Gravitational Vortex Water Turbine (GVWT) Conical Basin Design: the Effects of Cone Angle and Outlet Diameter on Vortex Characteristics

Erna Septyaningrum, Ridho Hantoro, Naufal Khoodi Mouti,
Wiwit Ria Rahayu
Department of Engineering Physics,
Sepuluh Nopember Institute of Technology, Surabaya, INDONESIA

Sutardi[*]
Department of Mechanical Engineering,
Sepuluh Nopember Institute of Technology, Surabaya, INDONESIA
[*]*sutardi@me.its.ac.id*

ABSTRACT

A Gravitational Vortex Water Turbine (GVWT) seems to be a promising technology for low-head hydropower applications. The basin of GVWT plays an important role in the vortex formation as well as its performance. This study aims to analyze the effect of basin design on vortex characteristics in the GVWT. A conical basin is chosen due to the good quality of the vortex. The lab-scale experiment and Computational Fluid Dynamic (CFD) simulation was conducted to gain deeper insight into the effect of cone angle (θ) and basin diameter ratio (d/D) on the vortex characteristic, including air-core diameter, vortex height, and tangential velocity. Both parameters play a major role in regulating the water level and tangential velocity in the basin. The current work shows that the smaller the basin diameter ratio (d/D), the higher the vortex height, leading to a decrease in tangential velocity and vortex strength as the kinetic head is converted to a potential head. Moreover, basin parameters, including the vortex height and tangential velocity, fluctuate due to the variation of cone angle. According to this work, the conical basin with a large cone angle of 20° and a small d/D of 0.2 is suggested, since it produces a vortex with height tangential velocity and vortex height.

ISSN 1823-5514, eISSN 2550-164X

https://doi.org/10.24191/jmeche.v21i1.25366

Received for review: 2022-10-03
Accepted for publication: 2023-10-03
Published: 2024-01-15

Keywords: *Basin; Conical; Cone Angle; Diameter; Vortex*

Introduction

Along with the increase in energy demand, the government is expanding the share of renewable energy through various policy schemes, including those implemented by the Indonesian government [1]. The hydropower observation in Indonesia began in 1983, in collaboration with the Japan International Cooperation Agency (JICA), revealing that Indonesia has an enormous potential for Run of River (RoR) hydropower [2]. Based on the Indonesian Electric Energy Supply Business Plan, Indonesia's hydropower potential is approximately 94.3 GW, involving large-scale hydropower plants and micro-hydro facilities [3]. The utilization of low-head hydropower is still rare. Whereas most of the hydropower potential in rural areas is in the low-head category.

Several studies have proposed various low-head energy conversion techniques, ranging from 3 to 40 m of head, which can be selected based on energy resource capacity and topological considerations. These techniques include the Francis Turbine, Kaplan, Crossflow, Pump as Turbine, and Gravitational Vortex Turbine [3]-[6]. Among these techniques, the Gravitational Vortex Water Turbine (GVWT) seems to be a promising option due to its easy installation, operation, and maintenance. GVWT does not require the complex civil construction of a dam, making it a less expensive alternative. The development of GVWT can be traced back to 1930 when the prototype of the suction turbine and free-surface flow model were invented. In 2006, Zotloterer installed the first 10 kW GVWT on the Obergrafendorf Austria River [8].

The GVWT comprises an intake channel, basin, runner, blade, and supporting system, as illustrated in Figure 1. As GVWT harnesses energy from the vortex, it operates at low heads, i.e., 0.7 - 2.0 m [9]. The basin of GVWT plays a crucial role in vortex formation, as it is responsible for creating a high-quality vortex, with greater vortex height and tangential velocity. This is crucial for optimal power production, given that GVWTs are generally less efficient than other commercial turbines [7], [10]-[15]. Tangential velocity, which represents the kinetic head in the vortex, is the key consideration for effective runner design [16], as it has a major impact on hydrodynamic force formation [17].

Numerous studies have been conducted to improve basin design for GVWTs. In the early stage of GVWT, researchers utilized a square basin, which was still used by Zotloterer in 2004. A circular cross-section basin is more appealing due to its ability to generate induced tangential velocity, making it a popular choice for development [9]. Previous researchers have proposed various types of basins, including; (a) cylindrical basins, (b) conical basins [18], (c) stepped inlet [19], and (d) sloped inlet. The research conducted by Mulligan et al. [20] showed that the circulation number, which indicates vortex strength, is dependent on basin geometry. Basin design has a significant impact on the tangential velocity and vortex height [8]. The increase in tangential velocity enhances the runner's hydrodynamic force and performance.

Over 2012, many researchers have been inquiring about conical basins, due to their ability to produce stronger vortexes than cylindrical. Runners installed in conical basins exhibit higher efficiency than those installed in cylindrical basins [21]. Dhakal et al. [22] conducted a comparative analysis of the performance of cylindrical and conical basins. The study showed that conical basins produce vortexes with a higher average velocity at the same flow rate and head conditions. The decrease of the cross-section in the conical basin leads to an increase in velocity, following the continuity law. In this case, the vortex height is maintained in the constant value.

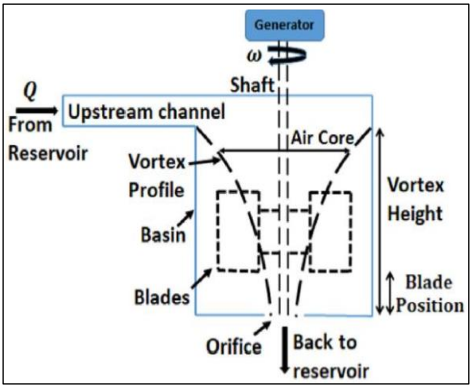


Figure 1: Schematic of GVWT [17]

The diameter ratio (*d/D*) and cone angle (for conical basin) are important design parameters that determine the vortex characteristic [22]. Mulligan et al. [20] conducted research in the cylindrical basin and found that the *d/D* ratio significantly impacts the formation of the vortex. According to this study, the cylindrical basin with 14% < *d/D* < 18% provides the maximum vortex strength. The research in the conical basin conducted by [23] showed that the optimum *d/D* is 0.167, which is influenced by the available head. The increase in the basin diameter (*D*) changes the air-core area, while the increase in the outlet diameter (*d*) leads to a larger depreciation area. The air-core diameter is a crucial factor as the dominance of air (represented by a larger air core) indicates low hydro energy and triggers some material issues for the runner, such as cavitation. Increasing the intake flow rate is another method for the reduction of air-core diameter while increasing the vortex height and tangential velocity [24].

The characteristics of the vortex in the conical basin are significantly influenced by the cone angle, which is the angle formed between the basin wall and the vertical plane. Research conducted by [25] varied the cone angle from 10^o to 18^o and it was found that the greatest tangential velocity is produced by a basin with a cone angle of 18^o, followed by the basin with a cone angle of 14^o. Similarly, research conducted by [22] found that the velocity in the conical basin is greater for a large cone angle, indicating an increase in kinetic head production. However, the velocity profile within the basin has yet to be fully described.

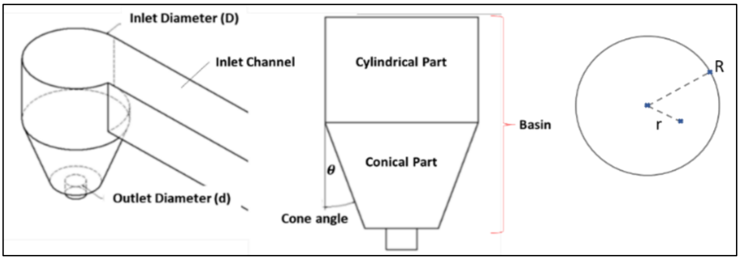
This study provides an analysis of the influence of diameter ratio (*d/D*) and cone angle on the vortex characteristics. This analysis is essential in determining the dimensions and design of the basin and provides an overview of the energy distribution within the basin. This information obtained from this analysis is used as a reference in determining the runner installation position and optimizing the energy extraction process, including for the installation of the GVWT multistage runner.

The novelty of this study lies in the detailed analysis of vortex characteristics in the conical basin, which has not been addressed in the previous research. The research results, especially the velocity profile and vortex height, are significant concerns in maximizing the energy extraction process.

The laboratory-scale experiment was carried out in conjunction with Computational Fluid Dynamic Simulation (CFD). As CFD is capable of accurately predicting the flow patterns, it provides data that cannot be obtained from the experiment. CFD has been widely employed in the prediction of renewable energy applications, such as wind turbines, hydro turbines, and the vortex flow [22], [26]–[30]. Validation and grid-independent study were carried out to ensure that data obtained from the simulation study was in accordance with the data obtained from the experiment.

Conical Basin Design and Parameters

The conical basin used in this study comprises an intake channel and the main basin, as illustrated in Figure 2. The upper part is a cylindrical shape, combined with the conical shape in the lower part. The basin has a diameter (*D*) of 0.35 m and a height (*H*) of 0.35 m. The outlet diameter and cone angle were varied as part of the experimental design.



(a) (b) (c)

Figure 2: Design of conical basin, (a) isometric view; (b) frontal view; and (c) top view (basin only)

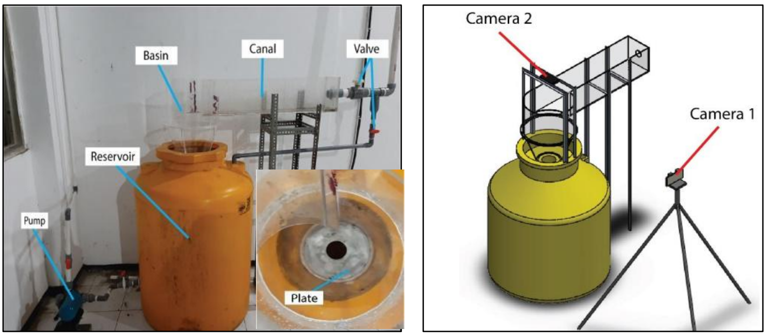
The diameter ratio (*d/D*) was varied, where *d* is the outlet basin diameter and *D* is the inlet basin diameter. The stepper diameter remained constant for the same cone angle. The cone angle, which is defined as the angle between the basin wall and the vertical plane as shown in Figure 2, was also varied. The effect of flow rate on the formation of the vortex was also investigated in this study.

Research Methods

Laboratories scale experiment setup

The laboratory-scale experiment was carried out at the Energy Engineering and Environmental Conditioning Laboratory (Laboratorium Rekayasa Energy dan Pengkondisian Lingkungan) of the Institut Teknologi Sepuluh Nopember. The experimental data was used as validation data for the experimental results, to ensure the conformity of the simulation results with the actual phenomenon. The GVWT experimental facilities are detailed in Figure 3. To model open channel flow under the influence of gravity, it was necessary to minimize the turbulence caused by the pump. To achieve this, the upper reservoir was installed at approximately 5 m above the basin. The upper water reservoir receives, and store water from the bottom reservoir (which is not visible in Figure 3(a)). The flow rate was controlled by a valve installed before the intake channel and the bypass valve was used to directly stream water to the lower reservoir, thus controlling the incoming flow. A pump was installed after the lower reservoir to recirculate water to the upper reservoir. The metal plate was used to modify the basin outlet diameter, while a support structure was necessary to support the intake channel and the piping system.

The research began with an experimental setup, which involved setting up the equipment as shown in Figure 3, and preparing the measuring instruments. Flowrate measurement was carried out before taking data on vortex height and air-core diameter. Flowrate measurement was conducted when the flow in the channel had reached a steady state. The measurement was repeated five times for each data collection. The flow rate measurement was conducted using a current meter. The calibration of the current meter was carried out to ensure its accuracy. This research varied three flow rate values, i.e., 0.00225, 0.00196, and 0.0009 m^3/s.



(a) (b)

Figure 3: Experimental set up; (a) basin set up, and (b) camera set up

The measurement of vortex height and air core diameter was conducted for each flow rate variation. Data collection was carried out when the vortex conditions were steady, indited by stable vortex height. The measurement is repeated five times for each data collection, both for vortex height and air-core diameter data. The measuring tape is attached to the basin wall to measure the vortex height. The additional camera is located beside the basin to capture the vortex structure, as given in Figure 3(b).

The standard deviation for the vortex height data is 0.435 cm for all flow rate variations. The standard deviation of vortex height measurement for each flow rate variation is 0.51, 0.38, and 0.41 cm, respectively. The uncertainty of the vortex height measurement is 0.05 cm. A camera is installed right above the center of the basin to record the vortex structure. The air-core diameter data was obtained from the observation of the recorded image/video. The standard deviation for the air-core diameter data is 1.23 cm for all flow rate variations, with the measurement uncertainty of 0.05 cm. The standard deviation of air core diameter for each flow rate variation is 1.3, 1.2, and 1.1 cm, respectively. The experimental studies documentation is given in Figure 4.

Simulation setup

ANSYS with CFX Solver was utilized to investigate the designed parameters. For this numerical simulation, a 3D computational domain was developed in the "Design Modeller" module. This domain consisted of a single stationary domain, representing the intake channel and basin, with dimensions similar to the experimental model.

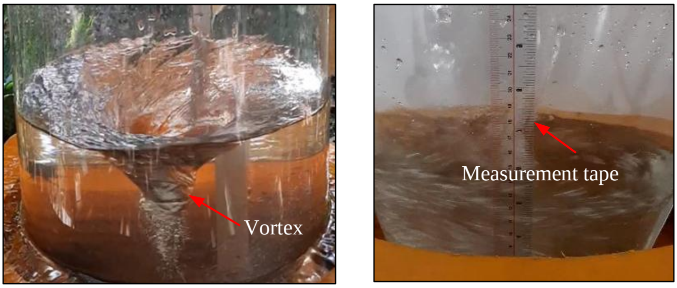


Figure 4: Experimental documentation

An unstructured tetrahedron mesh was generated for the simulation due to the complexity of the geometry. The quality of the mesh was monitored using the maximum skewness and minimum orthogonality parameters. The maximum skewness was recorded to be 0.84 with a minimum orthogonality of 0.16, which is considered acceptable for numerical simulation.

A Grid Independence Test (GIT) was conducted to determine the optimum number of elements for the simulation. The model was simulated in various numbers of elements aiming to obtain the optimum number of elements. The GIT revealed that the optimum number of grids was 1,566,230 elements, which produced simulation results that closely attained the experimental results, as shown in Figure 5. Insignificant changes were observed for a larger number of elements. Thus, this grid size was chosen to avoid excessive computational load.

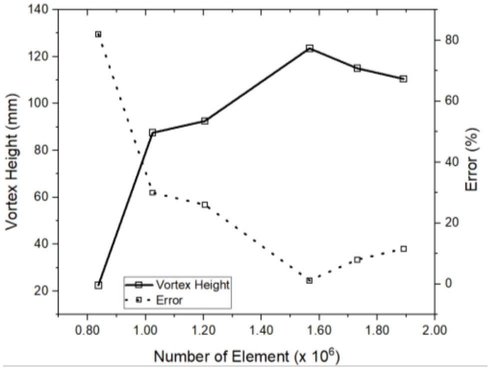


Figure 5: Grid Independence Test (GIT)

Reynolds Average Navier-Stokes (RANS) simulation, along with the Shear Stress Transport (SST) turbulence model, was utilized to get deeper information concerning the flow structure and velocity profile in the basin. The SST turbulence model is effective in predicting the flow in the viscous layer near the wall and is also well-suited to solving flow with a low adverse pressure gradient. It has been widely used for aerodynamics and hydroturbine simulations, and its ability to accurately predict aerodynamic phenomena has been demonstrated in the NASA Technical Memorandum [31].

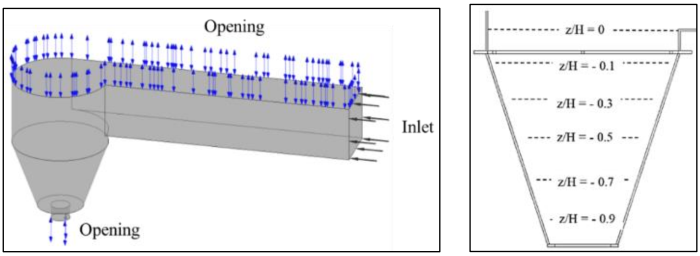
In this study, a multiphase based on the Volume of Fluid (VoF) model was carried out to describe the free surface of the flow. VoF is a reliable method to model a free surface. The interface on the free surface can be represented by the value of volume fraction, which is the volume fraction of the phase/material present in each cell/grid. The volume fraction can be expressed as in Equation (1) [32]. The free surface is located at $0 < F(x,t) < 1$ [33].

The standard free surface model was employed to model the interface layer between water and air. Water is the primary fluid, with a surface tension coefficient of 0.072 N/m. Meanwhile, the bulk mass flowrate inlet boundary condition was utilized in the inlet zone, together with the opening at the upper surface and outlet, as shown in Figure 6(a). The atmospheric pressure at the opening side is 1 atm.

$$F(x,t) = \frac{volume\ of\ fluid}{volume\ of\ element} = \begin{cases} fully\ filled & 1 \\ partially\ filled & 1 < F(x,t) < 0 \\ empty & 0 \end{cases} \quad (1)$$

In CFD simulation, validation is a crucial stage to ensure the accuracy of the boundary conditions used and to confirm the conformity of the simulation results with the actual phenomenon. The validation process involves evaluating the simulation data and comparing it with the experimental data to identify any discrepancies. The difference between the experimental and simulation data was 1.2%, indicating that the simulation accurately predicted the experimental results. The validation process focused on comparing the vortex height data obtained from the numerical simulation with the experimental data.

CFD simulation not only provides the core diameter and vortex height data, but also the flow velocity within the basin, including axial, radial, and tangential velocity components. Data sampling was conducted at different values of depth (z), as illustrated in Figure 6(b), and subsequently analyzed as a function of depth and distance from the center.



(a) (b)

Figure 6: Simulation setting and post processing; (a) boundary conditions, and (b) measurement point

Results and Discussion

The effect of cone angle on vortex characteristic

An experimental study was conducted to observe the effects of cone angle on vortex characteristics, providing information on vortex height and air-core diameter which represent the characteristics of the vortex. The session begins with an analysis of the experimental results, followed by the presentation of simulation results towards the end. Figure 7 provides a comparison of the experimental and simulation studies, demonstrating the correspondence between the two. In this session, the analysis is for a basin with $d/D = 0.2$.

The cone angle, which is a design parameter for the basin, has a significant impact on the shape of the vortex. The cone angle causes changes in the cross-section area and volume of the basin, which directly affect the water velocity and vortex height. Furthermore, this study examines the effect of water flow rate on the vortex height. The vortex height increases with the rise of flow rate as more water enters the basin. This phenomenon reveals that as more water is retained in the basin, it gains more static head, which has a positive impact on power extraction. The vortex height tends to increase linearly with the flow rate, with a slope of 0.0528 for a cone angle of 20°, as depicted in Figure 7. It is for basin with $d/D = 0.2$, but the phenomenon remains similar for other d/D. The experimental study shows a good correspondence with the simulation study as depicted in Figure 7, in which the vortex height increases with increasing flow rate, with an average simulation error of 10%. The simulation error tends to decrease at high flow rates. It also shows that the value of vortex height increases in a basin with a larger cone angle. At the same flow rate, a basin with a greater cone angle will generate a higher vortex height. This phenomenon is triggered by the narrower cross-section of the basin with a greater cone angle.

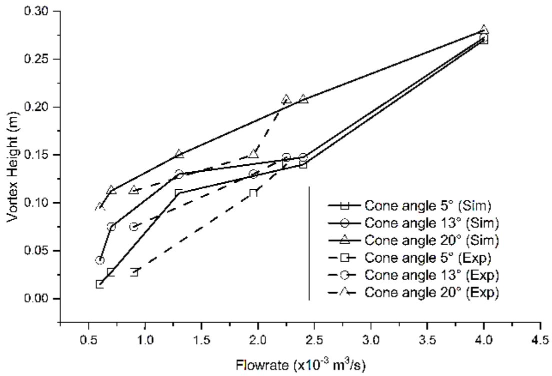


Figure 7: Effect Flowrate on the vortex height for *d/D* = 0.2 obtained from experimental and numerical study

As aforementioned, the cone angle is varied from 5° to 20° for every flow rate variation. In the beginning, the vortex height keeps changing until the inlet mass flow and the discharge mass flow are balanced, indicating a steady condition. It does not undergo significant changes after the steady state is reached. The value of vortex height is affected by the cone angle and flow rate. An increase in cone angle resulted in an increase in vortex height, and this trend was observed both in experimental and simulation studies. At low flow rates, increasing the cone angle leads to a significant increase in vortex height, particularly from a cone angle of 9° to 20°. However, the influence of the cone angle on the vortex height decreases as the flow rate increases, as given in Figure 8. The changes in the cone angle do not have a significant influent on vortex height at high flow rates (Q = 0.004 m^3/s). For every 4° increase in cone angle, the vortex height only rises approximately 0.0025 m, as water tends to exit the basin quickly, resulting in a high axial velocity. A significant impact is observed at low flow rates, as given in Figure 8.

The air-core diameter data is obtained from the experimental study. According to Mulligan et al. [29], the air-core diameter is a significant parameter that should be considered in conducting vortex analysis, both analytically and experimentally. The air core diameter contributes to the reduction of the discharge water flow rate reduction. The lab-scale experiment shows that the air core diameter tends to fluctuate with changes in flow rate, as given in Figure 9. Meanwhile, a basin with a large cone angle tends to produce a wider air core as compensation for the significant increase in vortex height at higher flow rates, to maintain energy balance in the system. The air-core diameter is a crucial consideration for determining the runner installation position. The air core diameter at the installation position should be smaller than the runner diameter to ensure that the surfaces of the runner are in contact with the water. A large air core diameter can inhibit the energy extraction process, as the runner is not completely submerged in the water. Hence the runner surfaces that are not exposed to water do not contribute to the formation of hydrodynamic forces, which directly affects the energy extraction process. Another drawback of a large air-core diameter is the risk of cavitation, especially in the runner, which can cause damage to the runner material, and reduce its lifetime.

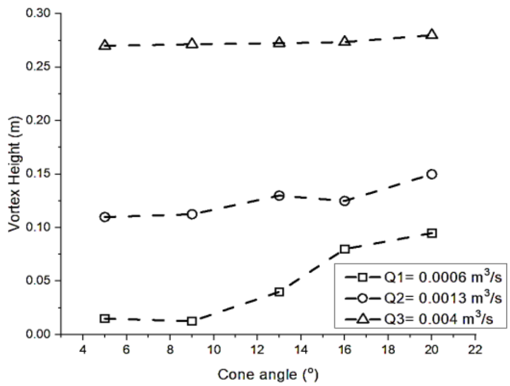


Figure 8: Effect of cone angle for *d/D* = 0.2

A simulation study is used to obtain detailed data which cannot be obtained through experimental study. This cost-effective method provides a detailed understanding of flow phenomena and is often used during the design phase or preliminary study. In this work, limitations in measuring tools have restricted the data to only vortex height and air-core diameter. However, simulation studies can still provide essential information regarding vortex characteristics, including tangential and axial velocities, as well as the shape of the vortex. More detailed information about the simulation results is explained below. In this session, the analysis pertains to a basin with a $d/D =$ 0.2 and an inlet flow rate of 0.004 m^3/s. Other basins with varying d/D are not shown, as they exhibit typical phenomena that do not differ significantly from those of the analyzed basin.

Experimental study does not provide a good description of the vortex structure that forms inside the basin, but numerical simulations can provide insight into this. Changes in cone angle shape affect the resulting vortex structure, as shown in Figure 10. It is an isosurface for water volume fraction 0.9 in a basin with d/D = 0.2 and a flow rate of 0.004 m^3/s. Based on this figure, the air core for basins with larger cone angles (16° and 20°) tends to be smaller, which indicates that a larger cone angle can trigger tangential velocity so that water does not immediately exit the basin and stays in the basin longer. As a result, a vortex shape with a narrower air-core is produced.

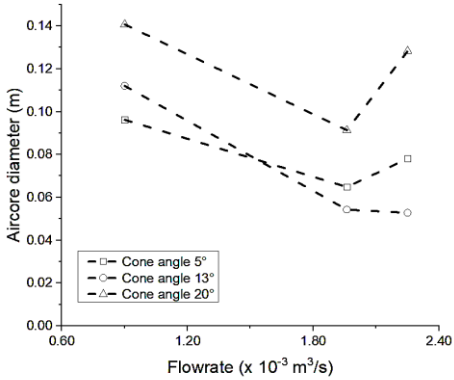


Figure 9: Change in air-core diameter as the effect of flowrate and cone angle

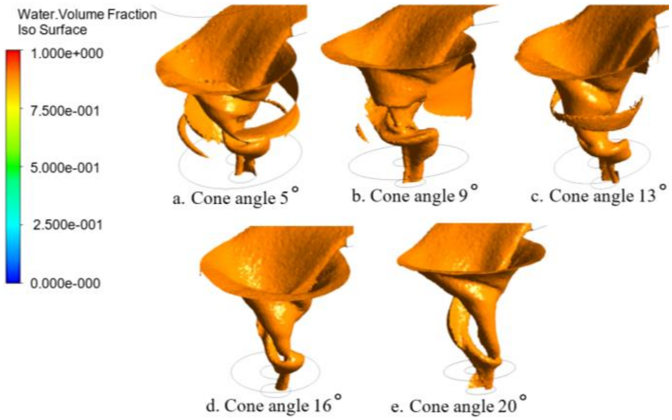


Figure 10: Change in vortex structure

The tangential velocity varies at different depths, as depicted in Figure 11. The notation z/H is used to indicate the depth, where z represents the depth position where the measurement is taken (referred to the Figure 7 and Table 1), and H is the height of the basin measured from the inlet channel bed to the basin outlet. Referring to the study conducted by [17] it shows that the GVWT runner installed at a depth of z/H -0.6 to -0.75 shows an increase in performance. Therefore, in this study, tangential velocity sampling was only carried out at depths of -1 < z/H < -0.5. Based on the provided figure, tangential velocity increases significantly at deeper positions. This phenomenon occurs for all cone angle variations. At deeper position, the tangential velocity increases due to the conversion of potential energy into kinetic energy. Furthermore, the narrower cross-sectional shape of the basin at greater z also contributes to the increase in tangential velocity, in accordance with the principle of continuity.

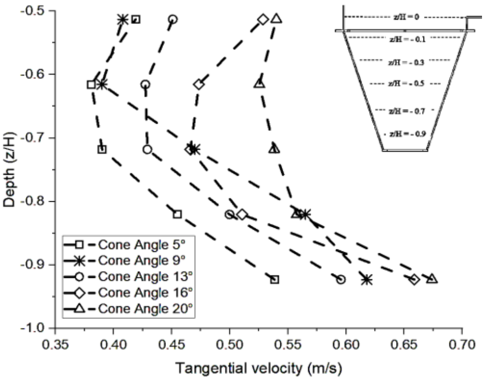


Figure 11: The effect of cone angle on tangential velocity

The cone angle has a significant influence on the tangential velocity in the basin. In the case of GVWT, tangential velocity plays an important role in the formation of hydrodynamic forces on the runner, particularly for the propeller runner. Since the radial velocity has no impact on the energy extraction process, this study focuses on the tangential velocity analysis. The tangential referred to in this study is the average tangential velocity inside the basin. A large cone angle (20°) results in a larger tangential velocity due to the smaller cross-sectional area, as shown in Figure 12. The effect of the cone angle on the tangential velocity is more pronounced at a large flow rate, which is indicated by a steeper slope. At a flow rate of 0.004 m^3/s, an increase in cone angle from 5° to 20° results in an approximately 50% increase in tangential velocity.

Figure 13 illustrates the changes in the tangential velocity profile due to variations in the cone angle. It shows significant differences in the tangential velocity, with the faster tangential velocity appearing on the left side of the basin. Additionally, the study notes that the tangential velocity profile is not axisymmetric. In the center of the basin, the tangential velocity is slower, indicating the presence of an air core region.

In order to obtain a general overview of the flow velocity conditions inside the basin, the average velocity parameter is employed. This parameter accommodates the tangential, axial, and radial velocity components. The contour of the average velocity inside the basin is depicted in Figure 14. Based on the figure, which is obtained for a d/D ratio of 0.2 and a flow rate of 0.004 m^3/s, the highest velocity is observed around the outlet area. Additionally, the figure indicates that the tangential velocity increases with a larger cone angle, and the increase in velocity is observed in the vicinity of the outlet.

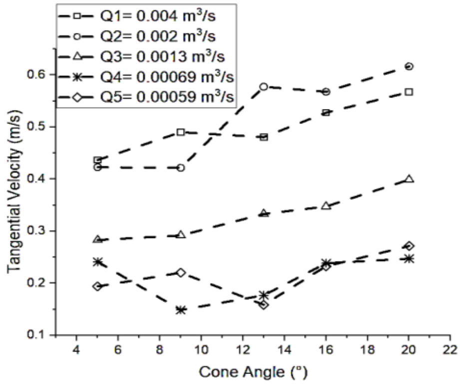


Figure 12: The impact of cone angle in tangential velocity

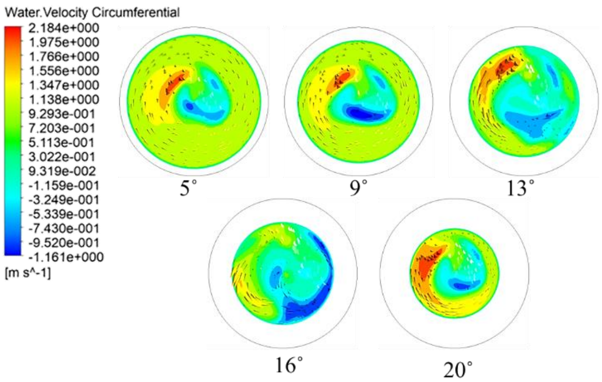


Figure 13: Tangential velocity contour measured at 0.06 m from the bottom (top view)

The determination of the cone angle is important due to its significant impact on the increase in vortex height and tangential velocity. However, the use of a large cone angle results in a smaller cross-sectional area in the outlet, thereby limiting the diameter of the installed runner. As the runner diameter decreases, the torque produced is also reduced, although it is typically associated with a higher rotational velocity. Furthermore, the adoption of a smaller cone angle leads to increased energy losses and a decrease in the total head.

The effect of diameter ratio (d/D) on vortex characteristic

The effect of (d/D) is analyzed through experimental and numerical/simulation studies. The experimental study aims to obtain data on vortex height and air-core. On the other hand, the simulation study provides data on tangential velocity and velocity contour. The vortex height and air-core diameter data obtained from simulation results are not presented since both simulation and experimental data have similar trends and phenomena.

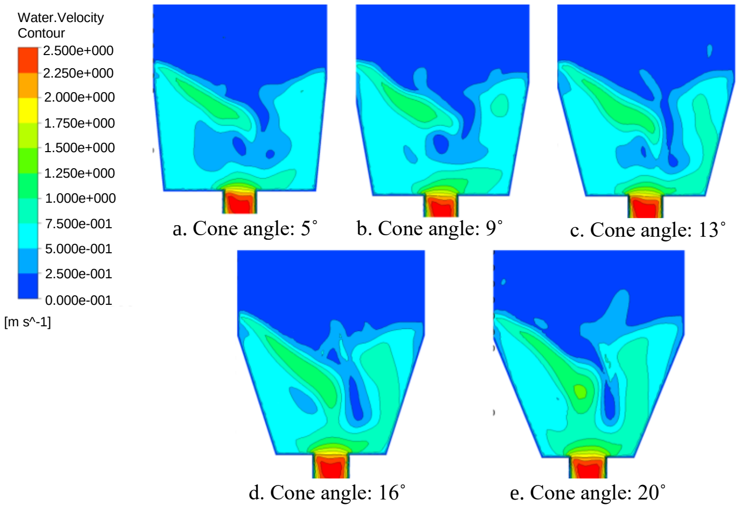


Figure 14: Velocity contour of the basin with $d/D = 0.2$ and flowrate of 0.004 m^3/s (front view)

The change in outlet diameter (indicated by d/D) affects the air-core diameter of the vortex. The air core diameter tends to increase with the increase of d/D. At large d/D of 0.4, the water flows directly through the exit and the vortex is not formed, as shown in Figure 15. The change of air-core diameter is insignificant for low flow rates. For higher flow rates, the air-core diameter tends to increase as the flow rate increases.

Changes in the diameter ratio (d/D) affect the vortex height, as illustrated in Figure 16. A basin with a large *d/D* ratio produces a low vortex height. In a conical basin with a cone angle of 20°, a d/D ratio of 0.2, and a flow rate of 0.0025 m^3/s, the vortex height reaches 0.19 m. However, at d/D 0.4, the vortex height decreased by 83% to only 0.032 m. This pattern is observed in other cone angle variations as well. This phenomenon suggests that using a basin with a $d/D > 0.4$ is not recommended, as the vortex formation is poor. The absence of vortex formation at $d/D = 0.4$ is also observed in simulations, indicated by a low vortex height. This is due to the large outlet diameter, which results in a large discharge mass flow rate. As a result, the flow does not rotate in the basin and exits straight out to the outlet [34]. This also leads to a depressed tangential velocity and a large axial velocity component. Consequently, the formation of hydrodynamic forces in the runner is affected.

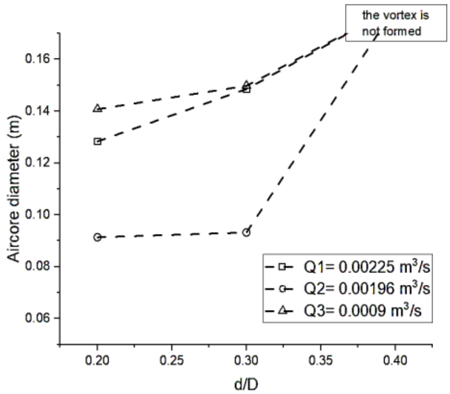


Figure 15: The effect of diameter ratio (d/D) on air-core diameter for conical basin with cone angle 20°

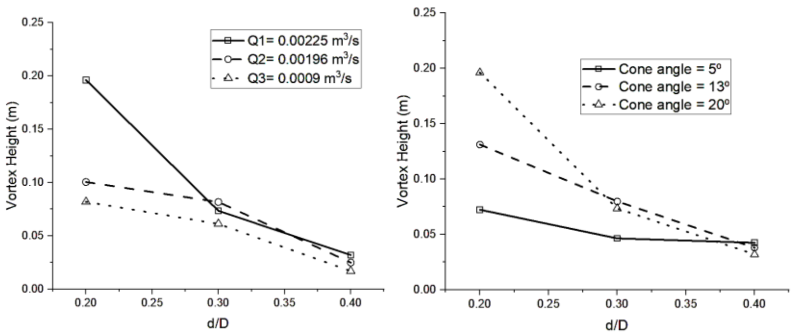


Figure 16: The impact of d/D on vortex height; (a) flowrate variation for conical basin with a cone angle of 20°, and (b) cone angle variation at a flowrate of 0.0225 m^3/s

The increase in vortex height due to the changes in the diameter ratio (d/D) is compensated by changes in the tangential velocity, as shown in Figure 17. The tangential velocity profile exhibits a fluctuating pattern. At a small d/D ratio, vortex formation does not occur, resulting in a depressed tangential velocity. Numerical simulation results suggest that the optimum *d/D* value for achieving maximum tangential velocity is 0.4. However, the vortex height is low in this condition, which limits the installation position and the diameter of the runner. Further consideration and analysis are necessary to determine the optimum d/D ratio that produces maximum power. Performance analysis of runners in various potential positions is crucial for this purpose.

The ratio of inlet and outlet diameter (d/D) has a significant impact on the structure of the vortex. Vortexs do not form in a basin with large d/D, as shown in Figure 18. A Large outlet diameter causes water to flow directly out of the outlet, leading to the domination of axial velocity and preventing the vortex formation. As a consequence, the tangential velocity is decreased. Moreover, a basin with a large outlet diameter produces a vortex with a large air-core diameter.

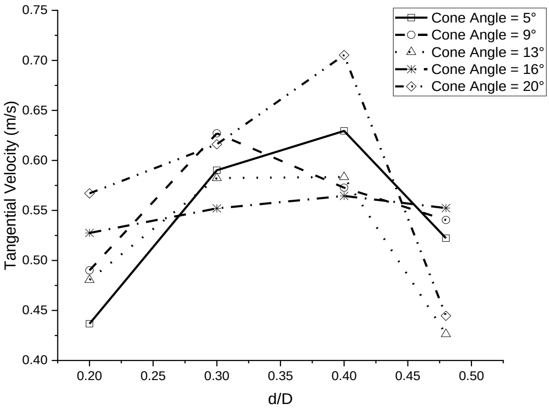


Figure 17: The impact of *d/D* on tangential velocity

According to the findings of the research conducted, the recommended design for a conical basin to be used in GVWT is a basin with a large cone angle of 20° and a small d/D ratio of 0.2. The basis for this recommendation is that the resulting vortex structure exhibits a high vortex height, a small air-core diameter, and a large tangential velocity.

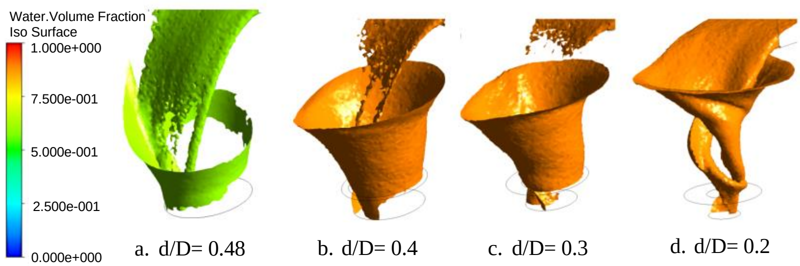


Figure 18: The impact of d/D on the vortex structure

Conclusion

Experimental and Numerical studies were performed to evaluate the impact of cone angle and d/D on the performance of conical basin for GVWT application. The validation was carried out to ensure the accuracy of numerical simulation in predicting the real phenomenon, by comparing the simulation and experimental results. The conclusions obtained from this study are as follows:

a. The flow rate significantly affects the vortex height and vortex strength, as it corresponds to the energy of water entering the basin. The vortex height and vortex strength increase at a higher flow rate.
b. Cone angle variation does not significantly affect vortex height, particularly at higher flow rates. However, increasing the cone angle results in a substantial increase in tangential velocity, while the air-core diameter fluctuates as the cone angle varies.
c. Basin diameter ratio (d/D) variation has a significant impact on vortex characteristics, such as air core diameter, vortex height, and tangential velocity). Change in d/D results in a significant change in vortex height and tangential velocity, except for air-core diameter. A smaller *d/D* leads to a greater vortex height compensating for the decrease in tangential velocity. According to the study, the recommended design for a conical basin to be used in GVWT is a basin with a large cone angle of 20° and a small d/D ratio of 0.2.

Contribution of Authors

The authors confirm the equal contribution in each part of this work. All authors reviewed and approved the final version of this work.

Funding

This work received no specific grant from any funding agency.

Conflict of Interest

All authors declare that they have no conflicts of interest.

Acknowledgment

The authors would like to express their sincere gratitude to the Ministry of Education and Culture and the Indonesia Endowment Funds for Education (PUSLAPDIK and LPDP) for providing financial support through a scholarship. The authors also wish to thank the Energy Engineering and Environmental Conditioning Laboratory and Mechanics and Fluid Machinery Laboratory for supporting this research.

References

[1] R. S. Roy and L. C. Lye, "ANSYS water-wind flow simulation to study pressure generated under various conditions to generate electricity using piezoelectric cells," *Journal of Mechanics Engineering*, vol. 17, no. 3, pp. 1–11, 2020.

[2] JICA, "Project for the master plan study of hydropower development in Indonesia," 2011.

[3] Suharyati, S. H. Pambudi, J. L. Wibowo, and N. I. Pratiwi, "Indonesian Energy Outlook 2019," 2019.

[4] J. Chen and A. Engeda, "Design and development of an ultra low head axial hydro turbine for electricity supply Part I," *Conference: ASME 2018 5th Joint US-European Fluids Engineering Division Summer Meeting*, vol. 3, pp. 1–8, 2018. doi: 10.1115/FEDSM2018-83015

[5] A. Muis, P. Sutikno, A. Soewono, and F. Hartono, "Design optimization of axial hydraulic turbine for very low head application", *Energy Procedia*, vol. 68, pp. 263–273, 2015. doi: 10.1016/j.egypro.2015.03.255

[6] M. Sinagra, V. Sammartano, C. Aricò, A. Collura, and T. Tucciarelli, "Cross-Flow turbine design for variable operating conditions," *Procedia Engineering*, vol. 70, pp. 1539–1548, 2014. doi: 10.1016/j.proeng.2014.02.170

[7] S. Wanchat, R. Suntivarakorn, S. Wanchat, K. Tonmit, and P. Kayanyiem, "A parametric study of a gravitation vortex power plant", *Advanced Materials Research*, vol. 805–806, pp. 811–817, 2013. doi: 10.4028/www.scientific.net/AMR.805-806.811

[8] A. B. Timilsina, S. Mulligan, and T. R. Bajracharya, "Water vortex hydropower technology: a state-of-the-art review of developmental trends", *Clean Technologies and Environmental Policy*, vol. 20, no. 8, pp. 1737–1760, 2018. doi: 10.1007/s10098-018-1589-0

[9] S. Wanchat and R. Suntivarakorn, "Preliminary design of a vortex pool for electrical generation," *Journal of Computational and Theoretical Nanoscience*, vol. 13, no. 1, pp. 173–177, 2011. doi: 10.1166/asl.2012.3855

[10] D. S. Edirisinghe, H.-S. Yang, S. D. G. S. P. Gunawardane, and Y.-H.

Lee, “Enhancing the performance of gravitational water vortex turbine by flow simulation analysis”, *Renewable Energy*, vol. 194, pp. 163–180, 2022. doi: 10.1016/j.renene.2022.05.053

[11] H. Feng Li, H. Xun Chen, Z. Ma, and Y. Zhou, “Formation and influencing factors of free surface vortex in a barrel with a central orifice at bottom,” *Journal of Hydrodynamics*, *Ser. B*, vol. 21, no. 2, pp. 238–244, 2009. doi: 10.1016/S1001-6058(08)60141-9

[12] S. Dhakal, S. Nakarmi, P. Pun, A. B. Thapa, and T. R. Bajracharya, “Development and testing of runner and conical basin for gravitational water vortex power plant,” *Journal of The Institute of Engineering*, vol. 10, no. 1, pp. 140–148, 2014. doi: 10.3126/jie.v10i1.10895

[13] R. Ullah, T. A. Cheema, A. S. Saleem, S. M. Ahmad, J. A. Chattha, and C. W. Park, “Performance analysis of multi-stage gravitational water vortex turbine”, *Energy Conversion and Management*, vol. 198, no. 1, p. 111788, 2019, doi: 10.1016/j.enconman.2019.111788.

[14] G. Slachmuyders, “A gravitational vortex water turbine assembly,” Patent, 2017.

[15] V. J. Alzamora Guzmán, J. A. Glasscock, and F. Whitehouse, “Design and construction of an off-grid gravitational vortex hydropower plant: A case study in rural Peru”, *Sustainable Energy Technologies and Assessments*, vol. 35, pp. 131–138, 2019. doi: 10.1016/j.seta.2019.06.004.

[16] V. J. Alzamora Guzmán and J. A. Glasscock, “Analytical solution for a strong free-surface water vortex describing flow in a full-scale gravitational vortex hydropower system”, *Water Science and Engineering*, vol. 14, no. 1, pp. 72–79, 2021. doi: 10.1016/j.wse.2021.03.004

[17] A. S. Saleem, T. A. Cheema, R. Ulah, S. M. Ahmad, J. A. Chattha, and B. Akbar, “Parametric study of single-stage gravitational water vortex turbine with cylindrical basin”, *Energy*, vol. 200, no. 4, pp. 1-18, 2020. doi: 10.1016/j.energy.2020.117464

[18] S. Mulligan and P. Hull, “Design and optimisation of a water vortex hydropower plant”, *Material Science and Engineering A*, vol. 6, p, 1, 2010.

[19] T. R. Bajracharya and R. K. Chaulagai, “Developing innovative low head turbine for free-flowing streams suitable for micro-hydro-power in flat (Terai) regions in Nepal”, *Kathmandu: Center for Applied Research and Development (CARD), Institute of Engineering, Tribhuvan University, Nepal,* 2012.

[20] S. Mulligan, J. Casserly, and R. Sherlock, “Effects of geometry on strong free-surface vortices in subcritical approach flows,” *Journal of Hydraulic Engineering*, vol. 142, no. 11, p. 1-12, 2016. doi: 10.1061/(asce)hy.1943-7900.0001194

[21] S. Dhakal, A. B. Timilsina, R. Dhakal, D. Fuyal, T. R. Bajracharya, and

H. P. Pandit, "Effect of dominant parameters for conical basin : Gravitational water vortex power plant specification of the appropriate boundary conditions at cells which coincide with or touch the domain boundary," *Proceedings of IOE Graduate Conference*, pp. 380–386, 2014. doi: 10.13140/RG.2.1.1455.7843

[22] S. Dhakal, A. B. Timilsina, R. Dhakal, D. Fuyal, T. R. Bajracharya, H. P. Pandit, N. Amatya, A. M. Nakarmi "Comparison of cylindrical and conical basins with optimum position of runner: Gravitational water vortex power plant", *Renewable and Sustainable Energy Reviews*, vol. 48, pp. 662–669, 2015. doi: 10.1016/j.rser.2015.04.030

[23] L. Velásquez, A. Posada, and E. Chica, "Optimization of the basin and inlet channel of a gravitational water vortex hydraulic turbine using the response surface methodology", *Renewable Energy*, vol. 187, pp. 508–521, 2022. doi: 10.1016/j.renene.2022.01.113

[24] N. H. Khan, "Blade optimization of gravitational water vortex turbine," Master Thesis, Faculty of Mechanical Engineering, Ghulam Ishaq Khan Institute of Engineering Sciences and Technology, 2016.

[25] R. S. Sreerag, C. K. Raveendran, and B. S. Jinshah, "Effect of Outlet Diameter on the Performance of Gravitational Vortex Turbine With Conical Basin," *International Journal of Scientific and Engineering Research*, vol. 7, no. 4, pp. 457–463, 2016.

[26] Y. Nishi, R. Suzuo, D. Sukemori, and T. Inagaki, "Loss analysis of gravitation vortex type water turbine and influence of flow rate on the turbine's performance", *Renewable Energy*, vol. 155, pp. 1103–1117, 2020. doi: 10.1016/j.renene.2020.03.186

[27] D. Powalla, S. Hoerner, O. Cleynen, N. Müller, J. Stamm, and D. Thévenin, "A computational fluid dynamics model for a water vortex power plant as platform for etho-and ecohydraulic research", *Energies*, vol. 14, no. 3, pp. 1–14, 2021. doi: 10.3390/en14030639

[28] A. B. Janjua, M. S. Khalil, M. Saeed, F. S. Butt, and A. W. Badar, "Static and dynamic computational analysis of Kaplan turbine runner by varying blade profile", *Energy for Sustainable Development*, vol. 58, pp. 90–99, 2020. doi: 10.1016/j.esd.2020.07.008

[29] S. Mulligan, J. Casserly, and R. Sherlock, "Experimental and numerical modelling of free-surface turbulent flows in full air-core water vortices," in Advances in Hydroinformatics, Springer, Singapore, 2016, pp. 549–569. doi: 10.1007/978-981-287-615-7_37

[30] A. Hoseyni-Chime and P. C. Malte, "Hydrokinetic turbines at high blockage ratio," *Proceedings of The 2nd Marine Energy Technology Symposium*, pp. 1–11, 2014.

[31] J. E. Bardina, P. G. Huang, and T. J. Coakley, "Turbulence modeling validation, testing and development", *Nasa Technical Memorendum*, no. 110446, pp. 1-101, 1997. doi: 10.2514/6.1997-2121

[32] S. Shin and W. Il Lee, "Finite element analysis of incompressible viscous

flow with moving free surface by selective volume of fluid method," *International Journal of Heat and Fluid Flow*, vol. 21, no. 2, pp. 197–206, 2000. doi: 10.1016/S0142-727X(99)00083-1

[33] R. Scardovelli and S. Zaleski, "Direct numerical simulation of free-surface and interfacial flow", *Annual Review of Fluid Mechanincs*, vol. 31, no. 1, pp. 567–603, 1999. doi: 10.1146/annurev.fluid.31.1.567

[34] M. G. Marian, T. Sajin, and A. Azzouz, "Study of micro hydropower plant operating in gravitational vortex flow mode", *Applied Mechanics and Materials*, vol. 371, pp. 601–605, 2013. doi: 10.4028/www.scientific.net/AMM.371.601