Journal of Advanced College of Engineering and Management

Computational and Experimental Analysis of Conical Draft Tube's Effect on Conical Basin of Gravitational Water Vortex Turbine

***Bishal Acharya[1], Raj Kumar Chaulagain[1]**

[1]Department of Automobile and Mechanical Engineering, Thapathali Campus, Institute of Engineering, Tribhuvan University

E-mail: bishalacharya13@gmail.com

ABSTRACT

This paper presents an insight into the performance of the Gravitational Water Vortex Turbine (GWVT), a type of Ultra-Low Head (ULH) turbine, with and without the implementation of draft tube. Till date, no attention has been given on the effect of draft tube on GWVT. A model of GWVT along with a conical basin is developed on CATIA v5, and then subjected to a series of computational studies on ANSYS 2020R2. A similar setup is constructed in which a straight divergent draft tube is installed at the exit of the conical basin. The numerical approach suggested that maximum efficiency of the setup in the former case was 72.0% at 53 RPM whereas the efficiency in the latter case was 79.0% at 65 RPM. The increase in the efficiency was calculated to be 9.72%. This conclusion was validated through experimental verification conducted at Himalaya College of Engineering, Chyasal on a pre-fabricated test-rig of capacity 480 L, where the maximum efficiency of 69.4% and 64.5% were obtained with and without the installation of the draft tube respectively. Consequently, an increase of 7.59% in the efficiency of GWVT as an effect of draft tube was observed experimentally. It can be concluded that draft tube plays a vital role in the efficiency of the Gravitational Water Vortex Turbine.

Keywords: *ANSYS, Gravitational Water Vortex Turbine, Draft Tube, Efficiency*

1. INTRODUCTION

In this era of rapid development, the statement that the demand for electricity is ever increasing is a mere platitude. In a country like Nepal, comprised of the steep gradient landscapes and where water is a major resource, the implications of Ultra Low Head (ULH) turbine lay colossal effect in the energy production of the nation. The source of water in the majority of cases is natural, ranging from streams flowing down mountains to rivers and canals flowing along plains. Although commercial turbines are used in power production, they demand huge investment, laborious works, detailed planning and multi-year timeframe. Furthermore, regular inspection and maintenance of the power plants become imperative, which further adds to the cost and time.

Gravitational Water Vortex Turbine (GWVT) is a system that can exploit the sites of low head and low discharge for the purpose of power generation. It is an ultra-low head turbine that can operate in low head up to 0.7m in which water passes through a large, straight inlet through the channel and then passes tangentially into a round basin, which forms a powerful vortex [1].

Austrian engineer, Franz Zotloterer is the pioneer of this low head power plant which makes the use of kinetic energy inherent in an artificially induced vortex [2].

Gravitational Water Vortex Power Plants deliver a high potential at sites having low heads. The main reason for this is that GWVT have the capability of maintaining high efficiency even when the head at the site tends to zero. On the other hand, conventional impulse and reaction turbines do not possess such a property. The reason why GWVT delivers high efficiency even at low head is that such a type of turbine requires head only to create an artificial vortex, and there are no other implications of higher head in the performance enhancement of this type of turbine. In GWVT, a change in momentum on the fluid is caused by high velocity fluid striking the runner, thereby causing and exerting a reacting force on the turbine runner. However, the design of GWVT is generally done in such a way that the water moves both horizontally and vertically. On the other hand, in other impulse turbines such as Pelton turbine, the fluid flows in one direction only [3].

It has simple structure with limited Hydraulic and Mechanical Components and can be installed in irrigation canal, navigation canal, and wastewater treatment plant and also in the manmade water flowing system. In GWVT, the flow of water is through a straight inlet. It is then followed by a round or conical basin in which the flow of water is tangential. The water will then form a powerful vortex, which exits the outlet at the centre bottom of the shallow basin (Rahman, 2017) [4]. However, in-depth research of this turbine is required to explore its optimum performance. Although there have been some studies on the geometry and basin structure of GWVT, there is no study of the effect of draft tube on this type of turbine till date. A draft tube may have a significant effect on the utilization of kinetic energy at the exit of the runner. The objective of this paper is to present the effect of draft tube on the performance of GWVT.

2. LITERATURE REVIEW

2.1. PREVIOUS STUDIES

One of the earliest studies in GWVT was conducted by (Marian G-M, 2012). Their study focused on the effects of basin shape on turbine's performance. For their study, they tested different sized Francis turbine at different levels of depths. It was found that a vortex proportional to the rotational speed was formed in the absence of turbine. However, when the turbine was installed, the vortex height changed, thereby lowering the efficiency [5].

The study was carried out in two phases. The first phase included the design and fabrication of two different turbines. The performance characteristics of the new turbines were compared to the old turbine. In the second phase, the conical basin was designed and fabricated. The research was mainly done on the geometry of basin, after which it was contended that conical basin was more efficient than the cylindrical basin. Up until 2015, the design study of basin was done extensively but the study of the runner itself was yet to be made [6].

R. Dhakal *et al*. developed three alternative runner designs based on basic hydrodynamic equations. According to CFD analysis, it was observed that a curved blade profile was the most suitable among the three runner designs. More specifically, maximum efficiency of 80% was calculated when the angle between the blade and the hub was 19°. Following this study and to validate its conclusions, an experimental test rig was constructed. From the experimental study, a measured efficiency of 70.9% was observed [3].

In the year 2013, an artificial vortex power generator prototype developed by (Aravind, 2013) consisting of eight inverted cone blades generated 150 W electricity. The study concluded that turbine's materials play a vital role in power generation. Likewise, the flow rate can be maximized by optimizing canal and notch angle [7].

According to a research conducted by T.R. Bajracharya, on 2014, vortex formation with conical basin was found to be stronger than the vortex formed with cylindrical basin when compared between the two basins. The vortex strength increased significantly when conical basin was used. Hence, it is preferable to use conical basins for gravitational water vortex turbines. Also, the measurements concluded that the bottommost position was the optimum position for the placement of the turbine. It is because the value of velocity head increases with the increase in depth [8].

However, the numerical and experimental study conducted by S. Dhakal on 2015 asserted that output power and efficiency is maximum in conical basin compared to that of cylindrical basin for all similar inlet and outlet condition with maximum power extraction at runner position of 65–75% of total height of basin from top position [9].

The observation was further bolstered by the conclusions of the research conducted by T. Bajracharya, 2018. It was found that the vertical position of the turbine in a modified vortex drop shaft type basin significantly affects the power output. The maximum efficiency occured when the runner submergence is about 65-75% of Basin height. Hence, blade formulation should not only consider horizontal but also consider the vertical component of velocity [10].

Christine et. al. found that in vortex power plants, the efficiency of a vertical axis turbine having different blade sizes significantly depends on parameters such as turbine's inlet height and flow rates. Nine different configurations of turbines were prepared with different sizes and blade numbers. It was observed that the overall efficiency of the vortex power plant varied from 15.1% to 25.36% depending on the geometry of the basin, volumetric flow rate, turbine position and blade geometry and number (Power C, 2016) [11].

A new approach for the design of the runner was adopted by (Regmi, 2019). In his study, CFD analysis of the basin was done. The velocity contour plots and streamlines obtained from CFD analysis of the basin were used as input for designing of runner blades. The design of blades were based on the assumptions that the maximum energy is extracted when water strikes the runner perpendicularly. Using the part design of CATIA V5R20, the final runner was achieved by arranging three blades each at an angle of 120°. The maximum efficiency of designed runner was found 7.93 % at 68 rpm from computational analysis while experimentally maximum efficiency was found to be 12.10 % at 66 rpm. However, comparison at the same runner speed of 66 rpm, the experimental maximum efficiency was found to be higher by 4.45% than computational efficiency 7.65%. Hence, this design was not adapted because of its low efficiency [12].

To enhance the efficiency of Gravitational Water Vortex Power Plant, the concept of booster runner was introduced. The numerical solution and experimentation of the same verified that output power and efficiency of the system increases by assembling booster runner with a single main runner for all similar inlet condition. The energy of water that falls to certain height after leaving the main runner was harvested by designing suitable booster runner that meet the necessary condition. The increase in efficiency is about 6% more than that of a single main runner [13].

3. METHODOLOGY

The methodology of this study is summarized in following points:

- Literature Review: Many research articles relevant to GWVT were referred to and significant findings were noted.
- CAD Model Preparation: A CAD model of the runner was prepared in CATIA v5 by referring to the design parameters of the pre-fabricated runner available at the experimental test site. The design and modeling of the setup including conical basin, draft tube, waterway, etc. was also done using the same software.
- Computational Analysis: The effect of draft tube on the performance of GWVT was studied through numerical approach using ANSYS 2020R2. Model of the turbine and the experimental setup designed in CATIA v5 were imported to ANSYS 2020R2 and simulation was performed under pre-defined boundary conditions.
- Experimental Analysis: Experimental study of the effect of draft tube on GWVT was performed using the test rig available at Himalaya College of Engineering. The findings obtained from numerical approach were compared to the results obtained from experimental analysis.

- Findings: The results obtained from the experimental study and computational study were compared and valid conclusions were drawn. The deviation from expected outcomes are addressed through recommendations for future study.

4. DESIGNS AND DRAWINGS

4.1. DESIGN OF BASIN

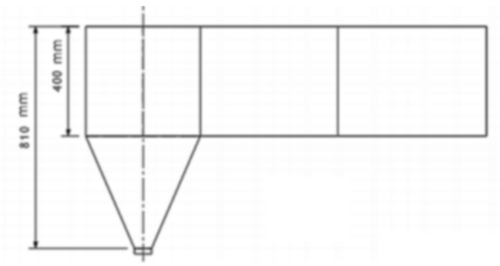


Fig 1: Front View of Conical Basin

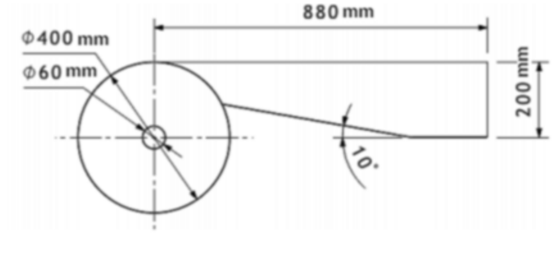


Fig 2: Top View of Conical Basin

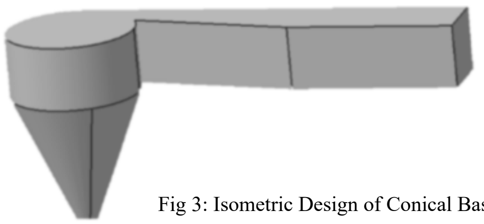


Fig 3: Isometric Design of Conical Basin

R. Dhakal *et al*. had published a research paper that included two phases of study. The first phase included the design and fabrication of two different turbines after which their performance characteristics were compared to the old turbine. In the second phase, conical basin was designed and fabricated. The performance of the setup having conical basin was compared to that of the setup having cylindrical basin through experimental tests. It was found that the value of velocity head increased with the increase in depth. As a result, the efficiency was found be the highest at the bottom-most position. Likewise, it was found that as the number of blades in a turbine decreased, the values of efficiency increased. In case of the turbine configuration with greater number of blades, even with smaller loads, a significant distortion of vortex was observed. Similarly, the efficiency of turbines were found to decrease with increasing radius of the blades. It was further observed from the tests that in the conical basin, the vortex strength of water was higher than the vortex strength in the cylindrical basin. As a result, the turbine efficiency in the conical basin was found to be higher than in the cylindrical basin [3]. For this study, the basin design studied by R. Dhakal was taken as reference.

4.2. DESIGN OF RUNNER

A Gravitational Water Vortex Runner is fabricated by R. Dhakal in his study with the below-mentioned design parameters.

- Inner Diameter: 100 mm
- Outer Diameter: 180 mm
- Height of Runner: 75 mm
- Radius of curvature: 35 mm
- No. of blades: 6
- Diameter of Shaft: 30 mm

Fig 4: Design of Runner

For the design parameters, the runner installed at the test rig available at Himalaya College of Engineering, Chyasal was taken as reference.

4.3. DESIGN OF DRAFT TUBE

Draft tube is a connecting pipe of uniformly varying area which connects the runner exit to tailrace. The fluid exiting from the runner exit still has residual appreciable kinetic energy and swirl even after passing through the turbine runner. To recover some of this kinetic energy that would otherwise be wasted, draft tube is used such that the fluid enters an expanding area of the diffuser. As a result, it turns the flow horizontally which in turn slows down the flow speed. Meanwhile, the pressure of the fluid is increased before it is discharged into the tailrace. The main role of the draft tube is that it allows the turbine to be connected with tailrace without being immersed in water [14].

The flow path of fluid is straight and divergent in straight divergent draft tube. The cross sectional area of the draft tube increases uniformly from beginning to ending. It is most suited for turbines having a low specific speed and configured in a vertical shaft. The cone angle of this type of draft tube is less than 10°. If the cone angle is higher than this value, the problem of cavitation is observed. Efficiency of this type of draft tube is about 90% [15]. To determine the pressure difference between the section 1 and section 3, Bernoulli's equation can be used, the mathematical expression of which is:

$$\frac{P_3 - P_2}{\rho g} = \frac{V_2^2 - V_3^2}{2g} + (z_2 - z_3) \qquad (1)$$

The above equation infers that pressure difference will be greater if the difference between the velocity heads is maximum, i.e. if the outlet velocity of draft tube is minimum. Hence, this situation is obtained by the diverging section of the draft tube, which ensures that the outlet velocity is minimum.

Till date, there is no any defined set of analytical procedures or numerical methods available for design of draft tubes, however they are designed based on the intuition and experiment of designers and are confirmed with experimental test study [16].

The primary purpose of any draft tube including the straight divergent draft tube is the recovery of head. The head recovery increases as the length of the draft tube increases. However, after the length exceeds 19D, the head recovery is not justifiable with the increase in length. Cone angle also plays a vital role in the head recovery. At small length of the draft tube, the rate of change in recovery is more. Khare observed a gradual increase in efficiency with the increase in L/D ratio, but after the ratio reached 19, increase of efficiency was very less due to small increase in recovery. Therefore, increasing the length beyond19D would not by economically unjustifiable but also cause problems like cavitation in the turbine. Majority of the hydro power plants have used straight conical draft tube of length around 19D and having diffuser angle 3.6° to 6° [17].

In order to design the draft tube, the inlet diameter of draft tube (Di) is taken constant, which is given by the diameter of exit hole of the turbine. The design parameters are listed below:

- Draft Angle (θ) = 6°
- Inlet diameter of draft tube (Di) = 80mm
- Length of draft tube accommodated by the turbine test rig = 500 mm
- Outlet diameter of draft tube (Do) = 213mm

Fig 5:Design of Draft Tube

5. COMPUTATIONAL ANALYSIS

Computational Fluid Dynamics (CFD) is a branch of science that involves use of computers in order to predict and produce information regarding fluid flow.The purpose of the simulation is to determine the torque developed by the runner in the given setup at a given flow rate and angular velocity. This helps us to calculate power produced by the runner which in turn helps us in determining the efficiency [18].

CFD comprises of three basic steps, i.e. pre-processing, solver, and post-processing. Pre-processing involves describing the geometry in the best possible manner. It is important to identify the fluid domain of interest. It is then further divided into smaller segments known as mesh, which is called the mesh generation step. The user then identifies the problem physics, defines the fluid material properties, sets the flow physics model, and sets boundary conditions using a computer. Post processor comprises of analyzing the results with methods like streamlines, vector, plots, contour plots, and so on to generate reports.

Two domains, Rotating and Stationary were created in ANSYS Workbench, where the respective geometry were imported. Boolean Subtract Operation was implemented in the rotating domain as shown in the figure below:

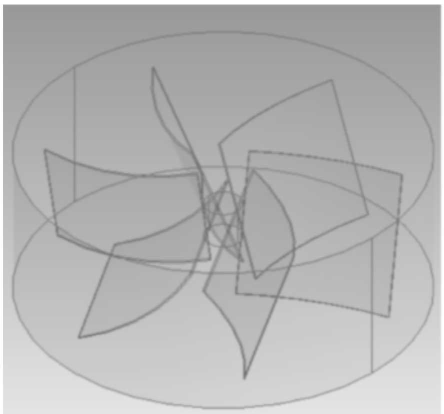


Fig 6: Boolean Subtraction of Rotating Domain

Fig 7: Meshing of Stationary Domain

Two different stationary domains were created, one without draft tube at the exit of the conical basin and another with draft tube installed.

The details of the domains are presented below:

1. Without Draft Tube:
    a. Mesh Type: Tetrahedron
    b. Number of Nodes: 114782
    c. Number of Elements: 345385
    d. Element Size: 10 mm
2. With Draft Tube:
    a. Mesh Type: Tetrahedron
    b. Number of Nodes: 139650
    c. Number of Elements: 371199
    d. Element Size: 10 mm

Boundary conditions along with the initial conditions are crucial in finding the exact solution. The type of partial differential equation and the way it has been discretized are what defines the boundary conditions that is to be used on that equation. However, when solving fluid flow problems with computers, some common boundary conditions are met. These can be classified in two ways. One, in terms of the numerical values that have to be set and the other in terms of the physical type of the boundary condition

Regarding boundary conditions for the stationary domain, the inlet velocity was set as 0.125 m/s and the pressure at outlet was set at atmospheric pressure. Shear Stress Transport model was selected as turbulence model. Likewise, the maximum number of iterations was set to be 300 with RMS Residual Value being 0.0001.

In case of setup without draft tube, six different values for torque were observed against six different values of initial angular velocity: 99 RPM, 90 RPM, 85 RPM, 72 RPM, 53 RPM and 300 RPM.

In case of setup with draft tube, six different values for torque were observed against six different values of initial angular velocity: 118 RPM, 115 RPM, 110 RPM, 90 RPM, 65 RPM and 58 RPM.

The reference runner was secured to the shaft with the help of screw that functions as a key. A digital tachometer was used to measure the RPM of the runner.

The input power (P_{in}) was calculated taking the reference of the experimental setup.

$P_{in} = \rho gQH$ (2)

where,

ρ = density of water

$g = 9.81\ m/s^2$

$H = 0.305$ m (without draft tube)

= 0.54 m (with draft tube)

For the value of Q, the formula to calculate flow over weir as suggested by Pritchard was used [19]

$Q = 1.36h^{2.5}$ (3)

where,

h = 0.11 m as measured from the base of the notch to the water surface level

Hence, the input power was calculated as 27.4 W and 14.71 W with and without draft tube respectively.

The formula to calculate the output power (P_{out}) is

$$\mathbf{Pout} = \frac{2\pi NT}{60} \quad (4)$$

where,

T = Output Torque

N = Initial Speed of the Runner

5.1. WITHOUT DRAFT TUBE

The simulation was run in six different values of speed where the initial speed of the runner was changed from 99 RPM to 48 RPM. The value of torque was then calculated for each value of initial speed. The velocity profile of the aforementioned setup is presented below:

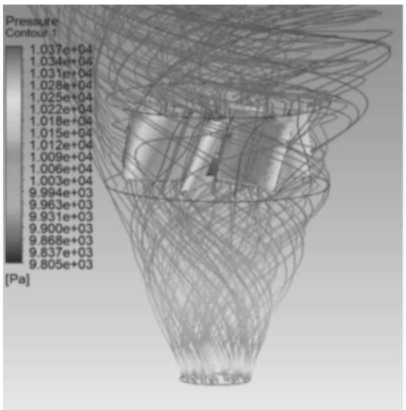


Fig 8: Pressure Contour of Runner (Without Draft Tube)

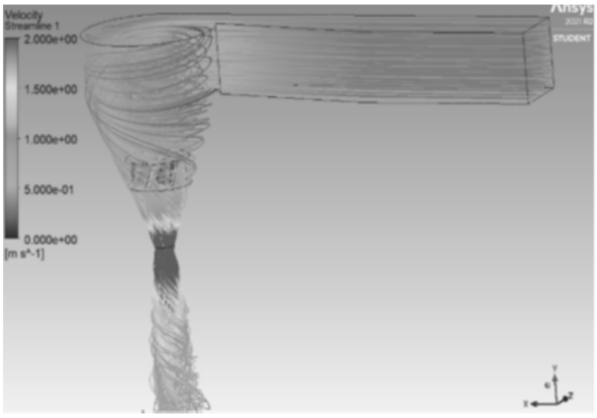


Fig 9: Velocity Profile (Without Draft Tube)

5.2. WITH DRAFT TUBE

Similar to the above case, the numerical analysis of the setup having draft tube for six different values of speed suggested the following results:

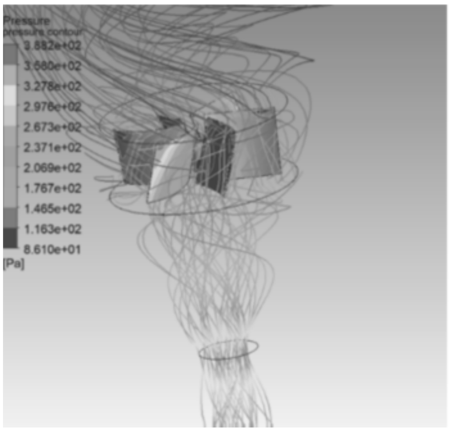


Fig 10: Pressure Contour of Runner (With Draft Tube)

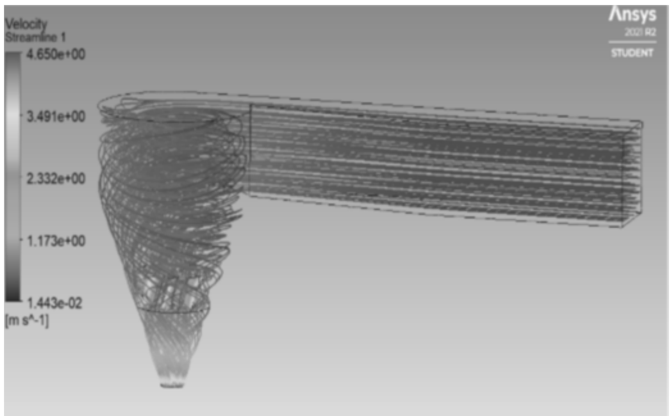


Fig 11: Velocity Profile (With Draft Tube)

From the computational analysis, it can be observed that the velocity of fluid near the runner and magnitude of pressure in the runner is higher in the setup consisting of draft tube. This may be due to the recovery of kinetic energy at the exit of the runner. The maximum efficiency of the runner has increased from 72.0% to 79.0%, which is an increment of 9.72%.

6. EXPERIMENTAL VERIFICATION

The experimental verification of the results obtained from computational analysis was performed with the help of the experimental test rig available at the Himalaya College of Engineering, Chyasal.

Rope brake dynamometer was used for the measurement of torque produced by the shaft at different RPM. For this purpose, a jute rope having diameter of 7 mm was wound around the wheel at the top of the shaft where one end of the rope was attached to a digital spring balance and the other end was attached to another digital spring balance where variable loads were to be applied. The rotational speed of the shaft was measured by optical type tachometer. The torque produced by the shaft can be calculated by the formula:

$$T = (W_1 - W_2) \times g \times R_{eff} \quad (5)$$

where,

W_1 = Weight displayed at the spring balance

W_2 = Counterweight added at the other end of the rope

R_{eff} = Effective radius of the system

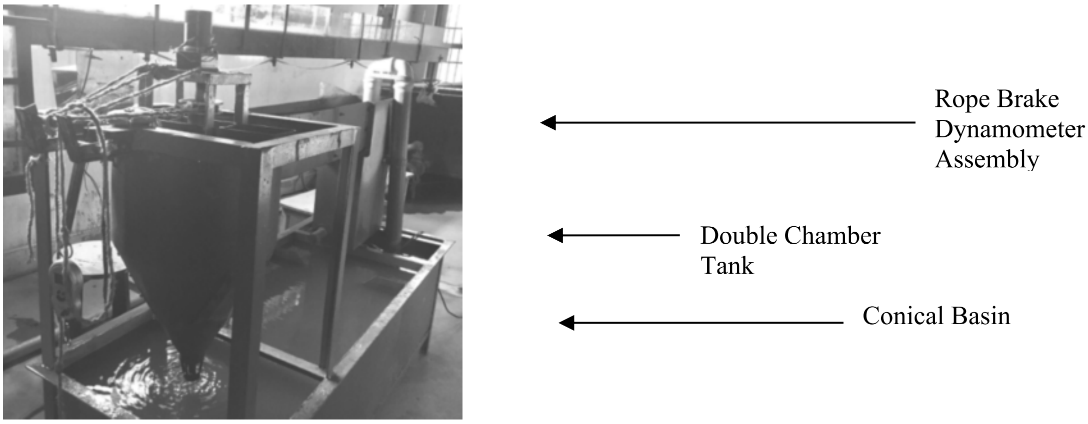


Fig 12 Test setup

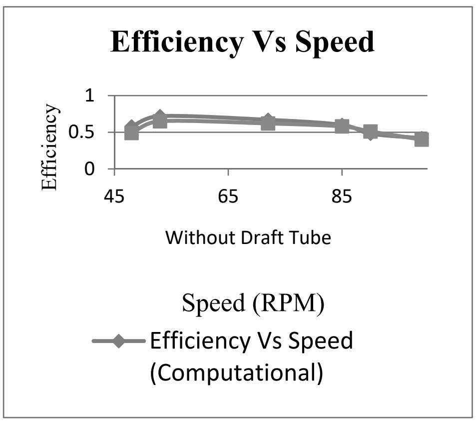
6.1. WITHOUT DRAFT TUBE

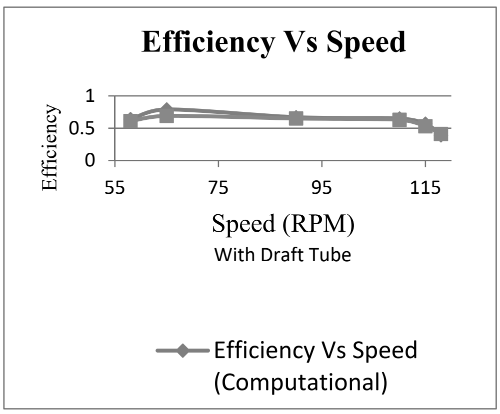
As in the computational analysis, the experiment was run in six different values of speed and the corresponding torque was calculated.

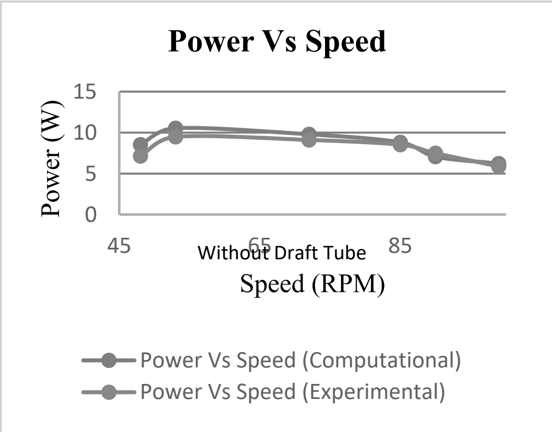
6.2. WITH DRAFT TUBE

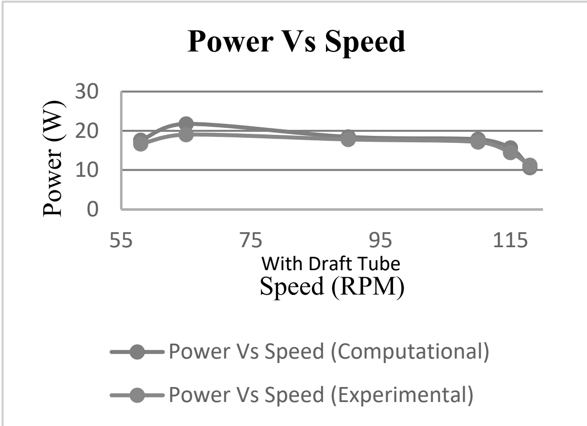
A draft tube of the dimensions mentioned above was fabricated at a local workshop. A pipe clamp was used to secure the position of the draft tube at the exit of the conical basin.

7. RESULTS AND CONCLUSION









As per numerical analysis, the maximum efficiency of 72% at 53 RPM was observed in the setup without draft tube whereas in the setup where draft tube was installed, the maximum efficiency was observed to be 79% at 65 RPM. The increment in efficiency was calculated to be 9.72% as an effect of draft tube.

Similarly, the maximum efficiency observed experimentally in the setup without draft tube was observed to be 64.5% at 53 RPM whereas in the setup with draft tube attached at the exit of the conical basin, the maximum efficiency was observed to be 69.4% at 65 RPM. This was an increment of 7.59% in maximum efficiency.

Hence, it can be concluded that installation of draft tube has a positive effect in the increment in efficiency of Gravitational Water Vortex Turbine. It can also be concluded that unlike common understanding, reaction force also acts on Gravitational Water Vortex Turbine apart from impulse force. The draft tube attached at the exit of the basin converted the kinetic energy to pressure head, thereby causing higher rotational speed of the runner and higher torque production. The efficiency of GWVT was found to increase at first with increasing speed. However, it decreased after reaching a maximum value. This may be due to the inverted 'V' characteristic of torque vs speed in hydraulic turbines.

From the experimental analysis, the maximum efficiency of GWVT has increased from 64.5% to 69.4% as an effect of draft tube. This is an increment of 7.59% in maximum efficiency

The recommendations for future work are presented as follows:

- Effect of draft tubes besides straight convergent draft tube should be studied.
- Finer mesh metrics should be implemented to make the results of computational study more accurate.
- Torque transducers should be used instead of rope brake dynamometers to obtain accurate values of torque produced by GWVT.

ACKNOWLEDGMENT

I would like to express my sincere gratitude to the Department of Automobile & Mechanical Engineering, Thapathali Campus for providing this wonderful opportunity to explore the prospects of Gravitational Water Vortex Turbine. This project has broadened my horizons in the field of ultra low head turbine.

I feel truly indebted to Himalaya College of Engineering for providing me the test rig to conduct experiments as a part of my project. I did not have to search for financial assistance because of the technical support provided by this institution. No any financial support has been received for the conduction of this research.

REFERENCES

1. Bajracharya, T. R. (2020). Effects of Geometrical Parameters in Gravitational Water Vortex. Journal of Renewable Energy, 02.
2. Bajracharya, T. R. & Chaulagain, R. K. (2012). Developing innovative low head water turbine for free-flowing streams suitable for micro-hydropower in flat (Terai) regions in Nepal. . Center for Applied Research and Development (CARD).
3. Dhakal, R. & Bajracharya, T. B. *Computational and Experimental Investigation* of Runner for Gravitational Water Vortex Power Plant.
4. Rahman, M. (2017). A Review on the Development of Gravitational Water Vortex Power Plant as Alternative Renewable Energy Resources. IOP Conference S*eries: Materials Science and Engineering*, (p. 2).
5. Marian G-M, S. T.-I.-N. (2012). World Energy System Conference., (pp. 219-226).
6. Dhakal, S. (2015). Comparison of cylindrical and conical basins with optimum position. El Sevier
7. Aravind. (2013). IEEE Global Humanitarian Technology Conf.: South Asia Satellite., (pp. 53-57).
8. Bajracharya, T. R. (2014). Development and Testing of Runner and Conical Basin Gravitational Water Vortex Power Plant . Centre for Energy Studies (CES) , 5-6
9. Dhakal, S. (2015). Comparison ofcylindrical and conical basins with optimum position. El Sevier
10. Bajracharya, T. (2018). DESIGN AND PERFORMANCE ANALYSIS OF WATER VORTEX POWERPLANT IN CONTEXT OF NEPAL. *20th International Seminar on Hydropower Plants.* Vienna.
11. Power C, M. A. (2016). *Journal of Clean Energy* Technologies, 112-119.
12. Regmi, N. (2019). *Analytical Modeling and Performance Evaluation of Gravitational Water Vortex.*
13. Gautam, A. (2016). *Study on Effect of Adding Booster Runner in Conical Basin: Gravitational Water Vortex Power Plant: A Numerical and Experimental Approach.* IOE Graduate Conference.
14. Wadibhasme, S. J. (2016). HYDRAULIC TURBINE DRAFT TUBE: LITERATURE REVIEW. *International Journal of Science, Engineering and Technology Research (IJSETR)* .
15. Neenu. (2019). Retrieved from https://me-mechanicalengineering.com/hydraulic-turbines-introduction-and-classification/
16. Soni, V. (2010). DESIGN DEVELOPMENT OF OPTIMUM DRAFT TUBE FOR HIGH HEAD FRANCIS TURBINE USING CFD. *37th International & 4th National Conference on Fluid Mechanics and Fluid Power*, (p. 1).
17. Dr. Ruchi Khare, D. V. (2012). Design Optimization Of Conical Draft Tube Of Hydraulic Turbine. *International Journal Of Advances In Engineering, Science And Technology (IJAEST), ISSN: 2249-913x*.
18. Zuo, W. (n.d.). *Introduction of Computational Fluid Dynamics* . JASS 05, St. Petersburg .
19. Pritchard, P. J. (2010). *Fox and McDonald'sIntroduction to Fluid Mechanics.* New Jersey: JOHN WILEY & SONS, INC.