Effect of Dominant Parameters for Conical Basin: Gravitational Water Vortex Power Plant

Sagar Dhakal [1], A. B. Timilsina [1], R. Dhakal [1], D. Fuyal [1], Tri Ratna Bajracharya [1], Hari Prasad Pandit [2]

[2] *Department of Mechanical Engineering, Central Campus - Pulchowk, Institute of Engineering, Nepal*

[2] *Department of Civil Engineering, Central Campus - Pulchowk, Institute of Engineering, Nepal*

Corresponding email: sgrdhkl64@gmail.com

Abstract: This study is the analysis of several geometric parameters of conical basin design and their effect on vortex formation and energy. Here we have studied effect of various parameters individually on vortex velocity by using Computational Fluid Dynamics . For a given flow and head the different geometrical parameters that can be varied of conical basin for gravitational water vortex power plant are: (i) basin opening, (ii) basin diameter (iii) notch length iv) Canal Height and v) Cone Angle . Different basin configuration were developed using Solid Works software and model was simulated by Commercial CFD code ANSYS Fluent. Each parameter was varied individually and corresponding velocity in region of interest was noted.

Keywords: Conical Basin; Gravitational Water Vortex Power Plant; Basin Diameter; Basin Opening; Notch Length; Cone Angle; Canal Height

1. Introduction

1.1 Gravitation Water Vortex Power Plant

Gravitational water vortex turbine is an ultra-low head turbine which can operate in as low head as 0.7m with similar yield as conventional hydroelectric turbines used for production of renewable energy characterized with positive environmental yield (Zotlöterer, 2014). The water passes through a large, straight inlet through the channel and then passes tangentially into a round basin, which forms a powerful vortex. An exit hole is made at the bottom of the basin through which the vortex finds its outlet (Mulligan & Hull, 2010). The turbine does not work on pressure differential but on the dynamic force of the vortex. Said aim is achieved by as hydroelectric power plant which supports the formation of a stable gravitational vortex which tends to be formed also in the upper reaches directly in front of the turbine inlet of conventional river stations as a lost vortex and is therefore prevented as much as possible there. The inventive hydroelectric plant, however, ensures that the necessary current-related conditions are fulfilled for reinforcing the rotational movement of the water, which is created when the water flows off, in an unimpeded manner into a stable gravitational vortex without using pressure lines and directing devices. A turbine that rotates in a coaxial manner within the gravitational vortex and is impinged upon along the entire circumference thereof withdraws rotational energy from the gravitational vortex, which is converted into electric power in a generator (Zotlöterer, 2014). In addition, gravitational vortex power plant is found to be advantageous due to the following properties of water vortex:

a. Increases the water surface area.

b. Maximizes the velocity of flow on the water surface area.

c. Disseminates homogenously contaminants in the water.

d. Increases the contact surface of the disseminate contaminants for microorganisms and water plants.

e. Aerates the water naturally, because of the high velocity of the flow on the water surface area.

f. Increase the heat of evaporation so water can reduce the temperature itself at rising temperatures in summer.

g. Concentrates dense water (water at 40C) in the ring shaped center to ensure the survival of microorganisms as long as possible. (Zotlöterer, 2013).

h. The BOD removal efficiency of aerobic biological treatment processes depends on a number of factors including (but not limited to): influent BOD loading, F:M ratio, temperature, nutrient levels, and dissolved oxygen (DO) concentrations. (Peroxide, 2014) . Through the creation of vortex dissolved oxygen concentration can be improved.

1.2 Computational Fluid Dynamics

Computational Fluid Dynamics can be defined as the field that uses computer resources to simulate flow related problems. To simulate a flow problem you have

to use mathematical physical and programming tools to solve the problem then data is generated and analyzed.

In order to provide easy access to their solving power all commercial CFD packages include sophisticated user interfaces to input problem parameters and to examine the results. Hence all codes contain three main elements: (i) a pre-processor, (ii) a solver and (iii) a post processor. We briefly examine the function of each of these elements within the context of a CFD code.

Pre-Processor

Pre-processing consists of the input of a flow problem to a CFD program by means of an operator-friendly interface and the subsequent transformation of this input into a form suitable for use by the solver. The user activities at the pre-processing stage involve:

- Definition of the geometry of the region of interest: the computational domain

- Grid generation-the sub-division of the domain into a number of smaller, non-overlapping sub-domains: a grid (or mesh) of cells (or control volumes or elements).

- Selection of the physical and chemical phenomena that need to be modeled.

- Definition of fluid properties.

- Specification of the appropriate boundary conditions at cells which coincide with or touch the domain boundary.

Solver

There are three distinct streams of numerical solution techniques: finite difference, finite element and spectral methods. In outline the numerical methods that form the basis of the solver perform the following steps:

- Approximation of the unknown flow variables by means of simple functions.

- Discretization by substitution of the approximations into the governing flow equations and subsequent mathematical manipulations.

- Solution of the algebraic equations

The main differences between the three separate streams are associated with the way in which the flow variables are approximated and with the discretization processes. Here we have employed finite volume method.

Post-Processor

As in pre-processing huge amount of development work has recently taken place in post processing field. Owing to the increased popularity of engineering workstations, many of which have outstanding graphics capabilities, the leading CFD packages are now equipped with versatile data visualization tools. These include

- Domain geometry and grid display

- vector plots

- Surface plots

- Particle Tracking etc. (Versteeg & Malalasekera, 1995)

1.3   Study of past researches

Mulligam and Casserly (2010) did their research project on "Design and Optimization of a Water Vortex Hydropower Plant" carried out at the Institute of Technology, Sligo in Civil Engineering. This research concludes that optimum vortex strength occurs within the range of orifice diameter to tank diameter ratios (d/D) of 14 % - 18 % for low and high head sites respectively. Thus, for cylindrical basin, to maximize the power output, the range of orifice diameter to basin diameter ratios lies within 14% - 18%.

Bajracharya & Chaulagai (2012) focused on developing innovative low head water turbine for free flowing streams suitable for micro-hydropower in Terai region of Nepal. In the study, water vortex was created by flowing water through an open channel to a cylindrical structure having a bottom whole outlet. The research concluded that for a fixed discharge condition, the height of basin, diameter and bottom exit hole are fixed. i.e. The basin geometry depends on the discharge supplied. This study suggests that, in sufficient flow condition, vortex minimum diameter is at bottom level and is always smaller the exit hole.

Wanchat & Suntivarakorn (2011) studied the effect of basin structure in formation of water vortex stream. This study indicates the important parameters which can determine the water free vortex kinetic energy and vortex configuration and they include the height of water, the orifice diameter, conditions at the inlet and the basin configuration. It was found that a cylindrical tank with an orifice at the bottom center with the incoming flow guided by a plate is the most suitable configuration to create the kinetic energy water vortex.

Wanchat, et al., (2013 studied the analysis and design of basin structure which has ability to form a gravitational vortex stream. The study investigated the suitable outlet diameter at the bottom center of the

vortex basin. In the case of 1m diameter cylindrical vortex basin, computational fluid dynamics (CFD) and experiment using the model indicate that the suitable outlet diameter was in range of 0.2+0.3m. The operating head of the free vortex was in the range of 0.3-0.4m. The maximum power output was 60 W at 0.2 m outlet diameter and the head of the free vortex was at 0.4 m. The total efficiency of the model system was 30%.

Yunliang CHEN,et al., (2012) shows that RNG k-ε model is more suitable than standard k-ε model to the rapidly strained and great curving streamline flows .

2. Model Development and Solution Procedure

In nearly all the previous researches, the air-core vortex was considered based on the assumption of steady, axisymmetric and incompressible flow. The continuity equation and the Navier-Stokes equations in cylindrical coordinates are described as following

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

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

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

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

Where, V_θ, V_r and 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 equations, it's extremely difficult to get an analytical solution directly. (Wang, Jiang, & Liang, 2010)

Computational fluid dynamics (CFD) has becomes a cost effective tool for predicting the performance of Fluid machines and also the fluid flow through a region of interest. In the present study, the simulation has been carried out for flow visualisation in the basins and velocity distribution in the plane where the turbine has extracted more power. For the purpose of simulation the fluid flow domain was modelled as shown in figure using CAD software, Solid Works. The model was then imported in a commercial CFD code ANSYS Fluent and was simulated.

The modelling and meshing of the proposed model is done using software ICEM CFD for fluid analysis. Firstly denser mesh was taken near wall and at air core region but later the grid was refined and uniformly dense mesh was generated. This was done because the vortex velocity was found to be maximum in region between the air core and wall. As this velocity was

subject of concern for basin optimization the CFD model was re-meshed. The canal and outlet region was modelled with Cut cell mesh and Tetrahedron was used for basin modelling. The grid convergence graph is shown below.

The computational domain used is shown in figure below and the Mesh independency is also shown in figure below.
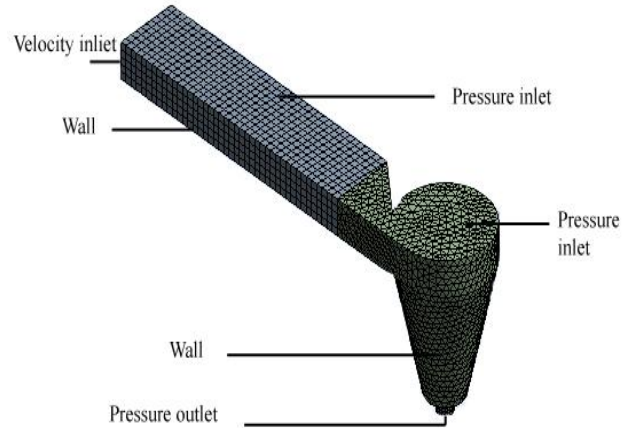

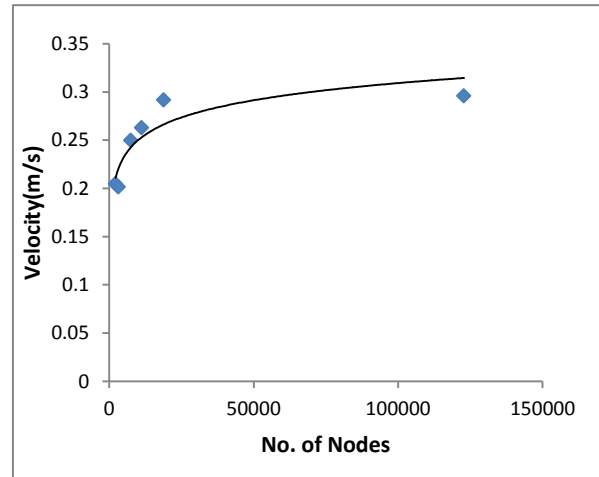**Figure 1: Boundary Conditions**

Figure 2: Convergence graph for conical basin

The simulation has been done for a steady flow to investigate the performance of different basin geometry on vortex velocity distribution. The main assumptions include a steady flow, no slip conditions. The working fluid, water is assumed as an incompressible fluid with density of 998.2 kg/m^3 and viscosity of 0.001003kg/m-s. The RNG k-Ɛ turbulent model was used to investigate the flow pattern of the system.

The Computational Fluid Dynamics simulation was run with no-slip conditions at the wall and pressure outlet condition at the outlet. The inlet was velocity inlet with initial inlet velocity of fluid (water) flow is set to be 0.1 m/s. The upper surface was subjected to atmospheric

pressure. The no of elements in the final computational domain used for the simulation was 308851 and minimum element size, maximum element size was $3*10^{-5}$m, $6*10^{-2}$m respectively.

The initial inlet velocity of fluid (water) flow is set to be 0.1 m/s and the outlet was pressure outlet with wall of the fluid flow domain stationary. As it doesn't have any drastic change on vortex structure whether the canal was open or closed; so, we have simulated the cylindrical basin by considering it as closed channel flow. We have considered the fluid was flowing through the area having hydraulic diameter of 0.2667m.
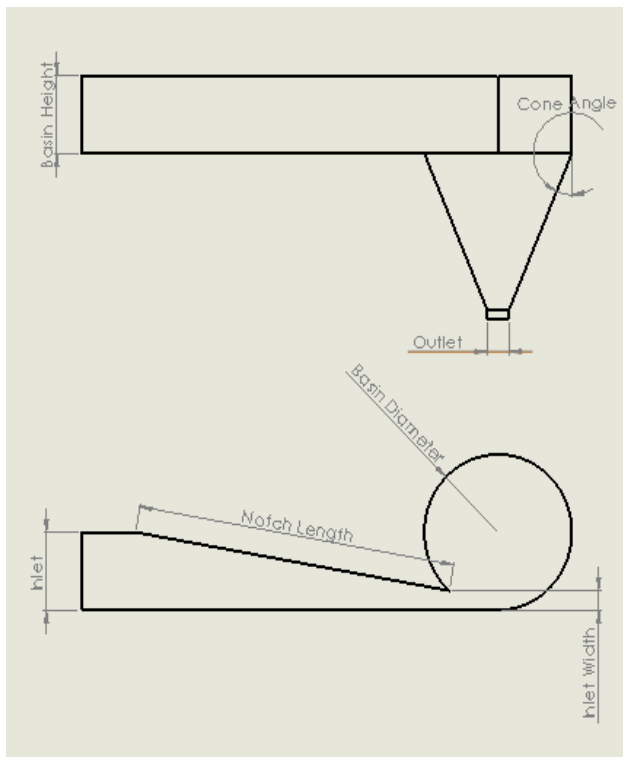

Figure 3: Different parameters of conical basin

The governing equations are discretized by the finite volume method (FVM) using the commercial CFD package ANSYS FLUENT 14.5. FVM is used to discretize the governing equations with suitable discretization schemes for each governing equation. To solve the discretization equation, steady pressure based segregated solver with double precision and implicit scheme is used. The second order method is used for the steady terms.

A SIMPLE method was used to solve the discretised equations. The second order up-winding method is used for the discretization of the momentum equation and other equations. This method provides a proper view of the physics of flow. The convergence criterion for all the equations is 10^{-4}.

Figure 2 shows the velocity distribution in confined vortex chamber. The velocity contours obtained seems to agree with these plots.

For the process of design optimization, at first simulation of existing basin was done. Then series of tests were done on the existing basin with and without Runner. Then the position of the turbine where it extracts maximum power was determined. The result of these tests found to agree with the tests done by previous researchers with some variation in power output. Then the simulation results were improved with better turbulence modelling and improved mesh. For understanding the effect of each parameter on vortex velocity and its distribution, each parameter was varied keeping other parameter constant and results were obtained. Empirical model and relationship building was then tried by Response Surface Methodology.

3. Experimental Setup and Data Collection

The preliminary design of basin was carried out using general laws of Fluid mechanics like Continuity Equation, Bernoulli Equation in reference of (Bajracharya & Chaulagai, Developing Innovative Low Head Water Turbine for Free-Flowing Streams Suitable for Micro-hydropower in Flat (Terai) Regions in Nepal, 2012). Series of test runs were done by varying various parameters that effect vortex formation in the basin. The best found Experimental set up was fabricated and was tested by (Bajracharya & Chaulagai, Developing Innovative Low Head Water Turbine for Free-Flowing Streams Suitable for Micro-hydropower in Flat (Terai) Regions in Nepal, 2012) and also by (Bajracharya, Thapa, Pun, Dhakal, & Nakarmi, 2013). The purpose of the research was to extract energy form thus formed vortex. For the purpose of power extraction a robust runner was fabricated and tested.

The inlet velocity was measured by using a simple methodology i.e. float method as no other devices for measuring inlet velocity was available. In this method an object was dropped in the canal. The time taken by the float to cover 1m distance was measured for several times and average was calculated. Then the average velocity of top surface of water was approximated as reciprocal of the average time taken. Later the correction factor of 0.85 was chosen as wall of flow channel are smooth. Thus, the mean velocity was approximated to be 0.1m/s.

4. Results and Discussion

Here, we have modeled the different basin with different parameters involved i.e. notch angle, notch

inlet width, canal height, diameter and cone vertical angle. This involved variation in the parameters of basin and consequently the simulation using commercial CFD code, ANSYS fluent and velocity contour were observed at the plane of interest. For the variation of one parameter all other parameters were kept constant.
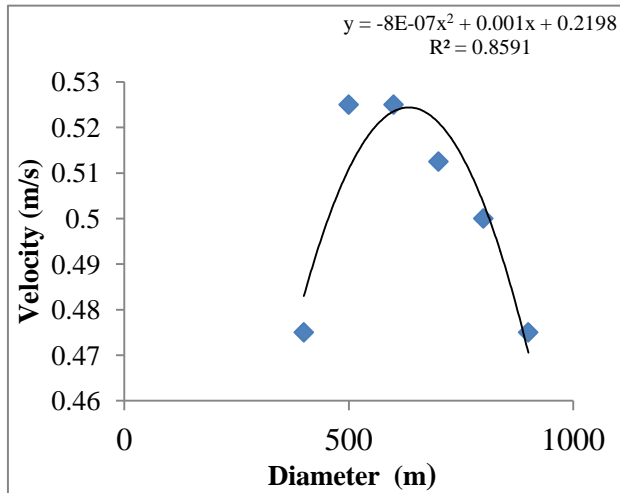
4.1 Effect of Basin Diameter

y = -8E-07x^2 + 0.001x + 0.2198
R^2 = 0.8591

Figure 4: Effect of Basin Diameter vs. Velocity for conical basin.

For vortex creation we have: $\frac{\Gamma}{\omega} = \frac{\omega A}{LA} = \frac{\omega}{L}$ = constant. As a vortex stretches, L increases, and since the volume is constant, from continuity equation of fluid mechanics, A and R decrease, and due to the conservation of the angular momentum, ω increases (MIT, 2011) . Therefore for a fixed height of the cone if the diameter is increased then cone angle gets increased thus ω increases. Here we have selected the basin diameter as 800mm so that a greater variation in runner position and diameter can be studied and vortex formation was smooth compared to that of other diameters.

4.2 Effect of Notch Angle

For various notch lengths (notch angles), the fluid flow was simulated and the variation is plotted in figure below. The velocity was found to increase in small range when the notch angle was varied from 10 degree to 70 degree.
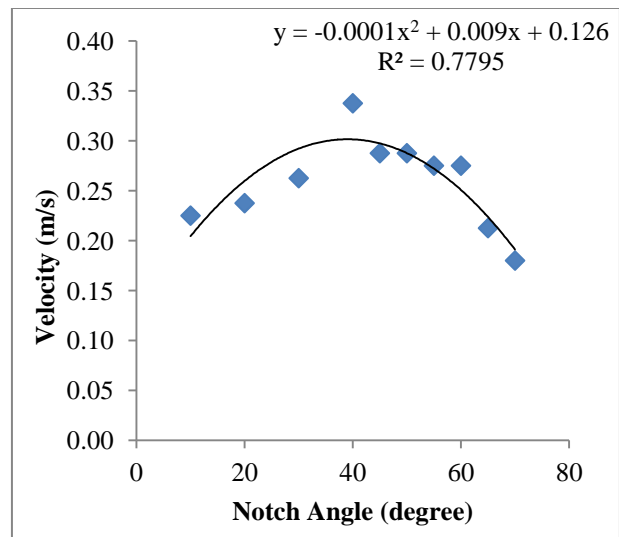

y = -0.0001x^2 + 0.009x + 0.126
R^2 = 0.7795

Figure 5: Notch Angle Vs Velocity

4.3 Effect of Notch Inlet width (Basin Opening)

The graph in figure 4.11 was obtained by simulation of existing basin geometry of conical basin keeping all other parameters constant but variation in basin opening. The graph depicts that as the basin opening increases, the velocity was found to decrease. Maximum velocity is obtained when the basin opening was kept 0.1m. The small opening area increases the inlet velocity of water into the basin. Thus, there is increase in velocity at region of interest.
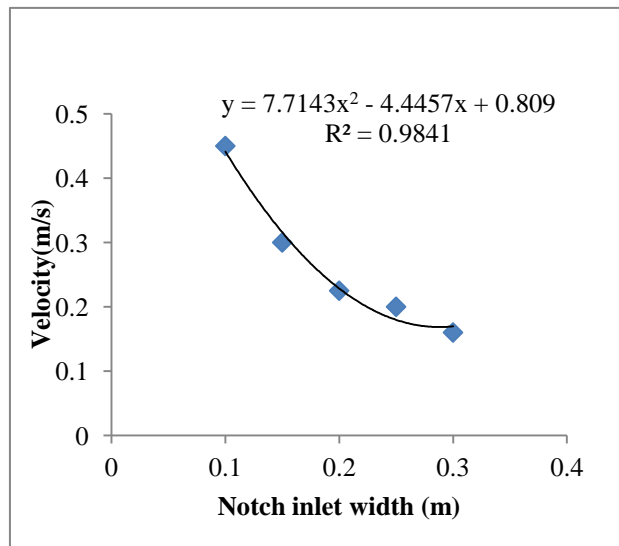

y = 7.7143x^2 - 4.4457x + 0.809
R^2 = 0.9841

Figure 6: Basin Opening Vs Velocity

4.4 Effect of Cone angle

Mulligan (2010), did his final year project in GWVPP and from his research he concluded that the optimum vortex strength occurs within the range of orifice

Effect of Dominant Parameters for Conical Basin: Gravitational Water Vortex Power Plant

diameter to tank diameter ratios (d/D) of 14 % - 18 % for low and high head sites respectively.

Now here for the given basin diameter of 800mm, the maximum cone angle that can be achieved is 23° as outlet hole diameter was fixed and also the velocity was also found maximum at this angle.
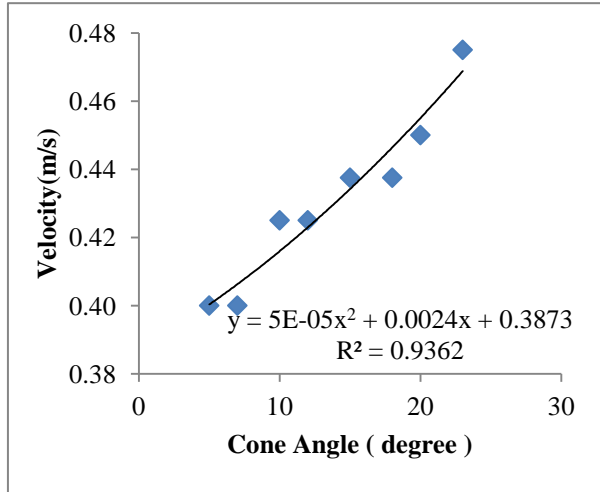

Figure 7: Cone Angle Vs Velocity
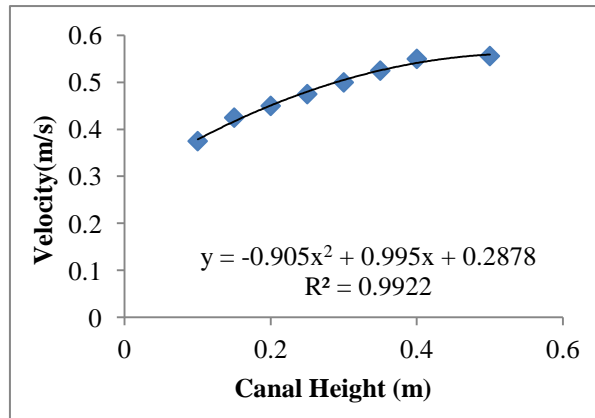
4.5 Effect of Canal height:

Figure 8: Canal Height Vs Velocity

The graph above was obtained by simulation of existing basin geometry of conical basin keeping all other parameters constant but variation in canal height (depth of canal). The graph shows that as the canal height is increased, the velocity also increases for a certain limit. When the canal height is increased beyond 0.4m, no significant variation in velocity was found.

5. Sensitivity Analysis:

The sensitivity analysis was done for the four parameters that affect the design of basin of GWVPP.

The four independent variables considered were the notch angle, basin opening, and canal height and cone angle. The dependent variable taken was the average vortex velocity at the plane of interest.
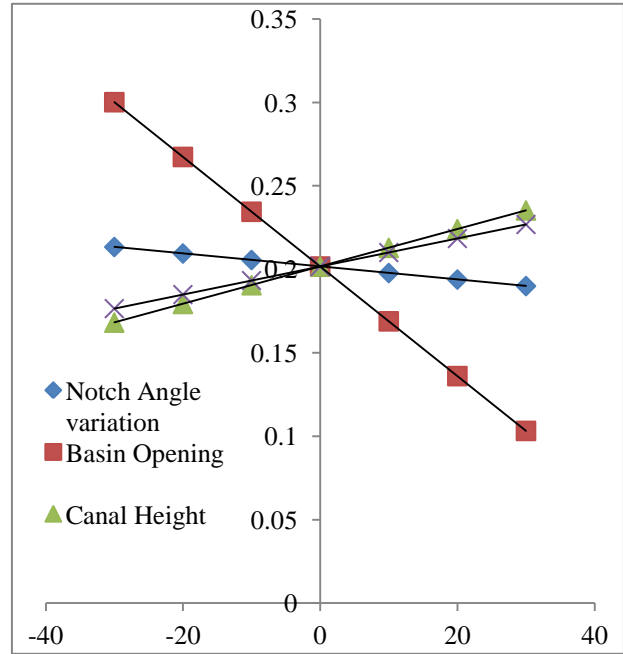

Figure 9: Sensitivity Analysis

The sensitivity graph shows that there is dominant effect of basin opening in the velocity. When the basin opening is gradually increased, the vortex velocity decreases. So, it is advisable to keep basin opening small. The optimum value of basin opening that was seen from optimization was 0.1m. The slope of basin opening is steeper than the slope of other parameters. Moreover, the graph depicts that there is no significant effect of notch angle in the velocity. When, the notch angle is varied from 100 to 700, there is change in velocity in a small range only. Similarly, when the value for canal height is increased, the vortex velocity also increases gradually as shown in figure. For a given basin of a certain diameter, the outlet tube diameter is within the range of 14 % to 18 % of the basin diameter. The value of velocity also increases as the cone angle is increased. But, the cone angle can't be increased beyond the certain limit. The optimum value of cone angle for basin diameter of 800mm was seen 23^0. Thus, the sensitivity analysis graph suggests that basin opening; canal height and cone angle should be taken into account for the design of GWVPP.

6. Conclusion

Various parameters that could be varied for the maximum power output was detected with their significance which was verified experimentally. Thus

the parameters were varied individually and its relationship with velocity at region of interest was studied. It was found that basin opening was most important factor to be considered during basin design of Gravitational water vortex power plant. Also notch length should not be decreased beyond some critical value as it creates unwanted turbulence at inlet region of basin. It is recommended to keep long notch length as it would gradually increase basin inlet velocity and thus will prevent unwanted losses. This study has determined the range up to which each parameter can be varied keeping all other parameters constant so that vortex structure will be such as to extract energy. This will help us to formulate constraints for the purpose of design optimization which is planned as future work.

References

Aravelli, A. (2014). Multi-objective Design Optimization of Engineering Systems: Uncertainty Approach and Practical Applications. University of Miami.

Bajracharya, T. R., & Chaulagai, R. K. (2012). Developing Innovative Low Head Water Turbine for Free-Flowing Streams Suitable for Micro-hydropower in Flat (Terai) Regions in Nepal. Kathmandu: Center for Applied Research and Development(CARD), Institute of Engineering, Tribhuvan University, Nepal.

Bajracharya, T. R., Thapa, A. B., Pun, P., Dhakal, S., & Nakarmi, S. (2013). DEVELOPMENT AND TESTING OF RUNNER AND CONICAL BASIN FOR GRAVITATIONAL WATER VORTEX POWER PLANT. Kathmandu: Institute of Engineering, Central Campus, Pulchowk.

Chen, Y., Wu, C., WANG, B., & Du, M. (2012). Three-dimensional Numerical Simulation of Vertical Vortex at Hydraulic Intake. Sichuan: Elsevier.

F. Trivellato, E. B. (1999). Finite volume modelling of free surface draining vortices. Journal of Computational and Applied Mathematics , 103, 175-185.

Inc., A. (2013). ANSYS Fluent User's Guide. Southpoints,Canonsburg, PA 15317: ANSYS Inc.

Kuch, T. C., Beh, S. L., Rilling, D., & Ooi, Y. (2014). Numerical Analysis of Water Vortex Formation for the Water Vortex Power Plant. International Journal of Innovation, Management and Technology , 5, 111-115.

Makky, A. A. (2014). Coding Tutorials for Computational Fluid Dynamics.

MIT. (2011). Marine Hydrodynamics . Lecture 8 .

Mulligan, S. &. (2010). The Hydraulic Design and Optimisation of a Free Water Vortex for the Purpose of Power Extraction. Sligo, Ireland: The Institute of Technology, Sligo.

Mulligan, S., & Hull, P. (2010). Design and Optimisation of a Water Vortex Hydropower Plant. Sligeach: Department of Civil Engineering and Construction, IT Silgo.

Ogawa, A. Vortex Flow.

Ogawa, A. (1993). Vortex Flow. Boca Raton, Florida: CRC Press INc.

Peroxide, U. (2014). Technology for a clean environment. Retrieved August 9, 2014, from http://www.h2o2.com/industrial/applications.aspx?pid=104&name=BOD-COD-Removal

Rout, S. K., Choudhury, B. K., SoHo, R. K., & Sarangi, S. K. (2014). CFD based Multi-objective parametric optimization of Inertance type pulse tube refrigerator using response surface methodology and non-dominated sorting genetic algorithm. Cryogenics(2014) , 5-6.

Rout, S., Choudhary, B., SoHo, R., & Sarangi, S. (2014). CFD based Multi-objective parametric optimization of Inertance type pulse tube refrigerator using response surface methodology and non-dominated sorting genetic algorithm. Cyrogenics .

Versteeg, H. K., & Malalasekera, W. (1995). An Introduction to Computational Fluid Dynamics. In H. K. Versteeg, & W. Malalasekera, An Introduction to Computational Fluid Dynamics (pp. 2-7). New York: John Wiley & Sons Inc.

Wanchat, S., & Suntivarakorn, R. (2011). Preliminary Design of a Vortex Pool for Electrical Generation. Khon Kaen: Department of Mechanical Engineering, Khon Kaen University.

Wanchat, S., Suntivarakorn, R., Wanchat, S., Tonmit, K., & Kayanyiem, P. (2013). A parametric Study of a Gravitational Vortex Power Plant. Advanced Materials Research , 805-806, 811-817.

Wang, Y.-k., Jiang, C.-b., & Liang, D.-f. (2010). Investigation of air-core vortex at hydraulic intakes. Journal of Hydrodynamics , 673-678.

Zotlöterer. (2014, August). ZOTLÖTERER SMART - ENERGY - SYSTEMS. Retrieved August 09, 2014, from http://www.zotloeterer.com/welcome/gravitation-water-vortex-power-plants/zotloeterer-turbine/