EFFECT ANALYSIS OF ROTATIONAL SPEED CHANGES ON PROPELLER TURBINES ON THE POTENTIAL FOR CAVITATION FORMATION USING COMPUTATIONAL FLUID DYNAMIC METHOD (CFD)

Aditya Putra Widodo[1], Deni Shidqi Khaerudini[1,2]

[1]Department of Mechanical Engineering, Faculty of Engineering, Mercu Buana University, Indonesia
[2]Research Center of Physic, BRIN, Indonesia

E-mail: adityaputrawidodo@yahoo.co.id

Abstract—Water is an abundant resource, includes rivers that contributing significantly to Indonesia's Central Java Province water energy potential for micro hydro power plants (PLTMH). Utilizing water energy for electricity generation supports emission reduction initiatives as it produces no CO_2 emissions during turbine-driven electricity production. One of the problems experienced by turbines is damage caused by cavitation due to very low pressure. This research was conducted to determine the effect of rotational speed on the potential for cavitation. This research was conducted using the CFD method and using the ANSYS software program. In the simulation process, several variables are used, namely rotational speed (236, 367, 479, 527, and 627 rpm) and fluid velocity (3,5, 3,9, 4,3, 4,6 and 5,0 m/s). The conclusion of this study is the turbine at 627 rpm shows greater cavitation potential with the lowest pressure of -9,422 Pa, this is in accordance with Bernoulli's Principle. Suggestions for further research can be done with several modifications or variations in the winglet radius are necessary to achieve a better design and modifying the blade inclination angle could also reduce the formation of gas bubbles on the propeller blades due to pressure drops.

Keywords: ANSYS, CFD, cavitation, turbine propeller, winglet.

1. INTRODUCTIONS

Water is an inexhaustible resource, one of which is rivers [1]. In Indonesia, Central Java Province itself has water energy potential that can be utilized for micro hydro power plants (PLTMH) such as those spread across the areas of Brebes, Banjarnegara, Kendal, Banyumas, Pemalang, Temanggung, Pekalongan, Kebumen and Wonosobo [2]. Water energy can be utilized to generate electricity and support emission reduction programs, as it does not produce CO_2 emissions during the electricity production process using water turbines.

One of the problems that turbines experience is damage caused by cavitation, also known as cold boiling [3]. Cavitation is a problem characterized by the appearance of bubbles due to high speed or pressure drop [4] . This issue presents a challenge for hydropower plants, as it can cause vibrations, decreased performance, and damage to turbine components [5]. Cavitation depends on the alignment of the turbine blades relative to the head, rotation speed, and blade profile shape [6].

To analyze the potential for cavitation in a stream, it is simplest to measure the amount of water vapor bubbles in the flow. The bubble content of water vapor is a critical parameter for predicting cavitation in the turbine. The quantity of water vapor bubbles ranges between 0 and 1, with a value of 1 indicating 100% steam bubbles [7].

To determine the water vapor bubble content, the Computational Fluid Dynamics (CFD) method can be employed, using applications such as ANSYS. CFD has been widely adopted because it reduces costs and time in analyzing physical phenomena occurring around and within test objects [8].

CFD can also provide performance predictions for fluid systems and visualizations of fluid flow patterns through the system that experimental methods cannot detect. Therefore, this study aims to analyze the effect of changes in rotational speed on propeller turbine for cavitation formation potential using CFD method by integrating two materials namely water and water vapor into a unified entity.

2. RESEARCH METHOD

The implementation methodology is a systematic procedure that outlines clear and sequential stages of the research process. Each stage or part, which determines the subsequent stage, must be executed carefully. This section will discuss the research sequence and the tools used for data collection.

The flowchart outlines the research implementation methodology. The research sequence starts with studying the literature from previous research, followed by designing turbine propeller images and setting boundary conditions. The simulation procedures include stages such as geometry creation, meshing, setup, solution, and results. After conducting the simulation, the data is analyzed, and conclusions are drawn. The research sequence is illustrated in Figure 1:

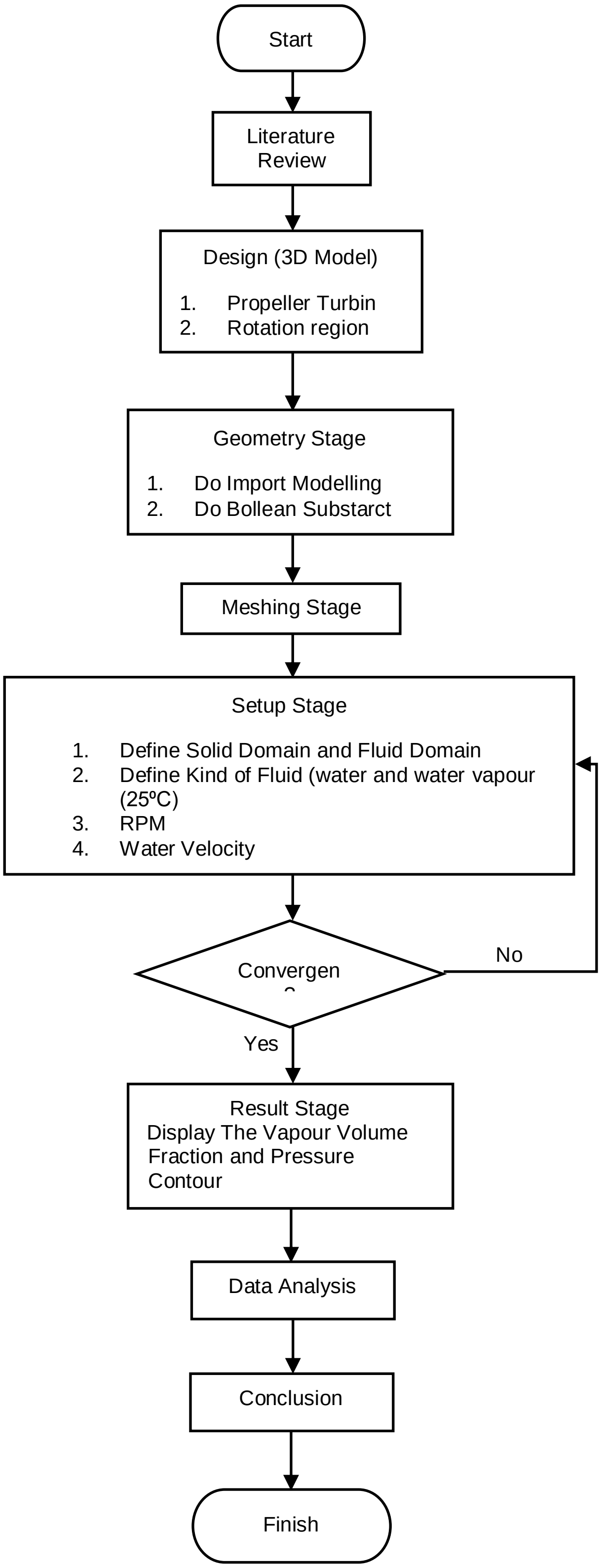

Figure 2. Cavitation Analysis Flow Chart

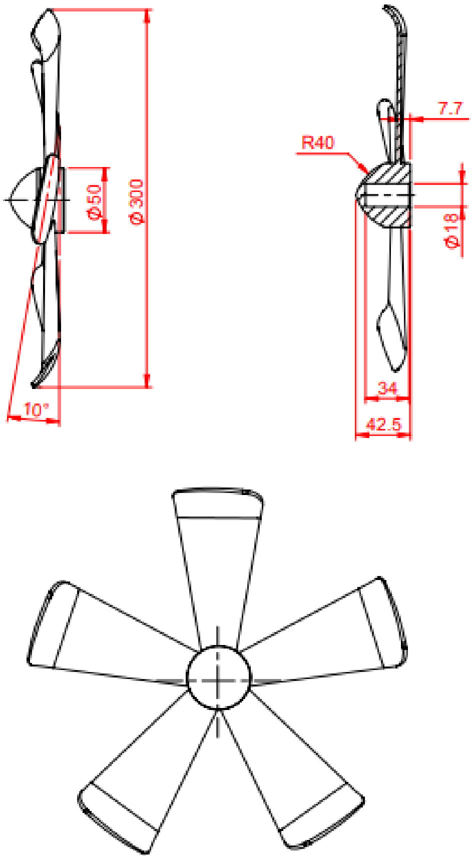

Figure 1. *Turbine propeller*

In the simulation process, the fluid and solid domains are defined. At the geometry stage, a boolean subtract is carried out to eliminate unnecessary areas, ensuring the simulation models only the water-filled regions. In the boolean subtract process, the pipe area is designated as the target body, while the rotation region, spindle area, and turbine shaft are set as the tool bodies. At the mesh stage, the definition of the dominant hex is applied to the meshed parts. Using a hex-dominant approach ensures a neater meshing result. The next stage is the setup. During the setup stage, the turbulence model used is Shear Stress Transport (SST), which is chosen based on the number of nodes and elements formed in the meshing section. SST is also capable of defining fluid flow very close to the boundary. In the fluid domain, the inlet and outlet are defined, while in the solid domain, the propeller (wall) is defined. The boundary conditions used in this simulation include the speed of the water on the entry side and the static pressure on the exit.

Cavitation analysis is performed with variations in rpm and water velocity to determine the potential for cavitation. For more details, see Table 1:

Table 1. Simulation parameter for setup stage

Case	Inlet Velocity (m/s)	Outlet Pressure (0 Pa)	Rotational Velocity (rpm)
5 blades	3.5		236
	3.9		367
	4.3		479
	4.6		527
	5.0		627

The cavitation simulation process uses a homogeneous model with two materials: water and water vapor at 25°C. To show the potential for cavitation. the Rayleigh-Plesset model is used with a saturation pressure of 3170 Pa. This analysis is conducted using steady-state analysis.

2.1 Turbin Propeller

In this study. a 5-blade propeller turbine is used. For more details. see Figure 2:

3. RESEARCH METHOD

In this chapter. we will explain the simulation results in the form of vapor volume fraction contours.

3.1 Contour Vapour Volume Fraction Turbin and Contour Pressure

After the setup and solution processes. the simulation results in the solution stage can be observed in the results stage to assess the potential for cavitation. Contours indicating the potential for cavitation in the turbine based on RPM and water velocity can be seen in Figures 3 to 7:

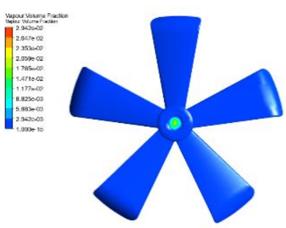

Figure 3. Contour vapour volume fraction turbine propeller 236 rpm

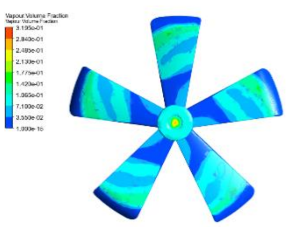

Figure 4. Contour vapour volume fraction turbine propeller 367 rpm

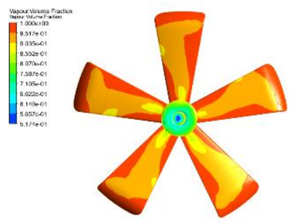

Figure 5. Contour vapour volume fraction turbine propeller 479 rpm

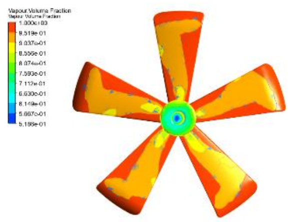

Figure 6. Contour vapour volume fraction turbine propeller 527 rpm

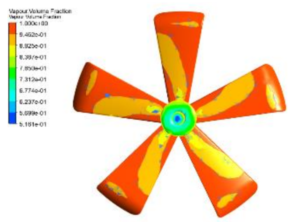

Figure 7. Contour vapour volume fraction turbine propeller 627 rpm

Figures 3 to 7 depict the vapor volume fraction contours on the propeller. The blue color indicates low vapor volume fraction values. while the yellowish-red color indicates high vapor volume fraction values. Higher vapor volume fraction values correspond to a greater potential for cavitation. Figures 3 and 4 clearly show noticeable differences or increases in their vapor volume fractions. Meanwhile. Figures 5. 6. and 7 appear almost similar when viewed visually. but the differences in their vapor volume fraction values are quite distinct. as shown in Table 2.

Table 2. Vapour volume values of turbine

rpm	vapour volume fraction (min)	vapour volume fraction (max)
236	1.000e-15	2.942e-02
367	1.000e-15	3.195e-01
479	5.174e-01	1.000e+00
527	5.186e-01	1.000e+00
627	5.161e-01	1.000e+00

When looking at Figures 3 to 7. the turbine operating at 627 rpm shows a higher potential for cavitation because the areas with high vapor volume fraction values are more extensive. This occurs due to the decrease in fluid pressure caused by the increased water velocity and turbine rotation. A comparison of turbine rotation speed and pressure can be seen in Figures 8 to 12.

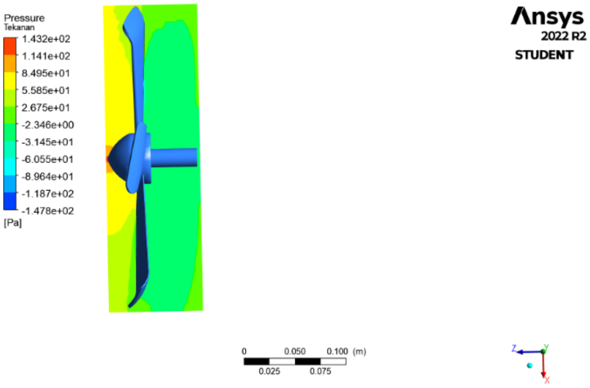

Figure 8. Contour pressure turbine propeller 236 rpm

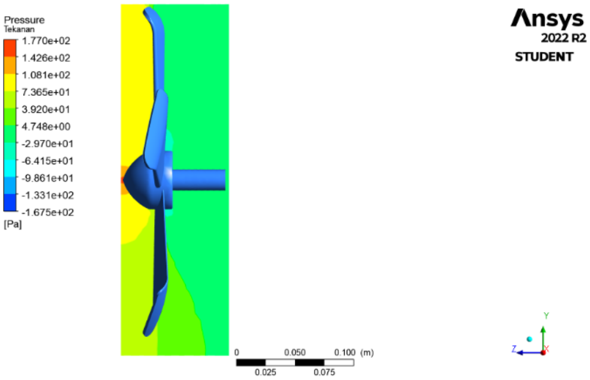

Figure 9. Contour pressure turbine propeller 367 rpm

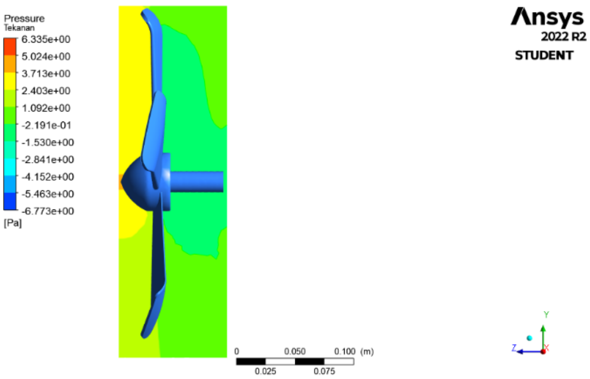

Figure 10. Contour pressure turbine propeller 479 rpm

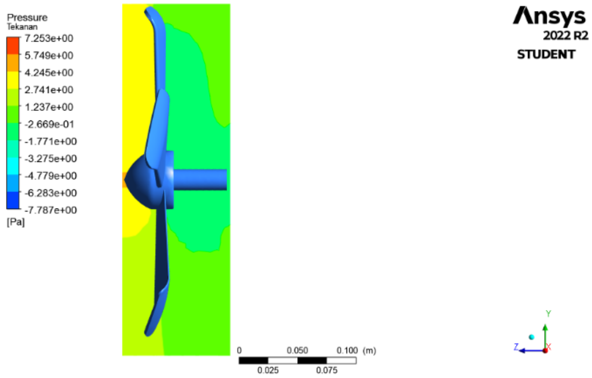

Figure 11. Contour pressure turbine propeller 527 rpm

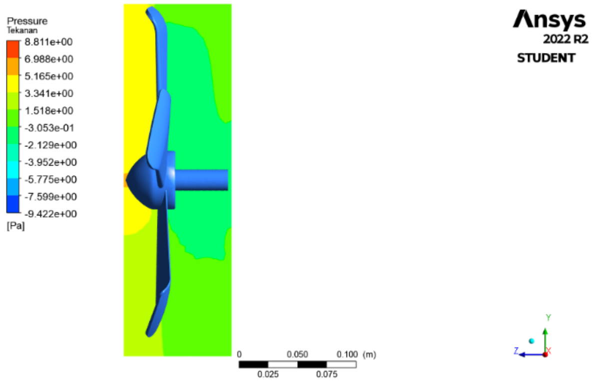

Figure 12. Contour pressure turbine propeller 627 rpm

Similar to the vapor volume fraction contours. blue color indicates low pressure and red color indicates high pressure. The turbine operating at 627 rpm has the lowest pressure. as can be seen more clearly in Table 3.

Table 3. Pressure (Pa) values vs rpm of turbine

rpm	Pressure Min (Pa)
236	-1.478
367	-1.675
479	-6.773
527	-7.787
627	-9.422

3.2 Validation Research

The validation of this study aims to compare its findings with previous research. thereby reinforcing or supporting them through relevant scholarly references. Previous research conducted by [9] mentioned that among the five valve opening variations and different pressure settings. varying cavitation results were obtained accordingly. Higher pressure drops corresponded to greater cavitation. aligning closely with the findings of the study by [9].

4. CONCLUSIONS

From the series of studies conducted using the CFD method with varying turbine speeds of 236. 367. 479. 527. and 627 rpm. it can be concluded that the propeller turbine speed of 627 rpm has a greater potential for cavitation due to having the lowest pressure of -9.422 Pa. This aligns with Bernoulli's principle. In this research. the propeller turbine used is a horizontal shaft propeller turbine with winglets added to the blade tips. Therefore. several modifications or variations in the winglet radius are necessary to achieve a better design. Additionally. in future research. modifying the blade inclination angle could also reduce the formation of gas bubbles on the propeller blades due to pressure drops.

REFERENCES

[1]. Hamdani. D.. (2023). Listrik Hijau dari Parit. Diambil dari website: https://pengabdian.lppm.itb.ac.id/terap/listrik_hijau_dari_parit.

[2]. Ferial.. (2015). Energi Air Terangi Jawa Tengah. Diambil dari website: https://ebtke.esdm.go.id/post/2015/03/02/793/energi.air.terangi.jawa.tengah.

[3]. Ramelan. S.. (2018). Analisis Pengaruh Perubahan Sudut Sudu Turbin Hydrocoil Terhadap Potensi Terjadinya Kavitasi Menggunakan Metode CFD (Computational Fluid Dynamic). S1 thesis. Universitas Mercu Buana Jakarta.

[4]. Effendy. M.. Wijianto. (2010). Aplikasi Response Getaran untuk Menganalisis Fenomena Kavitasi Pada Instalasi Pompa Sentrifugal. Jurnal Penelitian Sains & Teknologi. Vol. 11. No. 2. 2010: 191 – 206.

[5]. Celebioglu. K.. Altintas. B.. Aradag. S.. & Tascioglu. Y. (2017). Numerical research of cavitation on Francis turbine runners. International Journal of Hydrogen Energy. 42(28). 17771–17781. https://doi.org/https://doi.org/10.1016/j.ijhydene.2017.03.180

[6]. Himaran. Syukri. (2017). Turbin Air Teori & Dasar Perencanaan. Penerbit Andi. Jogjakarta.

[7]. Gohil. P. P. & Saini. R. P. (2016). Numerical Study of Cavitation in Francis Turbine of a Small Hydro Power Plant. Journal of Applied Fluid Mechanics. 9 (1). 357-365.

[8]. Luthfie. A. A. (2017). Analisis Pengaruh Perubahan Sudut Pipa Siphon Terhadap Performasi Turbin Hydrocoil Dengan Menggunakan Metode Computational Fluid Dynamic (Cfd). Jurnal Teknik Mesin Mercu Buana. 6(1). 41-47.

[9]. Aulia. A.. Nasrul. Z. A.. Sylvia. N.. Hakim. L.. & Bahri. S. (2022). Kajian Terhadap Kavitasi Dan Pressure Drop Pada Bukaan Control Valve Tipe Globe Valve Dengan Menggunakan Software Autodesk CFD (Computational Fluid Dynamics). *Chemical Engineering Journal Storage (CEJS)*. *1*(4). 57-66.

[10]. Cengel. Y. A.. &Cimbala. J. M. (2006). Fluid Mechanics: Fundamentals and Application. New York: McGraw-Hill.

[11]. Darmawan. D. B.. Chrismianto. D.. & Iqbal M. (2016). Analisa Kemiringan Sudut Propeller Tipe B-Series Pada Kapal Selam Tipe Menengah Untuk Mengoptimalkan Kinerja Kapal Selam Dengan Meotde CFD. Jurnal Teknik Perkapalan - Vol. 4. No. 2 April 2016. 352-361.

[12]. Febrianto. A.. Santoso. A. (2016). Analisa Perbandingan Torsi dan RPM Turbin Tipe Darrieus Terhadap Efisiensi Turbin. Jurnal Teknik ITS Vol. 5 No. 2. 517-521.

[13]. Lanzafame. R.. Mauro. S.. & Messina. M. (2013). Wind turbine CFD modeling using a correlation-based transitional model. Renewable Energy. 52. 31-39.

[14] Li. Y.. Paik. K. J.. Xing. T.. &Carrica. P. M. (2012). Dynamic overset CFD simulations of wind turbine aerodynamics. Renewable Energy. 37(1).285-298.

[15]. Wang. Y. F.. & Zhan. M. S. (2013). 3-Dimensional CFD simulation and analysis on performance of a micro-wind turbine resembling lotus in shape. Energy and Buildings. 65. 66-74.

[16]. Naveenagrawal. Stonecypher. Lamar. (2009). Cavitation in Hydraulic Turbines: Causes and Effects. Di ambil dari website: http://www.brighthubengineering.com/fluid-mechanics-hydraulics/27427-cavitation-in-hydraulic-turbines-causesand-effects/.