International Journal of Engineering Research and General Science Volume 2, Issue 6, October-November, 2014
ISSN 2091-2730

Flow Simulation and Performance Prediction of Cross Flow Turbine Using CFD Tool

Misrak Girma[1*] and Edessa Dribssa[2]

[1]School of Mechanical and Industrial Engineering, Addis Ababa University, Institute of Technology, Addis Ababa, Ethiopia, E-mail: misrak_mgh@yahoo.com

[2]School of Mechanical and Industrial Engineering, Addis Ababa University, Institute of Technology, Addis Ababa, Ethiopia, E-mail: Edessa_dribssa@yahoo.com

***Abstract*:** the need of small hydropower schemes and availability of small hydropower potential sites in Tanzania makes Dar Es-Salam University and NBCBN to work together towards designing and manufacturing of cross flow turbine for the rural population electrification program. The aim of this paper is to study the internal flow and performance characteristics of the cross flow turbine using CFD-Tools.

A 2D-CFD steady state flow simulation has been performed using GAMBIT for geometry creation and ANSYS FLUENT for analysis. Simulations were carried out using viscous, standard K-epsilon and standard wall function model. The velocity and pressure distribution within the internal surface runner of the cross-flow turbine were analyzed.

From the simulation results it was observed that there is a high pressure area inside the nozzle and near to the first stage runner blades. Then it is going to decrease at the second stage and the out let of the turbine. And even it can be negative at the region where there is no cross flow.

In general, the simulation using CFD-Tool is very important to show the velocity and pressure distribution inside the nozzle, runner and casing.

Keywords: Cross Flow Turbine, Simulation, ANSYS FLUENT, GAMBIT, Performance Prediction, CFD-Tool

1. INTRODUCTION

Recently, micro hydro become attractive because of its clean energy sources, renewable and has a good future development. However, the turbine type must be fit to the area conditions of the built turbine. The study or research against the effective and the relatively high production costs with a complex structure are the biggest obstacles to develop micro hydro. Turbine flow of latitude (Cross-Flow) is adopted because it has a relatively simple structure [5].

A classical cross-flow turbine consists of two main parts, a nozzle and a runner. The main characteristic feature of a cross-flow turbine is the water jet of rectangular cross-section, which passes twice through the blade cascade. Water flows through the runner blades first from the periphery towards the centre, and then, after crossing the internal space, from the inside outwards. This machine is therefore a double stage turbine and the water fills only a part of the runner at a time [6].

The energy from the water is transferred to the rotor in two stages, hence the name of the machine as the 'turbine of double effect'. The first stage on average transfers 70% of the total energy to the rotor whilst the second transfers the remaining 30%. As the flow enters the second stage, a compromise direction is achieved which causes significant shock losses.

Cross flow turbines are characterized by simple design and ease of construction and its main attraction lies in its low cost and potential to be used in small-scale operations. The Tanzanian research group works a research on "Design and Manufacturing of Cross Flow Turbine" which is a good example to show how much the design is simple and the construction is easy. In addition to this the machine is simple to operate and maintain. Cross flow turbine can operate over a range of water flow and head conditions. These characteristic features of cross flow turbine makes them suitable to address the rural electrification problem in developing countries [2].

However, the overall of efficiency of cross flow turbine is lower than conventional turbines, but remains at practically the same level for a wide range of flows and heads. Therefore, the objective of this paper is to study the internal flow and performance characteristics of cross flow turbine which is designed and manufactured by the Tanzanian research groups using commercial CFD software.

2. LITERATURE REVIEW

The cross flow turbine was originally designed and patented by the Australian engineer, in 1903. His work was further developed by Donat Banki and presented in a series of publications between 1917 and 1919. Banki work resulted in a theory of operation and experimental results indicating an efficiency of 80%. The popularity of the turbine increased after these publications, and it became known for its ability to be efficient at low head with a wide range of flow [6].

Many researchers have studied the flow and performance characteristics of cross flow turbines experimentally, theoretically and using CFD tools. Some of the studies /researches are reviewed here as follows:

Chiyembeke, Cuthbert and Torbjorn [7] they conduct a research on title “A Numerical Investigation of Flow Profile and Performance of A low cost Cross Flow Turbine” in 2014. This paper studied the flow profile in turbine at best efficiency point and at operating conditions that are away from best efficiency point, using numerical methods. And researchers numerically obtained flow pattern should positions where the flow gives maximum efficiency. According to this study, the researchers have got maximum efficiency 82% and they said that numerical method is a superior design tool for cross flow turbines.

Vincenzo, Costanza,Armando,Oreste and Tullio [8] they are focused on theoretical frame work for a sequential design of the turbine parameters, taking full advantage of recently expanded computational capability. To this aim, they describe two steps procedure for banki-michel parameter design. In the first step, some criteria design parameters were theoretically estimated, on the basis of some simplifying assumptions. In the second step, the influence of efficiency of the remaining design parameters was analyzed by means of CFD numerical testing. And they apply these new design procedures for a specific power plant, for a give design point. In this test case the turbine with 35 blades and an attack angle equal to 22º exhibited at the design point a high efficiency equal to 86 %.

Eve Cathrin Walseth [9] was conduct two different experiment to determine the flow pattern and torque transfer through the runner. A cross flow turbine manufactured by remote hydrolight in Afganistan was installed in The Waterpower Laboratory at The NTNU in September 2008 and efficiency measurement was performed on the turbine. A maximum efficiency of 78.6% was obtained at 5 meter head. The first experiment was to visualize the flow through the runner with use of the high speed camera. The second experiment measured the torque transfer to the runner by the use of strain gages. But the main objective was to determine the flow pattern and torque transfer through the runner.

In Tanzania, a research team at Daresalam University designed and manufactured a cross flow turbine. All the manufacturing process carried out at TDTC workshop with the exception of the standard parts such as bearings, bolts & nuts e.t.c. which were purchased. Manufacturing can be carried out by a team of three or four people, consisting of a trained mechanic, a skilled worker trained on the job, and semi skilled helper. Unfortunately, they have very limited resource to conduct performance test of the turbine, therefore only two simple experimental tests conducted but it is not satisfactory.

Therefore, this paper deals with the analysis of flow field through the cross flow turbine which is manufactured by research team at Darussalam University and predict its performance characteristics using commercial CFD software (Ansys Fluent).

3. DESCRIPTION OF CROSS FLOW TURBINE

A. *Hydraulic Parameter and Operation Principle*

The main characteristic of the Cross-Flow turbine is the water jet of rectangular cross-section which passes twice through the rotor blades -arranged at the periphery of the cylindrical rotor - perpendicular to the rotor shaft. The water flows through the blades first from the periphery towards the centre and then, after crossing the open space inside the runner, from the inside outwards. Energy conversion takes place twice; first upon impingement of water on the blades upon entry, and then when water strikes the blades upon exit from the runner [2].

Cross-Flow turbines may be applied over a head range from less than 2 m to more than 100 m (Ossberger has supplied turbines for heads up to 250 m). A large variety of flow rates may be accommodated with a constant diameter runner, by varying the inlet and runner width. This makes it possible to reduce the need for tooling, jigs and fixtures in manufacture considerably. Ratios of rotor width/diameter is normally from 0.2 to 4.5 have been made. For wide rotors, supporting discs (intermediate discs) welded to the shaft at equal intervals prevent the blades from bending [2].

The effective head driving the Cross-Flow turbine runner can be increased by inducing a partial vacuum inside the casing. This is done by fitting a draft tube below the runner which remains full of tail water at all time. Any decrease in the level creates a greater vacuum which is limited by an air- bleed valve in the casing. Careful design of the valves and casing is necessary to avoid conditions where water might back up and submerge the runner. This principle is in fact applicable to other impulse type of turbines but is not used in practice on any other than the cross flow; It has additional advantage of reducing the spray around the bearing by rending suck air into the machine [2].

B. Sizing of Main Elements

Table 1 below summarizes the different parts of the cross flow turbine and its sizes which is manufactured by a research team at Daresalam University.

TABLE 1: CROSS FLOW TURBINE DIAMENSION

Parameter	Specification
Runner Diameter	230mm
Runner Width	200mm
No. of Blade	30
Entry Angle	16 degree
Shaft Speed	354 rpm
Nozzle Width	180 mm
Power Calculated	2.5Kw
jet velocity	9.7 m/s
Nozzle Arc	73
overall dimensions	523X343X520
Area of the jet	0.00824m2
Bearing	SKF 62206C
source: Wakati (2010)	

4. NUMERICAL ANALYSIS OF FLOW THROUGH THE TURBINE

General purposes of numerical computation of the flow through cross flow turbines included determination of the fields of pressure and velocity. The analysis was conducted on two dimensional models in the whole area of flow –from the inlet stub pipe to the outflow part of turbine. Computations in the third (axial) dimension were omitted because of the invariability of the flow channel geometry in this direction. This step decreased somehow the accuracy of the results, but on the other hand reduced substantially the time of calculations. The analysis was performed using the ANSYS Fluent solver, which is based on the finite volume method.

A. Grid Generation

In the process of grid generation one software tools were used. The Gambit software was applied to build the initial geometry of the turbine and to generate the Grid. In the whole area of flow, triangular mesh is used. This decision resulted from substantial deformations of the structural grid in many crucial areas of the flow field. In the areas of higher gradients of analyzed parameters, higher density of the grid was used to obtain the acceptable level of solution.

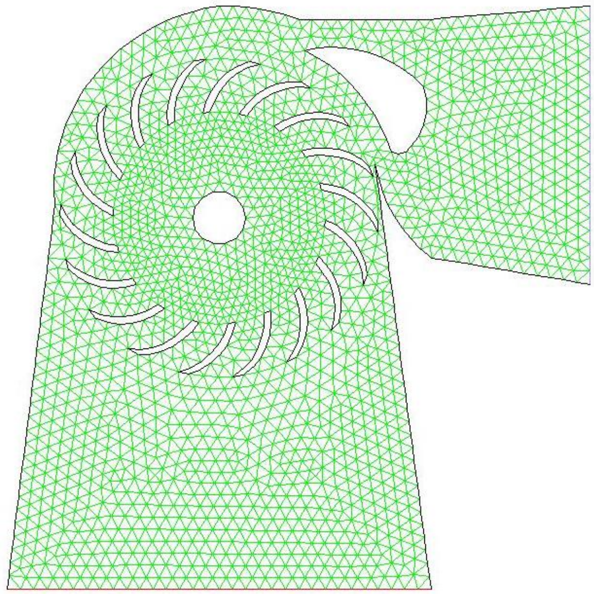

Fig. 1 Cross Flow Turbine Meshing

B. Boundary Conditions

Velocity inlet boundary conditions were used to define the fluid velocity at the flow inlet. In the incompressible flow, the inlet total pressure and the static pressure are related to the inlet velocity by Bernoulli's equation. Hence, the velocity magnitude and the mass flow rate could be assigned at the inlet boundary. Outflow boundary condition was defined at the outflow of the turbines.

C. Flow field in the rotating elements of the turbine

To analyze the flow in the rotating elements of the turbine, the Fluent Moving Reference Frame option was used. The calculations were performed in the domain moving with the runner. In this case, the flow was referred to the rotating frame of reference, which simplified the analysis. As no averaging process of the inflow parameters at the interface between the rotating and stationary zone was applied, computations were performed in the entire flow field.

D. Turbulence model

In the computation process the Renormalization Group (RNG) k-" Turbulence Model was used. Unlike the standard k-" model, the RNG-based k-" turbulence model is derived from the instantaneous Navier Stokes equations. The idea of this model is to eliminate the direct influence of small-scale eddies through some mathematical procedures. This treatment reduces computational requirements for solving the system of Navier-Stokes equations. In the presented examples the turbulence intensity and hydraulic diameters were used to describe the parameters of the model.

5. RESULT AND DISCUSSION

A. Introduction

The results of the analysis in ANSYS FLUENT can be shown by using graphical displays and numerical outputs. They are used to investigate the internal flow characteristics of cross flow turbine. The outputs here are to illustrate graphical displays and numerical outputs for a 2-D model of cross flow turbine. The graphical outputs can be displayed either in two dimensional (x-y) plots or in contour plot. In this chapter the graphical displays are shown and the performance characteristics of the cross flow turbine are also plotted using MatLab after processing the numerical results obtained from Fluent Analysis.

B. Graphical Display

The overall flow pattern in cross flow turbines can be clearly seen in graphical displays. The necessary graphical displays for flow in cross flow turbine are velocity vector colored by velocity magnitude, contour of static pressure, contour of velocity magnitude, and Tangential velocity, are shown below.

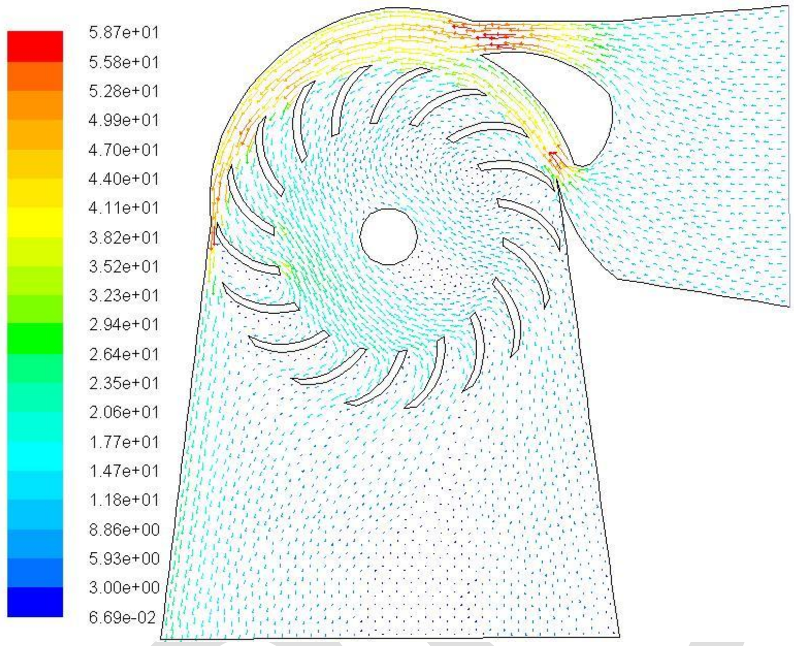

Fig. 2 Velocity Vector Colored by Velocity Magnitude

Figure 2 show the velocity vector colored by velocity magnitudes of cross flow turbine. The velocity magnitude is constant at the inlet of the nozzle and it is going to increases inside nozzle. The flow inside the nozzle is guide by the guide vane towards the runner blade at which the velocity becomes high.

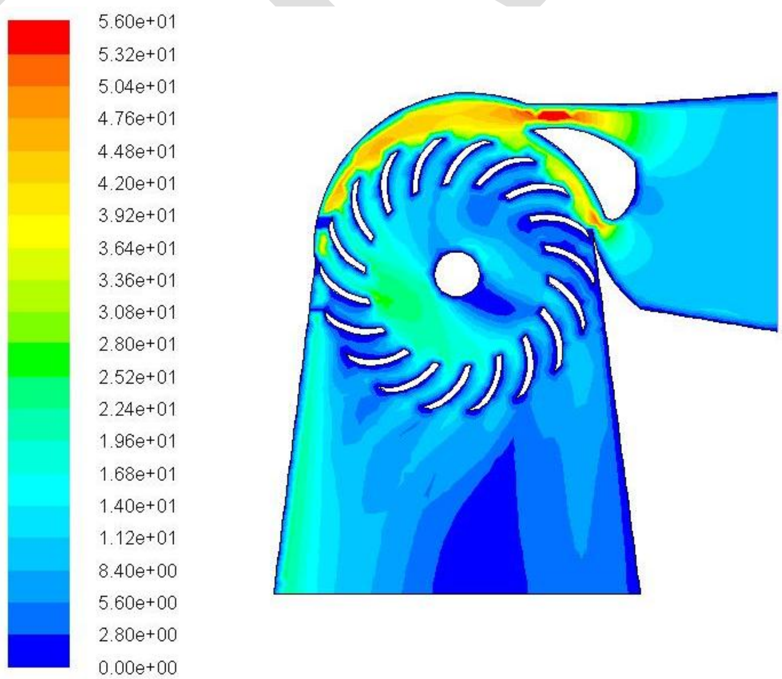

Fig. 3 Contours of Velocity Magnitude

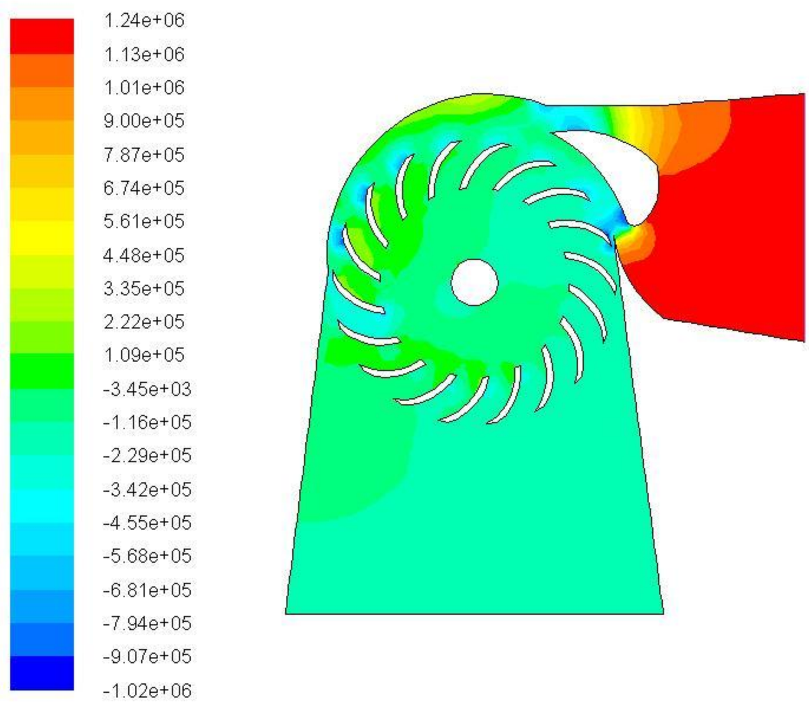

Fig. 4 Contour of Static Pressure

Figure 4 shows contour plot of static Pressure of cross flow turbine. According to the above figure the static pressure is maximum inside the nozzle at the inlet area and minimum at the outlet of the turbine or discharge area. The static pressure on runner blade is also divided in to two regions. The first region is called stage one in which the static pressure is with a certain value and the second one is stage two in which the static pressure is lease than stage one.

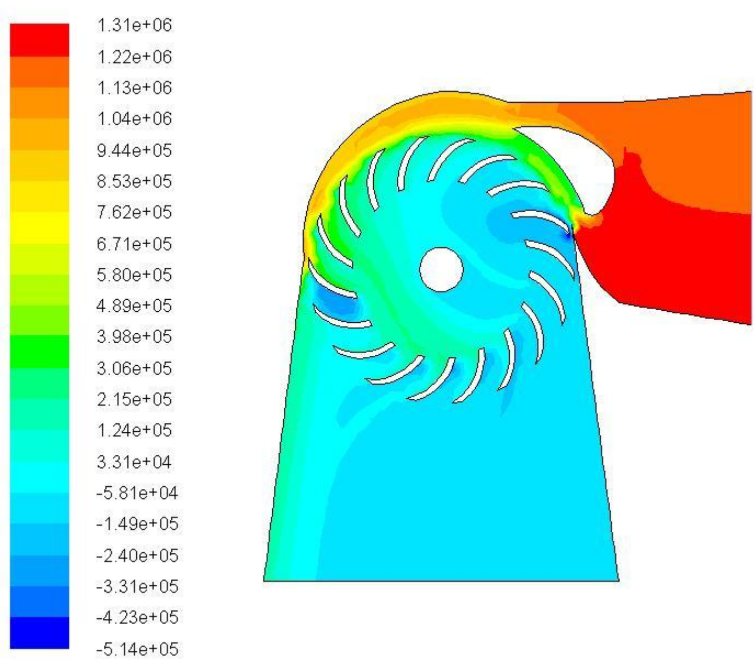

Fig. 5 Contour of Total Pressure

Figure 5 Show contour of Total pressure of cross flow turbine. The total pressure is high inside the nozzle area. And it's going to decrease in the first and second stage of the runner blade. According to the above figure the pressure is negative at the blade in which there is any cross flow.

C. Numerical Result

After convergence is reached printed quantitative results like flux balances, moments and surface integrated quantities are taken for the evaluation of design parameters. Reports of area-weighted average field variables on inlet and outlet surfaces, mass flow rate on inlet and outlet surfaces and moment about center (0, 0, 0) are illustrated below.

Area-weighted average field variables are the average value of field variables on inlet and outlet surfaces. They are computed by dividing the summation of the product of the selected field variable and facet area by the total area of the surface.

The total moment vector about center (0, 0, 0) is computed by summing the pressure and viscous moment vectors on the impeller wall. The z-component of total moment is then taken for the evaluation of shaft power.

The volume flow rate through the inlet and outlet boundaries is also computed by FLUENT. As described in the previous chapter there should be a volume flow rate balance at the inlet and out let boundaries.

6. *EVALUATION OF PERFORMANCE CHARACTERISTICS OF THE TURBINE*

Dimensional and non dimensional design parameters i.e. Head Coefficient, Flow Coefficient, power Coefficient, Head, Power and Hydraulic efficiency are evaluated from the numerical output results of ANSYS FLUENT. These parameters are used to compare the performance characteristics of different turbine models.

Both the operating characteristics and the non-dimensional characteristics are plotted after processing the numerical results from Fluent using Mat-lab.

Operating characteristics are plotted after processing the numerical results from Fluent using Mat-lab.

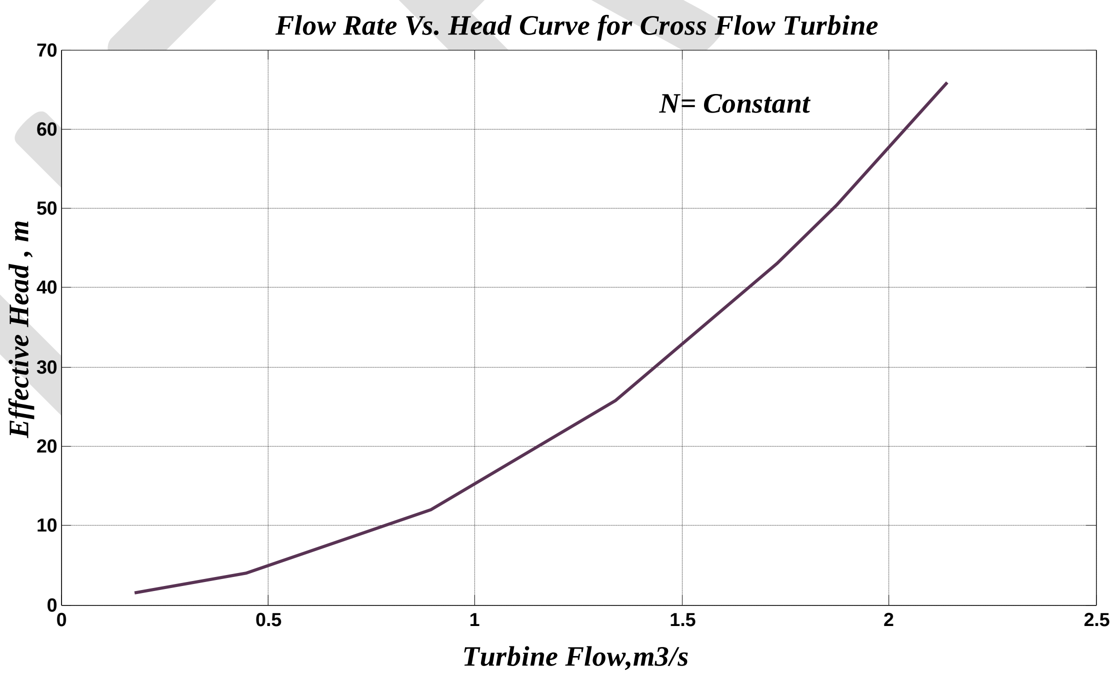

Fig. 6 Performance Curve of Cross Flow Turbine (Head vs Flow Rate)

Figure 6 shows the variation of head with flow rate. Theoretically it is expected that the head goes on increasing as the flow rate increases because the head is the difference in height between the intake and the turbine.

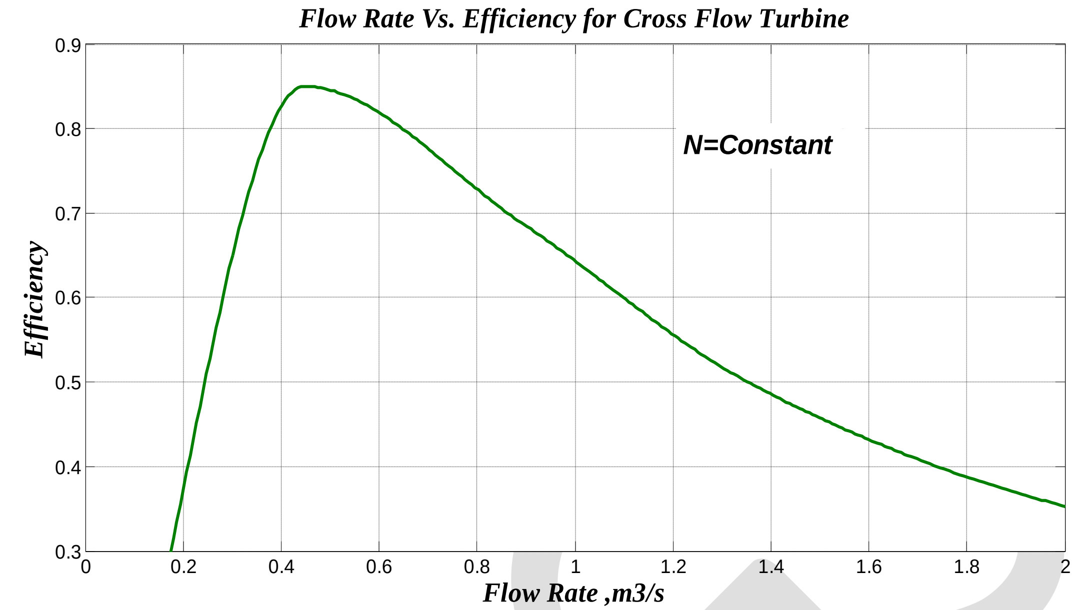

Fig. 7 Performance Curve of Cross Flow Turbine (Efficiency Vs. Flow Rate Curve)

Figure 7 show as the variation of Efficiency vs. flow rate, According to figure 7 the efficiency is maximum at 0.42m3/s after that the efficiency is dropped as the flow rate is increased.

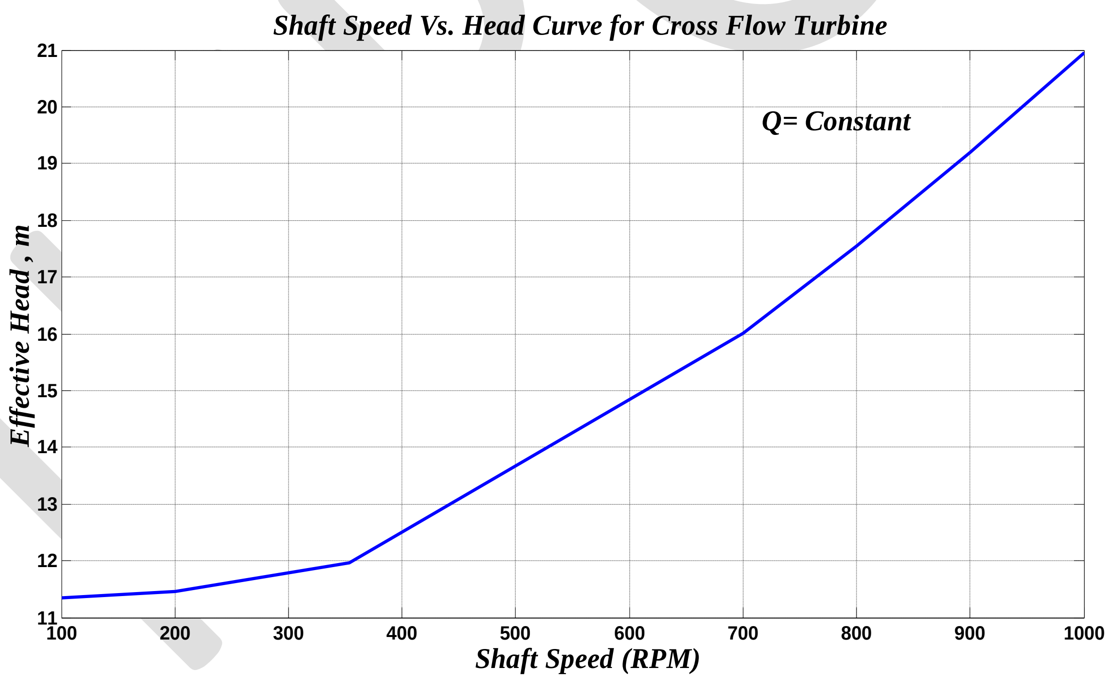

Fig. 7 Performance Curve of Cross Flow Turbine (Effective head Vs. Shaft speed)

According to figure 7 effective head and shaft speed are directly proportional, as the head increase the possible shaft speed also increase.

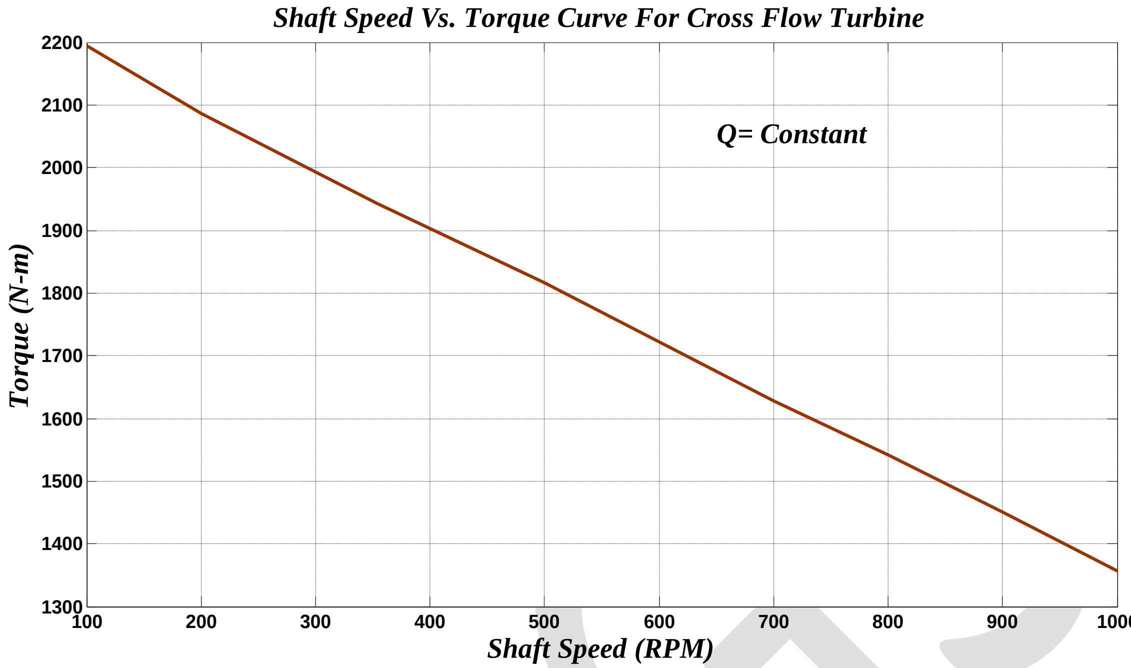

Fig. 8 Performance Curve of Cross Flow Turbine (Torque Vs. Shaft Speed)

As we have seen from figure 8 torque and shaft speed are inversely proportional, b/s as the shaft speed increases the torque produced is going to decrease.

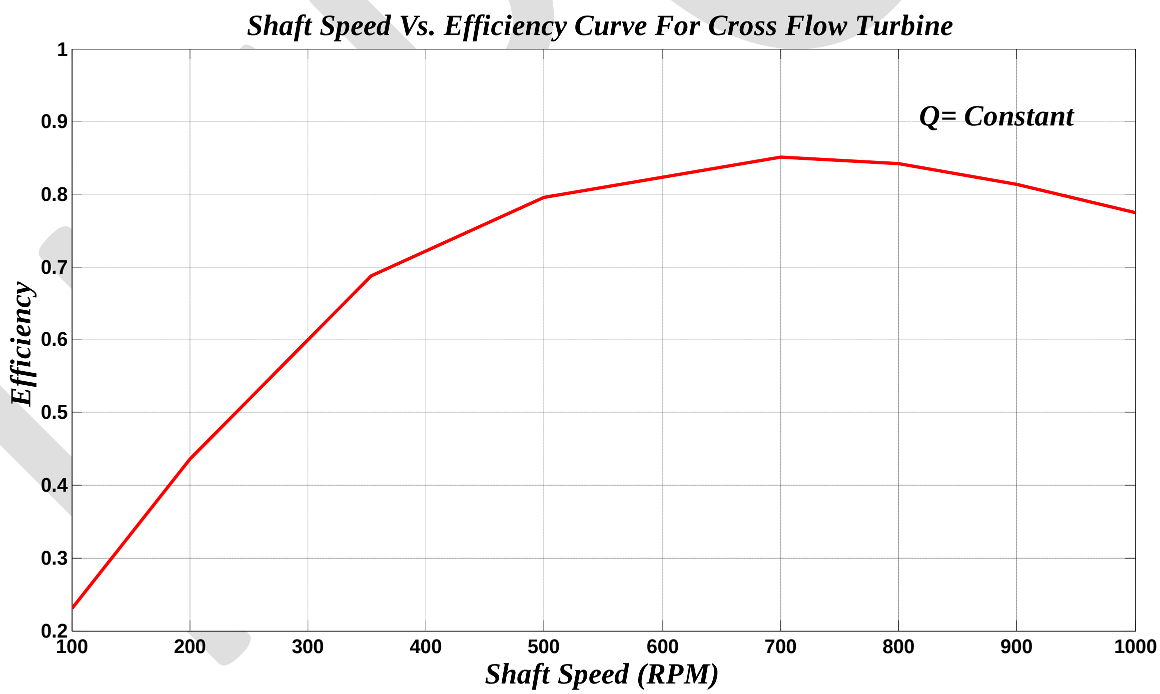

Fig. 9 performance curve of cross flow turbine (Efficiency Vs. Shaft Speed)

The above figure indicates that the maximum efficiency is occurring at 700 RPM at constant Q.

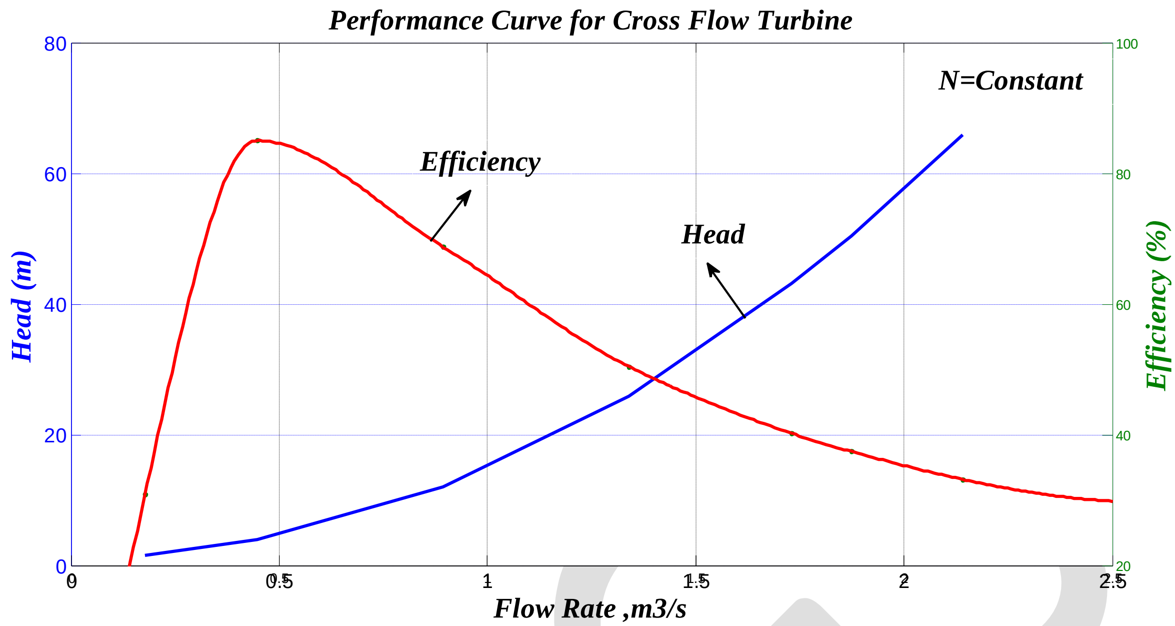

Fig. 5 Performance Curve of Cross Flow Turbine (Head and Efficiency Vs. Flow Rate)

CONCLUSION

In this study a steady state CFD analysis of a 2-D model of forward curved 18 blades cross flow turbine is carried out. The contour and vector plot of pressure and velocity distributions in the flow passage are displayed. And the operating characteristics of the turbine are also computed from fluent numerical results. Different performance curves for the Darussalam University cross flow turbine are plotted using Mat-lab program by analyzing the numerical results from Fluent.

From the study it was observed that there is a high pressure area inside the nozzle and near to the first stage runner blades. Then it is going to decrease at the second stage and out let of the turbine. And even it can be negative at the region where there is no cross flow.

The flow velocity at the inlet of the nozzle is constant and it is going to increase inside the nozzle nearer to the first stage runner blades and then going to decrease at the second stage out let of the turbine. The static pressure also dropped after the flow passes the first stage regions and it becomes negative at the region in which there is no any cross flow.

REFERENCE

[1] Vincenzo Sammartano [1],*, Costanza Aricò [1], Armando Carravetta [2], Oreste Fecarotta [2] and Tullio Tucciarelli [1],*Cross flow (Banki-Michell) Optimal Design by Computational Fluid Dynamics Testing and Hydrodynamic Analysis*, ISSN 1996-1073, *Energies* 2013, *6*, 2362-2385; doi:10.3390/en6052362.

[2] Felix Mtalo, Ramadhani Wakati, A.Towo, Sibilike K.Makhanu, Omar Munyaneza and Biniyam Abate, *Design and Fabrication of Cross Flow Turbine*, Nile basin Capacity Buliding Network (NBCBN-SEC) Office, 2010.

[3] Bilal Abdullah Nasir ,*Design of High Efficiency Cross-Flow Turbine for Hydro-Power Plant*, International Journal of Engineering and Advanced Technology (IJEAT) ISSN: 2249 – 8958, Volume-2, Issue-3, February 2013.

[4] Mohammad Durali, *Design of Small Water Turbine for Farms and Small Communities*, Massachusetts Institute of Technology may 10, 1976.

[5] Jusuf Haurissa, Slamet Wahyudi, Yudy Surya Irawan, Rudy Soenoko. *The Cross Flow Turbine Behavior towards the Turbine Rotation Quality, Efficiency, and Generated Power.* Mechanical Engineering Doctorate Program, University of Brawijaya and Mechanical Engineering, Jayapura Science and Technology University, Jl. Raya Sentani Padang Bulan-Jayapura. Mechanical Engineering, University of Brawijaya, Jl. MT. Haryono 167, Malang. (INDONESIA).

[6] Jusuf Haurissa, Slamet Wahyudi, Yudy Surya Irawan, Rudy Soenoko. *The Cross Flow Turbine Behavior towards the Turbine Rotation Quality, Efficiency, and Generated Power.* Mechanical Engineering Doctorate Program, University of Brawijaya and Mechanical Engineering, Jayapura Science and Technology University, Jl. Raya Sentani Padang Bulan-Jayapura. Mechanical Engineering, University of Brawijaya, Jl. MT. Haryono 167, Malang. (INDONESIA).

[7] MACIEJ KANIECKI, (2002). *Modernization of the Outflow System of Cross Flow Turbine*, Institute of Fluid-Flow Machinery, Polish Academy of Sciences, Fiszera 14, 80-952 Gdansk, Poland kaniecki@imp.gda.pl.

[8] Chiyembeke, Cuthbert and Torbjorn. *Experimental study on a simplified cross flow turbine*, Waterpower Laboratory, Department of Energy and Process Engineering, Norwegian University of Science and Technology, Trondheim 7491, Norway. Department of Mechanical and Industrial Engineering, University of Dar es Salaam, P.O. Box 35091, Dar es Salaam, Tanzania. 2014.

[9] Vincenzo, Costanza,Armando,Oreste and Tullio. *Banki-Michell. Optimal Design by Computational Fluid Dynamics Testing and Hydrodynamic Analysis*, energies 2013, ISSN 1996-1073.

[10] Eve Cathrin Walseth. *Investigation of the Flow through the Runner of a Cross-Flow Turbine*, Master of Science in Product Design and Manufacturing, Norwegian University of Science and Technology, Department of Energy and Process Engineering, July 2009.

[11] Jes ´ us De Andrade, Christian Curiel, Frank Kenyery, Orlando Aguill ´ on, Auristela V´asquez, andMiguel Asuaje, *Numerical Investigation of the Internal Flow in a Banki Turbine*, July 7, 2011.

[12] Bryan Ho-Yan[1], W. David Lubitz[1], *Performance Evaluation of Cross-flow Turbine for Low Head Application*, world renewable energy congress 2011-sweden, 8-13 may 2011.

Incheol Kim,Joji Wata, M. Rafiuddin Ahmed, Youngho Lee, *CFD study of a ducted cross flow turbine concept for high efficiency tidal current energy extraction*, AWTEC 2012, Asian Wave and Tidal Conference Series, Nov. 27-30, Jeju island , korea