

Numerical Analysis of Turbine Prototype Cross-Flow and Kaplan Flow Speeds

Purwantono, Ahmad Halim Sidiq, Irzal, Hasanuddin, Refdinal and Arwizet K

Jurusan Teknik Mesin, Fakultas Teknik, Universitas Negeri Padang, Indonesia

Article Info

Article history:

Received June 08th, 2018

Revised Aug 09th, 2018

Accepted November 17th, 2018

Keywords:

Turbine

Cross Flow Turbine

Kaplan Turbine

Flow Speed

Numerical Analysis

ABSTRACT

Based on previous research conducted by Purwantono about the utilization of exhaust flow from conventional turbine flow prototypes which were used as an inlet in turbine Kaplan [1]. This research was conducted to see how the exhaust flow velocity of each turbine before and after was combined into one combination turbine. This numerical based study uses the Ansys 18.0 application by inputting a 3D design from a conventional turbine prototype which was used as the material for this study. The results obtained in this study show that the average of outlet velocity in the turbine that uses a velocity outlet from a cross-flow turbine is 0.3 m / s greater when it is combined, which is 8.33 m / s and after being combined to 0.38 m / s. The results of this study are expected to contribute to the development of conventional turbines later.

Corresponding Author:

Ahmad Halim Sidiq,

Jurusan Teknik Mesin, Fakultas Teknik, Universitas Negeri Padang

Jln. Prof. Dr. Hamka Air Tawar, Padang (25131), Sumatera Barat, Indonesia

Email: ahmadhalimsidiq@gmail.com

1. INTRODUCTION

Indonesia electricity supplies sourced from diesel power plants (PLTD) and hydropower plants (PLTA) tend to be increasingly limited. Electrical energy sourced from PLTD is very dependent on fuel, while hydropower also depends on the availability of abundant water to drive the generator, and in the dry season, a decrease in water discharge is very influential in the production of electricity produced, even stalled [2]. Dependence on fossil fuels is a serious problem faced, which can deplete oil reserves, and result in price increases/instability caused by greater demand from production [3]. The Indonesian government is fully trying to overcome the problem of dependence on fossil energy sources by utilizing new and renewable energy sources (EBT) which are considered more environmentally friendly [4].

The natural condition of Indonesia, which is surrounded by mountains, makes PT PLN unable to reach all regions in Indonesia to meet the need for electrical energy [5]. These problems can be overcome by the use of new and renewable energy, one of which is by building a micro hydropower plant (PLTMH) [6]. The factors that must be considered in the utilization of this water energy are in the form of the amount of water availability, the height difference that can be utilized and the distance of location to the residential area [7]. Turbine designs that are varied with the height of waterfall and the amount of water availability are interesting problems to be considered as research objects in order to find the right system, shape and size in an effort to obtain maximum turbine efficiency.

Determination of the type of turbine used in the design must pay attention to the two most influential design parameters, namely the amount of water discharge and the height of falling water [1]. The selection of cross-flow turbines that can be used at high falls of 3-200m and Kaplan at 3-50m feels suitable to be the object of research [8]. The characteristics of high fall and water discharge in the Indonesian mountains are categorized at low and medium height and relatively large and medium discharge [1].

Turbin Pelton	$12 < n_s < 25$
Turbin Francis	$60 < n_s < 300$
Turbin Cross Flow	$40 < n_s < 200$
Turbin Kaplan	$250 < n_s < 1000$

Fig.1. Specific Speed of Water Turbines

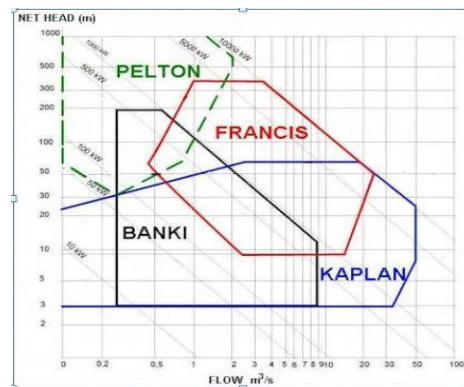


Fig. 2. Comparison of Characteristics of Water Turbines [9]

By considering these factors, the author will try to simulate the flow of a turbine prototype that is expected to provide input in the development of a type of combination turbine to get better performance.

2. MATERIAL DAN METHOD

The author uses the numerical method in this study, by entering the initial parameters and designs previously carried out by Purwantonono (Figures 3 and 4).



Figure 3. A prototype of the Cross-Flow Turbine and Kaplan

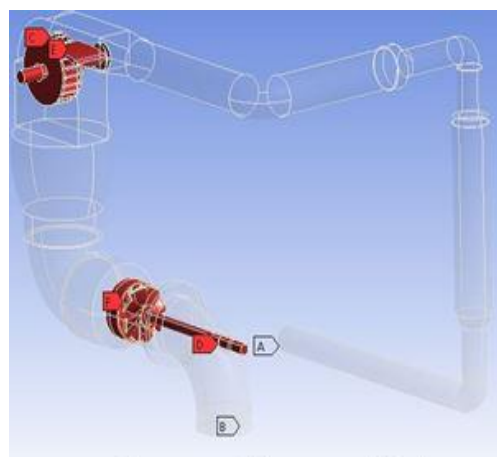


Figure 4. Design of Cross Flow Turbine and Kaplan 3D Prototypes

In this study, the authors used the Ansys18.0 application which is an application based on CFD (Computational Fluid Dynamics) for fluid simulation and analysis [10]. After previously carried out direct testing of the resulting prototype, some data was entered as general data which will be input into the application, including the maximum flowrate that can be produced by a prototype of 55 LPM (1.81m / s). Simulation is done using an application to obtain flow velocity values that can be generated during the simulation process using the software.

3. RESULT AND DISCUSS

The use of software in this analysis makes it possible to find out the flow velocity that can be generated in one system as a whole, and how when a single system is separated into parts that use data from the results of the previous part. The results of this numerical analysis can be illustrated by the assumption that the conditions that occur in the software are actual conditions. Flow velocity is a very important factor in this analysis, by inputting the initial speed on the cross flow turbine inlet and the output on the outlet is used as the turbine inlet data inlet.

3.1 ANALYSIS OF NUMERICAL TURBIN CROSSFLOW

The results of the analysis as in Figure 5 are in the form of fluid flow velocities which are described as symmetrical (streamlined) lines with different colour concentrations which describe the velocity of fluid flow in units of m / s. The highest fluid velocity flow is at the position indicated by arrow A with a value of 3.573 m / s, from figure 5 it can be seen that the change in speed is caused by changes in diameter from small diameters to large diameters. turbine outlet. Figure 6 shows the average flow velocity in the outlet section using the calculator function of 0.306 m / s.

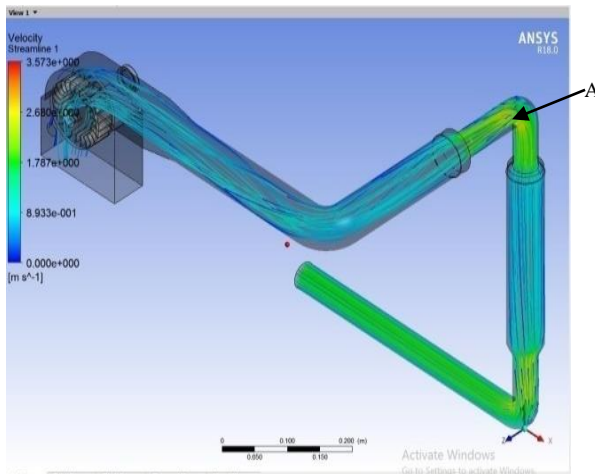


Fig. 5. Velocity Streamline Turbine Cross Flow

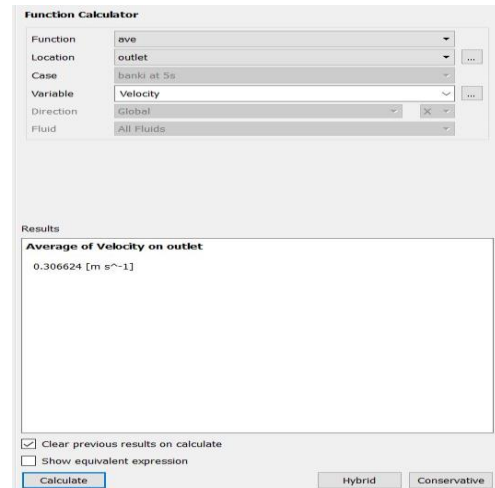


Fig. 6. Average of Velocity on Outlet Turbine Cross Flow

3.2 RESULTS OF ANALYSIS OF NUMERICAL TURBIN KAPLAN

The inlet velocity that is input to the Kaplan turbine is the result of the output in Figure 6 which is then calculated only on the turbine Kaplan to see the greatest speed that can be produced by the Kaplan turbine before being combined. Point B shows the largest flow velocity value, which reaches 12.41 m / s until the average outlet velocity obtained is 8.33 m / s.

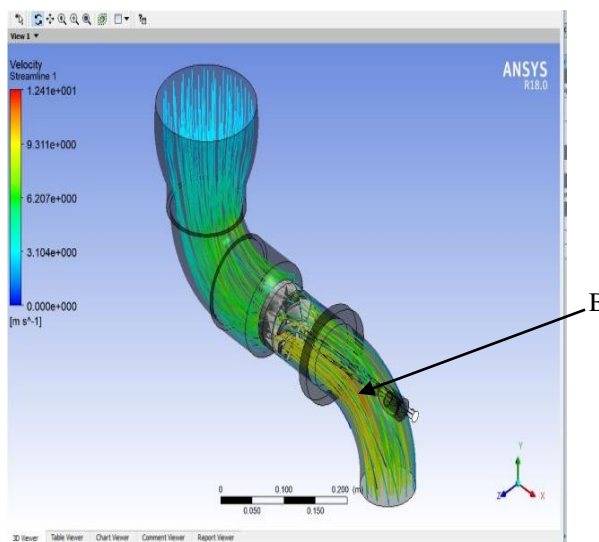


Fig. 7. Kaplan's Velocity Streamline Turbine

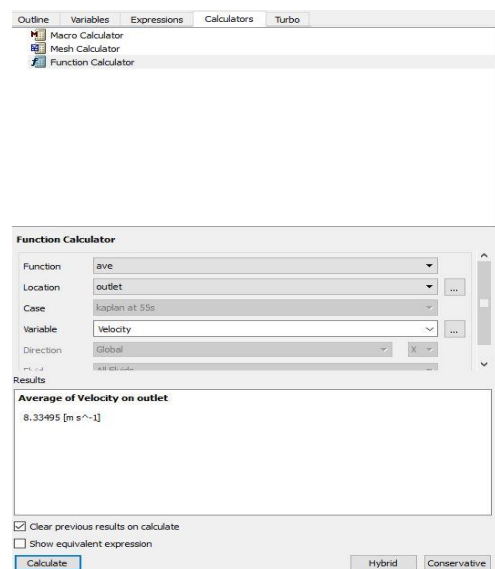


Fig. 8. Average of Velocity on the Kaplan Turbine Outlet

3.3 RESULTS ANALYSIS OF NUMERICAL COMBINATION TURBINES (CROSS FLOW AND KAPLAN)

Streamlined depiction of numerical turbine combination analysis can be seen in Figure 9 with the same inlet speed as in Figure 5, in the Kaplan turbine inlet section combined with the outlet section of the cross-flow turbine. This is done, in order to know the value of the overall outlet speed in the system, before and after the combined. The results obtained show a significant value of the outlet velocity in the turbine Kaplan. The average speed produced by this combination turbine on the outlet section is 0.38 m / s from the previous value of the average Kaplan turbine outlet is 8.33 m / s.

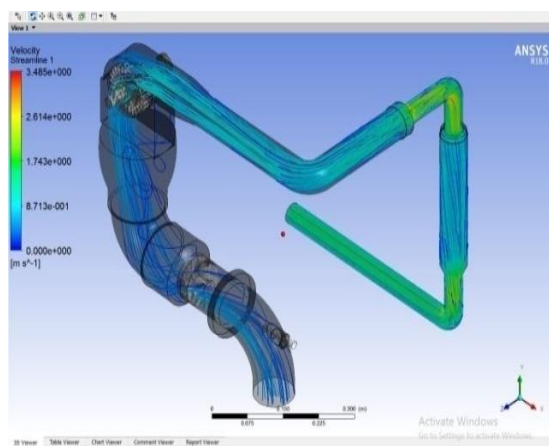


Fig. 9. Velocity Streamline Turbine Combination

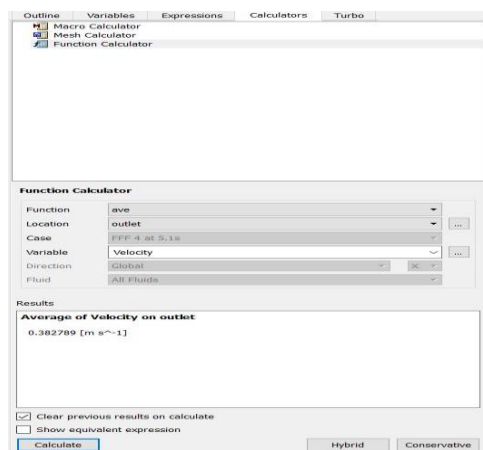


Fig. 10. Average of Velocity in Outlet Combination Turbines

4. CONCLUSION

This research is a continuation of the results of previous studies conducted by Purwantono to obtain flow velocity when the two turbines are combined and before being combined. The parameters of the results of the prior analysis used in this study are the inlet flow velocity in the inlet section. From the results of the research that the author has done, it can be concluded that the results of the analysis using Ansys 18 software, the flow velocity starting from the cross-flow turbine has a significant decrease, it is caused by different changes in cross section diameter, where the diameter of the inlet pipe is smaller than the pipe diameter positioned before the runner, this results in loss of speed on the system.

The Kaplan turbine that utilizes the flow velocity from the cross-flow turbine also changes when the turbine is combined and before merging, from 8.33 m / s to 0.38 m / s. This is due to the simulation using only the Kaplan turbine, the inlet pressure on the Kaplan turbine is in the entire inlet cross section (Figure 7), while in the combination turbine, from the analysis results it can be noted that the turbine inlet section only occurs in a small part from the turbine Kaplan inlet (figure 9). This research shows the results that can be used to refine the design that has been done in the previous research, in order to get the maximum flow velocity results, and is expected to get turbine rotation to the higher efficiency of course.

REFERENCES

- [1] Purwantono, Syahrul, and J. Adri, "Pengaruh Perubahan Debit Aliran Terhadap Putaran Turbin Banki dan Kaplan," *INVOTEK J. Inov. Vokasional dan Teknol.*, vol. 18, no. 1, pp. 13–18, 2018.
- [2] W. Friska, "No Title," 23 January, 2013. [Online]. Available: <https://www.scribd.com/doc/121931390/MAKALAH-ENERGI-docx>. [Accessed: 17-Jan-2018].
- [3] Arwizet, "Mesin destilasi pengolahan sampah plastik menjadi bahan bakar minyak menggunakan kondensor bertingkat dan pendingin kompresi uap," *INVOTEK J. Inov. Vokasional dan Teknol.*, vol. 17, no. 2, 2017.
- [4] H. Nurdin, Y. Fernanda, and M. Handayani, "Analysis of Tensile Strength the Fiber Bagasse Particles Board with Resin Adhesives," *J. Teknomekanik*, vol. 1, no. 1, pp. 1–5, 2018.
- [5] M. N. SyaFe'i, "Rancang Bangun Simulasi Turbin Air Cross Flow," *J. Pendidik. Tek. Mesinn*, 2016.
- [6] Adly Havendri and H. Lius, "Perancangan Dan Realisasi Model Prototipe Turbin Air Type Screw (Archimedean Turbine) Untuk Pembangkit Listrik Tenaga Mikrohidro Dengan Head Rendah Di Indonesia," *J. Tek. Mesin*, vol. 1–7, no. 31, p. 2, 2009.

- [7] P. T. D. Rompas, "Analisis Pembangkit Listrik Tenaga Mikrohidro (Pltmh) Pada Daerah Aliran Sungai Ongkak Mongondow Di Desa Muntoi Kabupaten Bolaang Mongondow," *J. Penelit. Saintek*, vol. 16, no. 2, pp. 160–171, 2011.
- [8] C. Penche, *Layman's Guidebook "on how develop a small hydro site."* European Small Hydropower Association (ESHA), 1998.
- [9] A. M. Harahap, "Analisis Numerical Aliran Fluida pada Rumah Water Vortex Turbine dengan Water Depth 200 mm dan Inlet Velocity 0.20 m/s Menggunakan Aplikasi Ansys Fluent Versi 16.2," Universitas Negeri Padang, 2017.
- [10] F. Tuakia, *Dasar-dasar CFD Menggunakan FLUENT*. Bandung: Informatika, 2008.