INTI JOURNAL | eISSN:2600-7320
Vol.2020:16

Numerical Simulation of Micro Gravitational Hydro-turbine for Alternative Renewable Energy Resources in Rural Area

Muhammad Fikri Musthapa[1], Azfarizal Mukhtar[2*], Mohamad Fariz Mohamed Nasir[3], Ahmad Faiz Tharima[4]

[1]Department of Mechanical Engineering, International Islamic University Malaysia, Jalan Gombak, 53100 Kuala Lumpur, Malaysia
[2]College of Engineering, Universiti Tenaga Nasional (UNITEN), Jalan IKRAM-UNITEN, 43000 Kajang, Selangor, Malaysia
[3]INTI International University, Persiaran Perdana BBN, Putra Nilai, 71800 Nilai, Negeri Sembilan, Malaysia
[4]Research and Planning Division, Fire and Rescue Department of Malaysia, PUSPEK, Lengkuk Teknologi, Kawasan Perindustrian Enstek, 71800 Nilai Negeri Sembilan, Malaysia

[*]**Email:** azfarizal@uniten.edu.my

Abstract

Hydropower is one of the most reliable and efficient sources for the clean generation of renewable energy. The increase of energy demand in developing countries has caused the extensive grow of hydropower development. In this light, the usage of low head hydro vortex turbine has been considered as a solution for the shortage of energy supply in rural areas. To alleviate the shortage of energy supply, it is essential to fabricate a Micro Gravitational Hydro-turbine which allows the conversion of energy in a moving fluid to rotational energy through the use of low head and low flow rate with a relatively simple structure. Thus, we attempt to analyse the maximum output power of the suggested model. The model was numerically simulated using commercial CFD software. Different geometrical basin designs are being developed as well as the simulations are varied with different flow rate parameter. The findings indicated that the maximum output generated via the conical basin is 630.01 W while the cylindrical basin is 27.47 W. It is shown that the conical basin design could increase power production and able to operate efficiently at a different flow rate. In general, the current work could be used as a replacement for the traditional small water turbine design in Malaysia.

Keywords

Computational Fluid Dynamics, Micro Gravitational Water Vortex, Rural Electrification, Geometrical Conditions.

International Conference on Innovation and Technopreneurship 2020
Submission: 8 July 2020; Acceptance: 3 August 2020

Introduction

Globally, about 32.9% consumed energy was from fossil fuels (Rahman et al., 2017). According to Abas et al., (2015) fossil fuels contributed about 65% of greenhouse gas emissions. In order to overcome these environmental issues, human has shown their interest in developing a renewable energy source especially hydropower. It is an economic and clean energy system allowing for the conversion of the low-head potential energy into kinetic energy to drive power turbines using a gravitation vortex pool (Rahman et al., 2017). Gravitational water vortex turbine is an ultra-low head turbine which can operate in as low head range of 0.7-2.0 m (Dhakal et al., 2015). The flowing water passes through into a basin which forms a vortex and hits the small-scale turbine forcing it to rotate thus converted into mechanical energy. The rotating shaft transferred the energy to generator and converted to electrical energy. The flowing water then will exit the basin through a hole at the bottom of the basin.

There are several prominent works related to Micro Gravitational Hydro-turbine. Dhakal et al., (2015) focused on the effect of runner position on cylinder and conical geometry basin structure of Gravitational water vortex power plant. Their findings found that the output power and efficiency is maximum in conical basin compared to the cylindrical basin for all similar inlet and outlet condition with maximum power extraction at runner position 65-75% of total height of basin from top position. Kumar et al., (2008) focused on developing low-head water turbine for free-flowing streams suitable for micro hydropower. It is found that the basin geometry depends on the discharge supplied. With the enough flow condition, vortex minimum diameter is at bottom level and is always smaller the exit hole. From the literature above, it appeared that the maximum power extraction is dependent on the geometrical aspects. Thus, in this study we attempt to analyze the suitability of turbine blade with different basin such as conical and cylindrical basin. The commercial CFD code ANSYS Fluent 2019 and SolidWork 2019 was used in this study to develop a simulation model for evaluating the performance of the basin and to investigate the configuration of the Micro Gravitational Hydro-turbine system. All analyses were performed by using the river flow data condition and dynamic state in Malaysia (Gasim et al., 2013).

Methodology

CFD Models Development

As shown in Figure 1, the design of turbine has a height of 53 cm and the diameter is 100 cm. While for the blades curvature angle is 15° and the number of blades used to find the efficiency were 5 blades. The selection of curved blades is based on three different design (i.e. twisted, straight, and curved blade structure). Based on the result of the simulation, it showed that the curved blade profile to be the most efficient profile, with a peak efficiency of 82%, compared to 46% for the straight blade runner and 63% for the twisted blade.

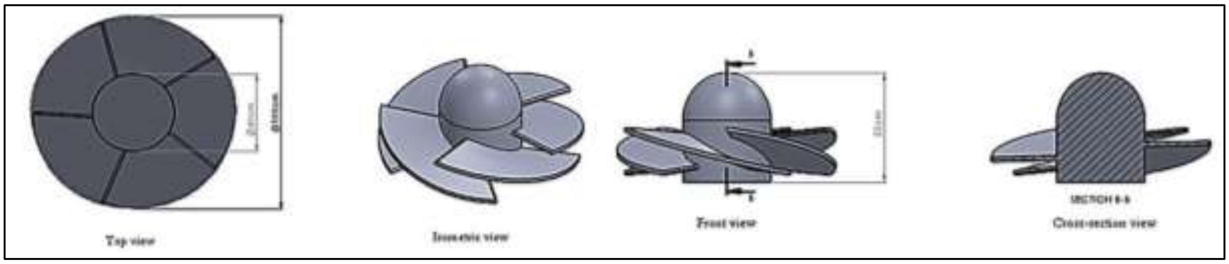


Figure 1: Design geometry of turbine

In this study, conical and cylindrical basin structure were applied in order to determine which of the structure that has a better efficiency and output power. Both basin's design has different geometrical parameters which they varied on its diameter of inlet, outlet and the wall body. The turbine placed inside both structures is the same (see Figure 2). The distance between the blades and the wall body also being kept less than 5 cm for both structures. The build-in turbine is placed in the middle of the 2m height for both basins.

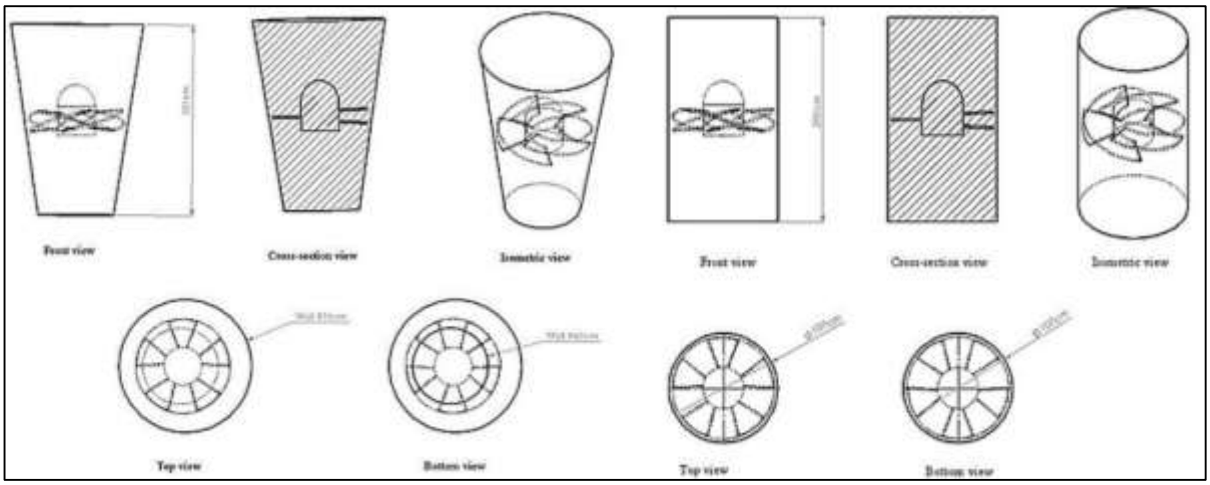


Figure 2: Design geometry of the conical and cylindrical basin

Governing equation (mass, momentum and energy)

The river flow through the basin was considered as steady, incompressible and turbulent. The continuity equation and Navier-Stokes equation in cylindrical coordinates is used to describe the flow (Dhakal et al., 2015). The general governing equations for the CFD used in this study are presented below:

$$\frac{\partial V_r}{\partial r} + \frac{\partial V_z}{\partial z} + \frac{V_r}{r} = 0 \tag{1}$$

$$V_r \frac{\partial V_\theta}{\partial r} + V_z \frac{\partial V_\theta}{\partial z} - \frac{V_r V_\theta}{r} = v\left(\frac{\partial^2 V_\theta}{\partial r^2} + \frac{\partial V_\theta}{r\,\partial r} - \frac{V_\theta}{r^2} + \frac{\partial^2 V_\theta}{\partial z^2}\right) \tag{2}$$

$$V_r \frac{\partial V_r}{\partial r} + V_z \frac{\partial V_r}{\partial z} - \frac{V_\theta^2}{r} + \frac{\partial p}{p\,\partial r} = v\left(\frac{\partial^2 V_r}{\partial r^2} + \frac{\partial V_r}{r\,\partial r} - \frac{V_r}{r^2} + \frac{\partial^2 V_r}{\partial z^2}\right) \tag{3}$$

$$V_r \frac{\partial V_z}{\partial r} + V_z \frac{\partial V_z}{\partial z} + \frac{\partial p}{p\, \partial z} = g + v\left(\frac{\partial^2 V_z}{\partial r^2} + \frac{\partial V_z}{r\, \partial r} + \frac{\partial^2 V_r}{\partial z^2}\right) \tag{4}$$

Where V_θ, V_r, V_z are tangential, radial and axial velocity components respectively, ρ is fluid density, g is gravitational acceleration and V is kinematic viscosity. Due to the complexity of the equation, it is extremely difficult to get the analytical solution directly (Dhakal et al., 2015).

Numerical tool, boundary conditions and computational mesh

This process involves quantification errors. We are estimating the discretization error of the model. It was quantified by schematic refinement space and time meshes. This study targeted the two or three levels of mesh refinement. The design that was modelled using SolidWork 2019 is then imported to ANSYS Fluent 2019 for meshing and simulation purposes. After modelling, the meshing of the proposed model was being conducted. For optimisation of the meshing process, the denser mesh was taken near the wall and put at the edge of the turbine's blade. Later the grid was refined and the uniformly dense mesh was generated. This process was implemented to obtain the maximum vortex velocity in the region between the turbine and the wall. The boundary condition and meshing for both basins are shown in Figure 3:

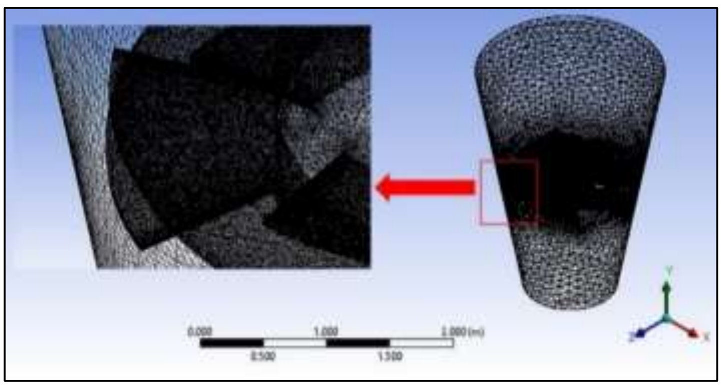


Figure 3: Mesh generation on the blade design

The maximum quality of mesh skewness for conical basin is 0.79934 and for cylindrical basin is 0.79963. Based on the table for the spectrum of skewness metric (see Table 1), each mesh is in good condition to be further simulated In term of the mesh orthogonal quality, the maximum mesh metric for the conical basin is 0.99542 and for the cylindrical basin is 0.99565, Moreover, based on the spectrum of mesh orthogonality metric table, each design is in excellent condition to be simulated (see Table 2).

Table 1: The spectrum of skewness metric

Excellent	**Very Good**	**Good**	**Acceptable**	**Silver**	**Degenerate**
0-0.24	0.25-0.50	0.50-0.80	0.80-0.94	0.95-0.97	0.98-1.00

Table 2: The spectrum of mesh orthogonality metric

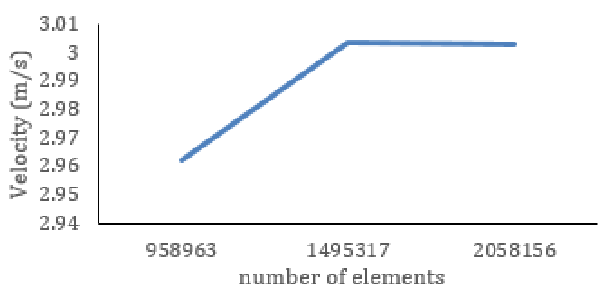
Unacceptable	**Bad**	**Acceptable**	**Good**	**Very Good**	**Excellent**
0-0.001	0.002-0.14	0.15-0.20	0.21-0.69	0.70-0.95	0.95-1.00

The 3D model was discretized using curvature and proximity element with tetrahedral grids where clustered near critical area of the domain water flow and turbine. The $k - \epsilon$ was applied to model the flow turbulence as it can capture general flow at reasonable accuracy (Mukhtar et al., 2019). SIMPLE was used for pressure-velocity coupling. Also, the use of higher order convective schemes are essential to ensure flow accuracy (Mukhtar et al., 2018). All governing equations were discretized using second-order upwind schemes for both accuracy and stability purposes. The convergence criteria were below 10^{-4}. The type of boundary condition used at the inlet is set as velocity-inlet and at the outlet as pressure-outlet.

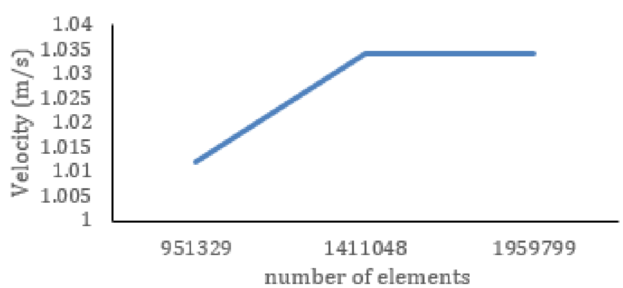
Results and Discussion

Grid Independent Test (GIT)

The simulation was conducted to investigate the performance and velocity output of different basin geometry with a built-in turbine. The simulation was run at steady flow with condition of no-slip at the wall and outflow condition at the outlet. The working fluid, water is assumed as an incompressible fluid with density of $998.2 kg/m^3$and viscosity of $0.001003 kg/ms$. The initial velocity at the inlet of fluid flow was set to be 0.5 m/s and the outlet was an outflow with wall of the fluid flow domain stationary. The upper surface was subjected to atmospheric pressure. Three different sets of number of elements is used to simulate to determine the value of variables. It is preferred that the grid refinement factor, $r = h_{coarse}/h_{fine}$ is greater that the ratio of 1.3 (Celik et al., 2008). The number of elements in the final computational domain used for the simulation of conical basin was 2057385 (see Figure 4(b)). While for the cylindrical basin, the number of elements used is 1959799 (see Figure 4(b)). Thus, the results can be regarded as grid-independent when the number of elements reaches the respective value. Also, it indicates that this model was successfully reliable for the next sequences of the variables that need to be investigated.



(a)



(b)

Figure 4: Graph of grid convergence for conical basin (a) and cylindrical basin (b)

Velocity contour

The initial velocity at the inlet is set at three different value which is 0.5 m/s, 1.25 m/s, and 2.0 m/s. These values were derived from the result of previous research and experiment conducted in the different river across Malaysia (Gasim et al., 2013; Saupi et al., 2018; Seyam & Othman, 2014). The flow simulation on both basins was simulated and the streamline and velocity contour can be seen from the Figure 5. From the velocity contour presented, it can be observed that the average velocity of flow in the conical basin is much better than the cylindrical basin. The velocity changed significantly after the flow passing through the turbine in the conical basin compared to the cylindrical basin. The decrease of flow area in conical basin results in greater vortex shedding in the region between the wall and the blade of the turbine and this subsequently leads to higher velocity.

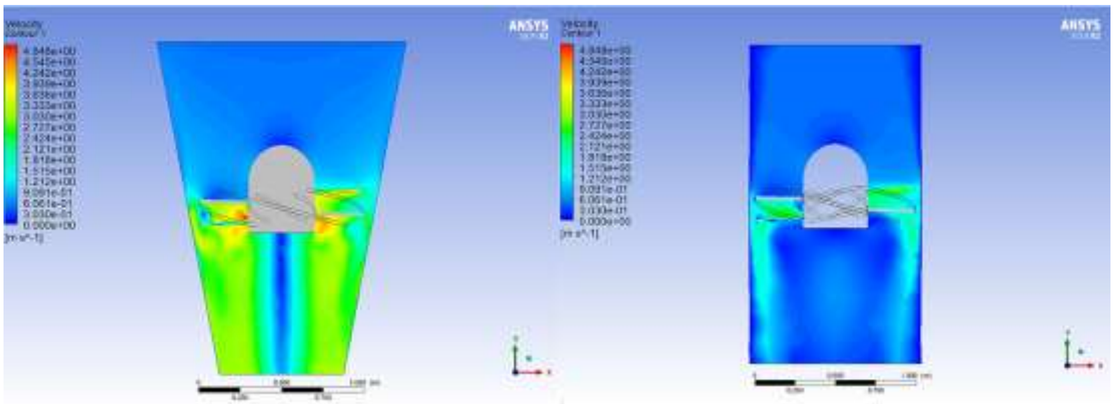


Figure 5: The contour of conical and cylindrical geometry structure

Power generation

Based on the Table 3, the output power of cylindrical and conical basin was found to be increasing as the input velocity increased. The same goes to other parameters as it increases with the increase of the input velocity. It can be seen that, the power generated from the conical basin shows a significant difference with power generated from the cylindrical basin, which indicates that the geometry structure of the conical is better to be used as the basin of hydropower plant. As the shape of the converging cone's diameter from the inlet to the outlet decrease, the vortex strength increases hence resulting an increase in power production.

Table 3: Simulation result of cylindrical and conical basin

	Input velocity (m/s)	Output velocity (m/s)	Input pressure (KPa)	Output pressure (KPa)	Mass flow rate (Kg/s)	Power generated (W)
Cylindrical basin	0.5	0.8253	2.249	0.00695	1.278	0.435
	1.25	2.047	13.97	0.051	3.202	6.708
	2.0	3.275	35.66	0.1297	5.124	27.47
Conical basin	0.5	2.497	8.686	0.2896	3.119	9.723
	1.25	6.299	54.66	2.034	7.807	154.88
	2.0	10.04	139.8	5.096	12.5	630.01

Conclusion

The numerical result of this study shows that the maximum power output generated is higher in conical basin (630.01 W) compared to the cylindrical basin (27.47 W) for all conditions. Conical basin is an excellent option for power generation in remote areas where power supply from the grid line is hard to obtain. There are a few recommendations that can be made in for the future studies which is to improve the design of the blade in terms of its number and the angle of the blade, which is believed can provide greater output for power generation.

References

Abas, N., Kalair, A., & Khan, N. (2015). Review of fossil fuels and future energy technologies. *Futures*, *69*, 31–49.

Celik, I. B., Ghia, U., Roache, P. J., Freitas, C. J., Coleman, H., & Raad, P. E. (2008). Procedure for estimation and reporting of uncertainty due to discretization in CFD applications. *Journal of Fluids Engineering, Transactions of the ASME*.

Dhakal, S., Timilsina, A. B., Dhakal, R., Fuyal, D., Bajracharya, T. R., Pandit, H. P., … Nakarmi, A. M. (2015). Comparison of cylindrical and conical basins with optimum position of runner: Gravitational water vortex power plant. *Renewable and Sustainable Energy Reviews*, *48*(August), 662–669.

Gasim, M. B., Toriman, M. E., Idris, M., Lun, P. I., Kamarudin, M. K. A., Nor Azlina, A. A., Mastura, S. A. S. (2013). River flow conditions and dynamic state analysis of Pahang river. *American Journal of Applied Sciences*, *10*(1), 42–57.

Kumar Jha, A., Subedi, D. U., & Timilsina, A. B. (2008). Assessment of Gravitational Water Vortex Hydropower Plant in Nepal. *International Research Journal of Engineering and Technology*.

Mohammed, N., Danapalasingam, K. A., & Majed, A. (2018). Design, control and monitoring of an offline mobile battery energy storage system for a typical malaysian household load using PLC. *International Journal of Power Electronics and Drive Systems*, *9*(1), 180–188.

Mukhtar, A., Ng, K. C., Yusoff, M. Z., Tey, W. Y., & Tan, L. K. (2019). Performance assessment of passive heating and cooling techniques for underground shelter in equatorial climate. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*.

Mukhtar, A., Yusoff, M. Z., Khai Ching, N., & Mohamed Nasir, M. F. (2018). Application of Box-Behnken Design with Response Surface to Optimize Ventilation System in Underground Shelter. *Journal of Advanced Research in Fluid Mechanics and Thermal Sciences*.

Rahman, M. M., Tan, J. H., Fadzlita, M. T., & Wan Khairul Muzammil, A. R. (2017). A Review on the Development of Gravitational Water Vortex Power Plant as Alternative Renewable Energy Resources. *IOP Conference Series: Materials Science and Engineering*.

Saupi, A. F. M., Mailah, N. F., Radzi, M. A. M., Mohamad, K. B., Ahmad, S. Z., & Soh, A. C. (2018). An illustrated guide to estimation ofwater velocity in unregulated river for hydrokinetic performance analysis studies in East Malaysia. *Water (Switzerland)*, *10*(10).

Seyam, M., & Othman, F. (2014). The Influence of Accurate Lag Time Estimation on the Performance of Stream Flow Data-driven Based Models. *Water Resources Management*, *28*(9), 2583–2597.