ISSN 2229-5518

EFFECT OF OUTLET DIAMETER ON THE PERFORMANCE OF GRAVITATIONAL VORTEX TURBINE WITH CONICAL BASIN.

Sreerag S.R*., Raveendran C. K*.Jinshah B S*
Department of Mechanical Engineering, Government Engineering College, Kozhikode - 673005

Abstract--- **This study focuses on the formation of a water vortex stream by gravitation, which is a new technique in the field of hydro power engineering. The water passes through a large straight inlet, and then passes in to a vertical conical channel, forming a vortex, which finds its outlet at the centre bottom of the shallow basin. Turbine blade is allowed to rotate in the vortex which in turn produces electricity by means of a generator. This type of turbine can be called as gravitational vortex turbine. The advantage of this method for electrical generation is the capability of producing energy using low heads (in the range of .7m to 3m).The turbine does not work on the pressure difference but on the dynamic force of the vortex.**

A computational fluid dynamic analysis (CFD) is to be carried out using ANSYS Fluent 14.5. The tangential and radial velocities at different planes in the flow field for different outlet diameter will be plotted. From the analysis an optimal outlet diameter will be predicted. To validate the results obtained are to be compared with a previous study.

After the analysis and validation of the result , it is found that maximum tangential velocities are obtained when outlet diameter approaches 30 % of basin diameter. So a when Gravitational vortex turbine with a conical basin is to be built, an outlet diameter which is 0.3 times the basin diameter (0.3D) gives the maximum output.

Keywords--- **Gravitational Vortex, Conical basin, Computational fluid dynamics**

INTRODUCTION

Energy exists freely in nature. Some of them exist infinitely which never run out, called renewable energy. Rest have finite amounts and will run out one day, called non-renewable energy. For years men have been using non-renewable energy for his needs. The adverse environmental impact and non-renewable nature of fossil fuels made the researchers around the world to turn towards renewable energy resources to meet the growing demand for electricity. None of CO_2 free technologies which are technically mature today and in the near future tackle the problem. So an efferent storage and transmission system are also key factors. Water power is one of the strong candidates towards sustainable future: Water power with wind power, are the most effective renewables

For thousand years hydel energy is being used for various purposes including electricity generation. As a result of continuous research, significant improvement in operating efficiency is achieved, which makes them economically competitive with other renewable energy generation techniques. Along with the need for sustainable energy techniques, hydro electric energy systems are increasing in their market share faster than any other renewable energy systems. Due to variability in availability for hydel energy, it cannot aim to sole electricity source for a single country, making it a competitive source of renewable electricity. It is also a flexible source of electricity since the amount produced by the station can be changed up or down very quickly to adapt to changing energy demands.

Various methods are adopted so far for the electrical energy generation from hydro energy which can be classified as large hydropower (capacity of more than 30 Megawatts), small hydropower (capacity of 100 Kilowatts to 30 Megawatts), and micro hydropower (capacity of up to 100 Kilowatts) .Gravitational vortex power generation belongs to micro hydro-power plants which is a new technology emerged recently.

I. BRIEF OVERVIEW OF GRAVITATIONAL VORTEX TURBINE

While trying to find a method for aerating water without energy input, Austrian engineer Franz Zotlöterer stumbled across the idea of a mini-power plant which is simple to construct and has a turbine efficiency of 80% but which is safe for fish due to low turbine speed and improves water quality by oxygenation. Needless to say, he quickly applied for the patents. The technology can be applied with water drop as little as 0.7 meters. To say about the specification the vortex basin has a diameter of 5.5 meters. In the first year of operation, the plant has yielded 50,000 kWatt-hours of electricity. Some advantages of gravitational vortex power plant are listed below.

1. Easy to maintain,
2. Compact.
3. Inexpensive.
4. Their construction will restore the body of water (e.g. A river) on which they are built, benefiting nature and the local community.
5. They pose no threat to the fish population, as they are able to bypass the rotor downstream and upstream.
6. Improved cleaning efficiency of natural micro-organisms thanks to the higher oxygen levels resulting from the regular aeration of the water.
7. Water vortex plants have little impact on the local environment as most of the construction work is below ground.
8. With correct planting, the basin itself can be hidden, rendering the entire facility virtually invisible.

II. STUDIES SO FAR

Sujate 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 centre of the vortex basin. In the case of 1m diameter cylindrical vortex basin, computational fluid dynamics (CFD) and experiment using

Sreerag S R, Student, Mechanical Engineering, Govt Engineering college, Kozhikode, India. E-mail: sreerag1989@gmail.com
Raveendran C.K, Associate Professor, Mechanical Engineering, Govt Engineering college, Kozhikode, India, E-mail: raveendran.ck@gmail.com
Jinshah B S,Assistant professor ,UKF College of Engineering and Technology,Kollam,Kerala,India ,Email-jinshah.b.s@gmail.com

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%.

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.

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 .

Ratchaphon Suntivarakorn et al (2012) studied 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 centre with the incoming flow guided by a plate is the most suitable configuration to create the kinetic energy water vortex.

Subash Dhakal et al (2012), have compared conical basin and cylindrical basin for the power plant. Conical basin for the turbine and found that vortex formation was aided by conical basin. Use of conical created a significant increase in vortex strength as shown in the vortex strength comparison table. Tests of the same turbine on conical basin provided the maximum efficiency of 29.63% which is significantly greater than the values provided by all the tests on cylindrical basin. Hence it is preferable to use conical basins for gravitational water vortex turbines.

Rabin Dhakal et al also have done Comparative numerical and experimental study of cylindrical and conical basins of Gravitational water vortex power plant the numerical and experimental study on this plant asserted that output power and efficiency is maximum in conical basin compared to that of cylindrical basin for all similar inlet and outlet condition

III. COMPUTATIONAL FLUID DYNAMICS-CFD

CFD is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyse problems that involve fluid flows. Computer resources to simulate flow related problems.CFD includes three elements to do one analysis. (i) a pre-processor, (ii) a solver and (iii) a post processor. They are briefly explained below.

Pre-Processor

Pre-processing is the first step in CFD in which we give the inputs to the CFD program means of a user interface and the subsequent transformation of this input into a form suitable for use by the solver. Pre-processor involves following tasks

- Definition of the geometry or domain. (Can be imported from other CAD software or can be created inside the software)
- Discretization of the domain in to small elements (Finite element, or finite volume).It can be also called as meshing.
- Select the physical or chemical process which is to be modelled
- Definition of fluid properties.
- Giving appropriate boundary conditions.

Solver

There are three methods of discretization of complex equations. Finite difference, finite element and finite volume method. 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

Post-processor helps to show the results obtained after the analysis. Post processor includes.

- Domain geometry and grid display
- vector plots
- Surface plots
- Particle Tracking etc.

IV. MODEL DEVELOPMENT.

In order to investigate the velocity vector flow field, a solid model is built in Solidworks with dimensions as that of previous study.Dimensions are given in figure 1. The model is a conical tank with an orifice at the bottom centre. The incoming flow is guided by a plate. Tank size is 0.5m in diameter and 1m in height. And the orifice diameter is varied for 3 different values .Cone angle is assumed which is fixed to a certain value (14 degree) after first analysis. The upper surface is set open to the ambient air. There are no-slip conditions at the wall and there is a pressure outlet at the orifice. The incoming velocity was set at 0.1m/s.

ISSN 2229-5518

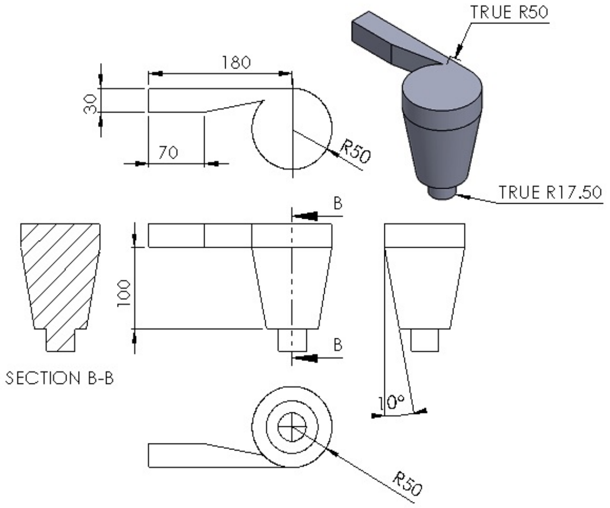


Figure.1 Dimensions of the basin and runner

V. ANALYSIS TO FIND OUT OPTIMUM CONE ANGLE

Before going in to the analysis for finding out optimum outlet diameter for the basin we need to fix the cone angle. For that first an arbitrary outlet diameter is selected and then performed the CFD analysis with boundary conditions by varying the different cone angle from 10 degree to 18 degree cone angle Figure 2 and tangential velocities at different heights as shown in the figure 3 are plotted.

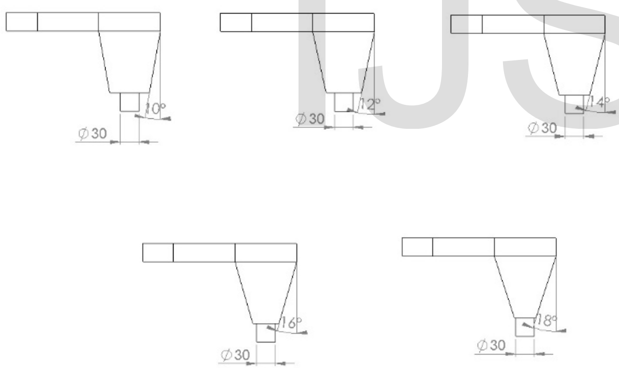


Figure.2 varying cone angles of the basin.

Boundary Conditions:

- Inlet: Velocity Inlet (0.1m/s)
- Wall: No slip wall.
- Pressure outlet with gauge pressure zero.

Figure 3 shows the graphs plotted along the line drawn at 0.1 meters form the bottom face in which 18 degree cone angle has given the highest value of tangential velocities among different cone angles

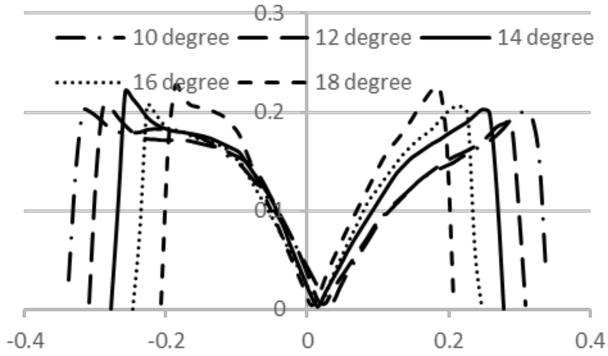


Figure.3 Tangential velocity distribution along the line drawn at 0.1 meters form bottom face for different cone angle.

Same results are obtained for heights 0.3 ,0.5,0.7 and 0.9 meters form the bottom face.and it can be seen that for 18 degree cone angle, tangential velocity has got higher values. Second highest value is not for 16 degree but 14 degree cone angle. Given below figure 4 and figure 5 is the analysis results obtained for the basin with and without a base plate. For the same dimensions except that of base plates.

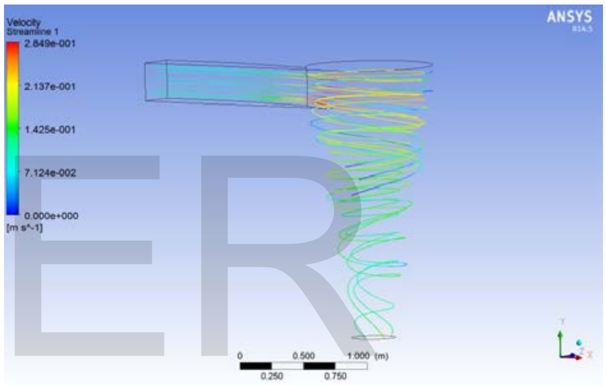


Figure.4 Stream lines without base plate.

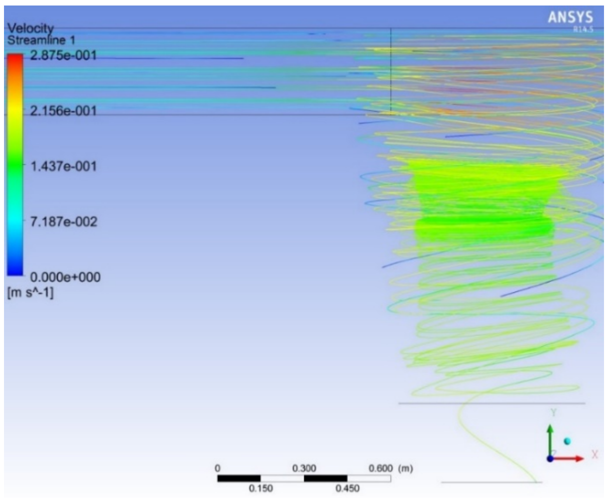


Figure.5 Stream lines with base plate.

It can be seen that model with base plates gave a strong stable vortex figure 5. From figure 2 one can observe that as the cone angle approaches 18 degree, base plates vanishes .Since base plate is a good candidate for stable vortex generation one can select second option of 14 degree cone angle which gave second largest values for tangential velocities.

VI. ANALYSIS TO FIND OUT OPTIMUM OUTLET DIAMETER

For this purpose a model is taken from literature in which dimensions are specified for cylindrical basin except cone angle. Cone angle is selected as 14 degree from our previous analysis. Fluid domain of the same is given below with the dimensions in which all dimension are in centimetres.

Continuity equation and Navier-Stokes equations are governing equations. These two equations in cylindrical coordinates are expressed as.

$$\frac{\partial V_r}{\partial t} + V_r\frac{\partial V_r}{\partial r} + V_z\frac{\partial V_r}{\partial z} - \frac{V_\theta^2}{r} = 2\omega_0 V_\theta - \frac{1}{\rho}\frac{\partial P}{\partial r} + \nu\left(\frac{1}{r}\frac{\partial V_r}{\partial r} + \frac{\partial^2 V_r}{\partial r^2} + \frac{\partial^2 V_r}{\partial z^2} - \frac{V_r}{r^2}\right) \quad (1)$$

$$\frac{\partial V_z}{\partial t} + V_r\frac{\partial V_r}{\partial r} + V_z\frac{\partial V_z}{\partial z} - \frac{V_\theta^2}{r} = -\frac{1}{\rho}\frac{\partial P}{\partial r} + \nu\left(\frac{1}{r}\frac{\partial V_z}{\partial r} + \frac{\partial^2 V_z}{\partial r^2} + \frac{\partial^2 V_z}{\partial z^2} - \frac{V_z}{r^2}\right) \quad (2)$$

$$\frac{\partial V_\theta}{\partial t} + V_r\frac{\partial V_\theta}{\partial r} + V_z\frac{\partial V_\theta}{\partial z} - \frac{V_\theta V_r}{r} = 2\omega_0 V_r - \frac{\partial P}{\partial r} + \nu\left(\frac{1}{r}\frac{\partial V_\theta}{\partial r} + \frac{\partial^2 V_\theta}{\partial r^2} + \frac{\partial^2 V_\theta}{\partial z^2} - \frac{V_\theta}{r^2}\right) \quad (3)$$

$$\frac{1}{r}\frac{\partial}{\partial r}(rV_r) + \frac{\partial V_z}{\partial z} = 0 \quad (4)$$

Where
V_r is the velocity along radial direction (m/s)
V_θ is the velocity along θ direction (m/s)
V_z is the velocity along Z-direction (m/s)
ω Angular velocity (rad/s)

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. 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 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.

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 discretized 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}.

When one does CFD analysis with an arbitrary mesh refinement size, It ends up with a particular set of solutions which may or may not be accurate. Further if the refinement level is increased resulting in more accurate set of solutions, which implies that the first refinement level is not sufficient. To estimate the refinement level of the elements we do mesh independence study in which element size is being reduced until a constant value of solutions are obtained for which further reduction in the element size gives least change from the previous value. It is to be noted that as the refinement size increases, number of elements are increased .Steps involved in the study are given in the appendix

Because your solution is changing with the refinement of mesh, you have not yet achieved a mesh independent solution. You need to refine the mesh more, and repeat the process until you have a solution that is independent of the mesh. You should then always use the smallest mesh that gives you this mesh independent solution.

For this work, a graph, Figure 6 is plotted for maximum tangential velocity against number of elements. As the number of elements increased from 90,000 to higher values by increasing refinement level, tangential velocity value seems to be increasing and it changed until the 2, 53,788 after which increase in number of elements has least no effect since graph showed a constant line. So a value of 2, 53,788 is taken as the number of elements for the entire analysis.

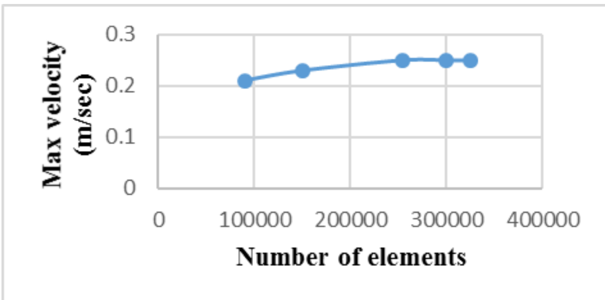


Figure.6 Mesh in depend study

ISSN 2229-5518

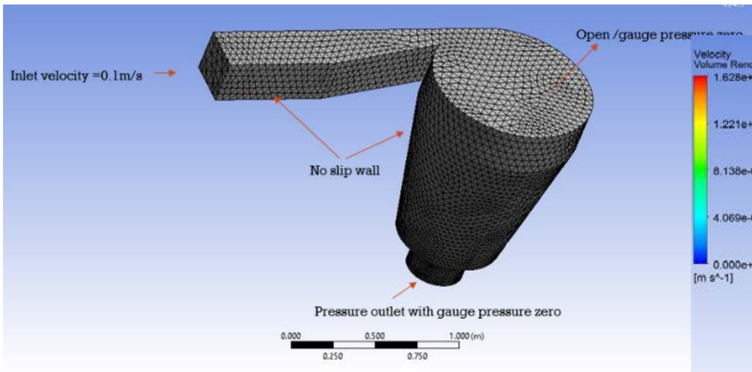


Figure.7 Meshed model with boundary conditions

Pressure distribution

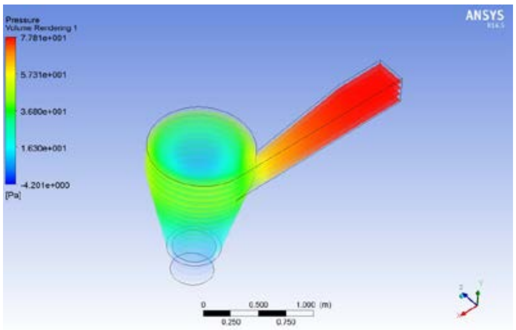


Figure. 8 Pressure distribution for 400 mm outlet diameter

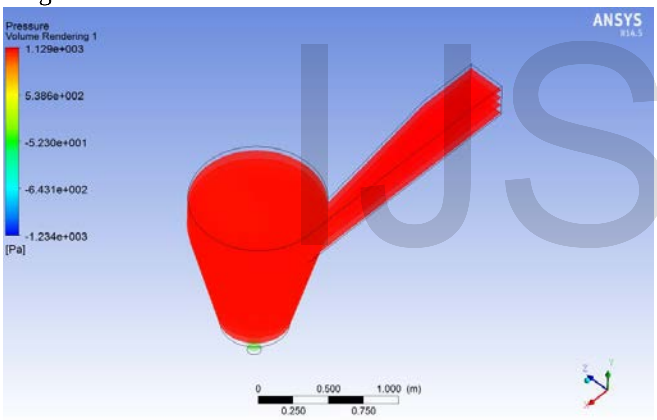


Figure. 8 Pressure distribution for 100 mm outlet diameter

It can be observed that as outlet diameter decreases, pressure developed increases. Figure () shows that pressure developed is too highs as an indication of hindered flow.

Velocity distribution

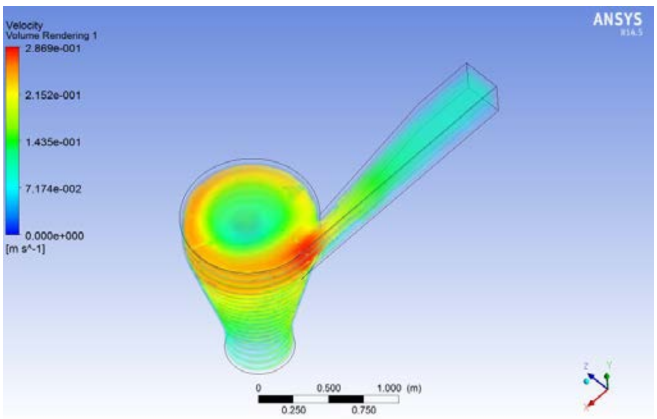


Figure.9 Velocity distribution for500mm Outlet diameter

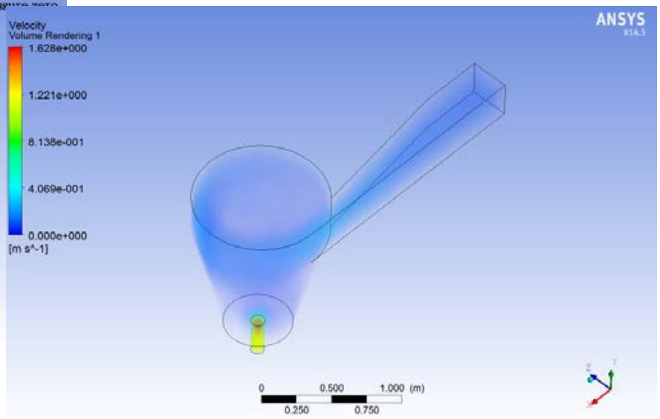
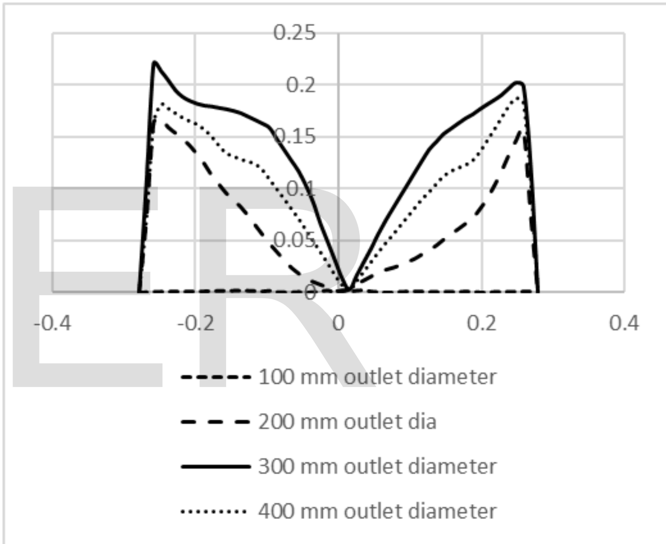


Figure.10 Velocity distribution for 100 mm Outlet diameter

In velocity distribution results one can observe that velocity values are decreased outlet diameter is decreased.

Tangential velocity distribution for different outlet diameter

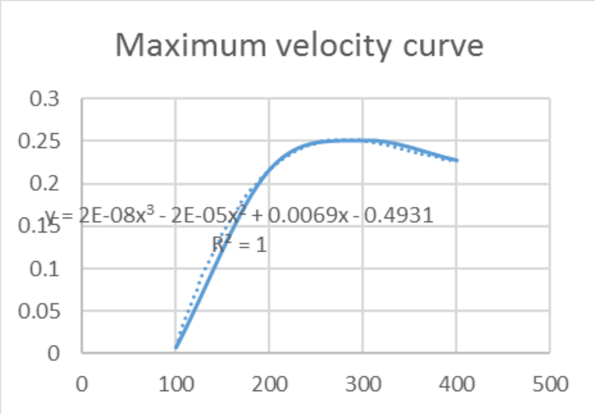


From Graph () one can observe that as outlet diameter increases from 100 mm to 400 mm tangential velocity values reaches a maximum value and then decreases. Here we can see one gets maximum tangential velocity for 300 mm outlet diameter (0.3 D) where D is the diameter of basin.

Same trend can be seen for all the heights measured from bottom plate.

Maximum velocity curve

Maximum velocity points are taken from each results for different outlet diameter and plotted a maximum velocity curve is plotted and an equation is generated as given below



$V_t = 2E\text{-}08*do_3 - 2E\text{-}05*do_2 + 0.0069do - 0.4931$
Where,
V_t = tangential velocity
do = orifice diameter

On analyzing this equation one will see the highest tangential velocity is 0.25m/s for an outlet diameter of 277.92 mm which can be approximated as 0.3D where D is basin diameter.

Same analysis is done on scaled up and scaled down model.For scaled up model all the dimensions and inlet velocity conditions are doubled. And for scaled down model all the dimensions and inlet velocity conditions are halved except cone angle.cone angle is kept 14 degree through out the analysis.

Maximum velocity curve for scaled up and scaled down model are plotted and the results are tabulated as given in the table

VII. RESULTS AND DISCUSSION

Basin diameter Outlet (D) mm	Orifice diameter (0.3 D) mm	Tangential velocity m/s
1000mm	300 mm	0.25m/s
500 mm(Scaled down)	150 mm	0.12m/s
2000 mm(Scaled up)	600 mm	0.54m/s

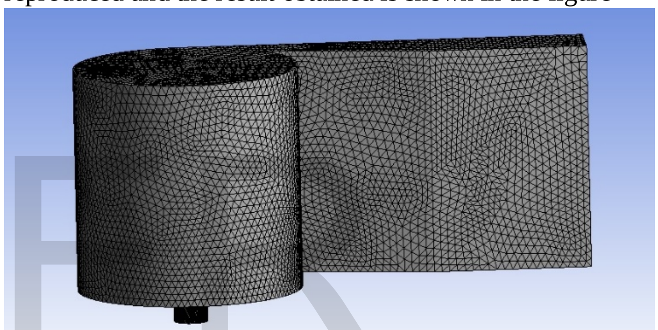
Basin diameter Outlet (D) mm	Orifice diameter (≈0.3 D) mm	Tangential velocity m/s
1000mm	277.92 mm	0.251m/s
500 mm(Scaled down)	166.6 mm	0.1244m/s
2000 mm(Scaled up)	681 mm	0.543m/s

Table 7.1 Highest tangential velocity for different scales of model.

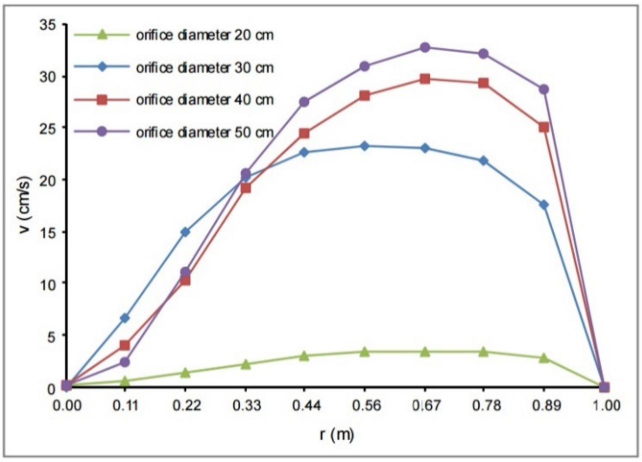
Results obtained from least square regression method- Orifice diameter and tangential

VIII. VALIDATION.

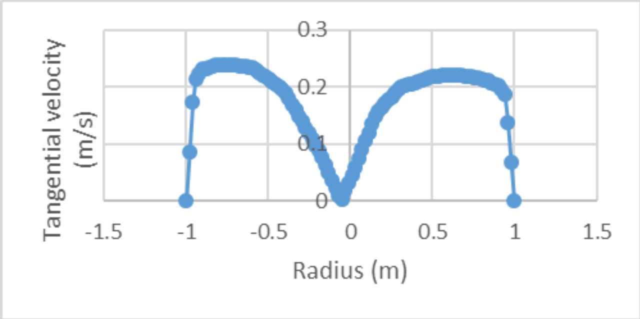
In order to validate the CFD analysis , a previous work done by Sujate Wanchat and Ratchaphon Suntivarakorn [1] , with the same boundary conditions and mesh quality (Skewness:0.35,Orthogonality:0.89) ,which is already validated experimentally is selected. Their work is reproduced and the result obtained is shown in the figure



Meshed model for the cylindrical basin



Tangential velocity distribution for different orifice diameters [1]



Tangential velocity for 300 mm outlet diameter from analysis.

On comparing the graphs, a maximum error of 2 % is obtained

IX. CONCLUSION

CFD analysis is conducted on Gravitational Vortex turbine with conical basin and effect of outlet diameter on the performance of the turbine based on tangential velocity is studied. A model with 1 meter basin and 1meter height is modelled, dimensions of which are taken from the literature studies except cone angle. Prior to the optimization of the outlet orifice diameter an analysis is conducted to fix the cone angle. 14 degree cone angle for the basin is opted from 10 degree, 12 degree, 14 degree, 16 degree and 18 degree, on the reason that it gave a strong stable vortex along with high tangential velocity.

After fixing the cone angle of the basin CFD analysis is performed to optimize the outlet orifice diameters. Analysis is conducted on the models with orifice dimensions varying from 0.1 D, 0.2D, 0.3D, 0.4D, 0.5D, Where D is the diameter of basin. From the analysis it is found out that 0.3 D has given maximum tangential velocity which is component relevant to drive the turbine blade, and more vortex stretching which is in proportional to the power output. Analysis reveals that the maximum tangential velocity is related to the outlet orifice diameter with a polynomial of order 3.The results agrees with scaled down and scaled up model.

CFD result obtained is validated with previous work on cylindrical basin done by Sujate Wanchat et al. Their work is reproduced same conditions and mesh quality. And result obtained is in accordance with their results with 2% error. The trend is in accordance with Monji's experiment for CFD-based Evaluation of Interfacial Flows

So the result is concluded as, for Gravitational vortex turbine with conical basin an outlet orifice of diameter 30% of basin diameter gives highest tangential velocity and thus maximum output.

Suggestions for Future work: Building a real time model and experimental study on conical basin can be conducted on the work.

Same studies can be done on different basin shapes other than conical and cylindrical shapes.

X. REFERENCES

[1]. Sujate Wanchat , Ratchaphon Suntivarakorn, "Preliminary Design of a Vortex Pool for Electrical Generation" Advanced Science Letters, Volume 13, Pages 173–177 ,30 June 2012.

[2]. Kei Ito1, Hiroyuki Ohshima1, Takaaki Sakai and Tomoaki Kunugi, "CFD-based Evaluation of Interfacial Flows" ISBN 978-953-7619-59-6, pp. 420, January 2010, INTECH, Croatia.

[3]. Sagar Dhakal,Ashesh B. Timilsina,Rabin Dhakal , Dinesh Fuyal , Tri R. Bajracharya , Hari P. Pandit , Nagendra Amatya , Amrit M. Nakarmi , "Comparison of cylindrical and conical basins with optimum position of runner: Gravitational water vortex power plant" Renewable and Sustainable Energy Reviews 48 (2015) 662–669.

[4].Sujete Wanchat ,Sujin Wanchat,Ratchaphon Suntivarakorn "A parametric study on gravitational vortex power plant"Advanced Materials Research Vols. 805-806 (2013) pp 811-817 (2013) Trans Tech Publications, Switzerland.

[5]. Subash Dhakal, Susan Nakarmi, Pikam Pun, Arun Bikram Thapa, Tri Ratna Bajracharya, "Development and Testing of Runner and Conical Basin for Gravitational Water Vortex Power Plant" Journal of the Institute of Engineering, Vol. 10, No. 1, pp. 140–148

[6]. Tze Cheng Kueh, Shiao Lin Beh, Dirk Rilling, and Yongson Ooi, "Numerical Analysis of Water Vortex Formation for the Water Vortex Power Plant"International Journal of Innovation, Management and Technology, Vol. 5, No. 2, April 2014.