Proceedings of IOE Graduate Conference, 2019-Winter
Peer Reviewed
Year: 2019 Month: December Volume: 7
ISSN: 2350-8914 (Online), 2350-8906 (Print)

Design and Analysis of Gravitational Water Vortex Basin and Runner

Nipesh Regmi [a], Hari Dura [b], Shree Raj Shakya [c]

[a, b, c] *Department of Mechanical Engineering, Pulchowk Campus, Institute of Engineering, Tribhuvan University, Nepal*
Corresponding Email: [a] regmi.nipesh@pcampus.edu.np, [b] duraharis@pcampus.edu.np, [c] shreerajshakya@ioe.edu.np

Abstract
Being new and not well developed technology, Gravitational Water Vortex Power Plant (GWVPP) needs optimization either in its structure or runner for enhancing power extraction from low pressure head. The study is focused on Computational Fluid Dynamics (CFD) analysis of basin and existing runner for GWVPP. The tapered ratios of the basin were varied with and without changing exit hole and corresponding variations in flow parameters were observed. The CFD analysis of the existing runner for baseline basin geometry was performed and found to have a maximum efficiency of 7.93%.

Keywords
GWVPP – Runner – CFD

1. Introduction

Electricity supply in rural areas of Nepal remains one of the most challenging issues due to high infrastructure cost and losses, lack of road, and poor returns compounded by constrained government budgets resulting from geographical profile of Nepal [1]. So, there is a need for most cost-effective off-grid renewable energy system for enhancing rural electrification.

The hydropower plants with higher water pressure are economically feasible whereas mini and micro hydro power plant (100-1000 kW) with lower water pressure (0.70 m up to 2.00 m) are not economical with conventional turbines [2]. Francis and Reaction turbines are not suitable when hydraulic head is lower than 2 m [3]. Kaplan and Pelton turbines can be scaled down for smaller hydro power plant but typically limited to hydraulic heads greater than 3m [4]. Hence, an alternative option to harvest energy from low head water resources is essential.

Gravitational Water Vortex Power Plant (GWVPP), under the category of micro-hydro scheme, has been considered as one of the best technologies for harvesting electricity from low hydraulic head (0.7 m to 2 m) using the energy available in vortex flow. For the very first time, this technology was invented by Austrian Engineer, Franz Zotloterer while he was looking for an efficient way to aerate the water in 2007 [5, 6]. In GWVPP, water is channelled through a large, straight inlet, and then passed tangentially into a round basin, forming a powerful vortex (whirlpool). The basin has a hole at the center bottom that acts as an outlet. A vertical axis turbine is placed at the centre of vortex which withdraws rotational energy from gravitational vortex. The turbine does not work on pressure differential but on the dynamic force of the vortex. This type of power generation system is suitable in areas with low velocity water flow such as small rivers and existing agricultural irrigation canals.

1.1 Problem Statement

It has only been a few years since the development of Gravitational Water Vortex Power Plant system started. It is a new and not well-developed technology for power extraction from low pressure water energy sources. Basin optimization has been very popular amongst the researchers following its development to achieve maximum power output. Limited literature are available on design, fabrication and physical geometry of vortex turbine and generator [7].

The past studies had found that the parameters including turbines, inlet height and flow rates have significant effects on the efficiency of vortex power plant. Turbine, being most important component of GWVPP system needs to be optimized for maximizing power output [6, 8].

Experimental and more often hit and trial methods are widely used for designing runner profile. Some latest studies however have tried in designing runner profile via understanding the flow pattern inside basin. But their studies assumed basin top surface as wall during CFD analysis. These assumptions limit the analysis of basin in their studies as a completely closed channel analysis thereby deviating from actual conditions. As basin top is open to atmosphere, it is therefore necessary to analyze the basin with top surface open so that an interaction with atmosphere can be taken into consideration.

This study is focused on CFD analysis of basin and existing runner for baseline basin geometry ensuring that the top surface of basin is open to atmosphere resembling to actual conditions.

1.2 Related Works

The study of [9] recommended for helical turbine steps with hydrofoil profile. The study further showed that, due to large variation along the peripheral velocity, there is a need for twisting blade.

The experimental study by [8] found that maximum energy can be extracted when the runner is placed near outlet and these results were validated with theories.

An experiment was performed by [10] and found that the conical basin has greater vortex strength than that of cylindrical basin. Greater efficiency was found at the bottom most position and agreed with the results of [8]. Smaller number of blades produces greater efficiency while increment in blade radius decreases the efficiency of turbine. The maximum efficiency recorded was 25.36%.

Similarly, experimental study of [6] found that the optimal position of turbine is 65% to 75% of GWVPP's height and verified that the conical basin has higher overall power output with a maximum efficiency of 36.84%.

The efficiency was found to increase with increase in number of blades in the experimental study performed by [11]. This finding however goes against the results of [10]. This indicate that there might be an optimal number of blades for turbine. The maximum efficiency of the vortex power plant was found to be 15.1%.

A free vortex power generation system model was designed and tested by [2] under different water pressure and turbine parameters at Material and Mineral Research Unit Laboratories, Faculty of Engineering, University Malaysia Sabah. The study concluded that the tangential velocity at the vortex free surface was highest for 0.12 m water head and maximum efficiency of about 43% was achieved with three blades and 0.027 m turbine outer diameter. Also, the study found that in case of vortex power generation system the maximum hydraulic efficiency was recorded when the turbine rotating speed was half of the vortex tangential velocity. A very week relation is observed between turbine speed and hydraulic efficiency.

The study on runner of the GWVPP with conical basin was performed by [12] to improve its efficiency. The study takes into consideration of different runner parameters like impact angle, inlet and outlet blade angle, number of blades, taper angle of blades, surface area of blades, blade profile etc. The study was carried out with the assumption that only impulse action of the water is responsible for the rotation of the turbine. Twenty different runners have been developed computationally and tested experimentally. The impact angle, inlet and outlet blade angles, number of blades, tapered angle for conical profiles were optimized. A linear relationship of power output with flow rate were formulated both computationally and experimentally.

The study of [13] focused on the optimization of the runner to improve the efficiency of the GWVPP. CFD analysis was carried out on three different designs of runner with straight, twisted and curved blade profile. The ANSYS CFX was used to analyze the fluid flow through the channel, basin, turbine hub and blade whose results were used for evaluating the efficiency of each runner. The study found that the curved blade profile had higher efficiency with peak efficiency 82.4% compared to 46.31% for the straight blade runner and 63.54% for twisted blade profile. Experimental analysis showed the peak efficiency point of 71.01% at 0.5 m head for curved profile runner. The assumptions for the study was similar to that of [12] i.e., only the impulse action of flow was considered.

A review paper by [7] presented with the past literatures related to GWVPP suggested that there is a need for optimized shape of basin and blade profile to achieve maximum efficiency.

2. Methodology

2.1 Analysis of Conical Basin

2.1.1 Baseline Geometry

The conical basin as suggested by the study of [6, 10] was selected during this study and is shown in Figure 1.

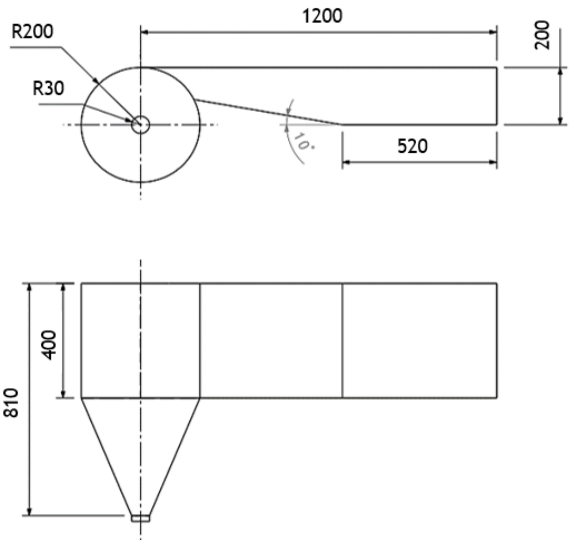

Figure 1: Selected basin with major dimensions

The basin has inlet canal length of 1200 mm and inlet canal height of 400 mm. The basin was modeled in CATIA V5R20 and then imported to ANSYS CFX for CFD analysis.

2.1.2 Governing Equations

The governing equations include the following conservation laws:

Continuity Equation

$$\frac{\partial \rho}{\partial t} + div(\rho \mathbf{u}) = 0 \tag{1}$$

Momentum Equations

$$\rho \frac{Du}{Dt} = \frac{\partial(-p+\tau_{xx})}{\partial x} + \frac{\partial \tau_{yx}}{\partial y} + \frac{\partial \tau_{zx}}{\partial z} + S_{Mx} \tag{2}$$

$$\rho \frac{Dv}{Dt} = \frac{\partial \tau_{xy}}{\partial x} + \frac{\partial(-p+\tau_{yy})}{\partial y} + \frac{\partial \tau_{zy}}{\partial z} + S_{My} \tag{3}$$

$$\rho \frac{Dw}{Dt} = \frac{\partial \tau_{xz}}{\partial x} + \frac{\partial \tau_{yz}}{\partial y} + \frac{\partial(-p+\tau_{zz})}{\partial z} + S_{Mz} \tag{4}$$

where, u, v & w are the velocity components along x, y & z axis respectively, p is a compressive stress, τ is a tensile stress, S_M is the body forces and $\frac{D}{Dt}$ is total derivative.

2.1.3 Physics Setup for Conical Basin

The analysis was based on multiphase Eulerian fluid approach where two fluids, air and water, occupy same domain. The temperature of the domain was set to 25°C and the reference pressure is set to 1 atm. The buoyancy reference density was set to the density of air at 25°C which is equal to 1.2 kg/m^3.

The built-in fluid models were used for both fluids. In the production of air-core vortices, the buoyancy plays a vital role, therefore, buoyancy was also included in the analysis. Turbulence model used was RNG k-ε [6, 12] as it is more suitable for the simulation of air core vortices due to great curving streamline flows [14]. The boundary conditions used for analysis of conical basin is shown in Figure 2.

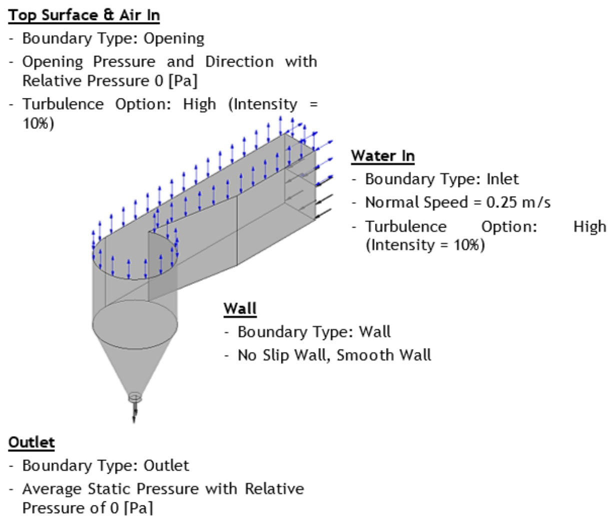

Figure 2: Boundary conditions for basin analysis

The model included advection scheme as "High Resolution" and turbulence numeric was "High Resolution". Timescale was set to "Physical Timescale". The number of iterations was set to 100000 with a residual target of 0.0001.

2.1.4 Physics Setup for Conical Basin with Runner

The setup was similar to basin analysis. The buoyancy was also included in the analysis.The RNG k-ε turbulence model was used.The basin domain was fluid domain and stationary domain. The runner being the rotating component in the assembly, the runner domain was rotating fluid domain and its rotational speed was input parameter for the analysis. The water inlet velocity was fixed at 0.25 m/s. The boundary conditions used for analysis of conical basin with runner is shown in Figure 3.

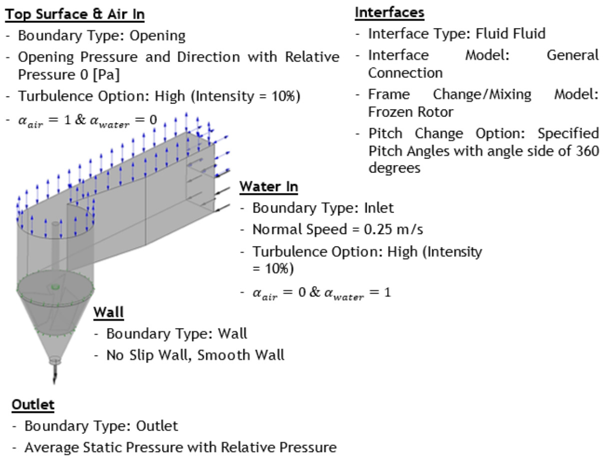

Figure 3: Boundary conditions for runner analysis

2.2 Mesh Independence Study

Prior to performing final CFD simulations for a problem, it is necessary to carry out mesh independent study to make sure that the solution is independent of the mesh resolutions. This allow us to reduce the mesh size without losing accuracy and, therefore, reducing the computational effort.

Seven different mesh sizes with the number of elements varying from 112954 to 1039630 were used for mesh independence study. Torque acting on the wall of conical basin was observed. The graph was plotted between number of elements and torque as shown in Figure 4.

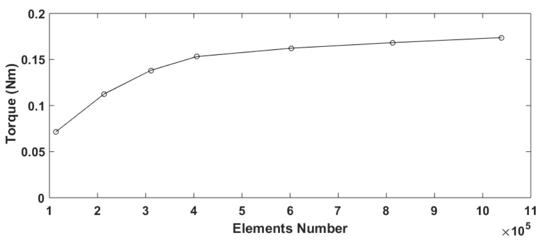

Figure 4: Mesh independent study for basin

As depicted in figure, when the number of elements were higher than 813080, the torque on the conical basin wall became nearly unchanged and remained within 3.15% of value than when the elements number was 1039630. Hence total 813080 was used for all subsequent computations.

2.3 Canal Length Independent Study

Five different simulations were carried out by varying the length of canal in the range from 880 mm to 3000 mm. The torque acting on the conical basin wall was considered as key parameters. The mesh density was used in proportion for longer canal length as obtained from mesh independent study. The graph was plotted between canal length and the torque as shown in Figure 5.

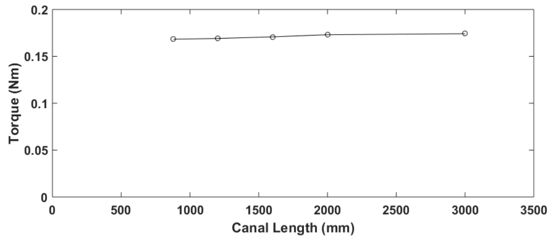

Figure 5: Canal length independent study for basin

The figure showed that, increasing the length of canal has no significant effect on the torque developed. So for the entire study, the canal length was chosen to 1200 mm as no significant changes were observed in the developed torque.

2.4 Effect of Air Domain Height

The inlet canal heights for the basin were varied keeping water portion height constant at 200 mm. The air portion heights were increased from 0 mm to 400 mm and then corresponding torque developed on the conical portion of basin was observed. The graph was plotted between air domain height and the torque as shown in Figure 6.

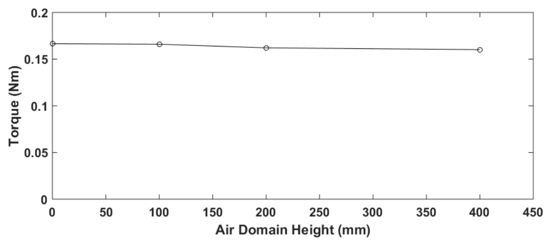

Figure 6: Air domain height independence study for basin

It was observed that changes in developed torque when the air domain heights were 200 mm and 400 mm were almost constant. Hence, air domain height of 200 mm was considered for further study.

3. Results and Discussions

3.1 Effect of Tapered Ratio with Constant Outlet Area

The diameter of the exit hole of the basin was kept constant of diameter 60 mm while the tapered ratio of the basin was varied by varying cone bottom diameter in steps of 60 mm, 100 mm and 150 mm as shown in Figure 7.

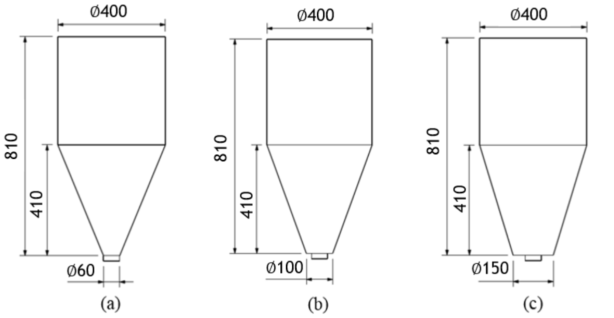

Figure 7: Basin with fixed outlet diameter and bottom cone diameter of (a) 60 mm (b) 100 mm & (c) 150 mm

Air Volume Fraction

Air volume fraction was observed in a longitudinal plane that lied along the center of the inlet canal length. The air volume fraction for different bottom cone diameters are shown in Figure 8.

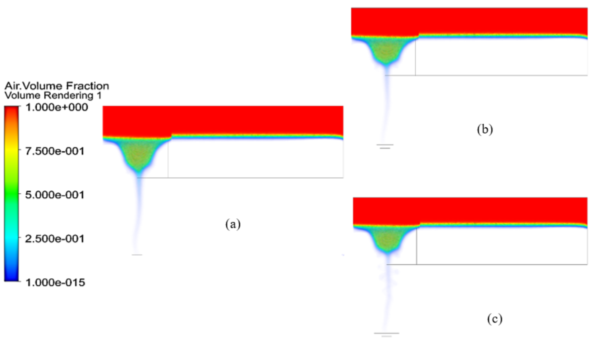

Figure 8: Variations in air volume fraction for fixed outlet diameter and bottom cone diameter of (a) 60 mm (b) 100 mm & (c) 150 mm

It was observed that, a conical region of the air was formed. This may be due to the presence of both fluid i.e., air and water in the same domain. However, with increase in bottom cone diameter, a slight reduction in the depression of air region were observed. The air region throughout the conical basin started to vanish with increasing bottom cone diameter.

Water Volume Fraction

Water volume fraction was observed in a transverse plane that passes through the central axis of the conical basin. The water volume fraction for different bottom cone diameters are shown in Figure 9.

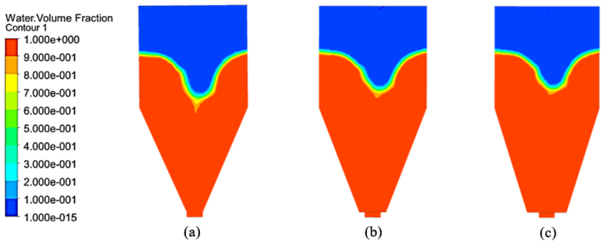

Figure 9: Variations in water volume fractions for fixed outlet diameter and bottom cone diameter of (a) 60 mm (b) 100 mm & (c) 150 mm

The reduction in the depression of air region with increased bottom cone diameter resulted in increasing water region throughout the conical basin portion.

3.2 Effect of Tapered Ratio with Varying Outlet Area

The diameters of the basin exit hole were varied as 50 mm, 60 mm, 70 mm and 80 mm as shown in Figure 10.

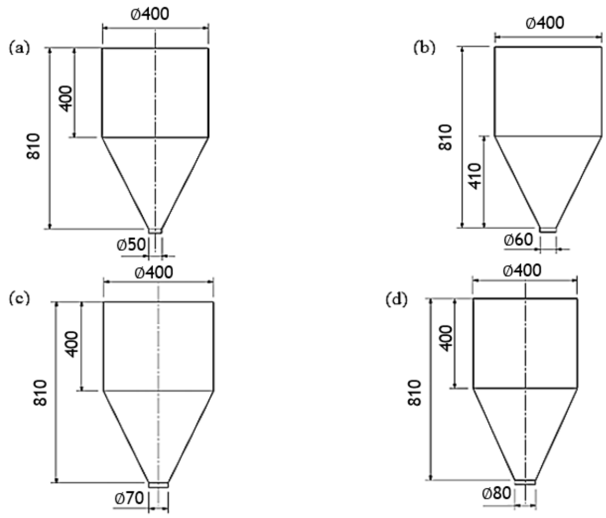

Figure 10: Basin with varying outlet diameter of (a) 50 mm (b) 60 mm (c) 70 mm & (d) 80 mm

Air Volume Fraction

The air volume fraction for different exit hole diameters are shown in Figure 11.

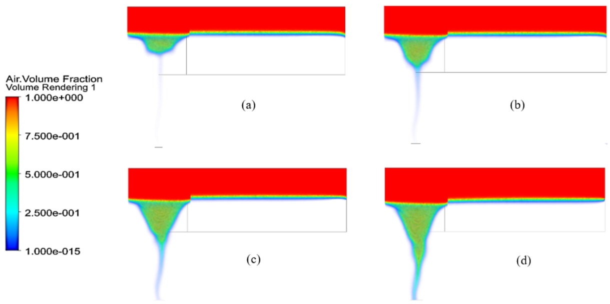

Figure 11: Variations in air volume fraction at exit hole diameter (a) 50 mm (b) 60 mm (c) 70 mm & (d) 80 mm

Here, it was observed that, increasing the outlet diameter of the basin resulted in the formation of dense conical region of air and extending upto the exit hole of the basin.

Water Volume Fraction

The water volume fraction for different exit hole diameters are shown in Figure 12.

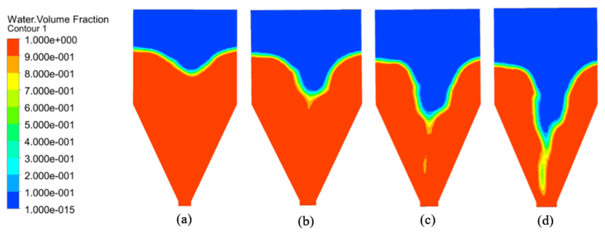

Figure 12: Variations in water volume fractions at exit hole diameter (a) 50 mm (b) 60 mm (c) 70 mm & (d) 80 mm

As the exit hole diameter was increased, air region depression started to dominate water region throughout conical basin thereby reducing the water portion inside the basin. This may be be due to lower velocity of water through the inlet canal and resembles to pouring of water in the vessel.

3.3 Performance Evaluation of Existing Runner

The CFD analysis of the existing runner for baseline geometry was performed. The speed of the runner was varied from 55 rpm to 76 rpm and the corresponding torque and power developed by the runner blades were obtained as shown in Figure 13 and Figure 14 respectively.

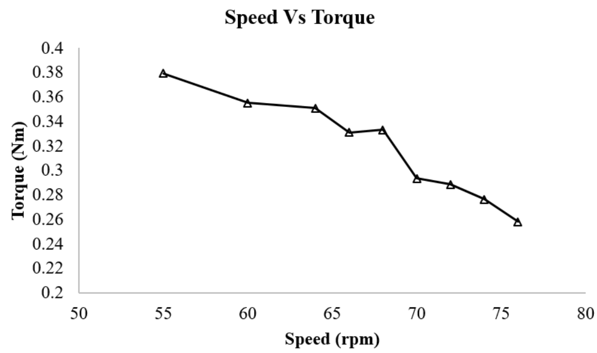

Figure 13: Variations of torque with speed of runner

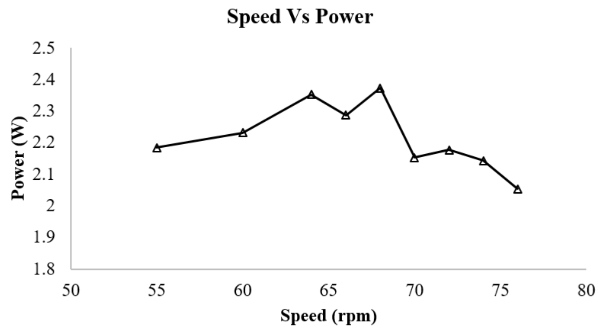

Figure 14: Variations of power developed with speed of runner

As depicted in the figure, the maximum power of 2.373 watt was developed by runner at 68 rpm.

The formula for determination of input power is given by

$$P = \rho g H Q$$

Here,

P = Input power ,

ρ = Density of water = 1000 kg/m^3,

g = Acceleration due to gravity = 9.8 m/s^2,

H = Head upto runner mean height = 0.305 m &

Q = Flow rate = 0.01 m^3/s.

The input power was calculated to be 29.9205 watt. The efficiency of the runner was then calculated for different runner speed as shown in Figure 15.

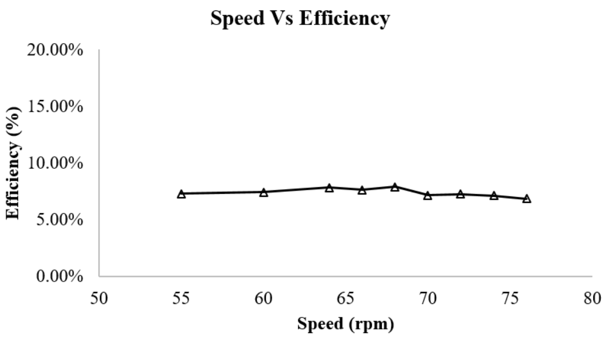

Figure 15: Efficiency of runner at different runner speed

As can be seen in figure, the maximum efficiency of 7.93% was observed when the runner speed was 68 rpm.

4. Conclusions

The GWVPP basin was analyzed by varying tapered ratios with and without changing exit hole diameter and corresponding flow parameters were observed. The CFD analysis of the existing runner for baseline basin geometry was performed and found a maximum efficiency of 7.93% with maximum output power of 2.373 watt at 68 rpm.

5. Recommendations

The number of blades in the study has been limited to three which can be changed for achieving more efficiency. The runner can be designed by varying different basin structure and the comparative study on the efficiency of the runner can be done. Further, the results can be validated from experimental analysis of the runner.

Acknowledgments

The authors would like to acknowledge and extend heartfelt gratitude to Mr. Sudeep Bhattarai for his continuous help and suggestions to keep the study in track. The authors are grateful to Institute of Engineering [IOE], Department of Mechanical Engineering, Pulchowk Campus for providing access to computer workstations.

References

[1] Sudeshna Ghosh Banerjee, Avjeet Singh, and Hussain Samad. Power and people: The benefits of renewable energy in nepal. Technical report, The World Bank, Washington, DC, 2011.

[2] Md. Mizanur Rahman, Tan Jian Hong, Raymond Tang, Ling Leh Sung, and Fadzlita Binti Mohd Tamiri. Experimental study the effects of water pressure and turbine blade lengths & numbers on the model free vortex power generation system. *International Journal of Current Trends in Engineering & Research*, 2(9):13–17, 2016.

[3] Dragica Jost, Aljaz Skerlavaj, and Andrej Lipej. Improvement of efficiency prediction for a kaplan turbine with advanced turbulence models. *Journal of Mechanical Engineering*, 60(2):124–134, 2014.

[4] Tasneem Abbasi and S. N. Abbasi. Small hydro and the environmental implications of its extensive utilization. *Renewable and Sustainable Energy Reviews*, 15:2134–2143, 2011.

[5] Sujate Wanchat, Ratchaphon Suntivarakorn, Sujin Wanchat, Kitipong Tonmit, and Pongpun Kayanyiem. A parametric study of a gravitation vortex power plant. *Advanced Materials Research*, pages 811–817, 2013.

[6] Sagar Dhakal, Ashesh Babu Timilsina, Rabin Dhakal, Dinesh Fuyal, Tri Ratna Bajracharya, Hari Prasad Pandit, Nagendra Amatya, and Amrit Man Nakarmi. Comparison of cylindrical and conical basins with optimum position of runner: Gravitational water vortex power plant. *Renewable and Sustainable Energy Reviews*, 48:662–669, 2015.

[7] M. M. Rahman, J. H. Tan, M. T. Fadzlita, and A. R. Wan Khairul Muzammil. A review on the development of gravitational water vortex power plant as alternative renewable energy resources. *IOP Conference Series: Materials Science and Engineering*, 217:1–9, 2017.

[8] Marian Marius Gheroghe, Sajin Tudor, and Azzouz Abdelkrim. Study of micro hydropower plant operating in gravitational vortex flow mode. *Applied Mechanics and Materials*, 371:601–605, 2013.

[9] Gheorghe-Marius Marian, Iulian Sajin, Tudor; Florescu, Dragos-Iulian Nedelcu, Constantin-Narcis Ostahie, and Catalin Birsan. The concept and theoretical study of micro hydropower plant with gravitational vortex and turbine with rapidity steps. *World Energy System Conference*, (3):219–226, 2012.

[10] Subhash Dhakal, Tri Ratna Bajrachary, Arun Bikram Thapa, Pikam Pun, and Sushan Nakarmi. Development and testing of runner and conical basin gravitational water vortex power plant. Pulchowk, Lalitpur, 2014. Center for Energy Studies (CES), Institute of Engineering, Tribhuwan University, Nepal.

[11] Christine Power, Aonghus McNabola, and Paul Coghlan. A parametric experimental investigation of the operating conditions of gravitational vortex hydropower (gvhp). *Journal of Clean Energy Technologies*, 4(2):112–119, 2016.

[12] Anil Sapkota, Ankit Gautam, Jhalak Dhakal, and Subash Neupane. Design study of runner for gravitational water vortex power plant with conical basin. Technical report, Department of Mechanical Engineering,Pulchowk Campus, IOE, Pulchowk, Lalitpur, 2016.

[13] Rabin Dhakal, S. J. Williamson, Kshitiz Khanal, Binod Babu Kumal, Laxmi Prasad Devkota, Shree Raj Shakya, Tri Ratna Bajracharya, and Rajendra Shrestha. Computational and experimental investigation of a runner; gravitational vortex power plant, 2016.

[14] Chao WU, Bo WANGa, and Min DU Yunliang CHEN. Three-dimensional numerical simulation of vertical vortex at hydraulic intake. *Elsevier*, 28:55–60, 2012.