Proceedings of 8th IOE Graduate Conference
Peer Reviewed
ISSN: 2350-8914 (Online), 2350-8906 (Print)
Year: 2020 Month: June Volume: 8

Design Simulation for Runner of Horizontal Spiral Turbine applicable in Hydrokinetic flow

Alisha Rajbanshi [a], Raj Kumar Chaulagain [b]

[a, b] *Department of Mechanical Engineering, Thapathali Campus, IOE, TU, Nepal*
Corresponding Email: [a] alisha.techno@gmail.com, [b] rajkrc12@gmail.com

Abstract

Horizontal Spiral Turbine is able to efficiently intercept kinetic energy from water. This study focuses whether its well-connected blades are suitable for low velocity, in a small-sized water receiving area or not. The analytical analysis was based was based on velocity triangle method. The geometry of the test turbine was designed in Solid Works and the meshing or discretization procedures were carried out in ANSYS Mesh 15.0. The computational simulation was carried out by CFX 15.0 solver. The earth's tides, waves, free flowing rivers and oceanic current all contains clean renewable resources which produce hydrokinetic energy. The hydrokinetic systems have relatively small scale power production with lower power coefficients. The maximum efficiency that can be reach is 59.3% which is also known as Betz limit. This paper presents the effect of water velocity on turbine efficiency for a spiral horizontal axis hydro turbine. Modeling and a computational fluid dynamic (CFD) analysis were performed in order to determine the maximum torque of the turbine. The water velocities were varied as 1, 1.5, and 2 m/s and the force, torque, power and efficiency was observed. The force obtained at the velocity of 1m/s, 1.5m/s and 2m/s was found to be 6.498N, 14.612N and 25.974N respectively, torque obtained was 0.0725Nm, 0.163 Nm and 0.291Nm for 1m/s, 1.5m/s and 2m/s respectively and efficiency of turbine obtained was found to be 37.8125%, 25.2509% and 18.9676% for 1m/s, 1.5m/s and 2m/s respectively from the CFD simulation of the turbine. The efficiency observed is within the Benz limit. This research can be conducted following same parameter but in turbulent flow or varying the different parameter like blade angle, twist angle, etc. in future to optimize the runner of the horizontal spiral turbine.

Keywords

Horizotal Spiral Turbine, Computational fluid dynamic, Turbine effficiency

1. INTRODUCTION

1.1 Background

The rising demand for energy, hikes in oil prices, depletion of fossil fuels, and the increasing concern for environmental issues have challenged researchers to develop new technological processes to obtain clean and sustainable energy mainly through the utilization of renewable energy sources. [1] Hydro energy is clean and environment friendly energy resource. Maximum amount of energy can be generated from the water as compared to other renewable resources. There are mainly two approaches to harness energy from water, namely, hydrostatic and hydrokinetic methods. Hydrostatic approach is the conventional way of producing electricity by storing water in reservoirs to create a pressure head and extracting the potential energy of water through suitable turbo-machinery [2]. In hydrokinetic approach, the kinetic energy inside the flowing water is directly converted into electricity by relatively small scale turbines without impoundment and with almost no head [3]. Nepal claims snowy mountains (Himalayan range) in the North which acts as a perennial source for many free flowing rivers establishing the country as second richest in water resources in the world after Brazil [4]. Due to steep gradient and mountainous topography, Nepal is blessed with the abundant hydro resources. The country's three major river systems and their smaller tributaries offer Nepal to produce economically and technically feasible nearly 50,000 MW power. Nepal can potentially generate over 90,000 MW hydropower [5]. At present, Nepal's total power generation is around 24223 MWh of which Nepal Electricity Authority (NEA) generates 10690 MWh and

Independent Power Producers (IPP) generates 13533 MWh. The electricity demand is 25659 MWh. The remaining electricity of 1436MWh is imported to fulfill the demand. [6] .Government of Nepal has prioritized energy sector as a key driver for meeting the government's slogan of "Prosperous Nepal – Happy Nepali". The government has set a generation target of 15,000 MW in the next ten years and aims to make the country energy secure. There are opportunities for regional connectivity allowing for energy trading. After many years of facing energy shortage, Nepal is presently well on its way to being an energy surplus nation in the coming year. Projects with over 2,500 MW capacity are under construction, and 2,900 MW capacity projects are ready for construction. Similarly projects with about 18,000 MW capacity are under study [7]. In Nepal, hydrokinetic approach for off-grid energy generation in remote areas is favored. This research focuses on design simulation for runner of horizontal spiral turbine applicable in hydrokinetic flows. The blade of horizontal spiral turbine follows the Archimedes spiral to generate torque and power. Archimedes' spiral is an Archimedean spiral with polar equation

$$r = a * \theta \quad (1)$$

in which r is the radius from the center, a is the constant and θ is the angular position. This spiral was studied by Conon, and later by Archimedes in On Spirals about 225 BC. Archimedes was able to work out the lengths of various tangents to the spiral. Archimedes' spiral can be used for compass and straightedge division of an angle into n parts (including angle trisection) and can also be used for circle squaring. [8]

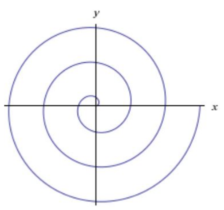


Figure 1: Archimedean Spiral at constant Speed

Aubin, et al [9] found that the horizontal spiral pattern turbine was able to efficiently intercept kinetic energy in the fluid. This kind of turbine is also safe for marine animals, and it blends well with the original ecosystem. Having a small radius creates little traction force. The water receiving area and the well-connected blades were suitable for low velocity, so this type of turbine was stronger and more suitable than the others which had a small sized water receiving area. Ratchaphon, et al. [10] found that the horizontal spiral pattern turbine designed by applying the Golden Ratio [11] as a function of blade radius expansion had optimal performance compared to a turbine with 3 blades and a turbine diameter to turbine length (D/L) ratio of 2/3. Such a turbine was able to generate electricity efficiently with a water velocity range of 0.5 – 2 m/s. Wiroon et al. [12] found that turbine efficiency with a collection chamber is higher than that of free-flow and the blade angle has significant effect on turbine efficiency for a spiral horizontal axis hydro turbine using modeling and a computational fluid dynamic (CFD) analysis to determine the maximum torque of the turbine. Till now all researches on the spiral horizontal turbines have focused on the optimum number of blades, blade radius-spindle length ratio, different blade angles, torque efficiency with and without collection chamber but there is no any study to compare the efficiency of turbine in low inlet velocity of water and low angular velocity in the steady flow condition. The comparative study of numeral analysis and ANSYS fluent will be able to address the above problem.

2. RESEARCH METHODOLOGY

2.1 Mathematical Model Development

The spiral turbine blade shape will be designed following the Archimedean circle to whirl around the core using the Golden Ratio function [10] which is a general natural Mathematical serial number. The ratio to the shape of the turbine was the number of blades. When it turns one full round around the axle or 360 degrees with a stable length.

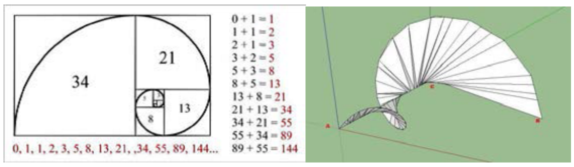


Figure 2: Spiral Water Turbine Blade of Golden Ratio Function Feature [10]

In case of the Archimedes spiral turbine, the flow of water at the inlet is axial i.e. parallel to the axis of turbine and the flow of water at outlet is radially outward. Since the flow of water is radially outward, centrifugal force created during the rotation of the runner is positive which boost the rotational motion.

Thus the speed control is not easy.

2.2 Velocity Triangle at inlet and outlet of the blade

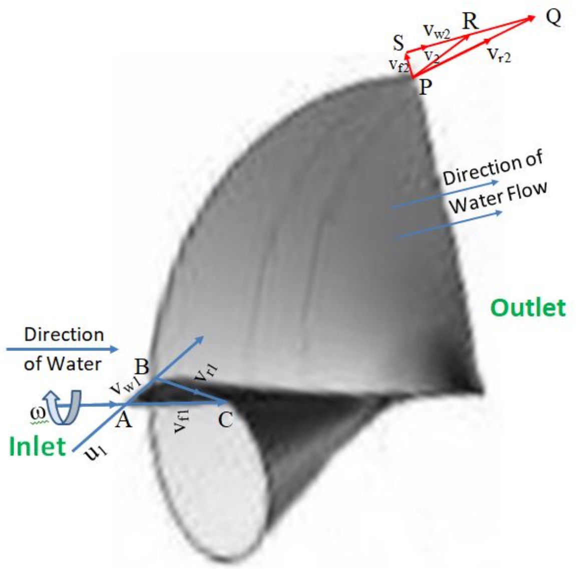


Figure 3: Velocity triangle at inlet and outlet

The different velocity component in the inlet and the outlet of the blade of horizontal spiral turbine forming the velocity triangle are as follow:

u_1= tangential velocity at inlet(m/s)

u_2= tangential velocity at outlet (m/s)

v_1= velocity of water at inlet (m/s)

v_2= absolute velocity of water at outlet (m/s) v_{r1}= relative velocity of water at inlet

v_{w1}= component of relative velocity at inlet along the tangential velocity

v_{f1}= component of relative velocity at inlet perpendicular to tangential velocity.

v_{r2}= relative velocity of water at outlet

v_{w2}= component of relative velocity at outlet along the tangential velocity.

v_{f2}= component of relative velocity at outlet perpendicular to tangential velocity.

α= absolute angle or angle made by velocity of water at inlet with axis of turbine.

θ= Blade angle or angle made by relative velocity with tangential velocity at inlet.

β= absolute angle or angle made by absolute velocity of water at outlet with tangential velocity.

ϕ= blade angle at outlet

2.2.1 Mathematical calculation and formula

$$u_1 = \omega * R1 \tag{2}$$

Momentum of water at inlet striking the blade per second along axis (pi)

$$p_i = \rho * Av_1 * v_1 \tag{3}$$

Angular Momentum of water at inlet per second (Li)

$$L_i = \rho * Av_1^2 * R_1 \tag{4}$$

Momentum of water at outlet (p_o)

$$p_o = \rho * Av_1 * v_{w2}\cos(\phi) \tag{5}$$

Angular Momentum of water at outlet per second (L_o)

$$L_o = \rho * Av_1 * v_{w2} Cos\phi R_2 \tag{6}$$

where, R_2= Radius of blade at outlet and ρ is density of water

Torque exerted by water(τ)= Rate of change of angular momentum

$$\tau = \rho * Av_1^2 * R_1 + \rho * Av_1 * v_{w2}\cos(\phi) R_2 \tag{7}$$

Work Done per second on turbine (W) W= Torque*Angular velocity
$W = \rho * Av_1^2 * \omega R_1 + \rho Av_1 * \omega R_2 v_{w2}\cos(\phi)$
$W = \rho * Av_1(v_1 \omega R_1 + \omega R_2 v_{w2}\cos(\phi))$

$$W = \rho * Av_1(v_1 u_1 + u_2 v_{w2}\cos(\phi)) \tag{8}$$

Efficiency of Turbine= W/K.E.
$\eta = \frac{[\rho * Av_1(v_1 u_1 + u_2 v_{w2}\cos(\phi))]}{(1/2 * \rho * Av_1 * v_1^2)}$

$$\eta = 2[\frac{(v_1 u_1 + u_2 v_{w2}\cos(\phi))}{v_1^2}] \tag{9}$$

2.3 Design Modeling

The design was modeled in the Solidworks software. The radius of the shaft is 0.005m and the maximum radius of the turbine blade is 0.05m. The design is modeled in such a way that the inlet of the turbine is facing positive z-axis and the outlet of the turbine blade is facing the negative z-axis. The blade angle is 60°.

Table 1: Design Parameters of horizontal Spiral Turbine

S.N.	Parameters	Values	Remarks
1	Diameter	100mm	Constant
2	Blade Angle	60°	Constant
3	Angular Velocity	3,6,9,12 rpm	Variable
4	Water velocity	1, 1.5, 2 m/s	Variable

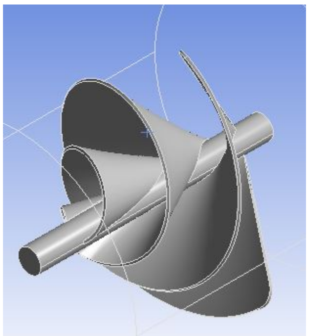


Figure 4: Isometric view of turbine blade in ANSYS

2.4 ANSYS CFX

2.4.1 Mesh Formation

The necessary domain of the computational field was made by using ANSYS Mesh v15.0. The findings of the research work will be presented in the form of formal thesis report and journal papers as per requirement of the guidelines of Department of Automobile and Mechanical Engineering, Thapathali Engineering Campus.

Table 2: Mesh Information in CFX

S.N.	Description	Quantity
	Default Domain	
1	Total number of Nodes	1717
2	Total number of Element	7,315
3	Total number of Faces	1,934
	Domain 1: Turbine	
4	Total number of Nodes	19,711
5	Total number of Element	100,976
6	Total number of Faces	11,446
	Global	
7	Total number of Nodes	21,428
8	Total number of Element	108,291
9	Total number of Faces	13,380

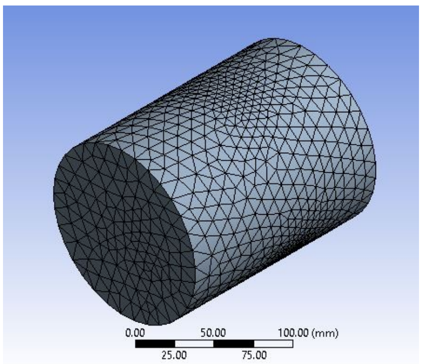


Figure 5: Mesh for CFD Analysis in ANSYS

2.4.2 Boundary Condition

The design model was set with the following boundary condition as shown in table below:

Table 3: Boundary Condition and Parameter in CFX Solver Manager

S.N.	Description	Condition Applied
1	Material	Water
2	Reference Pressure	1 atm
3	Reference Temperature	25°C
	Boundary : Inlet	
4	Normal Speed in m/s	Variable (1,1.5,2)
	Boundary : Outlet	
5	Pressure	Average Static Pressure
6	Pressure Profile Blend	0.05
7	Relative Presure	0 [Pa]
	Boundary : Wall	
8	Mass and Momentum	No Slip Wall
9	Wall Roughness	Smooth Wall
10	Buoyancy model	Non-buoyant
	Default Domain	
11	Domain Motion	Stationary
12	Reference Pressure	0 [Pa]
13	Turbulence Model	k-epsilon
	Domain 1 : Turbine	
14	Frame Type	Rotationg
15	Mass and Momentum	No Slip Wall
16	Wall Roughness	Smooth Wall
17	Buoyancy model	Non-buoyant
18	Angular Velocity in rpm	Variable (3, 6, 9 and 12)
19	Domain Motion	Stationary
20	Reference Pressure	0 [Pa]
21	Pitch Change	Specified Pitch Angles
22	Pitch Angles Side 1	360°
23	Pitch Angles Side 2	360°
	Convergence Control	
24	Length Scale Option	Conservative
25	Maximum no. of Iteration	20
26	Minimum no. of Iteration	1
27	Time Scale Control	Auto Timescale
28	Time Scale Factor	1.0

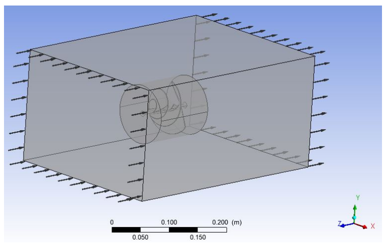


Figure 6: Solid Domain of Test-rig

3. RESULT AND DISCUSSION

3.1 Streamline Flow

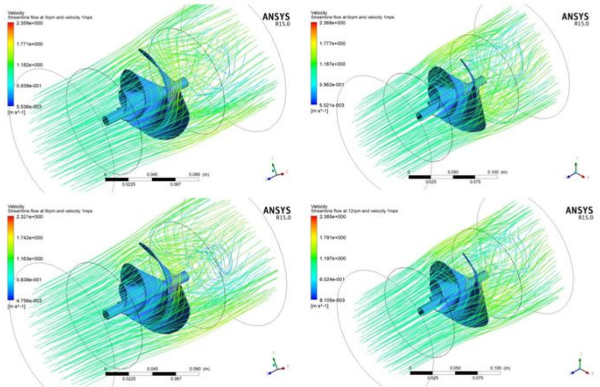


Figure 7: Streamline Flow at the velocity of 1m/s at 3rpm(top left), at 6rpm(top right), at 9rpm(bottom left) and 12rpm(bottom right)

The streamline flow of water in the CFD simulation at the velocity of 1, 1.5 and 2m/s in the variable angular velocity of 3rpm, 6rpm, 9 rpm and 12rpm is shown in the Figure 7, Figure 8 and Figure 9 respectively.

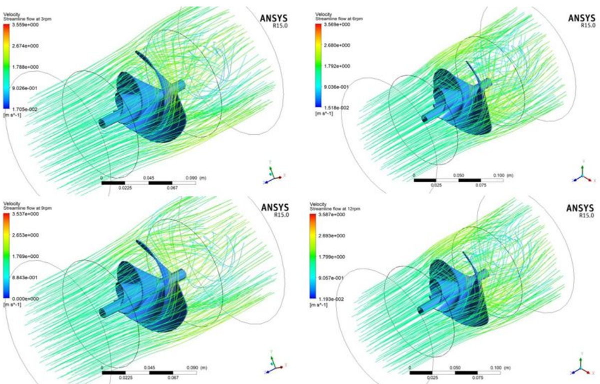


Figure 8: Streamline Flow at the velocity of 1.5m/s at 3rpm(top left), at 6rpm(top right), at 9rpm(bottom left) and 12rpm(bottom right)

The color pattern in the legend shows how streamline flow is affected while it is passing through the blade following the whirl motion. The maximum flow is obtain near the outlet of the blade near the shaft and the minimum flow is observed while the water passes through the blade and at default domain after the water is ejected from outlet.

3.2 Pressure Contour

The Figure 10, Figure 11 and Figure 12 shows the pressure contour in the blade of the turbine at the inlet velocity of water at 1, 1.5 and 2m/s respectively. The color pattern in the legend shows how the total pressure exerted on the blade varies at different position. The

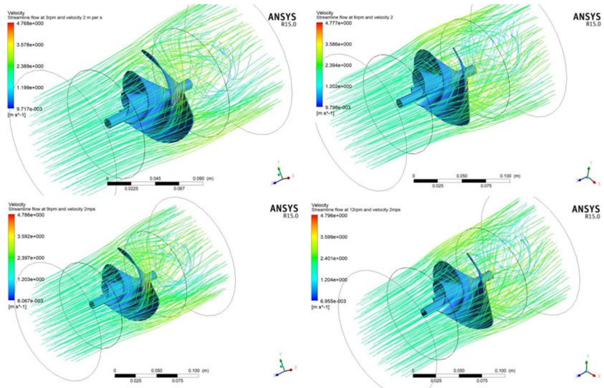


Figure 9: Streamline Flow at the velocity of 2m/s at 3rpm(top left), at 6rpm(top right), at 9rpm(bottom left) and 12rpm(bottom right)

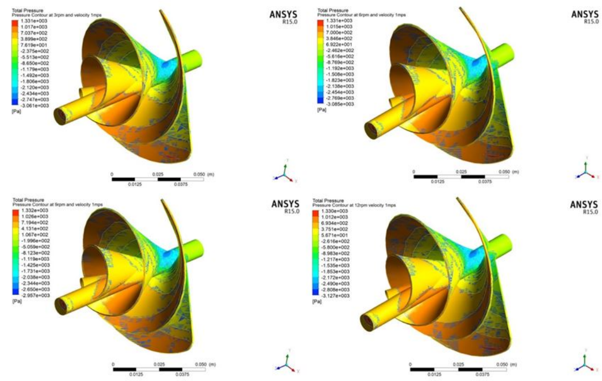


Figure 10: Pressure Contour at the velocity of 1m/s at 3rpm(top left), at 6rpm(top right), at 9rpm(bottom left) and 12rpm(bottom right)

maximum total pressure was towards the edge and gradually decreases from edge to center and negative value of pressure is obtained at the shaft. This negative pressure creates the suction pressure increasing the flow of water.

3.3 Velocity Contour

The Figure 13, Figure 14 and Figure 15 shows the velocity contour in the blade of the turbine at the inlet velocity of water at 1m/s, 1.5m/s and 2m/s respectively. The color pattern in the legend shows how the velocity with respect to the stationary frame of reference (Velocity in Stn Frame) exerted on the blade varies at different position.

It was observed that maximum velocity in stationary frame was before the outlet of the blade in the shaft and minimum velocity in stationary frame was towards the edge of the blade. Thus from the simulation it was observed that the velocity gradually increases from

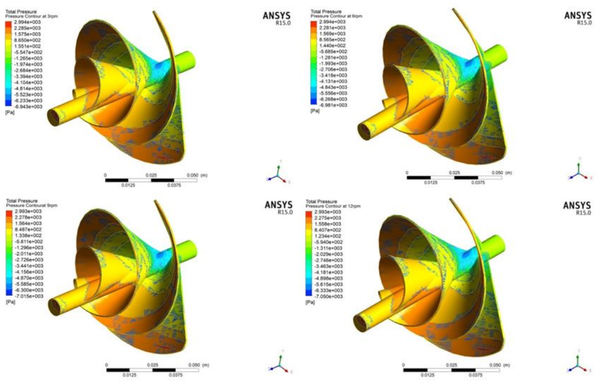


Figure 11: Pressure Contour at the velocity of 1.5m/s at 3rpm(top left), at 6rpm(top right), at 9rpm(bottom left) and 12rpm(bottom right)

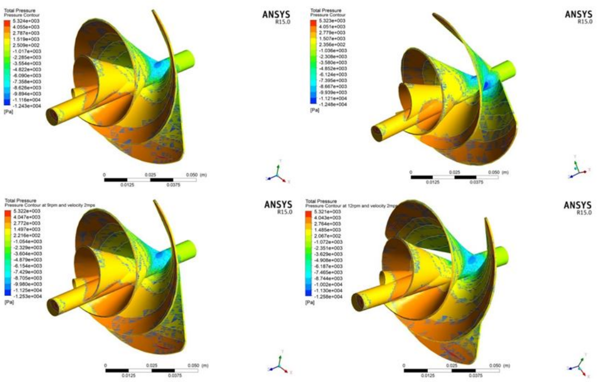


Figure 12: Pressure Contour at the velocity of 2m/s at 3rpm(top left), at 6rpm(top right), at 9rpm(bottom left) and 12rpm(bottom right)

edge of the blade to shaft of the runner as the water flow from inlet to outlet.

3.4 Force

The force obtained from CFD Simulation is plotted in graph 1. The graph represents the force on the blade in different inlet velocity of water of 1m/s, 1.5m/s and 2m/s respectively.

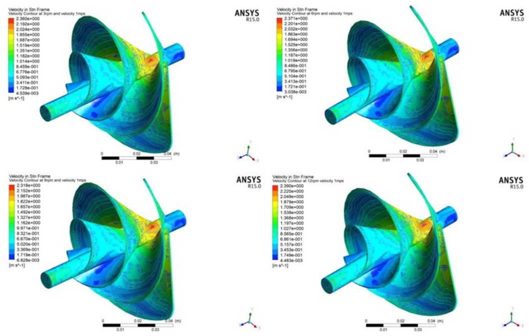


Figure 13: Velocity Contour(sth) at the velocity of 1m/s at 3rpm(top left), at 6rpm(top right), at 9rpm(bottom left) and 12rpm(bottom right)

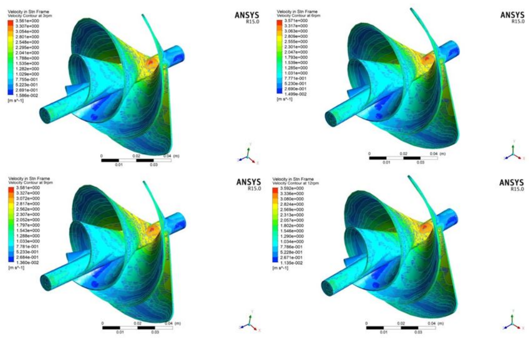
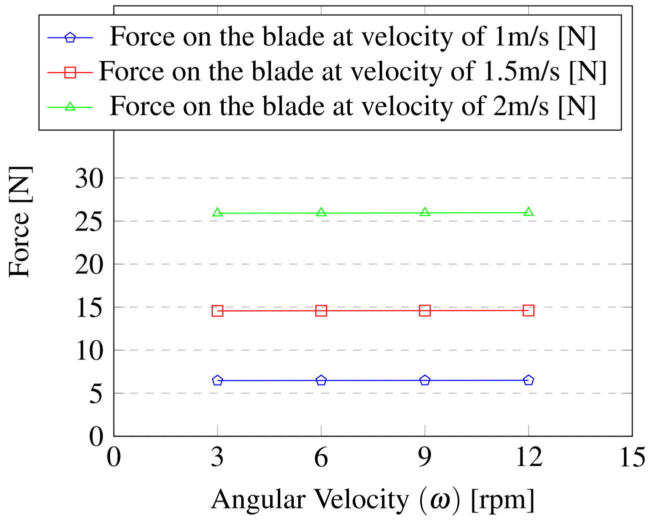


Figure 14: Velocity Contour(sth) at the velocity of 1.5m/s at 3rpm(top left), at 6rpm(top right), at 9rpm(bottom left) and 12rpm(bottom right)

Graph 1 : Normal Force on the Turbine Blade



3.5 Torque

The torque obtained from mathematical modeling and the CFD Simulation at the inlet velocity of 1m/s, 1.5m/s and 2m/s are plotted in Graph 2, Graph 3 and Graph 4 respectively. The blue line represent the torque from the mathematical modeling and the red

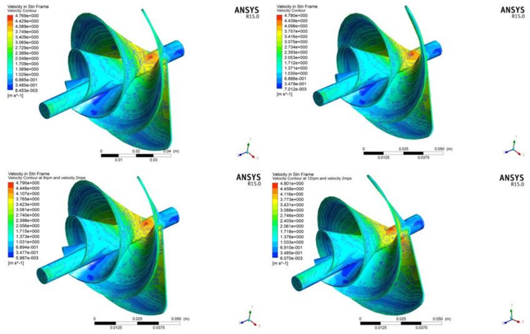
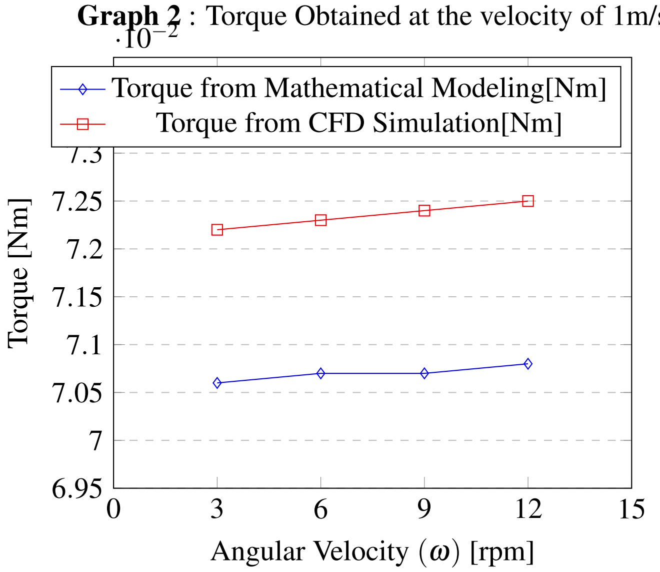


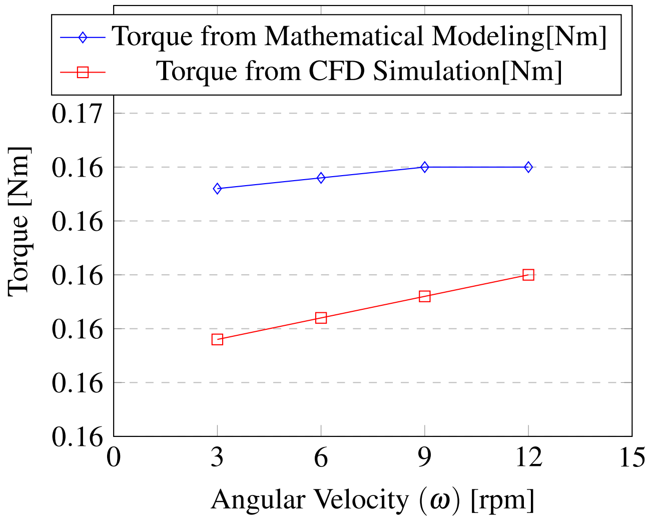
Figure 15: Velocity Contour(sth) at the velocity of 2m/s at 3rpm(top left), at 6rpm(top right), at 9rpm(bottom left) and 12rpm(bottom right)

line represent the torque from the CFD Simulation for each cases.

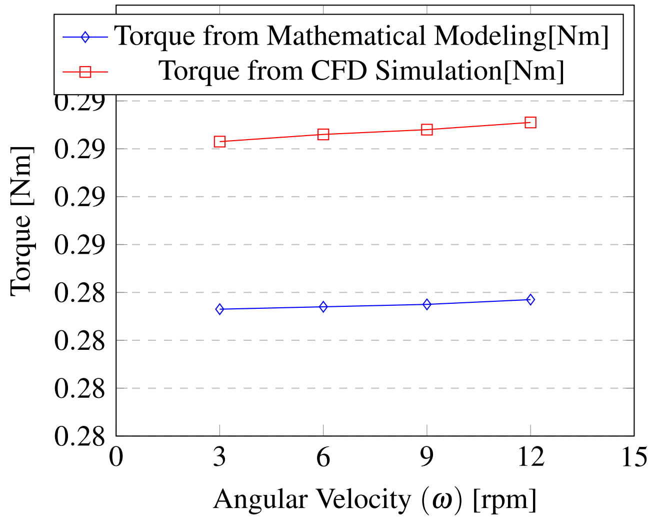
Graph 2 : Torque Obtained at the velocity of 1m/s



Graph 3 : Torque Obtained at velocity of 1.5m/s



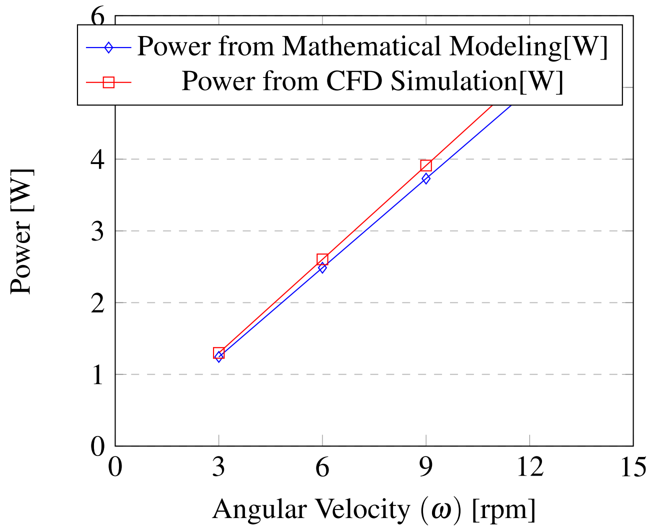
Graph 4 : Torque Obtained at the velocity of 2m/s



3.6 Power

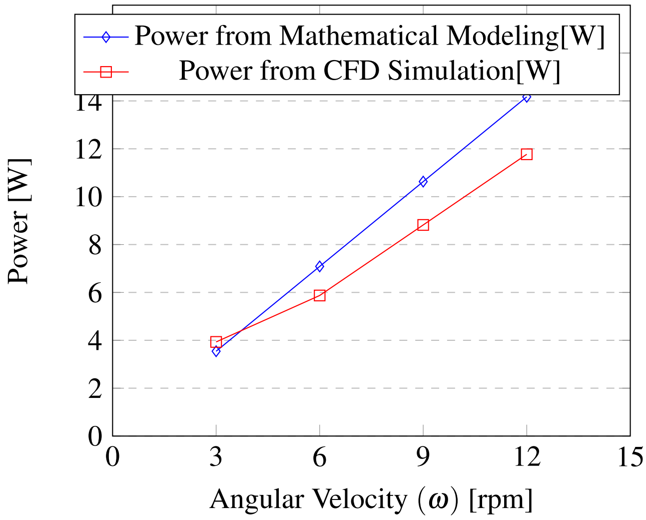
The power obtained from mathematical modeling and the CFD Simulation is plotted in graph 5 at the inlet velocity of 1m/s.

Graph 5 : Power Obtained at the velocity of 1m/s



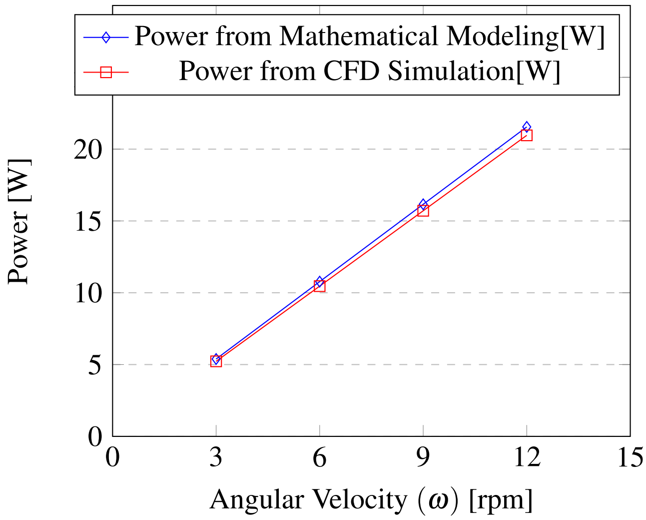
The power obtained from mathematical modeling and the CFD Simulation is plotted in graph 6 at the inlet velocity of 1.5m/s.

Graph 6 : Power Obtained at the velocity of 1.5m/s



The power obtained from mathematical modeling and the CFD Simulation is plotted in graph 7 at the inlet velocity of 2m/s.

Graph 7 : Power Obtained at the velocity of 2m/s



3.7 Efficiency

The figure 16 shows comparison of the efficiency of turbine in mathematical modeling and CFD simulation in varying angular velocity(ω) of 3, 6, 9 an 12 rpm at different inlet velocity of 1m/s, 1.5m/s and 2m/s. The efficiency of turbine at the velocity of 1m/s from the mathematical modelling is $9\%, 18\%, 27\% and 36\%$ and from the CFD Simulation is $9.408\%, 18.849\%, 28.318\% and 37.813\%$ for the angular velocity of 3, 6, 9 and 12rpm respectively. The efficiency of turbine at the velocity of 1.5m/s from the mathematical modelling is $7.6\%, 15.2\%, 22.8\% and 30.4\%$ and from the CFD Simulation is $6.291\%, 12.597\%, 18.917\% and 25.251\%$ for the angular velocity of 3, 6, 9 and 12rpm respectively. The efficiency of turbine at the velocity of 2m/s from the mathematical modelling is $4.875\%, 9.75\%, 14.625\% and 19.5\%$ and from the CFD Simulation is $4.729\%, 9.467\%, 14.213\% and 18.968\%$ for the angular velocity of 3, 6, 9 and 12rpm respectively.

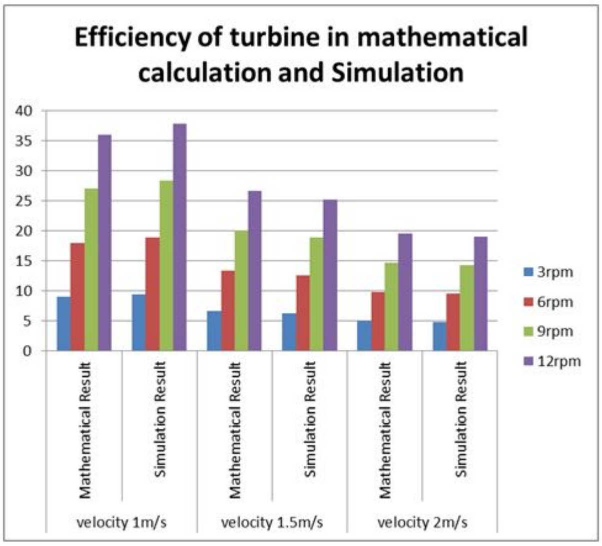


Figure 16: Efficiency of turbine in mathematical calculation and CFD Simulation

4. CONCLUSION

From the study of parameters like torque and power, it leads to the conclusion that, efficiency of horizontal spiral turbine is enhanced with the change in angular velocity and inlet velocity. The numerical modeling is performed at various range of inlet velocity and angular velocity is compared with analytical results. The findings of this study are as follows:

The force obtained at the velocity of 1m/s, 1.5m/s and 2m/s was found to be 6.49899N, 14.6123N and 25.9748N respectively

Torque obtained was 0.0725383Nm, 0.163487 Nm and 0.291094Nm for 1m/s, 1.5m/s and 2m/s respectively.

Efficiency of turbine obtained was found to be 37.8125%, 25.2509% and 18.9676% for 1m/s, 1.5m/s and 2m/s respectively from the CFD simulation of the turbine.

Maximum efficiency from numerical analysis was found to be 36.0001% at 1m/s and Maximum efficiency from analytical analysis was obtained to be

37.8125%. The deviation of 4.79% was obtained between analytical and numerical analysis.

Acknowledgments

The author gratefully acknowledged IOE, Thapathali Campus for technical support of this research.

References

[1] Owens J. M. Legrand R. Chynoweth, D. P. Renewable methane from anaerobic digestion of biomass. pages 1—-8, 2000.

[2] J.E.Quaicoe M. J. Khan, M.T.Iqba. River current energy conversion systems: Progress, prospects and challenges. *Renewable and Sustainable Energy Reviews*, pages 2177–2193, October, 2008.

[3] K.Kaygusuz M.S.Güneya. *Hydrokinetic energy conversion systems: A technology status reviewl*, volume 14. ELSEVIER, December, 2010.

[4] Sarba R. Khadka Neeraj N. Joshi. Crises, challenges and perspectives. 2009.

[5] Madeline Bergner. Developing nepal's hydroelectric resources: Policy alternatives. 2013.

[6] Nepal Electricity Authority, June 2020.

[7] 21-22 November,2019.

[8] Eric Weisstein, September 2019.

[9] Bertrand J Xueres C. Aubin J, Naude I. Blending of newtonian and shear-thinning fluid in a tank stirred with a helical screw agitator. November 2012.

[10] Sujate W Ratchaphon S and Wiroon M. An experimental study of electricity generation using a horizontal spiral turbine. September 2016.

[11] Spira M. On the golden ratioo.

[12] Wiroon Monatrakul and Ratchaphon Suntivarakorn. Effect of blade angle on turbine efficiency of a spiral horizontal axis hydro turbine.