The Numerical Study of the Effect of Blade Depth and Rotor-Basin Ratio on Vortex Hydro Turbine Performance

Warjito
Department of Mechanical Engineering, Universitas Indonesia, Indonesia Faculty of Engineering, Universitas Indonesia

Putrawan, Oimolala
Department of Mechanical Engineering, Universitas Indonesia, Indonesia Faculty of Engineering, Universitas Indonesia

Budiarso
Department of Mechanical Engineering, Universitas Indonesia, Indonesia Faculty of Engineering, Universitas Indonesia

Irwansyah, Ridho
Department of Mechanical Engineering, Universitas Indonesia, Indonesia Faculty of Engineering, Universitas Indonesia

他

https://doi.org/10.5109/4794187

出版情報：Evergreen. 9 (2), pp.556-562, 2022-06. 九州大学グリーンテクノロジー研究教育センター
バージョン：

The Numerical Study of the Effect of Blade Depth and Rotor-Basin Ratio on Vortex Hydro Turbine Performance

Warjito[1], Oimolala Putrawan[1, a)], Budiarso[1], Ridho Irwansyah[1], Sanjaya BS Nasution[1,b*]

[1]Department of Mechanical Engineering, Universitas Indonesia, Indonesia
Faculty of Engineering, Universitas Indonesia, Indonesia

a) Corresponding author: oimolalaputrawan@gmail.com
b) sanjayabaroarnasution@gmail.com

(Received February 11, 2022; Revised June 20, 2022; accepted June 20, 2022).

Abstract: In the end of third quartile of 2020, 98.9% Indonesia's national electrification ratio has been reached. The obstruction of 100% electrification is occurring in remotes areas. It is caused by demographical factors, including small population and incomes, accompanied by geographic and economic restriction on installing conventional power plant. To overcome this problem, gravitational water vortex turbine power plant can be an alternative to generate electricity even in most remotes area since it offers low cost in operating and maintenance as well as easy to manufacture compared to other type renewable energy power plant. It can generate electricity up to 5 kW capacities with low head and flow rate. In order to optimize the turbine performance, a study arranged to evaluate the effect of blade size and depth onto efficiency of the turbine. The study conducted numerically using Ansys Fluent within assumption of 1.03 m head and 0.16 m3/s flow rate. There are nine configurations in this study with varying rotor size of 200 mm, 350 mm, 500 mm and turbine depth of 270 mm, 340 mm, 410 mm. Result of study leads to conclusion that largest size and upper top position give highest performance of the turbine.

Keywords: numerical study; renewable energy; gravitational vortex; hydro turbine; depth-rotor ratio; efficiency

1. Introduction

Indonesia is known as one of the largest archipelago country which has more than 17.000 islands spread over the nation. The geographical condition sets an obstacle in progress of national electrification in many remote areas across the country. The country is targeting 100% electrification ratio implemented by 2025. Meanwhile, in the end of third quartile of 2020, Indonesian ministry of energy and natural resources stated that 98.9% national electrification ratio has been reached with Nusa Tenggara Timur regency being the least electrificated with 85.54% ratio[1)]. The obstruction of electrification in remotes areas is caused by demographical and geographical factors, including small population and incomes accompanied by geographic and economic restriction on installing conventional power plant[2)]. To overcome this problem, an easy to install and renewable energy-based power plant can be a solution. A pico hydro power plant can be an alternative to generate electricity even in most remotes area since it offers low cost in operating and maintenance as well as easy to manufacture compared to other type renewable energy power plant[3)4)5)]. It is estimated that 5% of the global small scale hydropower has been harnessed while its total potential capacity could reach 200 GW[6)]

Pico hydro power plant can generate electricity up to 5 kW capacity with low head and flow rate. There are many types of pico hydro turbines could be implemented in remote areas, which one of the most effective is gravitational vortex water turbine. The turbine utilizes the gravitational force and basin geometry for vortex formation which followed by energy extraction by blades.

There are several research had been conducted in gravitational vortex turbine in order to improve the efficiency of the turbine. Study done by Dhaka et al.[7)], investigates the basin geometry role in vortex formation and several basin design parameters are discussed. The study concluded that conical basin has higher efficiency over cylindrical shaped basin. Strong vortex formation in the basin of cylindrical configuration happens at the recommended ratio of orifice diameter to basin for low-head and high-head sites is 14%–18%, respectively[8)]. Christopher et al.[9)], conducted a study on comparison of straight blade over curved blade. The result showed that there is no significant improvement of power extraction with curved blade gives higher efficiency. The research of Shabara and Yaakob [10] related to using the CFD method to investigate the optimum configuration of water vortex formation, which showed that by reducing the outlet diameter, the velocity increased. The numerical modeling of a fluid dynamics problem implicates first a precise reading of the physical phenomena. All the relevant features of interest should be indicated at that first step, including geometry, materials, boundary conditions, to be defined in the simplest way, but without introducing extreme errors with the hypothesis. Nevertheless, a number of simplifications is always accepted, and is inevitable in order to model properly fluid dynamics problems . Another study on basin geometry is conducted by Muadz[9], which investigates the effect on ratio of orifices diameter to basin diameter. The study leads to conclusion that higher efficiency is achieved using smaller orifices diameter. Moreover, the study also implies the smaller orifices diameter generates smaller turbulent intensity. Suntivarakorn[11], clearly claimed baffle plate installation could improve efficiency of the turbine and stated that the increase in number of blades is related to the increase of efficiency with five blades is the optimum configuration. Subsequent increase in number of blades is followed by decrease in efficiency. The multi-stage gravitational vortex turbine research is conducted elaborately research is conducted elaborately by Ullah et al[12]. The draft tube (mostly elbow type) is an essential component of a reaction turbine for a high efficiency. Kumar[13] argue that use of draft the can increase the efficiency of turbine. The main function of the draft tube is the conversion of excessive kinetic energy (KE) at the runner exit to pressure energy. About 80% of the total recovery of the pressure takes place in the draft tube cone, which is around 10% of the total draft tube length[20]. The study discusses design parameters including ratio of rotor diameter and rotor position in multi-stage configuration along with the effect of number of stages on power generation. Then several performance parameters are measured, such as rotational speed (RPM), torque, and shaft power. The study declared that best output parameters occurred in the condition where all the blade in contact with the vortex which the x`blade surface is at the same height of the vortex surface. Study by Sritram showed that the number of blades and baffle plates attached to the blades turine affect the efficiency of turbne. It was seen that turbine with top and bottom baffle was 6.59% higher than that without baffle plate. The studies mentioned above are vary. Yet, the study of the blade size which allows optimum efficiency is still limited. There is a study conducted by Power et al regarding the mass of the blade[14]. The result of study shows that large mass runner is more efficient. However, after optimum mass there is decrease in power output. But the study didn't explore the relationship of size of blade and dept of submergence of the turbine. This study investigates the effect of size of blade on turbine performance with varying depth of turbine submergence.

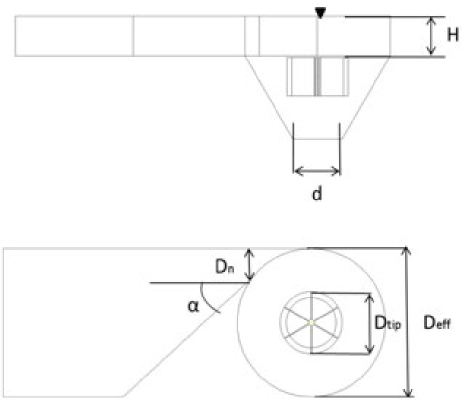


Figure 1 : The Geometry of Basin

Table 1. Parameters value of channel and basin.

Parameters	Value
Basin Diameter (D_{eff})	100 mm
Orifice Diameter (d)	140 mm
Approach Flow Angle (α)	40^0
Inlet Height (h_{in})	333.33
Basin Height	1033.33

2. Methodology

2.1 Basin Geometry

A channel used in this study was given by previous work of Muadz [9]. The difference between the Muadz study and this study is the dimensions of the blades and the position of the blades. The dimension of the blade that is changed is the diameter of the blade tip. In this study, the blade tip diameter was varied into 3 variations. Then, each different diameter variation will be placed in 3 different positions, so that in this study there are a total of 9 case variations. The dimension of the channel can be seen in Table 1. Then Figure 1 below visualizes the channel and basin geometry used for this study.

Blade tip diameter (Dtip) is varied with sizes of 200 mm, 350 mm, and 400 mm so that the rotor-basin ratio are 0.2, 0.35, and 0.5, respectively. Depth of submergence of the turbine also being varied with distance of 270 mm (H1), 340 mm (H2), and 410 mm (H3). Rotor- basin ratio can be seen in eq 1 below.

2.2 Simulaion Setup

Numerical simulation in this study is carried out by using Computational Fluid Dynamics (CFD) software ANSYS FLUENT 18.1[15]. The numerical simulation using Computational Fluid Dynamics (CFD) can be one of the tools which could provide a robust aerodynamics performance analysis. Nonetheless, the improvement could be very tricky with the optimization procedure and sizing technique[16]. The study conducted in 3-Dimensional analysis and using Multiple Reference

Frame (MRF) method. The multiple reference frame method was chosen due to existence of static cell zone and moving cell zone in this case, basin and turbine respectively. In multiple reference frames, rotation was dependent variable which include the variable as boundary condition[17]. In this study, rotation is varied into 7 variations which are 10, 30, 50, 70, 90, 110, and130 rpm. The variation of rotation is in order to find the optimum performance of turbine rotational speed.

The simulation is conducted using pressure-based solver assuming the flow is incompressible. Meanwhile, for turbulent transport model sensitivity, the turbine model is simulated using renormalized group (RNG), shear stress transport (SST) and standard k-ω. Ashwindran et al., reported that (SST) and standard k-ω exhibited trivial dissimilarities and stable numerical oscillationIntense inflation layers were applied to the elements adjacent to the model surface[18].

The numerical study in this study used the governing equations which were the mass conservation and momentum conservation[19]. The Navier-Stokes equation o be solved through CFD are as follow[20)21)] .

$$\frac{\partial\rho}{\partial t} + div(\rho u) = 0 \quad (2)$$

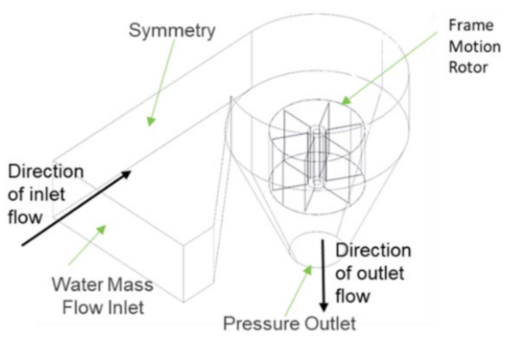
$$\rho(\frac{\partial V}{\partial t}) = -\nabla P + \rho g + \mu\nabla^2 V + S_m \quad (3)$$

There are three stages in CFD, namely (1) pre-processor, (2) solver, (3) post-processor. Pre-processing stage include 3D geometry creation, meshing, simulation setup. Solver stage includes numerical solving the equation by mean of numerical computation[22]. In this stage, computer solve the eq 2 and eq 3 using finite volume method. Post-processor stage includes looting the data from solver. At the stage of pre-processing, a 3D geometry is created using 3D CAD application. The geometry that had been created then is imported into fluent. In fluent, the important boundary faces and cell zones were being named in order to the parts being defined in the subsequent steps. This step is called name selection. Then grid generation was done to divide the geometry into chunks so that finite volume method could be carried out. Turbulent model of k-ε is applied to simulate the three dimentional flow[23]. The turbulent inlet intensity was in the value of 5%, considering that the blade is

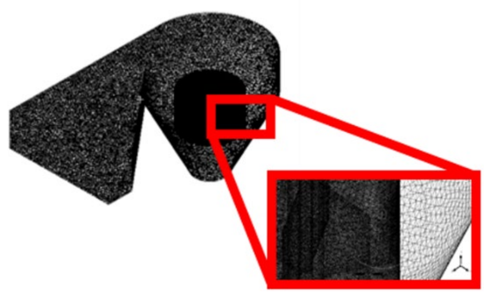
2.3 Independency Test

The independency test is done to verify the number of mesh elements which is also called verification. Verification is method to find the minimum error resulting from discretization of space. To do this, the number of meshes is varied. Generally, the higher the number of meshes, the more accurate the numerical calculation results will be. However, it also causes the calculation to take longer. Therefore, the percentage error in each variation in the number of meshes will be calculated so that the number of meshes is found which no longer gives a significant error. The CFD mesh for the flow domain was generated on the basis of unstructured hybrid 3D elements. Intense inflation layers were applied to the elements adjacent to the model surface. The value of y+ for the first cell above the model surface was conserved at below 1 (y+ <1) to capture the effect of boundary layer[20].

After finding the number of appropriate mesh, the final pre-processing step was setting up the boundary conditions and fluid properties. The boundary condition used for this study are mass flow inlet with 160 kg/s of water, pressure outlet with 0 Pa, and symmetry on above side of channel. For distinction, figure 2(a) and figure 2(b) visualize the geometry and mesh used in this study.



(a)



(b)

Figure 2 : (a) Geometry and boundary condition, (b) visualization of 4,478,843 elements

The simulation was done using steady, single phase, incompressible fluid and the turbulence model SST k-ω. This turbulence model was chosen because it can cover a standard bypass flow transition in a low free stream environment[24].

3. Results And Discussion

3.1 Independency Test Result

The mesh independency test was done using Richardson extrapolation [25]. The test achieved a result of mesh which were 2847785, 3283394, and 4478843 elements Torque at each number of elements was calculated to analyze the appropriate number of elements with GCI calculation method. The GCI calculation results are shown in Table 2 where the best number of elements is 4478843.

Table 2. Mesh Independence Test

Mesh Number	Grid Spacing	Torque	r	p	GCI
2847785	1.25	35.79	-		-
3283394	1.16	25.90	1.074	9.505	9.43%
4478843	1	24.01	1.168	13.598	0.20%

3.2 Turbine Performance

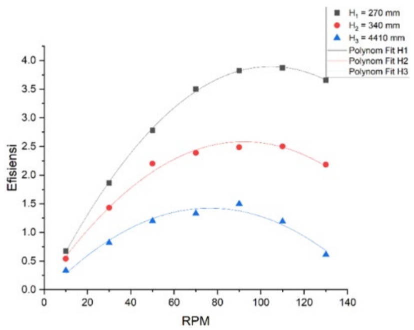
The turbine performance shown in this study is the power and efficiency of the turbine. Turbine power (Pin and Pout) and efficiency is obtained based on the following equation.

$$P_{in} = \rho g Q H \quad (4)$$

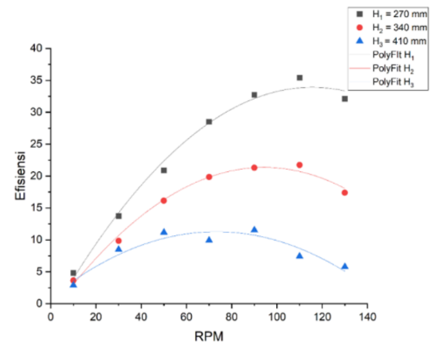
$$P_{out} = \tau \omega \quad (5)$$

$$\eta = \frac{P_{out}}{P_{in}} x100\% \quad (6)$$

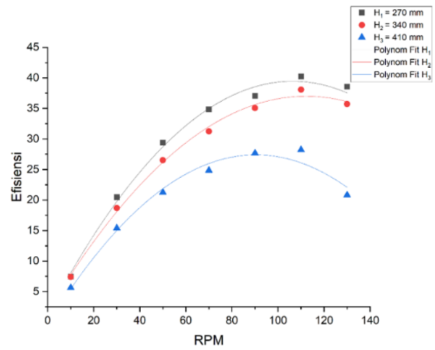
Figure 4 shows the hydraulic efficiency vs RPM of the 9 configurations of study. Figure 4a, 4b, 4c are distinguished by the tip of the blade diameter of 200 mm, 350 mm and 500 mm. From the figures can be seen that the highest performance is achieved when the turbine with diameter of rotor of 500 mm and depth of submergence of 210 mm with the value of the efficiency is 40.22%, while the lowest performance occurred on turbine having the smallest diameter of rotor of 200 mm and depth of submergence of 410 mm with the value of 1.12%. In addition, it can be seen that the blade position greatly determines the blade performance. The higher the depth of the blade, the lower the efficiency. At depth of submergence of 0.1Ht(270 mm), 0.2Ht (340 mm), and 0.3Ht (410 mm) from the water level the efficiency is 40.22%, 35.39%, and 3.87%, respectively. It also can be concluded that as the diameter of the rotor increases, the efficiency of the turbine is increased.



(a)



(b)



(c)

Figure 3. Efficiency vs Rpm, a) D_{tip}= 200 mm, b) D_{tip}= 350 mm, c) D_{tip}= 500 mm

3.3 Discussion

The phenomenon above can be analyzed by investigating the pressure contour of the turbine. Figure 4 – 6 show pressure contours for each configuration. Based on the figure can be seen that there is pressure distribution from the basin wall to the core of the vortex. As the vortex flow gets closer to the vortex core, the local pressure decreases. The velocity increase as the flow closer to the core is accounted for the decrease in pressure. In order to fulfill the conservation of angular momentum, the flow velocity near the vortex core has to be higher than the further one. The phenomenon above creates pressure distribution across the basin and the turbine.

In Figure 4-8 it can also be concluded that the smaller diameter blade has a more significant pressure distribution than the other two rotor sizes, so it can be concluded that the size of the rotor diameter and turbine position affect the vortex flow pattern. The larger pressure value on the smaller diameter rotor occurs because the water flow has a low speed because the area of water flow between the blade and the basin is quite large. In larger diameter rotors the pressure at the inlet side of the flow is quite low because the area of water flow between the blades and the basin wall is relatively small compared to before so the velocity is higher. Because kinetic energy is the main energy component that can be absorbed by the blades, the low that has a higher velocity can be extracted more energy by the blades than those with a higher velocity.

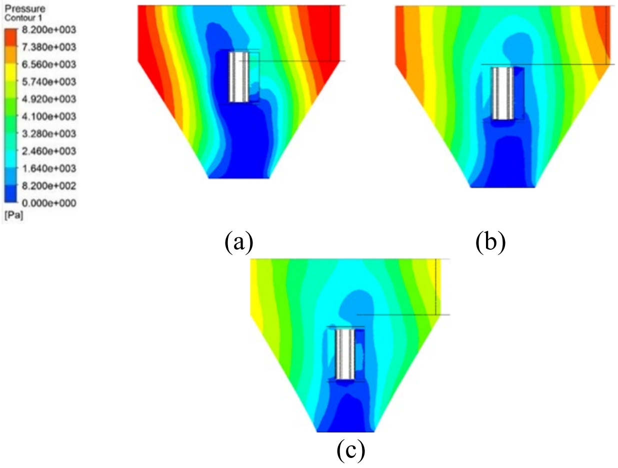


Figure 4. Pressure contour (a) H_1, (b) H_2, (c) H_3 at rotor size = 200 mm

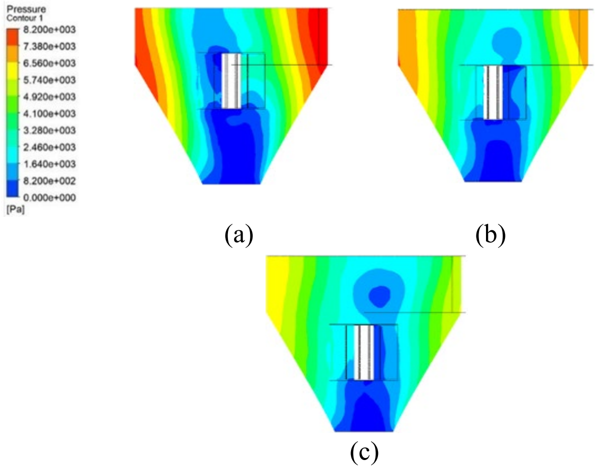


Figure 5. Pressure contour (a) H_1, (b) H_2, (c) H_3 at rotor size = 350 mm

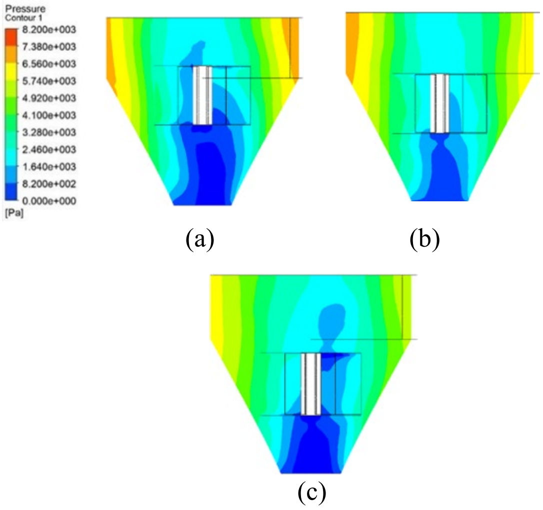


Figure 6. Pressure contour (a) H_1, (b) H_2, (c) H_3 at rotor size = 500 mm

Figure 7-9 visualizes the tangential velocity of the flow in the basin. The results inform that the smallest tangential velocity is occurred on the blade profiles which implicitly shows that the kinetic energy of the vortex flow is transferred fully into the blade. Also from the figures can be concluded that as the depth of submergence of the turbine gets deeper that the velocity of the flow imparting the turbine is decreased. The pressure difference of water flow at rotor level and the flow under the rotor is responsible for the difference in flow velocity imparting the turbine. This pressure difference generates suction effects so that the flow is accelerated while it flows directing to the exit through the orifice. Due to this phenomenon the turbines located in depth of 210 mm give better performance compared to other turbines installed in different location regardless the size of rotor turbine. The above results share similar trend with a study conducted by Muadz, which concluded increase in depth of submergence of turbine leads to performance drop.

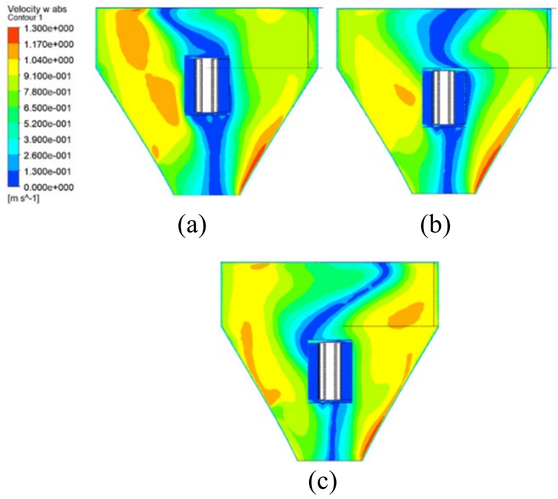


Figure 7. Velocity Contour (a) H_1, (b) H_2, (c) H_3 at rotor size = 200 mm

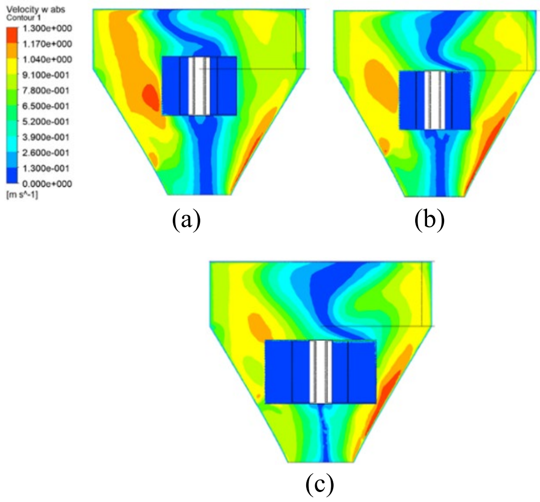
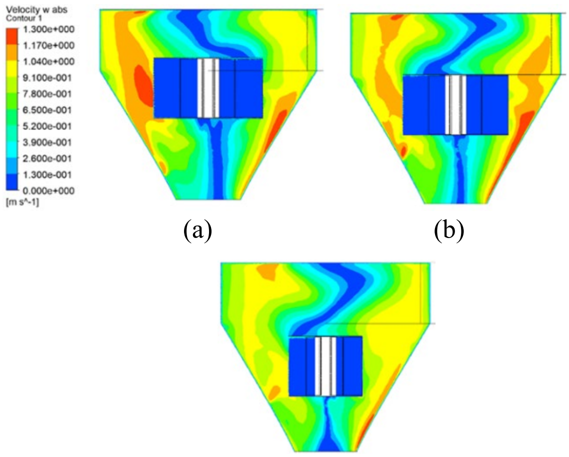


Figure 8. Velocity Contour (a) H_1, (b) H_2, (c) H_3 at rotor size = 350 mm



(a) (b)

(c)

Figure 9. Velocity Contour (a) H_1, (b) H_2, (c) H_3 at rotor size = 500 m

Larger size of rotor results in better performance of the turbine due to large area of the blade being imparted of momentum by water. This transfer momentum is followed by transfer and transform kinetic energy of fluid into rotational energy in blade of the turbine. From the figure 9 can be seen that the rotor is too small, thus only small area of the blade being struck by water which lead to small power output is achieved by rotor with size of 200 mm regardless of the position of the turbine. Larger size of rotor also has larger moment of inertia which giving better performance since the torque generated will be higher.

As a summary, table 3 shows the peak efficiency of all the cases. It shown that the highest efficiency is at point of Dt 500 mm and H 270 mm. However, the lower efficiency is at point Dt 200 mm and H= 410 mm.

Table 3. Peak efficiency at all cases

	H = 270 mm	H = 340 mm	H= 410 mm
D_t = 200 mm	3.87 %	2.52 %	1.47 %
D_t = 350 mm	35.39 %	21.72 %	11.53 %
D_t = 500 mm	40.22 %	38.08 %	28.22 %

Based on table 3, it can be seen that the larger the blade, the better the turbine performance and vice versa, the higher the blade depth, the lower the blade performance. Other studies show the same phenomena which larger size of rotor diameter gives higher power output[11]. In case of varying submergence of the turbine, this study gives similar results with the study of Dhaka et al. which showed that in conical basin vortex turbine, the performance of the turbine will decrease as the depth of submergence increased [7].

4. Conclusion

In this study, ratio of diameter of rotor-basin and depth of submergence of the gravitational vortex hydro turbine is investigated numerically using computational fluid dynamics (CFD). The steady simulation was conducted with multiple reference frame method. The performance and flow characteristics of the 9 configurations has been calculated and analyzed. Based on numerical result, rotor size and depth of submergence affect performance of the turbine. Turbine with largest ratio of diameter of rotor-basin gives better performance than the smaller ones with the efficiency of 40.22%. Then the turbine positioned at depth of submergence of 210 mm or top in this study results higher efficiency than those positioned at depth below the aforementioned.

Acknowledgements

Indicate This work was supported by the ministry of research, technology, and higher education (kemenristek dikti) of Republic of Indonesia with grant no: NKB-215/UN2.RST/HKP.05.00/2021.

References

1) "Statistik Ketenagalistrikan T.A 2017," 2018.
2) P. Maher, N.P.A. Smith, and A.A. Williams, "Assessment of pico hydro as an option for off-grid electrification in kenya," *Renew. Energy*, **28** (*9*) 1357–1369 (2003). doi:10.1016/S0960-1481(02)00216-1.
3) Warjito, E.P. Wijaya, Budiarso, S.B.S. Nasution, and A.F. Kurnianto, "Study on the application of particle image velocimetry with resin and rhodamine b as alternative seeding particles in pico hydro cross flow hydro turbine," *J. Adv. Res. Fluid Mech. Therm. Sci.*, **85** (*1*) 1–11 (2021). doi:10.37934/arfmts.85.1.111.
4) D. Adanta, M. Agil Fadhel Kurnianto, Warjito, S.B.S. Nasution, and Budiarso, "Effect of the number of blades on undershot waterwheel performance for straight blades," *IOP Conf. Ser. Earth Environ. Sci.*, **431** (*1*) (2020). doi:10.1088/1755-1315/431/1/012024.
5) B. Ho-Yan, "Design of a low head pico hydro turbine for rural electrification in cameroon," 1–175 (2012).
6) M.M. Rahman, T.J. Hong, R. Tang, L.L. Sung, F. Binti, and M. Tamiri, "Experimental study the effects of water pressure and turbine blade lengths & numbers on the model free vortex power generation system," *Int. J. Curr. Trends Eng. Res.*, **2** (*9*) 13–17 (2016). http://www.ijcter.com.
7) S. Dhakal, A.B. Timilsina, R. Dhakal, D. Fuyal, T.R. Bajracharya, H.P. Pandit, N. Amatya, and A.M. Nakarmi, "Comparison of cylindrical and conical basins with optimum position of runner: gravitational water vortex power plant," *Renew. Sustain. Energy Rev.*, **48** 662–669 (2015).
8) 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 Convers. Manag.*, **198** (*May*) 111788 (2019). doi:10.1016/j.enconman.2019.111788.
9) C.R. Christopher, and D. Adanta, "The effect of basin

geometry on gravitational vortex hydropower," in: IOP Conf. Ser. Mater. Sci. Eng., IOP Publishing, 2020: p. 12081.
10) H.M. Shabara, O.B. Yaakob, Y.M. Ahmed, and A.H. Elbatran, "CFD simulation of water gravitation vortex pool flow for mini hydropower plants," *J. Teknol.*, **74** (*5*) 77–81 (2015). doi:10.11113/jt.v74.4645.
11) S. Wanchat, and R. Suntivarakorn, "Preliminary design of a vortex pool for electrical generation," *Adv. Sci. Lett.*, **13** (*January 2011*) 173–177 (2012). doi:10.1166/asl.2012.3855.
12) R. Ullah, T.A. Cheema, A.S. Saleem, S.M. Ahmad, J.A. Chattha, and C.W. Park, "Preliminary experimental study on multi-stage gravitational water vortex turbine in a conical basin," *Renew. Energy*, **145** 2516–2529 (2020). doi:10.1016/j.renene.2019.07.128.
13) S. Kumar, M.J. Cervantes, and B.K. Gandhi, "Rotating vortex rope formation and mitigation in draft tube of hydro turbines – a review from experimental perspective," *Renew. Sustain. Energy Rev.*, **136** (*October 2020*) 110354 (2021). doi:10.1016/j.rser.2020.110354.
14) C. Power, A. McNabola, and P. Coughlan, "A parametric experimental investigation of the operating conditions of gravitational vortex hydropower (gvhp)," *J. Clean Energy Technol.*, **4** (*2*) 112–119 (2015). doi:10.7763/jocet.2016.v4.263.
15) "ANSYS FLUENT Theory Guide," 2011.
16) D.D.D.P. Tjahjana, I. Yaningsih, B.Y.L. Imama, and A.R. Prabowo, "Aerodynamic performance enhancement of wing body micro uav employing blended winglet configuration," *Evergreen*, **8** (*4*) 799–811 (2021). doi:10.5109/4742122.
17) M.M. Takeyeldein, T.M. Lazim, N.A.R. Nik Mohd, I.S. Ishak, and E.A. Ali, "Wind turbine design using thin airfoil sd2030," *Evergreen*, **6** (*2*) 114–123 (2019). doi:10.5109/2321003.
18) S.N. Ashwindran, A.A. Azizuddin, and A.N. Oumer, "Study of √2 conjecture in the construction of drag induced wind turbine blade morphology," *Evergreen*, **8** (*3*) 574–585 (2021). doi:10.5109/4491649.
19) Warjito, D. Adanta, Budiarso, S.B.S. Nasution, and M.A.F. Kurnianto, "The effect of blade height and inlet height in a straight-blade undershot waterwheel turbine by computational method," *CFD Lett.*, **11** (*12*) 66–73 (2019).
20) A.M. Halawa, B. Elhadidi, and S. Yoshida, "Aerodynamic performance enhancement using active flow control on du96-w-180 wind turbine airfoil," *Evergreen*, **5** (*1*) 16–24 (2018). doi:10.5109/1929723.
21) N.I. Ismail, H. Sharudin, M.M. Mahadzir, Z.M. Ali, A.A. Shariffuddin, and N.I. Kamel, "Computational aerodynamics study on neo-ptero micro unmanned aerial vehicle," *Evergreen*, **8** (*2*) 438–444 (2021). doi:10.5109/4480726.
22) O.M.A.M. Ibrahim, and S. Yoshida, "Experimental and numerical studies of a horizontal axis wind turbine performance over a steep 2d hill," *Evergreen*, **5** (*3*) 12–21 (2018). doi:10.5109/1957496.
23) H. Cheng, L. Zhou, Q. Liang, Z. Guan, D. Liu, Z. Wang, and W. Kang, "A method of evaluating the vortex rope strength in draft tube of francis turbine," *Renew. Energy*, **152** 770–780 (2020). doi:10.1016/j.renene.2020.01.020.
24) D. Adanta, Budiarso, Warjito, and A.I. Siswantara, "Assessment of turbulence modelling for numerical simulations into pico hydro turbine," *J. Adv. Res. Fluid Mech. Therm. Sci.*, **45** 21–31 (2018).
25) T.S. Phillips, and C.J. Roy, "Richardson extrapolation-based discretization uncertainty estimation for computational fluid dynamics," *J. Fluids Eng.*, **136** (*12*) 121401 (2014). doi:10.1115/1.4027353.