Scientific Journal of Impact Factor (SJIF): 5.71

e-ISSN (O): 2348-4470
p-ISSN (P): 2348-6406

International Journal of Advance Engineering and Research Development

Volume 5, Issue 03, March -2018

DESIGN AND SIMULATION OF CROSS FLOW TURBINE ON HULUKA RIVER IN ETHIOPIA

[1]Abubeker Negesa Gemeda

[1]*Department of Mechanical Engineering, Ambo University, Ethiopia*

Abstract- *Micro hydropower is an important source of renewable energy in the world because of its clean and abundant energy resource to develop. It is promising means of providing cost effective electricity to location with limited or no availability of grid supplied electricity. In this paper, discuss detail about the cross flow turbine which is considering the fluid parts of the machine. Also cross flow turbine is designed and simulated using commercial software ANSYS CFX. To design this machine, the actual parameters of micro hydroelectric plant such as the water flows, types of turbine, head and other parameters are specified. A horizontal rectangular nozzle is used with this type of turbine to drive a jet of water along the full length of the runner. The complete design parameters such as runner diameter, runner length, water jet thickness, blade spacing, and radius of blade curvature, turbine power, turbine speed and number of blades were determined at maximum turbine efficiency. Also low parameters like pressure and velocity distribution are discussed.*

Keywords: *Blade angle, CFX, Cross flow turbine, Horizontal rectangular nozzle, Micro hydro power*

I. INTRODUCTION

Micro hydropower (MHP) is an important source of renewable energy in the world today because of its clean and abundant energy resources to develop. It is a promising means of providing cost-effective electricity to locations with limited or no availability of grid-supplied electricity. MHP can be used decentralized and be locally implemented and managed. It is recognized as a viable option to electrify remote areas with regard to economic, environmental, and social perspectives. Unlike large-scale hydropower, there is low environmental impact with micro hydro systems, mainly due to the exclusion of large water containment. Associated large civil works and the displacement of habitats are not required for the commissioning of micro hydro systems [6].

Micro hydroelectric power is both an efficient and reliable form of clean source of renewable energy. It can be an excellent method of harnessing renewable energy from small rivers and streams. The micro-hydro project designed to be a runoff river type, because it requires very little or no reservoir in order to power the turbine. The water will run straight through the turbine and back into the river or stream to use it for the other purposes. This has a minimal environmental impact on the local ecosystem [5]. Cross flow turbine is the suitable primarily water turbine for this industry because of its simple structure and high possibility of applying to small hydropower. It is assembled with few components, and can work effectively at the comparatively low head and low discharge in the onshore and offshore without nature disruptions.

In this particular project this turbine was designed for the advantage of School, church, commercial centers, farming activities, health station, and others electrification. The machine is designed for Huluka River and finally simulated using ANSYS CFX.This section contains a review of the most relevant research performed on the cross–flow turbine using CFD commercial software with focus of flow analysis and performance analysis.

Young-DoChoi and Sung-WooSon [2011] have investigated the effect of inlet nozzle shape and Draft Tube on the performance and internal flow of a cross-flow turbine for small hydropower by commercial code of ANSYS CFX analysis. They considered two cases contracting passage of the upper wall of nozzle from the turbine inlet then become expanding passage and straight nozzle upper wall to observe their effect on the machine performance. In the same way to notice the effect of draft tube two cases in which the second case length of draft tube is double of another case is considered. The calculated result of output power P with contracting and narrow inlet nozzle shape generates more than that of upper wall of nozzle and there is no such change in pressure efficiency even if draft tube length was doubled.

Incheol Kim, JojiWata, M.RafiuddinAhmed,Youngho Lee [2012] have presented the influence on the turbine performance according to the variation of the duct size using the commercial ANSYS-CFX code. The external casing shape and external flow of duct was not considered but was taken as default from the conceptual design stage from previous research done by other researchers. A runner with blade length of 464mm and 30 in number are used in cross flow turbine simulation. The analysis was first calculated for each case of duct size (2.25A, 4A, 9A) at 14RPM. Results of the numerical analysis are presented in terms of pressure contours, streamlines, velocity vectors, power coefficient, and performance curves. From their result as the duct size increases, the flow speed is reduced and therefore it reduces the performance characteristics.

II. METHODOLOGY

In this paper discuss about the design components of cross flow turbine in fluid flow which is rotor, nozzle and housing. In design of cross flow turbine also include bearing, generator, and transmission but in this paper not design.

Table 1 Design Parameters of Cross Flow Turbine for the HulukaRiver

Parameters	Value	unit
Q	0.052	$\frac{m^3}{s}$
Hg	14	m
η	70	%
G	9.81	$\frac{m}{s^2}$
ϱ	1000	$\frac{kg}{m^3}$
Outer diameter(Do)	200	mm
Inner diameter(Di)	133	mm
Ratio of inner and outer diameter(m)	0.665	
Angle of attack(α)	17°	
Inlet angle(β1)	31°	
Shaft diameter(d)	34	mm
Nozzle cross section	182.5×170	mm
Radius of curvature	33	mm
Blade length	182.5	mm
Blade thickness	3.6	mm
Exit angle(β2)	90°	

Calculation of the net head (Hn):

$$Hn = Hg - Ht1 \quad [1]$$

The power of the turbine in Watt can be calculated as

$$Pt = \eta g Q Hn \varrho \quad [2]$$

$$Nshape = \frac{N\sqrt{Q}}{Y^{3/4}} \quad [3]$$

$$Y = gH \quad [4]$$

Table 2. Types of Runner (9)

Type of running rotor	10^{-3}Nshape
Slow running rotor	33 to120
medium running rotor	120 to 250
Fast running rotor	250 to 500
Axial flow running rotor	330 to 1500

Assume the type of rotor is slow running rotor for this design.

Take Nshape=50×10^{-3}

$$N = \frac{\text{Nsh } Y^{3/4}}{\sqrt{Q}} \quad [5]$$

Calculate of turbine torque (T):

$$T = 60P/2\pi N \quad [6]$$

Calculated from torque relation equation:-

$$T = \frac{\pi d^3 \tau}{16} \quad [7]$$

Diameter of shaft

$$d = \sqrt[3]{\frac{16T}{\pi\tau}} \quad [8]$$

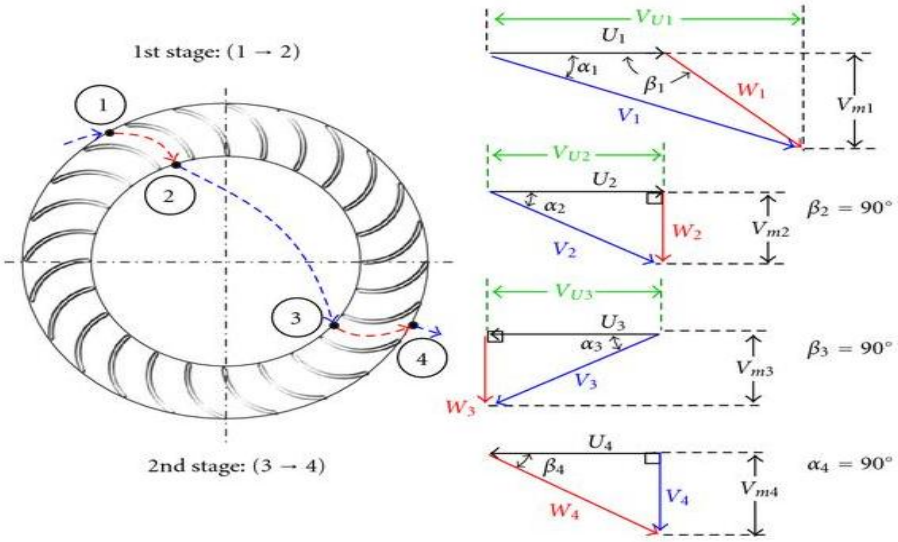


Figure 1. Velocity diagrams of different locations in cross-flow turbine

Calculate the absolute velocity (V1) of exit nozzle (which is coincident with the rotor inlet) is given by:-

$$V1 = kn\sqrt{2gHn} \quad [9]$$

Tangential velocity of the blade (U1) is given by

$$U1 = \frac{\omega Do}{2} \quad [10]$$

Speed in rad/s is calculated by:-

$$\omega = \frac{2\pi N}{60} \qquad [11]$$

Axial component of absolute velocity

$$Vm1 = V1\sin\alpha 1 \qquad [12]$$

Tangential component of absolute velocity

$$Vu1 = V1\cos\alpha 1 \qquad [13]$$

Calculate runner length (L)

$$L \times Do = \frac{2.627\,Q}{\sqrt{Hn}} \qquad [14]$$

Radius of curvature (ρ): The radius center runner to where blades are drawn. All the blades are actually arcs of a circle. To determine the radius of curvature of the blades: - [8]

$$\rho = 0.163 \times Do \qquad [15]$$

Blade spacing (s): Blade spacing is the distance between the tips of two adjacent blades on the outer periphery. This quantity is usually denoted by s. Evaluated by [8]

$$s = \frac{te}{\sin\beta 1} \qquad [16]$$

Determine the number of blade (Z): The number of the runner blades can be determined as:-

$$Z = \frac{\pi Do}{s} \qquad [17]$$

Calculate the thickness of blade: The blade thickness should be kept at an optimum value, to bear the stresses induced by the water. Blade thickness should not be too low to break down under the given load conditions.

Alternatively too great thinness of blade may cause interruption to the water admission. The following relation holds true for the blade thickness: [5]

$$t = K1 \times Do \qquad [18]$$

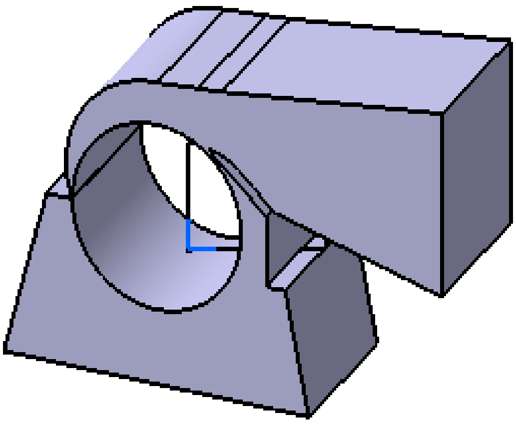


Figure 2. Housing of cross flow turbine

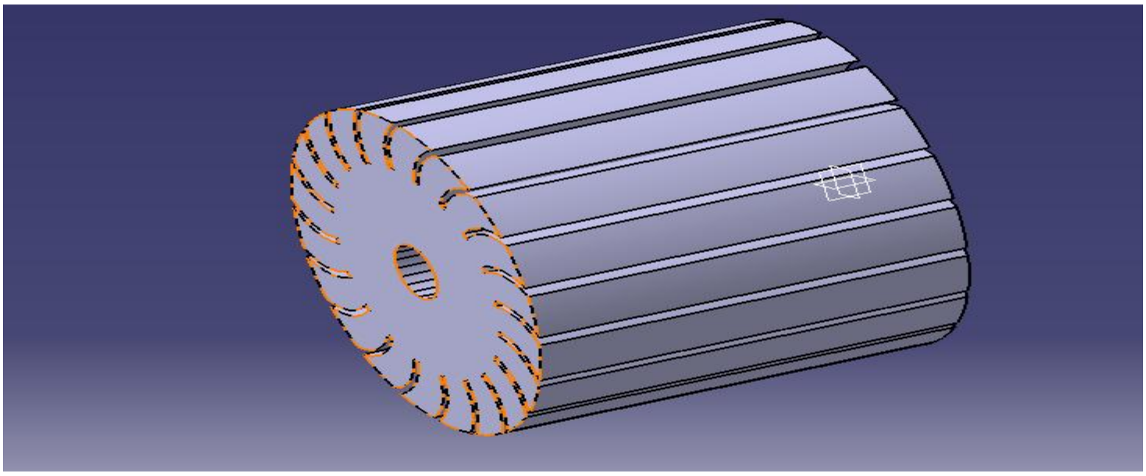


Figure 3. Rotor of cross flow turbine

CFD Simulation of Cross Flow Turbine in ANSYS CFX

CFD can be used to determine the performance of a component at the design stage, or it can be used to analyze difficulties with an existing component and lead to its improved design.ANSYS CFX consists of four software modules that take a geometry and mesh and pass the information required to perform a CFD analysis.

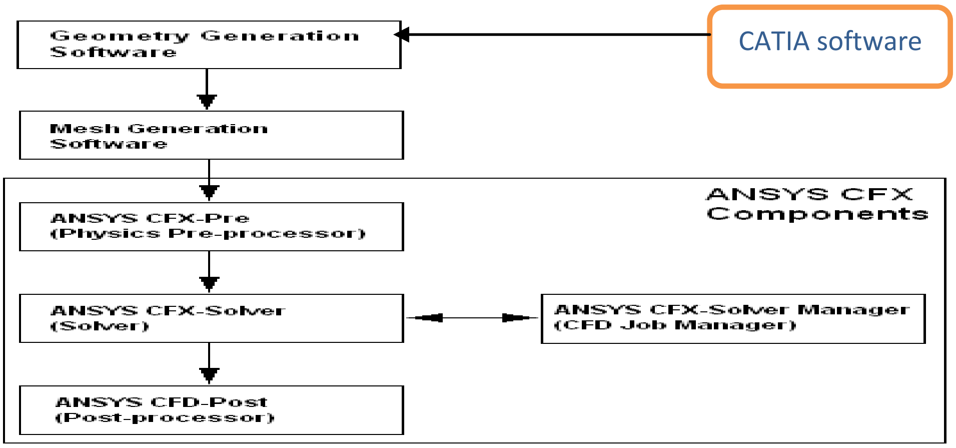


Figure 4. Structure of ANSYS CFX

In CFX, geometry can be imported from most major CATIA packages using native format, and the mesh of control volumes is generated automatically.

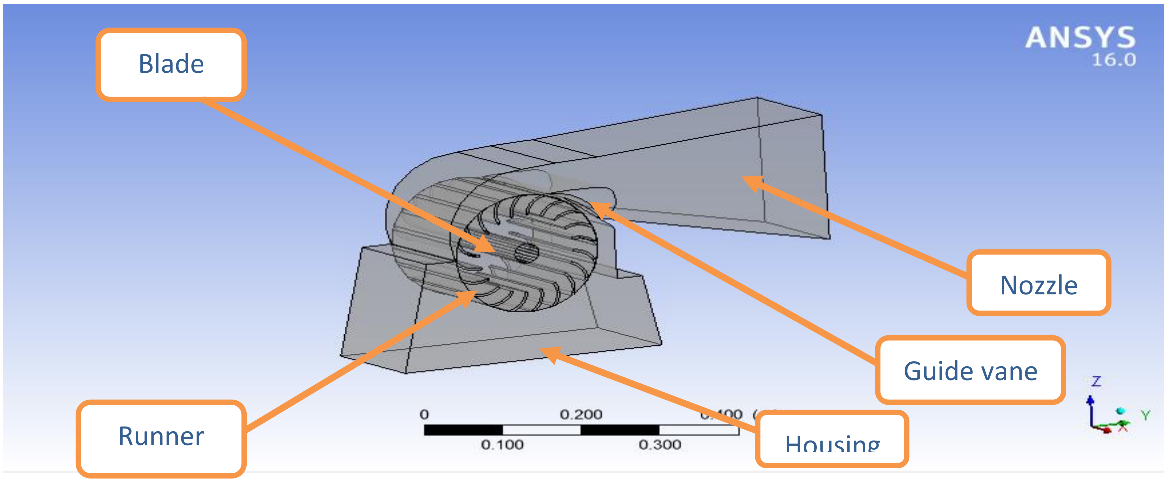
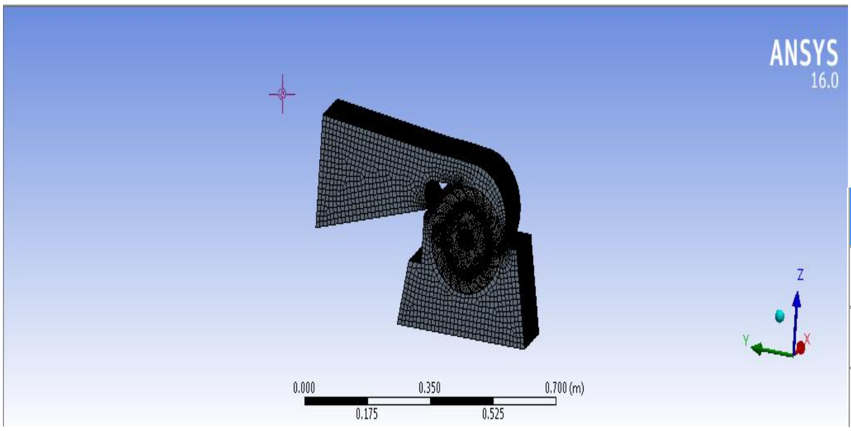


Figure 5. Mechanical model in geometry stage

The mesh was carried out separately for casing (nozzle, guide vane and housing) and runner using the ANSYS Mesh sub-program of the ANSYS CFX Project. 3D views of these domains including all the meshes are shown in Figure 6 and Table 3 presents the mesh statistics for these two domains.

Table 3.Mesh Statistics



Statistics	
Nodes	728797
Elements	644279
Mesh Metric	None

Figure 6. Mechanical model in meshing stage

Use the Setup cell to launch the appropriate application for that system. You will define your loads, boundary conditions, and otherwise configure your analysis in the application.

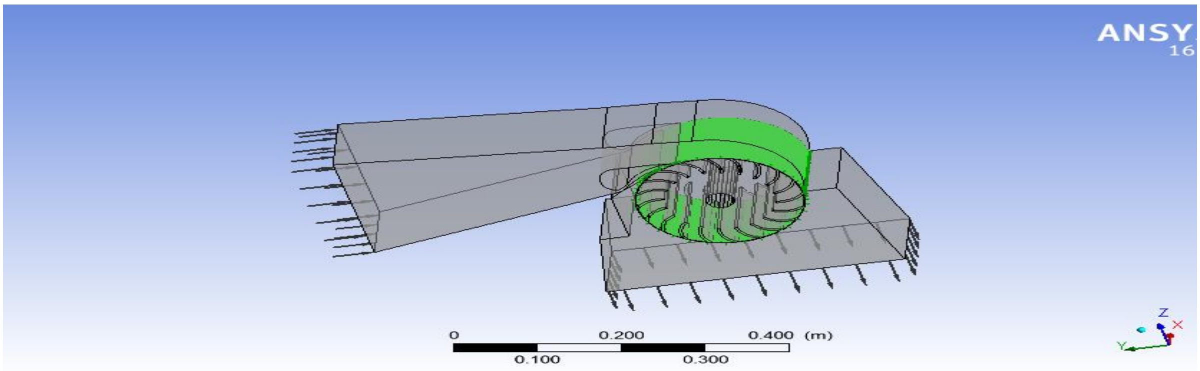


Figure 7. Setup stage

III. RESULT AND DICSUSTION

In this paper, the simulation results of designed cross flow turbine for the site were presented with discussions on the obtained results based on available literatures. The results obtained from CFD analysis in CFX are presented in two ways; the graphical display and numerical report. They are used to study the flow characteristics of selected cross flow turbine.The overall flow pattern and change of flow variables in CFT are shown using graphical display, after the calculation was calculated in CFX-solver

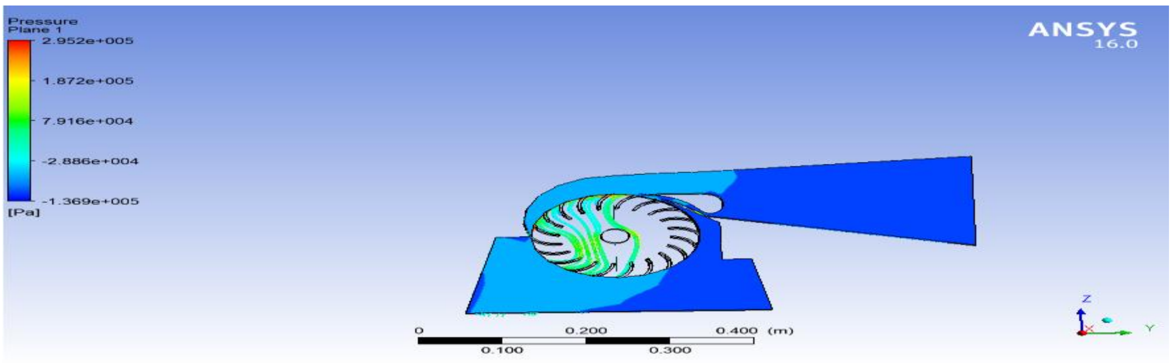


Figure 8. Pressure plane

This figures show the turbine inlet pressure will be decrease along the nozzle passage. At stage 1 the fluid pressure passing through the passage of the runner blade will be rapidly drops. At the center of the runner is relatively low pressure is occurred.

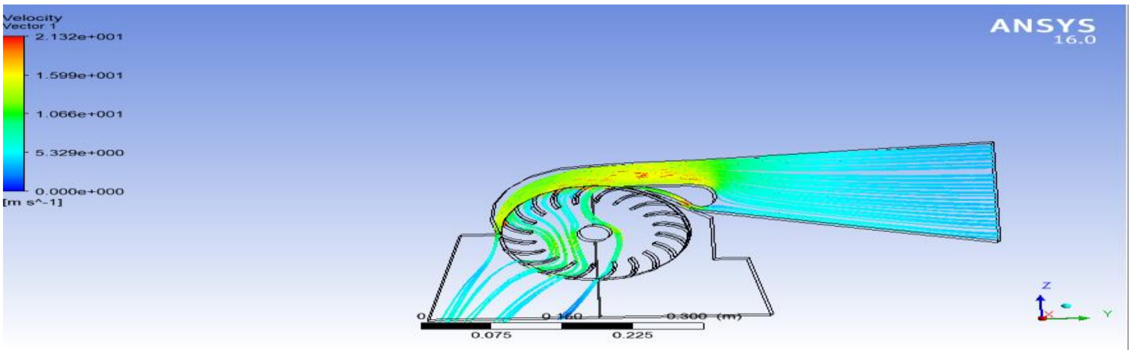


Figure 9. Velocity vector

As shown in figure the turbine inlet velocity passing through the nozzle is increasing with decreasing the pressure which the main function of nozzle to increasing the velocity and decreasing the pressure in fluid flow. At the first stage of turbine the kinetic energy will be changed to the rotational motion i.e., the torque is created which is the maximum required power is produce in this stage. Again the velocity of fluid is increase somewhat in the case of blade rotation. At the second stage the minimum required power is produce.

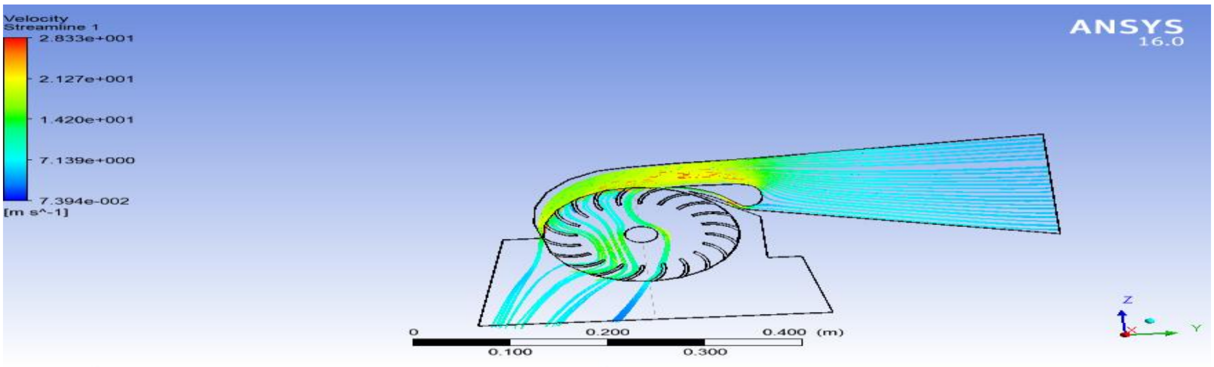
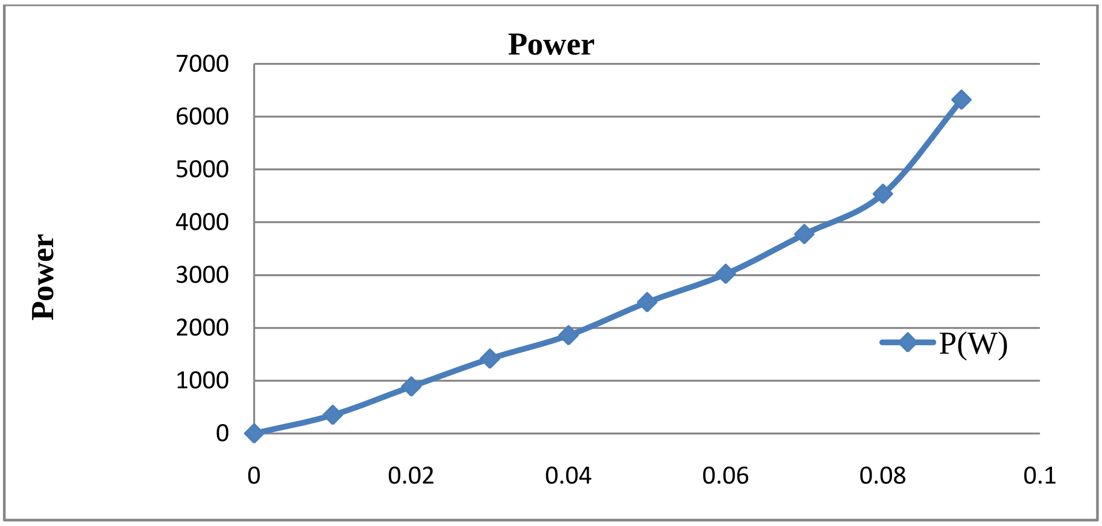


Figure10. Velocity stream line

Table 4. Values of Torque, Power and Efficiency When Speed is Constant

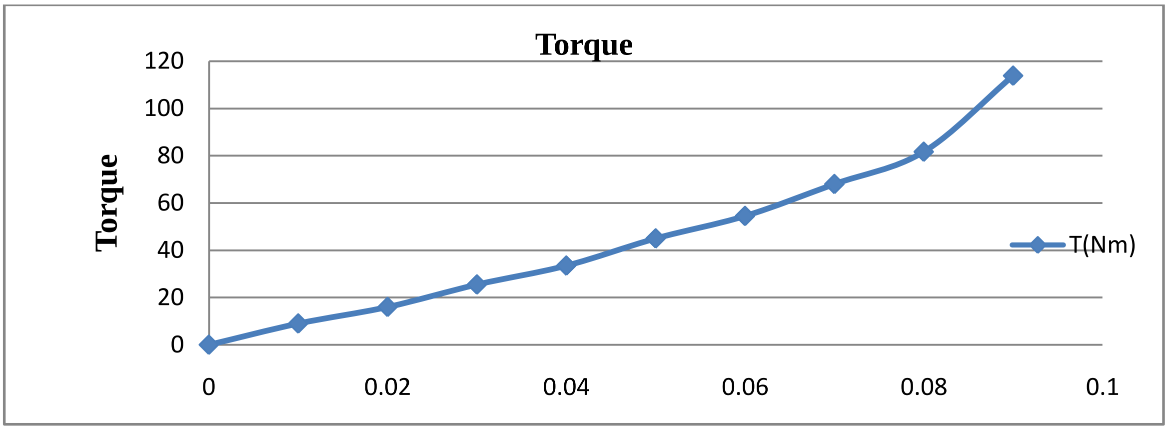
Q(m3/s)	T(Nm)	P(W)	Efficiency (%)
0	0	0	0
0.01	9	350	31.23
0.02	16	888	32.33
0.03	25.5	1415.68	33.359
0.04	33.49	1859	33.84
0.05	45	2485	34.79
0.06	54.5	3022	36.67
0.07	68	3774.75	39.26
0.08	81.68	4537	41.2
0.09	113.87	6320	42.3



Flow rate

Fig.11.Flow rate vs. power at speed constant

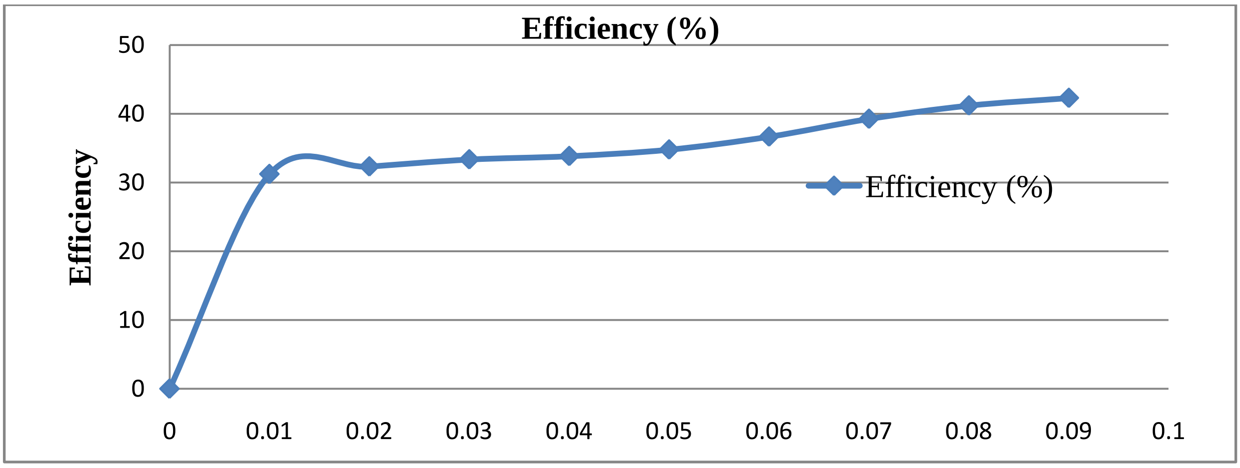
This graph shown the flow rate of the fluid is increasing the power produce will be increasing.



Flow rate

Fig.12.Flow rate vs. torque at speed constant

This graph shows the flow rate of the water is increasing the torque will be increasing.



Flow rates

Fig.13.Flow rate vs. efficiency at speed constant

The graph shows the flow rate of water is increasing the efficiency will be increasing and it is somewhat flat efficiency curve is occurred.

Table 5:Values of Power, Torque and Efficiency When Discharge is Constant

N (rpm)	P (W)	T(Nm)	Efficiency (%)
0	0	37	0
100	383	36.59	0.027
200	642	30	0.089
300	768	24	0.107
400	883	21	0.123
500	900	17.3	0.124
600	811	13	0.1136
700	680	9.27	0.095
800	511	6.15	0.072
900	309	3.28	0.0112
1000	36	0.06	0.008

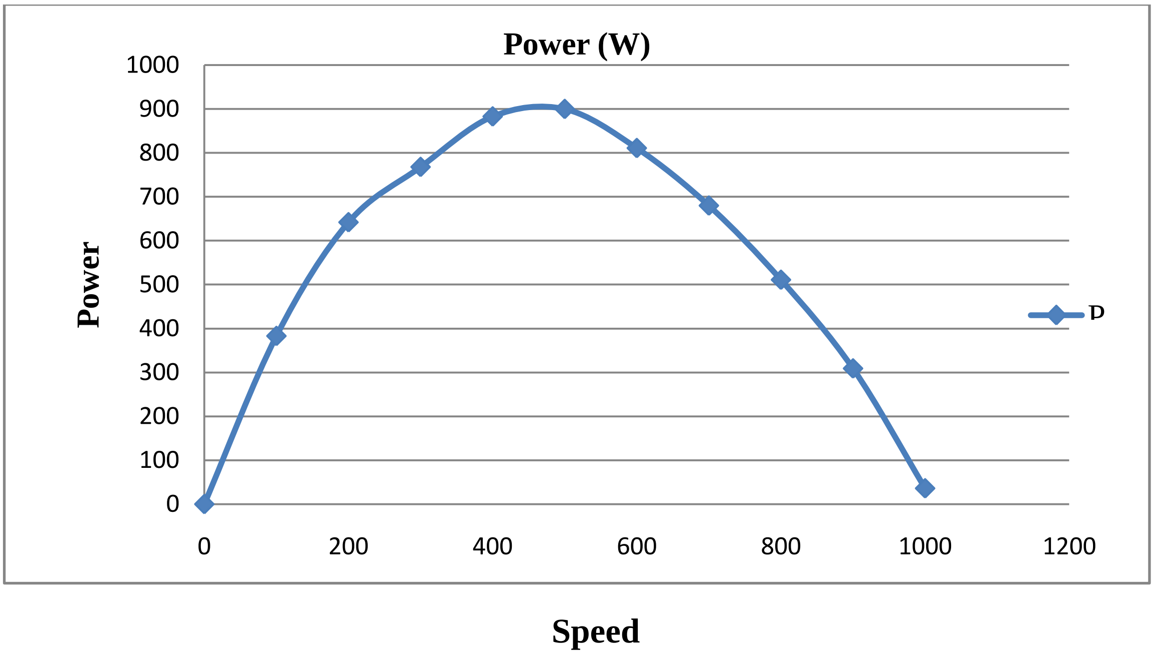


Fig.14. Speed vs. power at flow rate constant

This graph shows the turbine speed is increasing the power will be increasing until the maximum power is produce then after the power is decreasing.

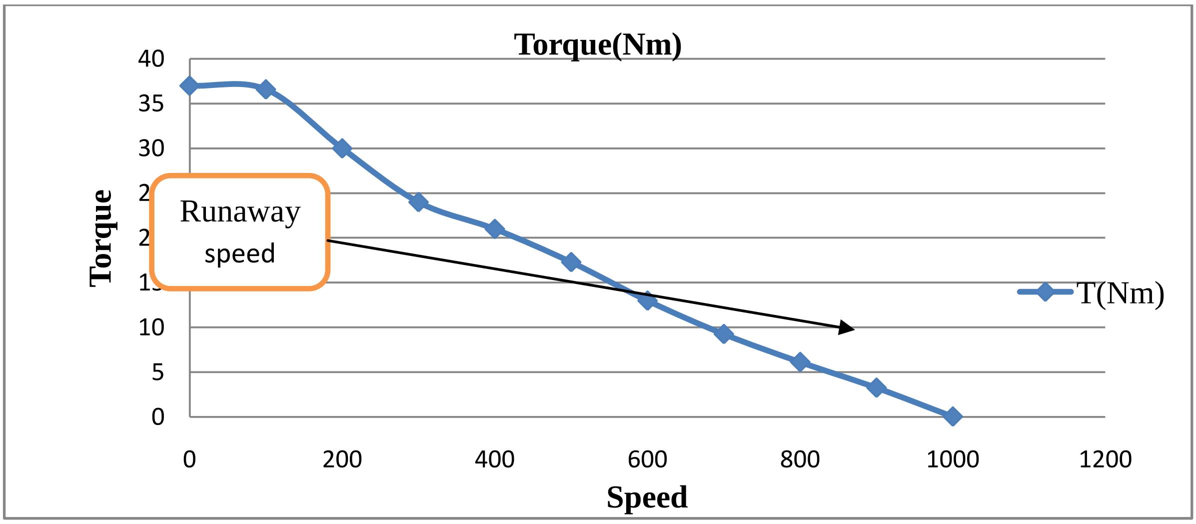


Fig.15. Speed vs. torque at flow rate constant

The graphs shown the speed of turbine increasing the torque is decreasing. The torque is inversely proportional to the turbine speed. The value of torque is zero at hydraulic power is at its maximum. The runaway speed unit can theoretically attain in case of load rejection when the hydraulic power is at its maximum. During this condition the turbine has to run without any mechanical problem.

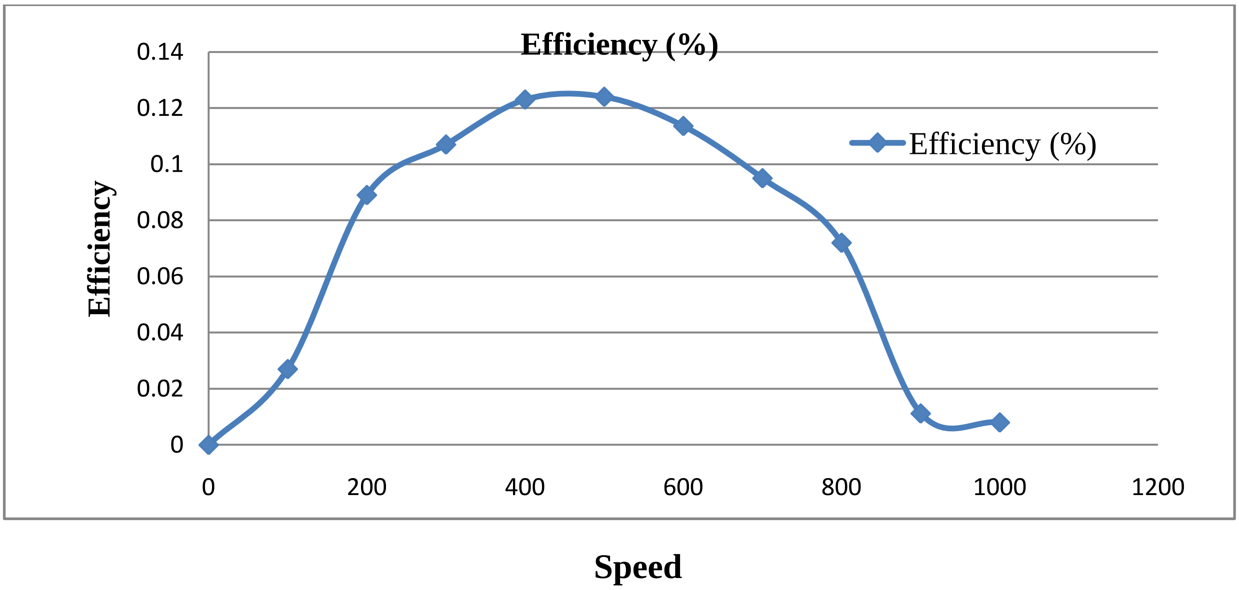


Fig.16.Speed vs. efficiency at constant flow rate

This graph shows the speed of turbine is increasing the efficiency will be increasing until the maximum efficiency is appear then after the efficiency is decreasing because the hydraulic power is increasing.

IV. CONCLUSIONS

Micro hydro power installations are usually run-of-river systems, which do not require a dam, and are installed on the water flow available on a year round basis. Cross flow turbine is considered one of the simplest turbine for micro hydro especially for low capacity hydropower plants. However, it has inherently low efficiency, which can be improved up to certain extent by modification in turbine.

The complete design parameters such as runner diameter, runner length, water jet thickness, blade spacing, and radius of blade curvature, turbine power, turbine speed and number of blades were determined at maximum turbine efficiency. Also low parameters like pressure and velocity distribution are discussed. The choice of turbine will depend mainly on the pressure head available and the water flow rate. Regulated turbines can move their inlet guide vanes or runner blades in order to increase or reduce the amount of flow they draw.

REFERENCE

[1] APCTT, " Micro Hydro Power Resource Assessment Handbook"

[2] ATamilChandran, G Anil, Jacob Chandapillai, "Development and testing of a cross flow turbine",AHEC, IITRoorkee, India. (Oct.21-23, 2010)

[3] Alphonsus Song HuaBing, "Design of micro-hydropower system for standalone rural village electrification",BorangPengesahan Status Thesis,University Malaysia Sarawak, (2010)

[4] Anil Kunwor, "Technical Specifications of Micro Hydropower System, Design and its Implementation Feasibility Analysis and Design of LamayaKhola Micro Hydro Power Plant", Nepal (2012)

[5] Bilal Abdullah Nasir, "Design of Micro - Hydro - Electric Power Station", International Journal of Engineering and Advanced Technology, (June 2013).

[6] Bilal Abdullah Nasir, "Design of High Efficiency Cross-Flow Turbine for Hydro-Power Plant", International Journal of Engineering and Advanced Technology, (February 2013).

[7] Brian Polagye1, Robert Cavagnaro, Adam Niblick, "Cross-flow turbine performance and wake characterization", University of Washington, Washington, D.C (April 10-11, 2013).

[8] Dr.ing. Wolf gang Scheer, "Turbo machinery", (January 1976).

[9] John Kobbina, "Assessment of a typical small hydropower site for rural electrification", Kwame Nkrumah University of Science and Technology, Ghana, (May, 2014)

[10] Mohammad Durali, "Design of small water turbines for farms and small communities", Massachusetts Institute of Technology Cambridge, Massachusetts 02139, (spring 1976).

[11] Mohammed TaihGatte and RasimAzeezKadhim, "Hydro Power", Ministry of Sciences and Technology, Babylon Department, Hilla, Iraq.

[12] Noor Azlizabt Ibrahim, "Modeling of micro hydroelectric system design", University Tun Hussein Onn Malaysia, (July 2012).

[13] Vincenzo Sammartano,CostanzaAricò, Armando Carravetta , OresteFecarotta and TullioTucciarelli, Banki-Michell,"Optimal Design by Computational Fluid Dynamics Testing and Hydrodynamic Analysis", University of Palermo, Italy, (2013).