CFD evaluation of performance of Gravitational Water Vortex Turbine at different runner position

Manil Kayastha

Prashant Raut

Nirmal Kumar Subedi

Sandesh Tamang Ghising

Department of Mechanical Engineering, Kathmandu University, School of Engineering, Dhulikhel, Kavre ,Nepal

Email: manilkayast@gmail.com

Rabin Dhakal

Department of Mechanical Engineering, Texas Tech University, Lubbock, Texas, USA

Email : Rabin.Dhakal@ttu.edu

***Abstract*— A free vortex is a region in which flow revolves around an axis line that requires a small head to form, about 0.7 m – 2 m. In Gravitational water vortex turbine, water assuming to be non-rotational and inviscid passes through an open channel and enters the basin tangentially where it forms a powerful vortex. Then, the dynamic force of water is transmitted by the vortex to the turbine via mixed flow, impulse and reaction, phenomena. It can be a promising cheap and effective solution to compliment recent strives for renewable energy technologies. This paper deals with design and development of a prototype Gravitational water vortex turbine and analysis with computational and experimental methods. Initially, the computational method focused on determining maximum tangential velocity of water achievable in the setup without turbine. And further computational analysis was carried out for the setup with turbine to determine performance characteristics by adjusting the runner heights in three positions. Then the experimental analysis was performed with runner placed at three different heights. The experimental and numerical analysis showed best efficiency and maximum RPM (revolutions per minute), for the given prototype, when runner was at lowermost position i.e. 84 cm from top of the conical basin.**

Keywords*— Gravitational water vortex turbine, Computational method, Experimental method, conical basin, impulse, reaction, Runner positions, efficiency

I. INTRODUCTION

Water energy being a clean, cheap and environment friendly source of power generation is of great importance for sustainable future. Beside the fact, still major of the hydro energy is under-utilized. Nepal is a developing country with the capacity of producing 83,000 MW Hydro Electricity but the large scale projects have not been broadly successful and have just produced 480 MW till date [1]. It had been challenge to provide electricity all over the country, especially in hilly areas of Nepal. Therefore small scale hydropower plants can play vital role to overcome this problem.

Gravitational Water Vortex power plant can be an efficient method for generating electricity. Gravitational water vortex turbine is an ultra-low head turbine which can operate in low head range of 0.7–2 m with similar yield as conventional hydroelectric turbines used for production of renewable energy characterized with positive environmental yield [2]. GVT is based on the principle of converting the potential energy of water with a powerful Gravitation Water Vortex (GWV) in a rotation tank to kinetic energy. This kinetic energy is focused as rotational energy of the turbine in the center of the GWV. The work produced by a turbine can be used for generating electrical power when combined with a generator. In fluid dynamics, a vortex is a region in a fluid in which the flow is rotating around an axis line, which may be straight or curved. Vortices are a major component of turbulent flow.

A. Computational Fluid Dynamics

A CFD analysis basically consists of the following three phases:

a) *Pre-processing:* In this phase the problem statement is transformed into an idealized and discretized computer model. Assumptions are made concerning the type of flow to be modeled (viscous/inviscid, compressible/incompressible, steady/non steady). Other processes involved are mesh generation and application of initial- and boundary conditions.

b) *Solving:* The actual computations are performed by the solver, and in this solving phase computational power is required. There are multiple solvers available, varying in efficiency and capability of solving certain physical phenomena.

c) *Post-processing:* Finally, the obtained results are visualized and analyzed in the post processing phase. At this stage the analyst can verify the results and conclusions can be drawn based on the obtained results. Ways of presenting the obtained results are for example static or moving pictures, graphs or tables.

B. *Work flow chart*

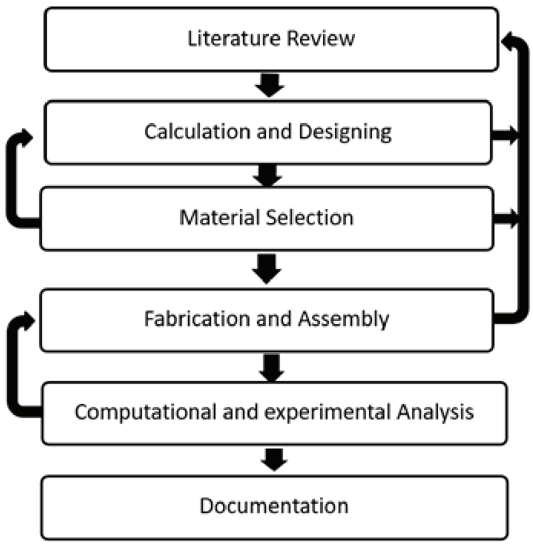

Fig. 1. Working Plan

In the first phase topic was selected as "Computational and Experimental Study of Gravitational Water Vortex Turbine". Literature review was done over the topic keywords and past researches over the topic was commenced. The literature review was continued throughout the course of the project. Moving into the calculation and design part, parameters such as cone inlet diameter etc. was acquired based on the past researches and some parameters were assumed on our own. Then a CAD model of the prototype was designed in SolidWorks based on the collected parameters. Then it was carried onto material selection where mild steel was selected as core material for our prototype. After acquiring parameters and material selection, fabrication and assembly was proceeded. The prototype was fabricated using technical level manpower and no advanced fabrication like casting, machining etc. was performed. The fabricated parts were then finally assembled for experimental testing. The prototype geometry was exported to ANSYS CFX for computational analysis. Torque and efficiency was then calculated for three different runner positions. The experimental setup was adjusted so as to resemble setup as for computational analysis. Then using rope brake dynamometer, torque was calculated for three different runner positions and finally efficiency was derived. Lastly the final documentation of the paper was prepared to publish.

II. Study of Past Research

R.Dhakal and colleagues did their research on "Computational and Experimental Investigation of Runner for Gravitational Water Vortex Power Plant" and came to conclusion that curved blade profile is the most efficient profile, with a peak efficiency of 82%, compared to 46% for the straight blade runner [3]. Research on "Comparison of cylindrical and conical basins with optimum position of runner: Gravitational water vortex power plant" by Sagar Dhakal and colleagues concludes that output power and efficiency is maximum in conical basin compared to that of cylindrical basin for all similar inlet and outlet condition [4]. In another research on "Technical and Economic Prospects for the Site Implementation of a Gravitational Water Vortex Power Plant in Nepal" asserted that vortex system can be easily installed in existing water infrastructure, with the pilot project costing $2725/kW, less than the average cost of micro and Pico hydropower installations reported [5]. Another research based on "Inlet and Outlet Geometrical Condition for Optimal Installation of Gravitational Water Vortex Power Plant with Conical Basin Structure." by R.Dhakal and colleagues concluded that the basin having d/D ratio between 20% to 25% is efficient for power production [6].

III. Design

A. *Design of Conical Basin*

A past research based on basin selection had concluded that conical basin was more efficient over cylindrical basin.

Water channel was designed for flow rate of 6.2 l/s and the inlet velocity of 0.27 m/s.

By continuity equation area was calculated as 0.022 m^2.

The width of the channel was taken 10 cm and we get the channel height of 22 cm.

So rectangular channel was fabricated with 25 cm height for compensating the overflow problem if encountered and width of 10 cm. Following are the design parameters:

- Cone Angle =9.89^{o}

- Cone Height =860 mm
- Height of Cylindrical Basin = 440 mm
- Total height =1300 mm
- Inlet Diameter = 400 mm
- Outlet Diameter = 100 mm
- Channel length = 1000 mm
- Channel width = 100 mm
- Channel height = 250 mm

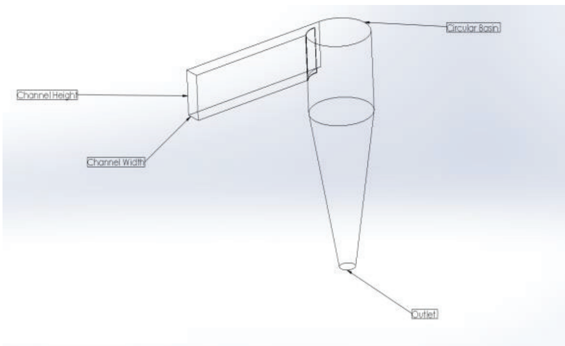

Fig. 2. Conical basin

Since one of our objectives was to test the rig at three different turbine positions, small cone angle was taken so as to increase the conical basin height.

From another research on Inlet and Outlet geometrical condition for optimal installation of conical basin, the CFD analysis showed that the basins having d/D ratio in the range of 20 % to 25 % gives maximum output power, so the inlet diameter (D) of basin was assumed and designed the outlet diameter as 25% of the inlet diameter.

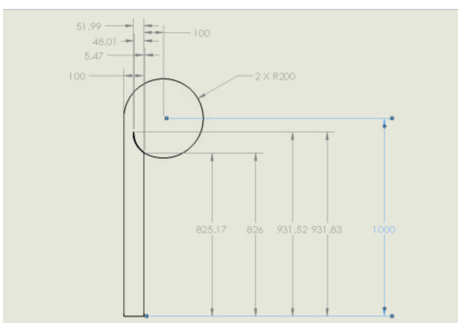

Fig. 3. Conical basin (Top view)

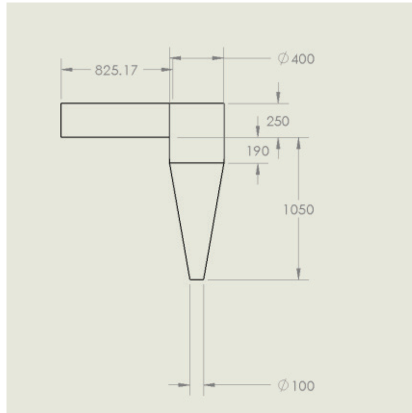

Fig. 4. Conical basin (Front view)

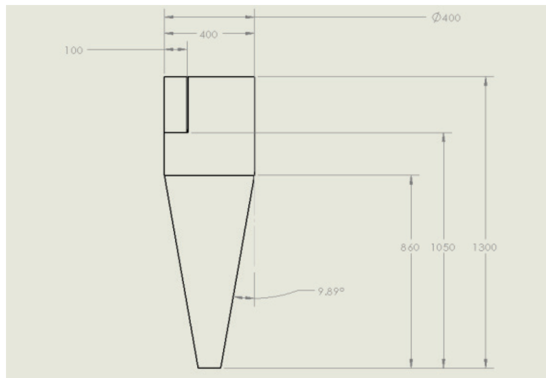

Fig. 5. Conical basin (Right side view)

B. Design of runner

Based on the past researches on the gravitational vortex turbine, there is no specific design for the turbine. Generally straight and curved blade turbines are used for GVT's. These turbines are of impulse type. However a small reaction part is also encountered.

1) Jet impingement on a set of flat plates mounted on a wheel

Consider a set of flat plates mounted on a wheel that is capable of rotating about its axis as in figure 5. A free jet of water strikes the plate and as the wheel moves, one plate or another intersects the jet continuously. Let V be the velocity of the jet of cross sectional area A. Let u= peripheral velocity of the wheel. Then the relative velocity of the plates will be (V-u), the total discharge Q= (AV). The jet impinges normally on the plate ($\theta=90^o$) and the plate moves with the uniform velocity u after the impact.

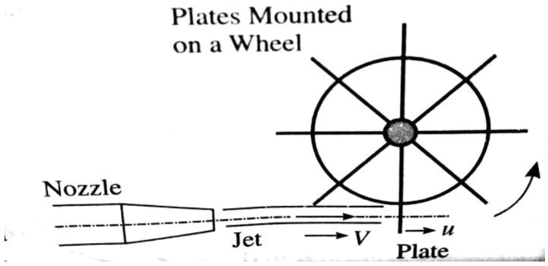

Fig. 6. Jet impingement on plates mounted on a wheel

Let R_x= x-component of the reaction of the plate on the control volume.

$$R_x = [0-\rho AV (V-u)] = -\rho AV (V-u) \qquad (1)$$

The idealized force F_x is equal and opposite to R_x. Thus the idealized force on the plate in the x-direction $F_x = \rho AV (V-u)$

Assuming zero losses, rate of work done by the assembly of plates = Power transmitted to the wheel

$$P = F_x.u = \rho AV(V-u).u \qquad (2)$$

Kinetic energy on the jet per unit of time = $\frac{1}{2}\rho AV.V^2$

$$\text{Efficiency, } \eta = \frac{\textit{Power transmitted to the wheel}}{\textit{Kinetic energy supplied by the jet per unit time}}$$

$$= \frac{\rho AV(V-u).u}{\frac{1}{2}\rho AV.V^2} = \frac{2(V-u).u}{V^2} \qquad (3)$$

For maximum efficiency, $\frac{d\eta}{du} = 0$

$$u = \frac{V}{2}$$

Hence the maximum efficiency $\eta_{max} = 50\%$

2) Jet impingement on series of curved vanes mounted on a wheel

This is the similar case as above except that the series of curved vanes are replacing the plates as shown in figure 6. Since one or other vanes will intercept all the flow from the jet, the entire discharge,Q issuing out of the jet is moved in the transfer of the power of the wheel. The relative velocity of the entering as well as existing jet is (V-u).

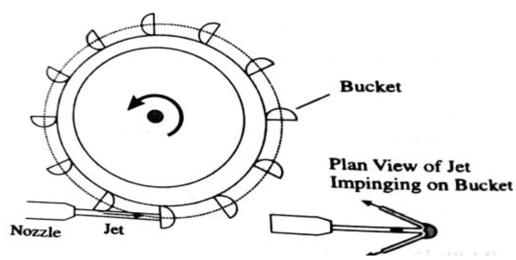

Fig. 7. Jet impingement on curved vanes mounted on a wheel

Consider the linear momentum equation in the x-direction:

$\sum(\text{Forces in x} - \text{direction}) =$ (Momentum flux in x direction) out – (momentum flux in y direction)$_{in}$

$$0-R_x = [-\rho AV(V-u)\cos\theta] - [\rho AV(V-u)]$$

$$R_x = \rho AV(V-u)(1+\cos\theta) \qquad (4)$$

If F_x be the force on the vane in the x-direction, F_x is equal and opposite to R_x.

Thus, the idealized force on the vane in the direction is

$F_x = \rho$ AV (V-u) (1+cosθ) acting in the positive x-direction.

By linear momentum equation in the y-direction:

$\sum(\text{forces in y} - \text{direction})$ = (momentum flux in y-direction) $_{out}$ – (momentum flux in y-direction) $_{in}$

$$0-R_y = [\rho AV (V-u) \sin\theta] - [\rho AV (V-u) \sin\theta] - 0 = 0$$

Hence Ry, the force in the vane in y-direction is zero.

Work done by the jet per second = Power transmitted to the wheel

$$= F_x.u = \rho AV(V-u)(1+\cos\theta).u \qquad (5)$$

Kinetic energy on the jet per unit time = $\frac{1}{2}\rho AV.V^2$

Efficiency of system,

$$\eta = \frac{\text{Power transmitted to the whee}}{\textit{Kinetic energy supplied by the jet per unit time}}$$

$$= \frac{\rho AV(V-u)(1+\cos\theta).u}{\frac{1}{2}\rho AV.V^2} = \frac{2u(V-u)(1+\cos\theta)}{V^2} \qquad (6)$$

Case a: For a given vane (θ = Constant) and fixed jet velocity (V=constant)

Thus for maximum efficiency, $\frac{d}{du}[2Vu - u^2] = 0$ which gives the condition,

$$u = \frac{V}{2} \qquad (7)$$

$$\eta_{max} = \frac{2\frac{V}{2}(V-\frac{V}{2})(1+\cos\theta)}{V^2} = \frac{(1+\cos\theta)}{2} \qquad (8)$$

Case b:

If θ is varied, maximum most efficiency of η_{max} = 100% is obtained for θ = 0°, that is for a semicircular vane.

So these theories suggest that curved blade profile is more efficient than flat profile [7].

From the past researches on the study of runner blade profile, it was concluded curved blade profile to be the most efficient profile, with a peak efficiency of 82%, compared to 46% for the straight blade runner. Therefore curved blade profile was chosen over straight blade profile. A curved blade profile runner of diameter 200 mm was designed. The reason behind choosing a small diameter turbine was to fit the runner at different position of tapering cross section conical basin. For the blades a flat plate of 90 mm*80 mm*2 mm was transformed into a curved plate having an arc radius of 60mm. Six such blades was fabricated and was then spaced equally on the circumference of a 25 mm diameter hollow mild steel pipe by welding.

Fig. 8. Runner designed dimensions

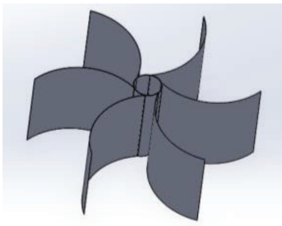

Fig. 9. 3D model of runner

IV. Computational and Experimental Analysis

A. Computational analysis of conical basin without runner

Firstly, the fluid domain (geometry) for the flow simulation without runner was created with CAD software using Solidworks 2016. Then the domain was imported into the ANSYS R15.0 in the CFX solver for the computational analysis.

Various meshes were created starting from minimum of 2195 to maximum of 26710 numbers of nodes through refinement by changing the mesh element sizes. Finally the mesh having 148403 nodes was used for the simulation whose minimum size was default (2.6479e-4 m) and maximum size of 11e-3 m.

The solver preference was CFX. The analysis type was made transient where simulation was run for 30 seconds with time steps 0.5 second and having the convergence criteria of residual target 1e-4. The minimum and maximum coefficient of loops for convergence control was 1 and 5 respectively. The working fluids were taken as air and water at ^{25oC}. The reference pressure of 1 atm and buoyancy reference density of 1.12 kg/m3, surface tension coefficient 0.072 N/m and the domain was of stationary type. The fluid model was of standard homogeneous model with turbulence model of k-Epsilon which gives great analysis in fully developed flow and having scalable wall function.

The simulation was run with no-slip conditions at the wall; bulk mass flow rate of 6.2 kg/s normal to both inlet and outlet boundary. The upper surface of water channel was subjected to the atmosphere and the whole domain was assumed to be filled with water.

1) Results

As for the results water velocities was measured at three different positions in the conical basin. Minimum velocity striking the runner blade was found at 44 cm from the inlet surface level which gradually increased towards core of the vortex.

Table I. Velocity at different position

Position no.	Vertical distance from upper surface (cm)	Velocity (m/s)
1	44	0.51
2	74	0.61
3	84	0.64

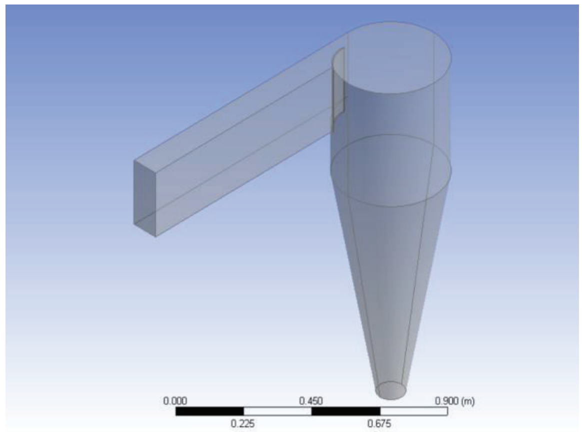

Fig. 10. Fluid Domain without runner

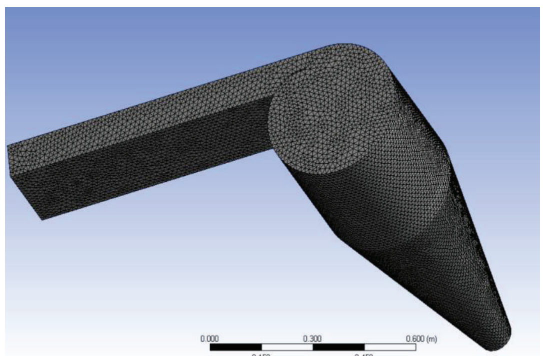

Fig. 11. Mesh of the domain without runner

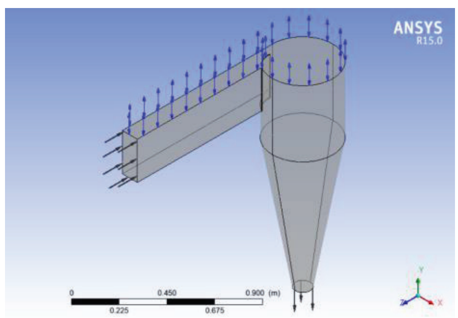

Fig. 12. Boundary conditions for flow simulation without runner

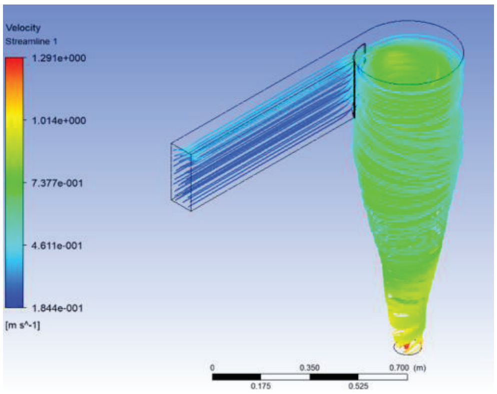

Fig. 13. Water velocity streamline

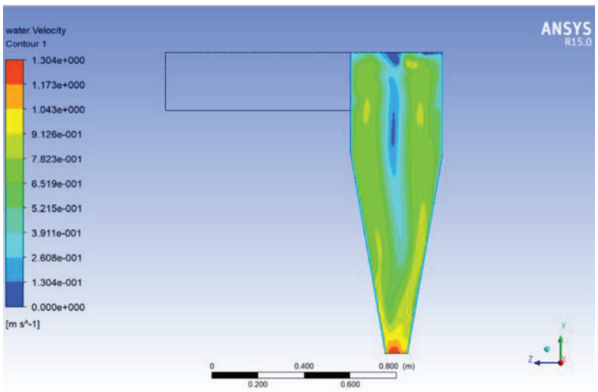

Fig. 14. Water velocity contour

2) *Mesh independence test*

The mesh independence test was performed for the grid convergence. Two functional values were taken for the test; they are inlet and outlet water velocity.

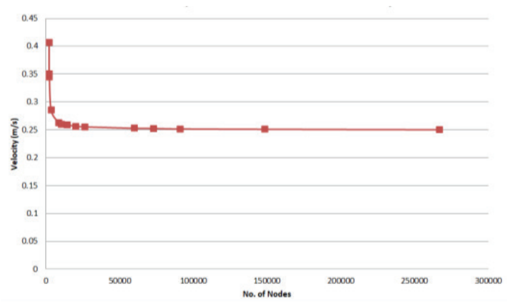

Fig. 15. Mesh independence test for inlet velocity

In the first test for inlet velocity the graph had shown mesh independence after reaching around 25000 number nodes but in the second test for outlet velocity the graph showed mesh independence after reaching around 100000 number nodes.

B. *Experimental analysis*

Our objective was to test the setup for a free flow. Since motor was used to pump the water, the results would have to include pump pressure as well. So a drop tank of dimensions 1ft * 1ft * 1ft was used and was attached to the rectangular channel to nullify the pressure of pump.

During the first phase of testing, there were complications in forming of water vortex. It was due lower cone angle and larger cone height. Due to this the high velocity was reached at outlet which made outlet discharge greater than the inlet flow rate. So two pressure valves was used at the outlet to maintain inlet and outlet flow rate as equal, which is important for the formation of water vortex. The experiment (test) was done with the runner at those three positions decided from the above analysis 4.1 which gave the result as torque and speed (RPM).

The speed was measured by tachometer which is available in the TTL whereas torque was measured using Rope Brake Dynamometer method.

The torque produced was calculated by using the simple mathematical relation of rope brake dynamometer method i.e. Torque (T) = r(W_1-W2)

Where W_1 and W_2 is load applied at end of the rope and the spring balance reading respectively and r is the radius of the brake drum (pulley) used which was found to be 29.25 mm.

Power in the shaft is calculated by:

$$P = \frac{2\pi NT}{60} \text{ Watts}$$

Where N is the RPM measured by tachometer.
The input power (hydraulic power) is calculated as:

$$P_{in} = \rho . Q . g . H$$

Where ρ is the density of water, 997 kg/m^3
Q is the flow rate, 6.2 l/s
g = 9.81m/s^2

Fig. 16. Experimental setup of Rope Brake Dynamometer

Table II. EXPERIMENTAL DATA FOR TORQUE, RPM AND EFFICIENCY

Position	Torque Measurement		Flow rate (l/s)	Head (m)	Torque(Nm)	RPM	P_{out}(watt)	P_{in}(watt)	Efficiency (%)
	W_1(kg)	W_2(kg)							
1	3.1	1.5	6.2	1.28	0.45	142	6.81	77.69	8.7

2	3.1	1.4	6.2	1.28	0.48	160	8.16	77.69	10.51
3	2.55	1.05	6.2	1.28	0.42	175	7.88	77.69	10.14

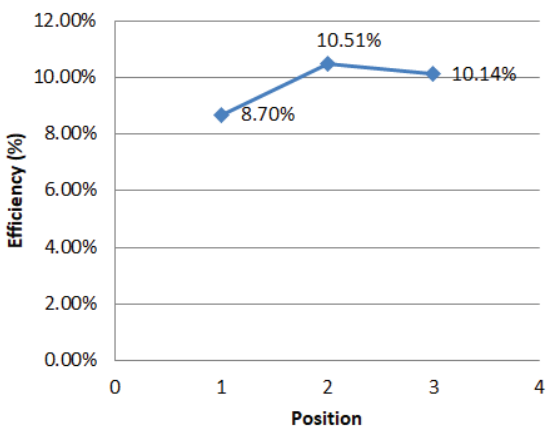

Fig. 17. Graph of Efficiency vs. runner position

C. Computational analysis of conical basin with runner

Similar to the computational analysis of conical basin without runner, the stationary and rotating fluid domains (geometry) for the simulation with runner at different position were created with CAD software using Solidworks 2016. The domains then were imported into the ANSYS R15.0 for the computational analysis.

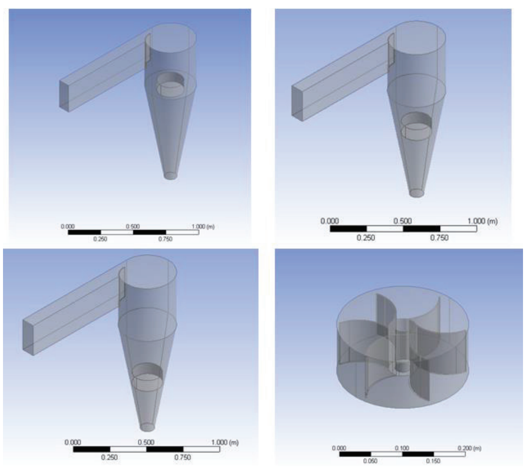

Fig. 18. Stationary and rotating domain

Various meshes were created for stationary and rotating domains and refinements were made by changing the mesh element sizes. Finally the mesh having 131864, 133404 and 205139 nodes was used for the simulation of conical basin with runner position 1st, 2nd and 3rd respectively. The minimum size was default (2.6479e-4 m) and maximum size was 13e-3 m for stationary domain and 5e-3 m for rotating domain of 1st and 2nd runner position simulation respectively whereas the minimum size was default (2.6479e-4 m) and maximum size was 10e-3 m for stationary domain and rotating domain of 3rd runner position simulation.

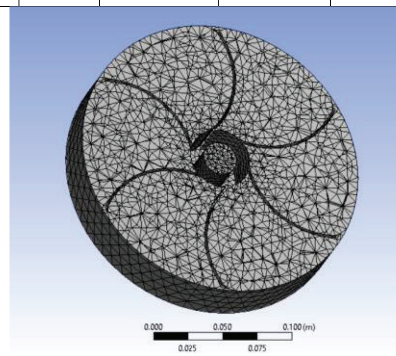

Fig. 19. Rotating domain mesh (cross sectional view)

The solver preference was CFX. The analysis type was made transient where simulation was run for 30 seconds with time steps 0.5 second and having the convergence criteria of residual target 1e-4. The minimum and maximum coefficient of loops for convergence control was 1 and 5 respectively. The working fluids were taken as air and water at 25^{o}C. The reference pressure of 1 atm and buoyancy reference density of 1.12 kg/m^3, surface tension coefficient 0.072 N/m and the domain was of stationary type. The fluid model was of standard homogeneous model with turbulence model of k-Epsilon which gives great analysis in fully developed flow and having scalable wall function. The interface model between the stationary and rotating domain was general connection with frozen rotor and specified pitch angle of 360^{o}.

The simulation was run with no-slip conditions at the wall; bulk mass flow rate of 6.2 kg/s normal to both inlet and outlet boundary. The upper surface of water channel was subjected to the atmosphere and the whole domain was assumed to be filled with water. The black arrows normal to the inlet channel and the outlet shows the bulk mass flow rate normal to the boundary whereas the blue double head arrow shows the channel is open to the atmosphere.

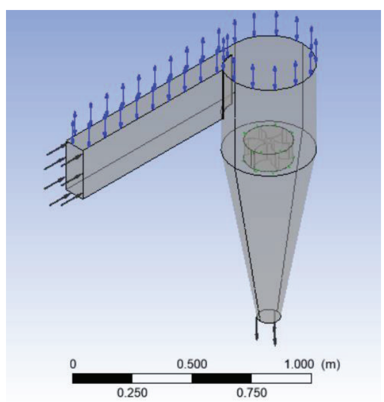

Fig. 20. Boundary conditions

1) Results

Table III. COMPUTATIONAL DATA FOR TORQUE AND EFFICIENCY

Position	Torque (Nm)	Flow rate (l/s)	Head (m)	RPM	P_{out}(watt)	P_{in}(watt)	Efficiency (%)
1	1.32	6.2	1.28	142	19.89	77.69	25.60
2	1.38	6.2	1.28	160	23.11	77.69	29.74
3	1.51	6.2	1.28	175	27.65	77.69	35.59

As for the result, minimum torque with minimum RPM was found at first runner position and their values gradually increased towards bottom of the conical basin. Therefore maximum efficiency was obtained when the runner is positioned at the core of the vortex which is the third runner position in our setup.

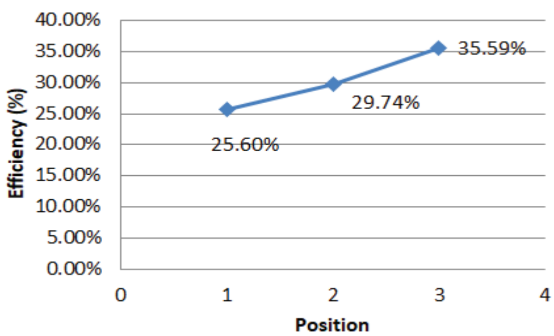

Fig. 21. Graph of Efficiency vs. runner position

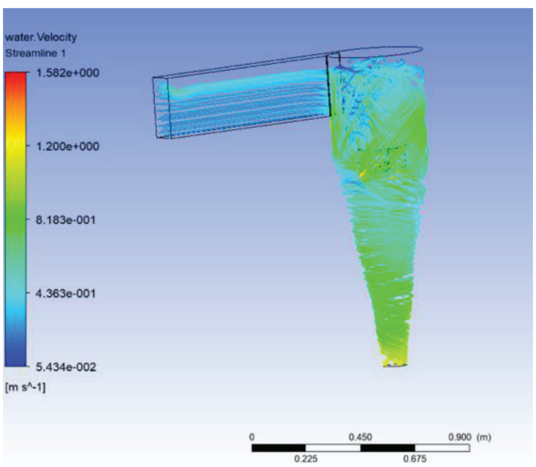

Fig. 22. Water velocity streamline at 1st runner position

2) Mesh independence test

The mesh independence test was performed for the grid convergence. Two functional values were taken; one from the rotating domain and other from the stationary domain, which were torque and outlet water velocity.

It showed mesh independence after reaching around 32000, 40000 and 60000 number nodes for first, second and third simulations respectively.

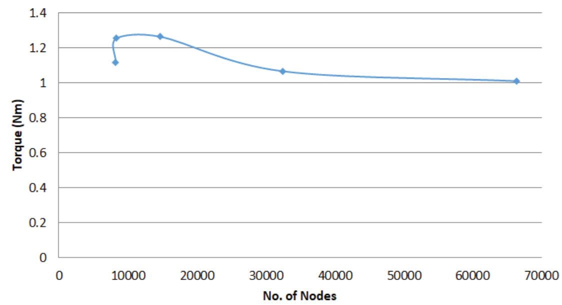

Fig. 23. Test for 1st runner position (torque)

D. Comparison of experimental and computational efficiencies

After the completion of experimental and the computational study of the efficiencies it was found that the experimental efficiencies are less than the computational efficiencies at the respective runner positions.

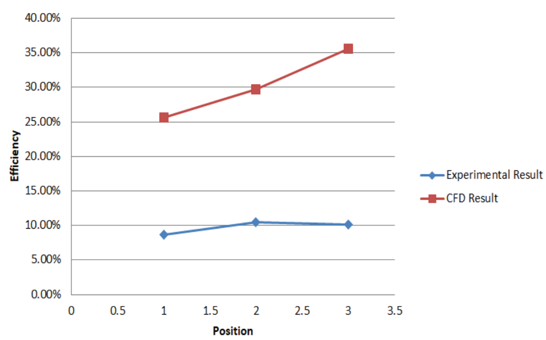

Fig. 24. Comparison of efficiencies

V. CONCLUSION

Gravitational Water Vortex Turbine can be a promising cheap and effective solution to compliment recent strives for renewable energy technologies. The experimental and numerical study on it showed that the efficiency increased as the runner was shifted down the axis of the shaft. It showed maximum RPM when the runner was 84cm, for the given prototype, from top of basin. Although there are larger deviations between results of computational and experimental analysis, both agreed to a point that the runner efficiency goes on increasing while moving towards the bottom of the vortex.

VI. Recommendations

The project can be continued for a larger cone angle which increases the vortex strength and thus more have more efficiency. The runner could be designed and fabricated more precisely using casting instead of manual fabrication. A fine meshing would be possible with high end processing computers, increasing the accuracy of the results.

Acknowledgment

We would like to express our gratitude towards Kathmandu University, School of Engineering for providing us with this opportunity to implement ideas practically.

Our project would not have been completed without the guidance and supervision of our supervisor Er. Atmaram Kayastha, and our project coordinator Er. Pawan Karki.

We are indebted to the Department of Mechanical Engineering for providing us the platform to robust our knowledge even more.

We are also very thankful to Er. Rabin Dhakal for guiding us and providing suggestions in improving and solving various problems encountered.

We are very grateful to Er. Aman Kapali and the team of TTL (Turbine Testing Lab) for helping us to monitor and provide guidance to our project.

We gratefully acknowledge Mr. Bhuwan Bhattarai for helping us with technical part of the project.

Our heartfelt gratitude goes to countless reader of this project report, who is the most important critic and commentator. We welcome constructive comments and suggestions which will help us to do better one.

At last we would like to thank those people who gave their helping hands directly or indirectly for this project

References

[1] Authority, N. e. (n.d.). Hydropower Grid Amount. Retrieved June 25, 2018, from http://www.nea.org.np/

[2] Zotlötere. (n.d.). GRAVITATION WATER VORTEX POWER PLANT. Retrieved June 22, 2019, from http://www.zotloeterer.com

[3] R.Dhakal, T. B. (2017). Computational and Experimental Investigation of Runner for Gravitational Water Vortex Power Plant. San Degio, CA, USA: 6th International Conference on Renewable Energy Research and Application.

[4] Sagar Dhakal, A. B. (2015). Comparison of cylindrical and conical basins with optimum position of runner: Gravitational water vortex power plant. Elsevier.

[5] R.Dhakal, A. N. (2016). Technical and Economic Prospects for the Site Implementation of a Gravitational Water Vortex Power Plant In Nepal. Birmingham, UK.

[6]Rabin Dhakal, S. S. (n.d.). Inlet and Outlet Geometrical Condition for Optimal Installation of Gravitational Water Vortex Power Plant with Conical Basin Structure.

[9] K. Mahadevan, K. B. (2013). Design Data Handbook for Mechanical Engineers. CBS Publisher & Distributors.