ISBN 978-602-53531-8-5

SIMULATION OF WATER FLOW DISTRIBUTION ON LOW-HEAD PICOHYDRO CAPACITY 2 kW

Agung M. Werdhana[1,*], Agus Hermanto[2]

[1]Graduate Student of Mechanical Engineering, Itenas, Bandung – INDONESIA

[2]Dept. of Mechanical Engineering, Itenas Bandung – INDONESIA

[*] Corresponding author e-mail: agungmukti@gmail.com

Abstract

The amount of potential water energy in irrigation channels in Indonesia has not been optimally utilized because it has a small flow and low head. This study will simulate a low-head picohydro plant with a net head of 1.5 m whice is easily applied without altering the civil structure of the irrigation canal. The phenomenon of water flow pattern in headtank and horizontal axial turbine intake have an effect on the potential energy of water flow. Potential energy of flow at 0.22 m3/s discharge and 1.86 m head will be simulated using CFD (Computational Fluid Dynamic). The CFD results show that the reverse flow occurs at several positions in the headtank affecting the uniformity of the flow pattern and reducing the potential for utilization of electric water. The simulation results show that the flow power potential for a low-head picohydro generating system is 3190.2 Watt with a fluid velocity passing through a turbine runner of 2.28 m/s.

Keywords: Design, Picohydro, Low Head, Flow Pattern, and CFD Simulation.

1. Introduction

The uneven distribution of electrical energy occurs in rural areas due to inaccessibility of the power grid to the countryside, it takes electricity procurement effort to fulfill rural energy necessity. One way to fulfill this necessity in order to cover the lack of distribution is utilizing the existing energy sources, water energy in irrigation channels having energy from flowing water and low head water. Unfortunately, no one has developed and applied a low head picohidro power plant system, therefore a design that can exploit the potential energy of water in the irrigation canals is absolutely needed. Generally the use of irrigation channels is limited to irrigating rice fields and plantations.

The picohidro power plant system is designed with capacity of 2 kW, net head of 1.5 m. It uses horizontal axial flow turbine utilizing irrigation channel without altering the existing civil structure, verificated with potential energy generated and recognizing the flow pattern on the generating system without involving the turbine blade used dynamic fluid flow computation software (CFD).

The working principle of picohidro generating system by utilizing irrigation channel is similar to hydro power in general, the difference is in the intake channel and head race directly using the irrigation channel. Thus it does not require lead or steering dam. The water flow in the irrigation channel will be accommodated in the head tank firstly, the collected water has a height difference between the upper and lower water levels it's called name head as the water comes out from the accommodated water turbine, channeled through a conductor channel which consist of nozzles and a small pipe in which the turbine will be placed properly so that it can rotate well. The turbine rotation generates torque and the torque is connected with spinning velocity, it will produce power that will be transmitted through the shaft, pulley and v-belt for rotating the generator that serveing as a power plan.

2. Design of Picohydro

The potential energy of water available for energy generation can be calculated based on the equation below:

$$\mathrm{P} = \rho \times g \times Q \times H_n \times \eta \quad (1)$$

Designing a 2 kW picohidro generating system requires several design criteria for design calculating design serving as limitations in the design, the required criteria are provided in table 1.

Table 1: Design criteria

Information	Design Criteria
Generated power	2 000 Watt
Net head	1.5 m
Turbin efficiency	75%
Generator efficiency	85%
System transmission efficiency	95%

The power that must be provided by the irrigation channel to generate 2 kW generation power using equation 1 is 3.3 kW and turbine output power of 2.48 kW with the requirement water discharge of 0.22 m^3/s.

Specific speed is an important parameter in designing a turbine used in a generating system such as to determine the type of turbine to be used, the turbine rotational speed during operation and determining the geometry of the turbine especially the diameter of the turbine. The specific speed equations used are as follows:

$$n_{QE} = \frac{n\sqrt{Q}}{E^{3/4}} \tag{2}$$

The specific speed of the propeller turbine has a limit value that has been developed, used and made as a reference for the designing of turbine propellers. Limitations of the specific velocity value using the ESHA (2004) reference is $0.19 \leq n_{qe} \leq 1.55$ with using equation 2 obtained turbine rotational speed at the time of operation has a minimum of 180.8 rpm and a maximum of 1474.9 rpm. The turbine turn speed of design to be used in this design is 500 rpm, this value is based on the design use of a generator having specification of operating speed on 1500 rpm simplifiying the designing transmission system to be used.

The dimensions of turbine diameter can be obtained using the equation of ESHA as follows:

$$\mathrm{D} = 84{,}5\left[0{,}79 + \left(1{,}602 \times \mathrm{n_{QE}}\right)\right]\frac{\sqrt{\mathrm{H_n}}}{60 \times \mathrm{n}} \tag{3}$$

Dimension of turbine diameter base on from calculating using equation 3 is obtained 0.34 m. It is used as reference to the dimension of pipe diameter for delivering water to the turbine.

The position of the turbine set on the generating system affects the cavitation phenomena during the turbine operation. The cavitation will occur when the dynamic hydro pressure in the flow stream falls below the vapor pressure of the water properties there will be the formation of the vapor phase. This phenomenon will result in the form of small bubbles that occur in the area of low pressure by the flow and when the bubble burst it will produce a high pressure. If it happens, it produces a noisy sound which will cause damage to the turbine such as pitting will occur on the runner causing the effects of vibration while turbine operates. Cavitation is a characteristic marked with the value of the cavitation coefficient called the thoma number (Celso Pence, 2004), defined by the following equation:

$$\sigma = \frac{NPSE}{gH_n} \tag{4}$$

in order to avoid the cavitation phenomenon it requires minimum pressure during the turbine to operate so it takes the ideal placement distance for the turbine that is the vertical distance from the bottom water surface level to the center of the turbine is also called a turbine suction head (Celso Pence, 2004), The turbine suction head can be calculated by the following equation:

$$NPSE = \frac{p_{atm} - P_v}{\rho} + \frac{V^2}{2} - gH_s \tag{5}$$

In order to obtain thoma numbers, the following empirical equation can be used :

$$\sigma = 1{,}5241\ x\ n_{QE}^{1{,}46} + \frac{V^2}{2\ x\ g\ x\ H_n} \tag{6}$$

Placement of the turbine to avoid cavitation, from the calculation results obtained thoma number of 0.92 with head suction of 8.2 m. The value of the head suction is positive knowned, position of the turbine can be placed at above the water surface with a maximum limit of 8.2 m from the water surface to the turbine center.

Total head loss (overall energy losses) is the value of major head loss and minor head loss (Fox, R. w., McDonald, A. T., 1998)head loss major in pipe is where the position of turbine is located and minor head loss is in accessories which is used in channel of delivery that is entrance between outside head tank with nozzle (gradual contraction), nozzle and diffuser (gradual enlargement) or called draft tube. The inlet channel on the inlet side to be used has a diameter of 0.7 m with a length of 0.3 m, using pipe diameter of 0.35 m with a length of 0.25 m and a diffuser with 0.7 m diameter outlet side with a length of 0.3 m, water velocity 2.42 m s. The flow of water in the pipeline undergoes turbulent flow, based on the reynold number of 947427.3, thus the major head loss experienced by the pipe is 0.0007 m, while the minor head loss is 0.36 m, the total head loss at system amount 0.3607 m. The required gross head of the picohidro generating system is 1.86 m.

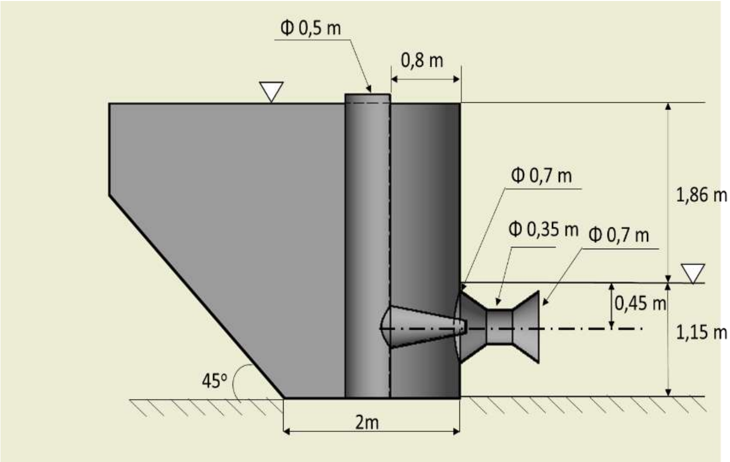

Fig. 1: Generating system design

Site surveys are conducted to record data and find to locate potential energy available according to the design criteria, data to be extracted from selected locations in the form of water debit, channel dimension and upper lower water level. The characteristics of irrigation channels has a constant flow rate and typical civil construction. One of the irrigation channel locations shown in figure 3, it is has the potential power generated amount 5022.7 Watt and average debit of 0.256 m^3/s. The generating system to be developed in this plan is based on the results of obeservation and the initial calculation is given as shown in figure 2.

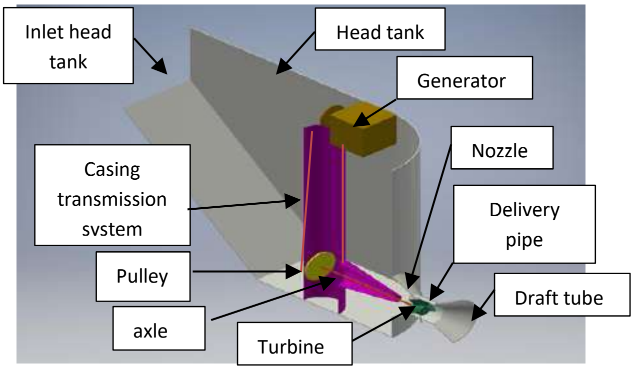

Fig. 2: Generator system design

Fig. 3: Location photos

3. CFD Simulation

This simulation is intended to verify the calculation result and to recognize the phenomenon of water flow pattern occuring in the picohidro generating system without involving the turbine blades, the simulation used with dynamic fluid flow computation method using software from ansys. the simulation process must go through three steps: preprocessing, solving and post processing (Tuakia.F, 2008). Preprocessing is the step in modeling or geometry in 3D as shown in figure 4.

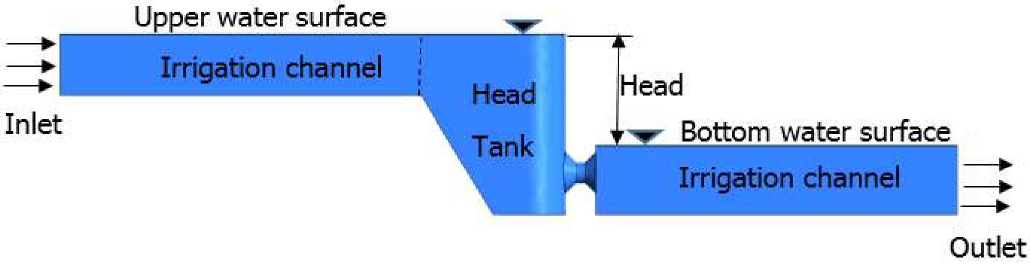

Fig. 4: Head tank model

The model is made based on the head tank geometry design created previously, the next step is making meshing, the field or volume of the model filled with the fluid divided into small volume cells, meshing process is made because in the simulation process uses calculation based on finite volume. In the meshing process applies the naming of boundary conditions on the model as shown in figure 4. The next step of the process of solving is the simulation process by entering the boundary conditions required during the calculation, conditions applied to the fluid flow model in laminar state with steady and gravity conditions. The boundary condition at the inlet side is rated of 220 kg/s on the type of mass flow with the type of water fluid, after all the data needed to perform the simulation process has been fed, the next step is calculating.

4. Simulation Results

After the calculations performed by the fluent software, the next step is postprocessing. It is the result of the simulation performed by fluent in the desired area and interpreting both in the form of vector images and contours with a variation pattern of certain color.

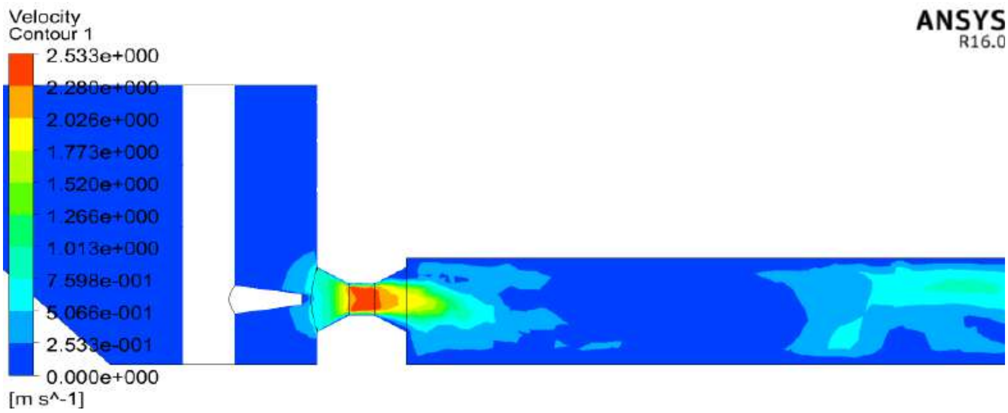

Fig. 5: Velocity Contour

In Figure 6 the high fluid velocity occurs in the area where the turbine will be placed that is in the pipe section, performed in red colour, the average velocity in the area is 2.28 m/s with a mass flow of 216.86 kg/s, by knowing data of speed and mass from simulation result hence the power potential using equation 1 from picohidro generating system is equal to 3190.2 Watt.

From the simulation results can know the pattern of water flow that occurs in the picohidro generating system by looking at the velocity velocity shown clearly in Figure 6, the figure can be interpreted in certain areas.

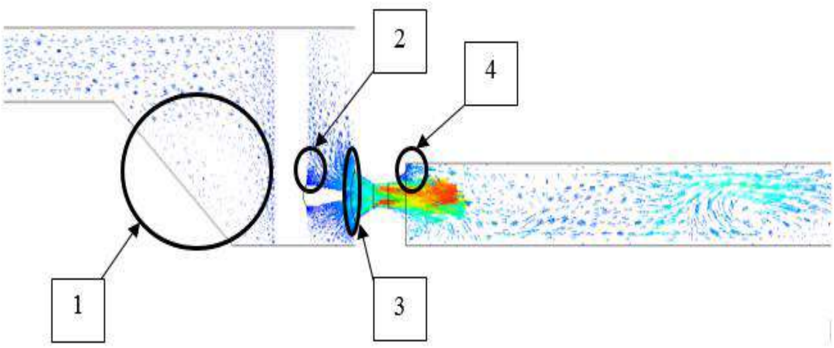

Fig. 6: Vector velocity on the generating system

In figure 7 the water flow is swirling in the sloping surface area close to the inlet from the irrigation canal to the head tank. It is due to the flow of water coming into the head tank can not go directly to the inlet to the turbine conductor as a result the flow of water coming from the inlet head tank stuck by the previous water and reverses direction causing a rotating flow in the area.

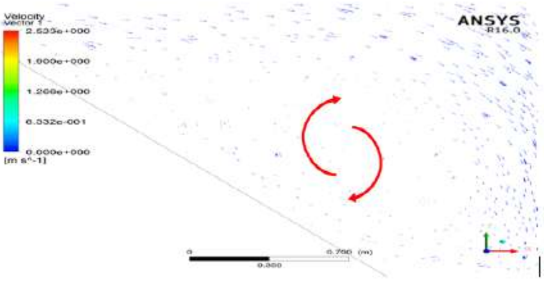

Fig. 7: Detail 1 vector velocity on generating system

In Figure 8 shows a swirling in an area having this angle caused by the moving water can not follow the angled path so the water hits the wall and the water flow split into two cousing swirling in the corner area, This swirling becomes a resistance to the moving flow.

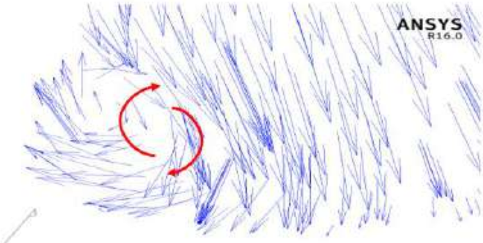

Fig. 8: Detail 2 vector velocity on generating system

In figure 9 shows at the inlet area of the nozzle, the water flow moves at the same time to enter the nozzle. It can be seen clearly from the intensity of vector in the area, caused by the water flowing from all directions try to enter the nozzle.

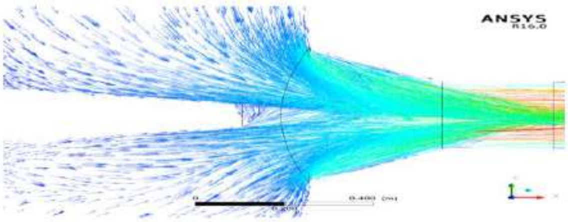

Fig. 9: Detail 3 vector velocity on generating system

In figure 10 shows a backflow in the draft tube region especially near the wall area cause the draft tube angle is too large this is due to the flow of water coming out of the draft tube is not fully developed resulting in the suction between the empty space with the flow of water on the outer side and this may cause resistance to the flow of water coming out of the turbine.

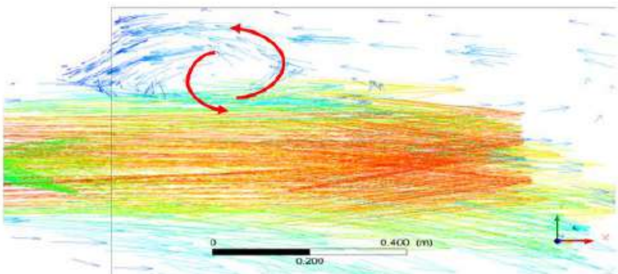

Fig. 10: Detail 4 vector velocity on generating system

Figure 11 shows the flow of water that swirling on the water surface due to the flow of water that is fast enough to hit the head tank wall and the flow back. It is crashed into a new stream from inlet the head tank caused to from swirling pattern.

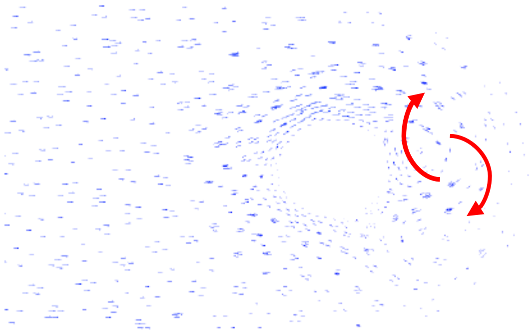

Figure 11: vector velocity on surface water generating system

5. Conclusions

The conclusions can be drawn from the calculation, simulation and analysis that has been done then can be conclude as follows

- Design and simulation results have identical tendency
- Backflow occurs at several positions within the generating system affecting the uniformity of the flow patterns in the system and reduces the potential power flow of the fluid that can be utilized
- Potential power flow simulation results for Pico hydro generating system is 3190.2 Watt with fluid velocity passing the turbine runner position of 2.28 m/s
- Centerline positioning of the turbine can be placed at above a the water level without experiencing cavitation phenomena as the turbine suction head is positive 8.2 m

The continuation and refinement of this research need some system geometry improvements with larger radius to maintain flow uniformity and minimize loss of fluid flow power.

6. Acknowledgment

This research is part of PUPT research funded by Direktorat Riset dan Pengabdian Masyarakat Kementrian Riset, Teknologi dan Pendidikan Tinggi Tahun Anggaran 2017.

7. Nomenclature

D	Diameter	m
g	Gravity	m/s^2
H_n	Head net	m
H_s	Head suction	m
N	Rotating velocity	rpm
N_{QE}	Specific velocity	
P	Power	Watt
P_{atm}	Atmospheric pressure	Pa
P_v	Water vapor pressure	Pa
Q	Water discharge	m^3/s
V	Water flow velocity	m/s

Greek letters

ρ	water density specific	kg/m^3
σ	Thoma number	
η	Efficiency	

8. References

C.C.Warnick, Howard A.Mayo, J., Carson, J. L., & H, L. (1984). Hydro Power Engineering. New Jersey: Prentice-Hall,Inc.

EM20, E. M. (1976). Selecting Hydraulic Reaction Turbine. Washington: U.S.Goverment Printing Office.

Data Rasio Elektrifikasi Jawa Barat, available at: www.esdm.jabarprov.go.id, accessed on june, 2016.

European Small Hydropower Association, Celso Pence. (2004). The Guide on How to Develop a Small Hydropower Plant. European: European Renewable Energy Council.

F.Dietzel, Sriyono, D. (1990). Turbin Pompa dan Kompresor. Jakarta: Erlangga.

Fox, R. w., McDonald, A. T. (1998). Introduction to Fluid Mechanics. Singapore: John Wiley & sons.

Indian Institute of Technology Roorkee. (1990). Standard/Manuals/Guidelines fo Selection of Turbine and Governing System for SHP. India: Ministry of New and Renewable energy Govt. of India.

JICA. (2009). Manuals and Guidelines for Micro-Hydropower Development in Rural Electrification. Japan: Energy Utilization Management Bureau.

Mott, R. L. (1994). Apllied Fluid Mechanics. United State of America: Prentice Hall.

Rencana Usaha Pengadaan Tenaga Listrik (RUPTL). available at: www.listrik.org, accessed on June, 2016.

Priyono Sutikno, I. K. (2011). Design, Simulation and Experimental of The Very Low Head Turbine with Minimum Pressure and Free Vortex Criterations. International Jurnal of Mechanical & Mechatronics Engineering IJMEE-IJENS, 8.

Sayers, A. T. (1992). Hydraulic and Compressible Flow Turbomachines. Singapore: MCGraw-Hill Book.

Susatyo, A., Subekti, R. A. (2016). Rancang Bangun Turbin Arus Sungai/ Head Sangat Rendah. LIPI, 8.

Tuakia, F. (2008). Dasar-dasar CFD Menggunakan Fluent. Bandung: Informatika.