental efficiency level around 77.9% compared to the 82% simulation efficiency. This showed the error occurred between the two values of efficiency is around 5%. Thus, same performance decrease characteristic due to error is expected to be happening if the designed turbine on this paper is fabricated and tested.

The lowest value of absolute pressure of this CFWT on 5 meters head operation is found to be at the level of 49867.87 Pascal which has the higher value than the selected saturated vapor pressure on the temperature 25ºC with its value 3169 Pascal. Based on that, we can know that there is no cavitation occur on the simulation result of the mentioned CFWT operation.

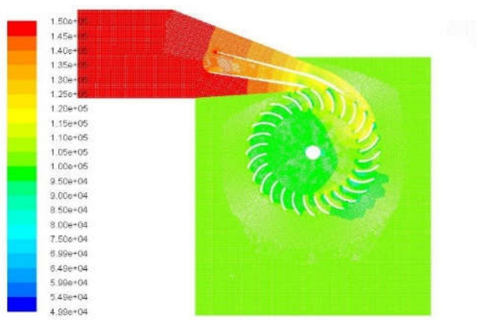
Fig. 13 a, b, c, each showed the velocity magnitude contour, vector velocity contour and absolute pressure contour of the designed CFWT turbine on 5 meters head and 400 RPM operation. Fig. 13. (a) showed that on 400 RPM condition with its highest efficiency value performed, there is no fluid flow which crashed into the shaft that explained its best performance result. Fig. 13 (c) showed that the lowest absolute pressure which existed on the turbine according to the simulation appeared on its area which had the highest value of velocity, around the blade of the runner.



(a)



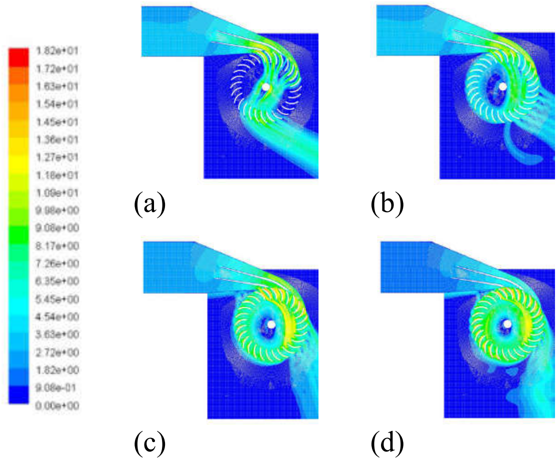
(b)



(c)

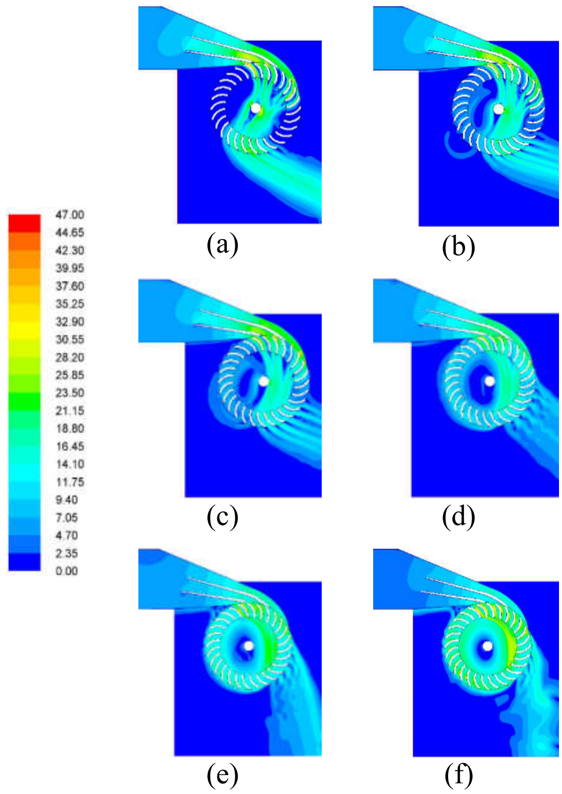
Figure 11. (a) Velocity Contour (b) Vector Contour (c) Absolute Pressure Contour

Fig. 14. a, b, c, d, e, f, showed the comparison between velocity contour of 5 meters head operation and 30 meters head operation. Those figs showed the best performance generated by each head operation had different angular velocity operation value. On 5 meters head, the best performance was generated by 400 RPM angular velocity operation value. On 30 meters head, the best performance was generated by 1200 RPM angular velocity value. This showed the angular velocity operation which generated the peak efficiency point shifted to the right (the higher value of angular velocity) along the increase of head operation value due to the higher absolute velocity entering the runner.



(a) (b) (c) (d)

Figure 12. Velocity Magnitude Contour (m/s) of CFWT on 5 m head operation on (a) 0 rpm, (b) 400 rpm, (c) 700 rpm, (d) 996 rpm



(a) (b) (c) (d) (e) (f)

Figure 13. Velocity Magnitude Contour (m/s) of CFWT on 30 m head operation on (a) 0 rpm, (b) 400 rpm, (c) 700 rpm, (d) 1200 rpm, (e) 1800 rpm, (f) 2440 rpm

All the performance parameters result generated by the simulation was illustrated on the same graphic as shown in Fig. 16 for the 15 meters head operation of the designed CFWT. The value of efficiency, head, and hydraulic power can be read by pulling straight line to the left y-axis and the output power, torque, mass flow value can be read by using the right y-axis.

The 3D Model of the designed turbine is displayed on Fig. 16 included the configuration of several turbine parts such as nozzle, body, bearings, additional duct and the runner itself.

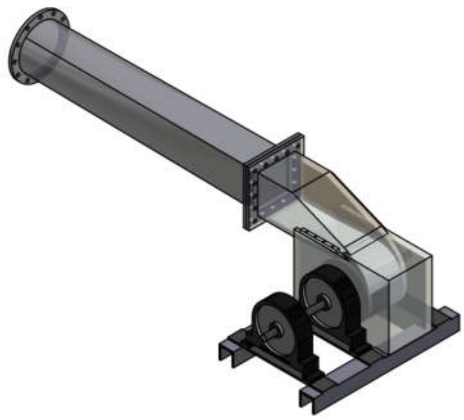


Figure 14. 3-D Model of the designed CFWT Turbine

Conclusion

In this paper, the designed CFWT turbine with three-way nozzle for various heads are constructed with the geometries list as follows:

- D1 = 175.2 mm and inner diameter
- D2 = 115.6 mm
- Number of blade = 28 number of blades
- Angle of attack (α_1) = 16º
- Blade's arc radius (ς) = 28.5 mm
- Discharge angle as design alternatives for nozzle design = 30º, 45º, 60º, 75º, 80º, 90º and 100º.

Maximum efficiency value was generated by the design with 90 discharge angle on the number of 81.73% for the angular velocity operation on 400 RPM and 5 meters head condition. The simulations for various heads had produced the biggest power output on head 30 meters with a number of 18.41 kW and efficiency value of 78.8% on 1000 RPM. For highest efficiency on this condition lays on 79.04% with the output power 16.96 kW on 1200 RPM.

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 head value variation simulations showed a stable performance of efficiency on the designed turbine with error between efficiencies below 6%. Hence this designed turbine would have performed a stable performance despite various head operations.

Reference

[1] Kementrian Energi Dan Sumber Daya Mineral, 2016. Rencana Usaha Penyediaan Listrik PT. Perusahaan Listrik Negara (PERSERO) Tahun 2016 s.d 2025.

[2] L. Nippon Koei Co., Project for the Master Plan Study of Hydropower Development in Indonesia, Kementrian Energi dan Sumber Daya Miner.

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

[4] Chandran, J. dkk., 2010. Development and Testing of A Cross Flow Turbine, *IGHEM: India*.