PERFORMANCE ANALYSIS OF CENTRIFUGAL PUMP OPERATING AS TURBINE FOR IDENTIFIED MICRO/PICO HYDRO SITE OF ETHIOPIA

Edessa Dribssa , Tilahun Nigussie, Bazen Tsegaye

Department of Mechanical Engineering, Addis Ababa Institute of Technology (AAIT), Addis Ababa, Ethiopia

tilnigu@yahoo.com

Abstract— The research on 'pumps as turbines' at this critical phase of developing decentralized small hydro systems is appropriate for many reasons. The conventional turbine technologies like pelton and cross flow turbines that have been implemented in the micro hydro sector have to be custom-made and are therefore more expensive. They also need expert design and precise manufacturing skills for a good performance. This often becomes a bottleneck especially for smaller projects below 20 KWs of installed capacity. An alternative approach of using well-known 'pump as turbine' technology can be contemplated and popularized. Pumps are readily available all over the world in every shape and size, mainly due to the ever-increasing demand for pumping application [7].

This research presents CFD based prediction of performance analysis of PAT for Dabis Hydro site which is located in West Shewa zone of Ethiopia, at 8°52'38.97" N latitude and 37°47'04.95" E longitude. According to secondary source data, the site is characterized by a minimum flow rate of 0.025 m^3/s and a gross head of 17m.The estimated output power for this site is 2.70 kW. The numerical study of a centrifugal pump running as turbine at different operation conditions was achieved using commercial CFD software Ansys CFX. 3D Navier–Stokes equations were solved using Ansys CFX. The standard $k-\varepsilon$ turbulence model was chosen for turbulence model. Using results from simulation, complete characteristic curves of the pump in normal and reverse modes were obtained. After that, the performance of PAT was estimated using various methods proposed by different researchers. Experimental data from AAIT is used to test the accuracy of various correlations that can be used to generate turbine-mode operation curves from pump curves. And the simulation result of the PAT at best efficiency point (BEP) was compared with results obtained using published empirical formulas and the prediction method proposed by **Stephanoff [15]** showed acceptable result. The efficiency of the pump is greatly improved beyond the BEP when the impeller tips is made round. As the impeller speed increase, impeller torque decreases. Whereas as the rotational speed of the PAT increases, the power outputs as well as the efficiency of the PAT increase until the impeller speed reaches 1400 RPM. Lastly, the effect of draft tube on PAT was simulated and studied. The results show that the PAT head increase when draft tube is added to the system.

Keywords— Pump As Turbine, CFD, Performance prediction, Efficiency, Impeller tip rounding, Draft tube, Impeller speed effect.

INTRODUCTION

The rising electricity cost in Ethiopia has caused rural areas to start investigating means of reducing their energy consumption. Micro/Pico-hydro power is the small-scale extraction of energy from falling water from a local river to power a small village in rural areas using turbine. Micro and Pico hydro power generation has particularly become a major focus of Ethiopian electricity sector for off-grid rural electrification in order to improve sustainable energy [3].

Centrifugal pumps are used to raise liquids from a lower to a higher level by creating the required pressure with the help of centrifugal action. But when the direction of flow is reversed, they can be made to operate in a turbine mode. The use of centrifugal pump as turbine for electricity power generation was inspired by several researchers [7, 9-14]. The problem aroused from the use of high cost convention turbine for micro-hydro projects can be successful solved by utilizing Pump as Turbine (PAT) as a solution [18].

CENTRIFUGAL PUMP AS TURBINE

The use of pump to transfer fluid from lower pressure to higher pressure in industrial and other areas has been applicable for several decades [5]. Recently; however the use of pump-as-turbine (PAT) systems has become popular. In such a system a pump is operated in reverse so that it functions as a turbine. According to **Chapallaz [6],** PATs are practically applicable in the areas of power generation. Standard pumps are now more and more used in MHP/PHP schemes (5 to 500 kW). For pumped storage scheme, pump-turbines are specifically used to operate in both modes; pumping water into an elevated storage lake over night at low tariff electricity and during the day, generating peak demand electricity through the same machine operating in turbine mode.

The appropriate operating range of a reversible pump-turbine depends on the available head and flow rate on a hydro site. **Chapallaz [6]** presented a selection chart based on more than 80 test results of pump working in turbine mode. Fig.1 shows the range of head and flow rate for various types of PAT. From fig.1, it can be observed that centrifugal pumps can be used in turbine mode for head range 10 - 150 m and up to 0.5 m^3/s flow rate.

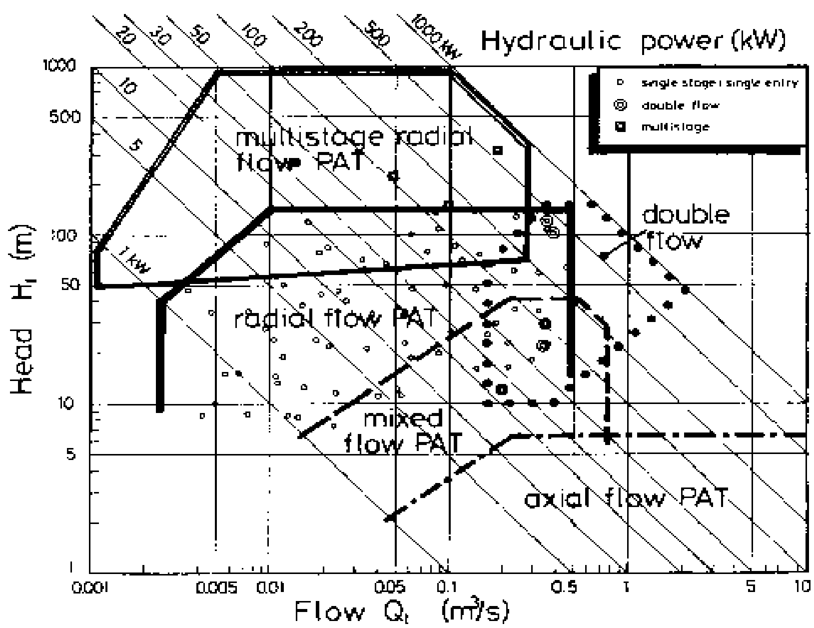

Fig.1: General range of application of different PAT types [6].

MODELLING OF CENTRIFUGAL PUMP AS TURBINE

Modeling of the geometry of a centrifugal pump running in turbine mode involves defining the impeller and volute components. Geometric modeling of draft tube is also considered since it is one of the major components in PAT system.

A. Creating of Impeller and Volute

The volute and the impeller are generated using CF turbo. CF turbo is software package used to model and design turbo machineries like pumps, ventilators, compressors, turbines interactively. The software is easy to use and does enable quick generation and variation of impeller and volute geometries. The selected pump specification to operate in turbine mode is tabulated in Table 1.

Table 1: Centrifugal pump impeller specification

Impeller Geometry Parameter	Inlet	Outlet
No. of Blades	5	
Eye Diameter	44 mm	
Impeller Diameter	101.5 mm	200 mm
Blade width	32 mm	20 mm
Blade angle	18^{o}	40^{o}
Blade thickness	2.4 mm	4.2 mm

Using CF turbo graphics user interface (GUI), two components of the pump; namely the impeller and volute casing are modeled. Due to the periodic nature of the impeller geometry, only a single blade passage of the complete impeller model of the original pump is considered, thus minimizing the computer resources required to obtain a solution. Fig.2 and 3 depicts the complete and segment (one fifth) fluid volume extracted from impeller model and the fluid domain within the pump volute casing respectively.

Fig.2: Complete and segment 3D CFD fluid domain of centrifugal pump impeller model.

Fig.3: 3D model of a centrifugal pump volute fluid volume.

B. Creating of Draft tube

Draft tubes are usually used on PAT to partially regain the velocity energy at the diffuser throat of the PAT outlet (pump inlet).Due to gradual expansion of area toward the outlet section of the draft tube; the kinetic energy of the water then can be transformed into pressure which will increase the net turbine head of the installation.

Fig.4 shows a typical design used to size draft tube for PATs. The dimensions of the draft tube are calculated according to Fig.4. Since centrifugal pumps are usually mounted with their impeller axis being horizontal and parallel to the ground and since the outlet of the draft tube must be submerged to be operational, the elbow – type draft tube design shown on right hand side of Fig.4 is selected. By making use of the dimension, the draft tube fluid domain is modeled. Fig.5 shows the fluid volume within the draft tube. From the figure, it can be seen that to make the flow through PAT more realistic, the fluid domain that exists within pump diffuser throat is added at the right end of draft tube domain.

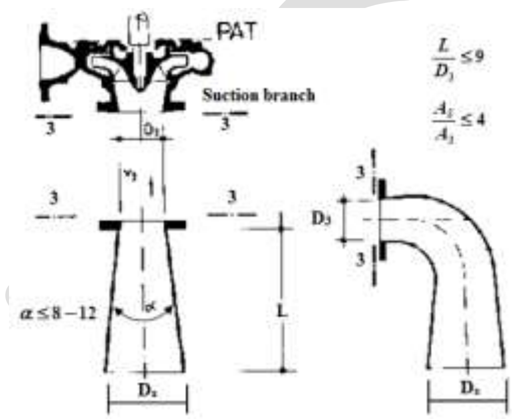

Fig.4: Typical draft tube design and main dimensions (according to Lein) [6].

Fig.5: Draft tube fluid volume.

GRID GENERATION

Mesh for different components of the PAT system like volute casing, impeller and draft tube fluid volumes are generated in Ansys ICEM. After modeling of the geometry of impeller and volute is finished, the two components were exported to design modeler in Ansys workbench using '.stp' format.

A. Computational Domain

The volute casing and impeller are imported together; they are meshed in a single mesh since the preparation of structured grid is very time consuming task, unstructured grid with tetrahedral elements has been used to discretize and generated all components of PAT fluid domains. The automatic meshing method used in Ansys mesh is sweep/patch conforming method. These methods are useful to mesh multi-body part at the same time when the mesh does not need to be conformal.

Table 2: Mesh statistics of different components of PAT system

	Tetrahedral elements	
Component	**Number of Elements**	**Number of Nodes**
Impeller	325878	8258
Volute Casing	138437	73301
Draft tube	83571	33732
Total	547886	117303

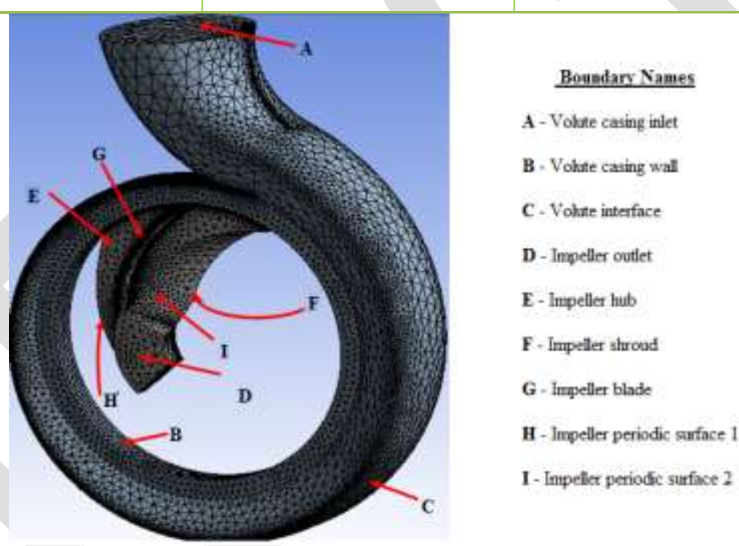

Fig.6: Meshed volute and impeller fluid domains with PAT boundary names.

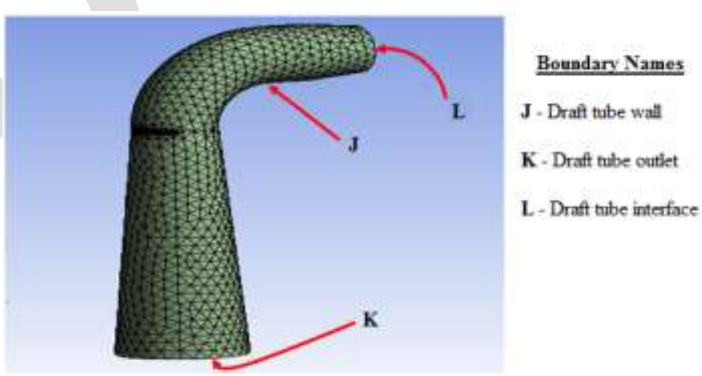

Fig.7: Meshed draft tube fluid domain with PAT boundary names.

Fig.8 shows the work flow used to simulate the PAT and draft tube. After the mesh file is transferred to the setup cell, preprocessing starts in Ansys CFX. The transferred mesh file contains information about the boundary name.

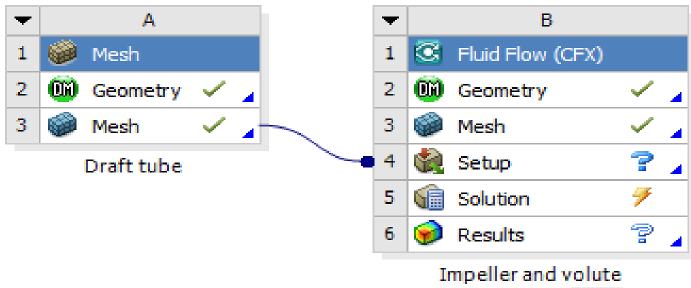

Fig.8: Overview of Ansys workbench workflow of PAT with draft tube simulation.

B. Boundary Conditions

Boundary condition is set using the '**Boundary**' icon on the toolbar. The inlet and outlet boundary condition were set by imposing a constant 0 Pa total pressure on the casing inlet surface and variable mass flow rate on the impeller outlet surface respectively. 5% medium turbulence intensity for the inlet conditions is considered. Fig.9 shows the PAT model after all boundary conditions imposed on the fluid domains.

Table 3: Summary of boundary conditions for PAT model

Domain Name	Domain motion	Surfaces	Boundary Condition
Volute	Stationary	Volute casing wall	Smooth no slip wall
		Volute casing Inlet	Inlet
Impeller	Rotating	Impeller Hub	Smooth no slip wall
		Impeller Shroud	
		Impeller Blade	
		Impeller Outlet	Outlet
		Impeller Periodic surface 1	Periodic
		Impeller Periodic surface 2	
Draft tube	Stationary	Draft tube wall	Smooth no slip wall
		Draft tube outlet	Outlet

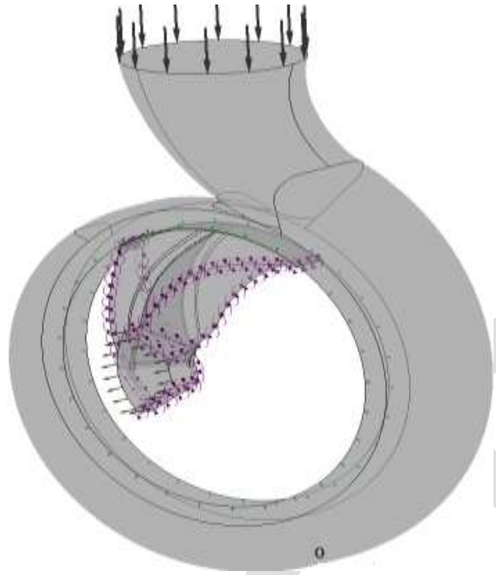

Fig. 9: PAT model with boundary conditions.

RESULTS AND ANALYSIS

Primarily, a description of the results of the selected pump with original geometry is made to show how it operates in normal operation mode (pump mode). Then the pump operating in reverse mode (turbine mode) is discussed. Then using the simulated result of the PAT at BEP, the deviation is discussed in contrast to predicted performance using published empirical formulas. After that, the effect PAT speed on performance is discussed. Lastly, the effect of adding draft tube on the discharge end of the PAT and the effect of impeller tip rounding on PAT performance is discussed in contrast to the original non modified geometry based on obtained results.

A. *Pump Mode Performance*

In order to begin the investigation of the PAT performance, it is mandatory to check the pump mode performance of the model under consideration whether it coincides with the experimental data. For this reason, the simulation of the model operating in pump mode is performed at the design rotation speed of 2960 RPM (310 rad/s). By using these results obtained from the simulation, the data are reported in Table 4. Finally comparison where made between the experimental data and numerical result.

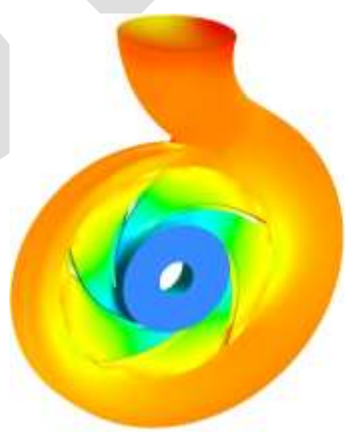

Fig.10: 3D Static pressure distribution for selected centrifugal pump model at Q = 0.025m^3/s.

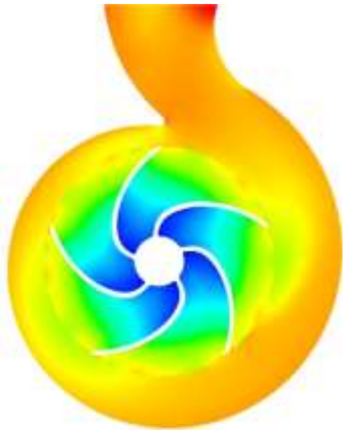

Fig.11: Mid plane static pressure distribution for selected centrifugal pump model at different flow rates.

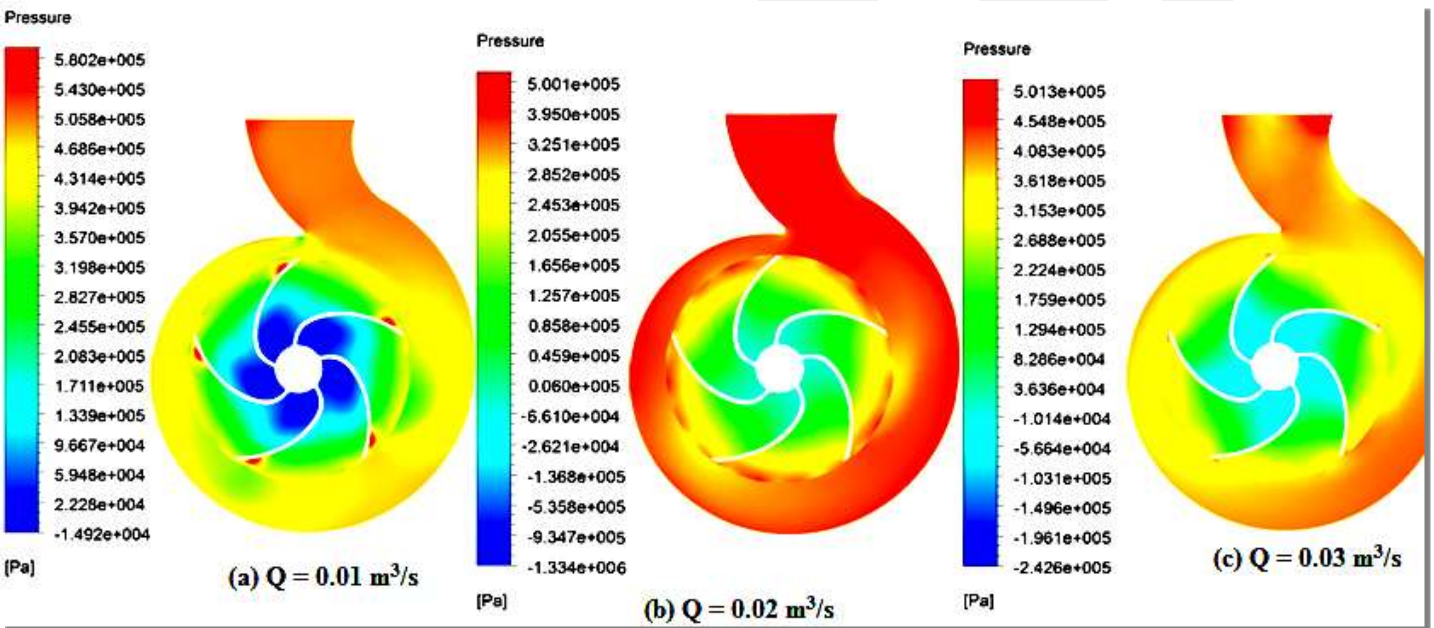

Fig.12: Mid plane static pressure distribution for selected centrifugal pump model at different flow rate.

In both Fig.13 and 14; the experimental result shows that beyond the nominal flow rate of the pump, the head developed by the pump and the efficiency of the pump rapidly drops down. This is because as the flow rate of the pump increases, the pressure of fluid at the suction side of the pump becomes so lower that bubble will start to form. The formation of bubble within the fluid causes cavitation in the pump.

When Fig.13 and 14 are closely examined, slight variation can be observed between CFD and experimental result curve. The reason for this is, the simulation of pump model carried out involves simplification from existing real model. These simplifications include neglecting the component losses that exists within the pump. To generalize these results, the deviation between the numerical and experimental data is within reasonable range.

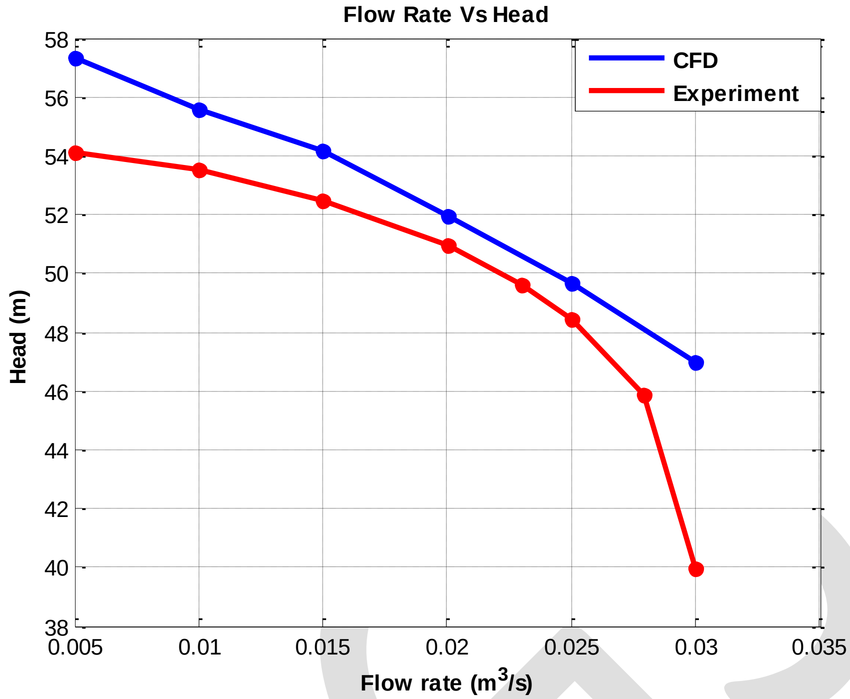

Fig.13: Pump head vs. flow rate characteristics curve of selected pump model at N=2960 RPM

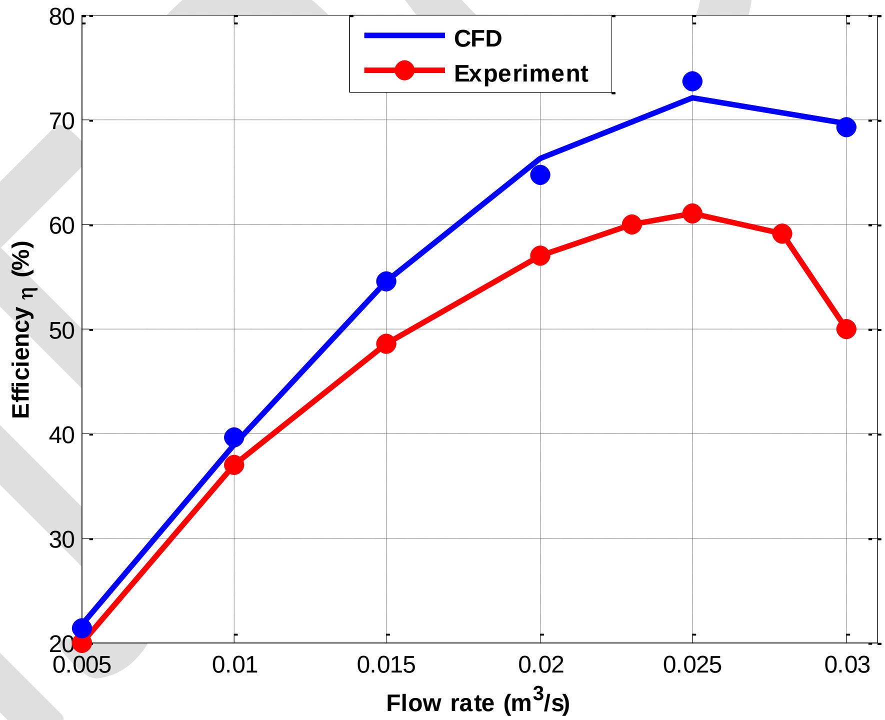

Fig.14: Pump efficiency vs. flow rate performance characteristics curve of selected pump model at N=2960 RPM

Table 4: Pump mode CFD output and experimental data for selected pump model at 2960 RPM

CFD Output								Experiment Data (Source: AAIT thermal lab tested data) [19]		
Flow rate, Q (m^3/sec)	Total pressure at Inlet, TP_{in} (pa)	Total pressure at Outlet, TP_{out} (pa)	Head (m), H $\frac{TP_{out}-TP_{in}}{\rho g}$	Water Power, P_{hy}(W), $=\rho gHQ$	Torque (Nm)	Brake Power, P_b(w) $T\times\omega$	Efficiency **η**	Flow rate, Q (m^3/sec)	Head, H (m)	Efficiency **η**
0.005	100243	662388	57.3033	2810.7269	42.61	13207.594	0.2128	0.005	54.1284	0.20
								0.01	53.516	0.37
0.01	100739	643609	55.338	5428.6578	44.453	13779.118	0.3939	0.015	52.497	0.48442
0.015	101083	633082	54.2303	7979.991	47.279	14655.094	0.54452	0.02	50.968	0.57
								0.023	49.592	0.60
0.02	101127	611414	52.017	10205.742	50.903	15778.427	0.6468	0.025	48.42	0.61
0.025	101156	589186	49.7483	12200.759	53.37	16543.124	0.7375	0.0279	45.8716	0.59
0.03	101195	538669	44.5947	13124.221	63.39	19650.020	0.6679	0.03	39.96	0.50

B. *Predicted Performance of the PAT*

After simulating the behavior of the pump operating in turbine mode at different flow rate values, the results obtained are graphically and numerically displayed. The PAT is simulated in accordance to the selected synchronous speed of a generator. Thus all the simulations of PAT were carried out at fixed rotational speed of 1500 RPM (157.08 rad/s) and with same computational procedure as done for pump mode operation.

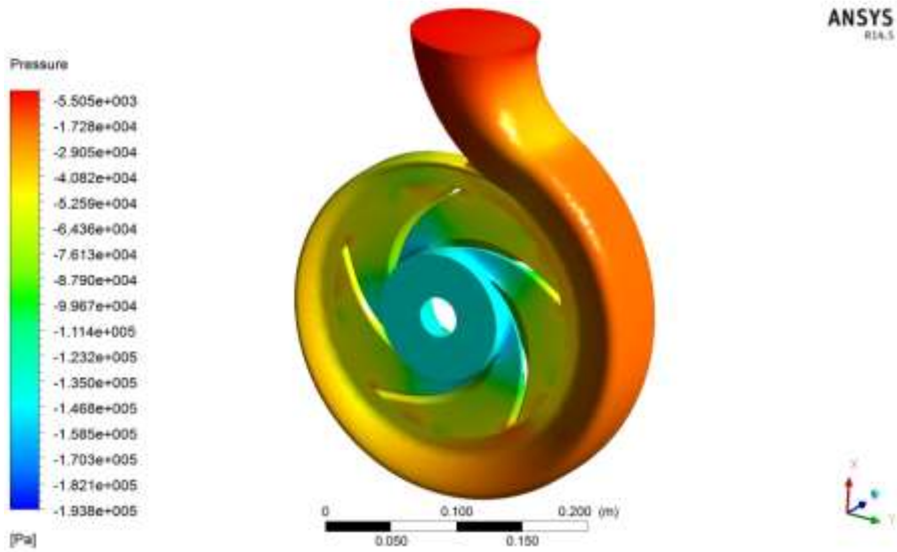

Fig.15: 3D Static pressure distribution for selected PAT model at Q = $0.025m^3/s$

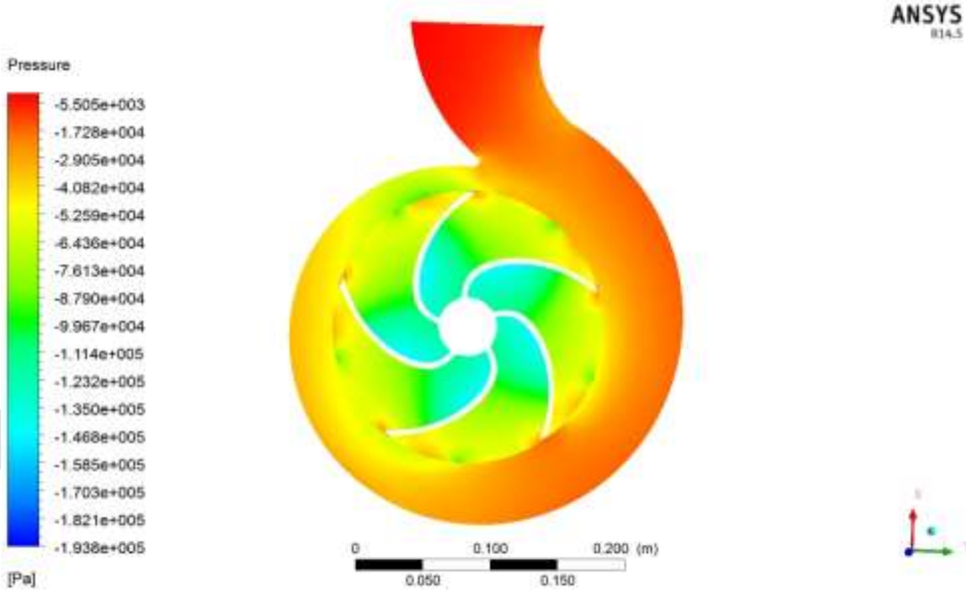

Fig.16: Static pressure distribution for selected PAT model on impeller mid plane at Q = $0.025m^3/s$

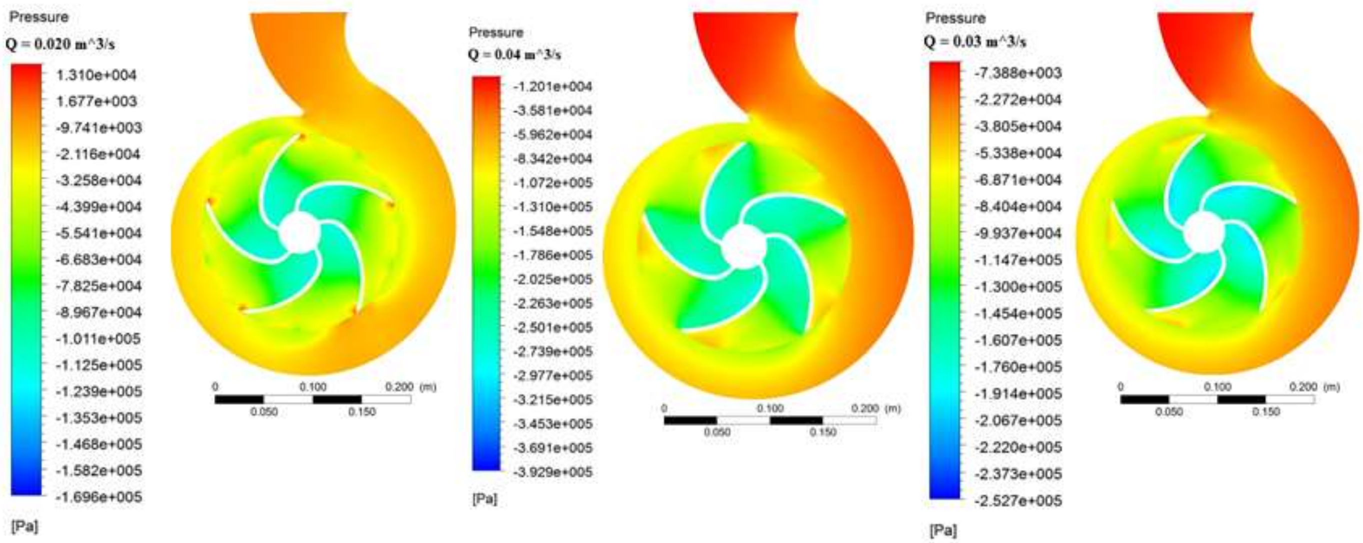

Fig.17: 2D Contour plot colored by static pressure distribution for selected PAT model at different flow rates

The fig. 18 shows the relationship that exists when the PAT is installed at different sites with available heads and the flow rate required by the PAT to operate at that particular site. It is observed that when the PAT is made to operate from low to high head condition, the required flow rate at the site increases. This implies that, if the PAT is required to operate at a site with a given head, it is required that the flow rate at that site should match the available head.

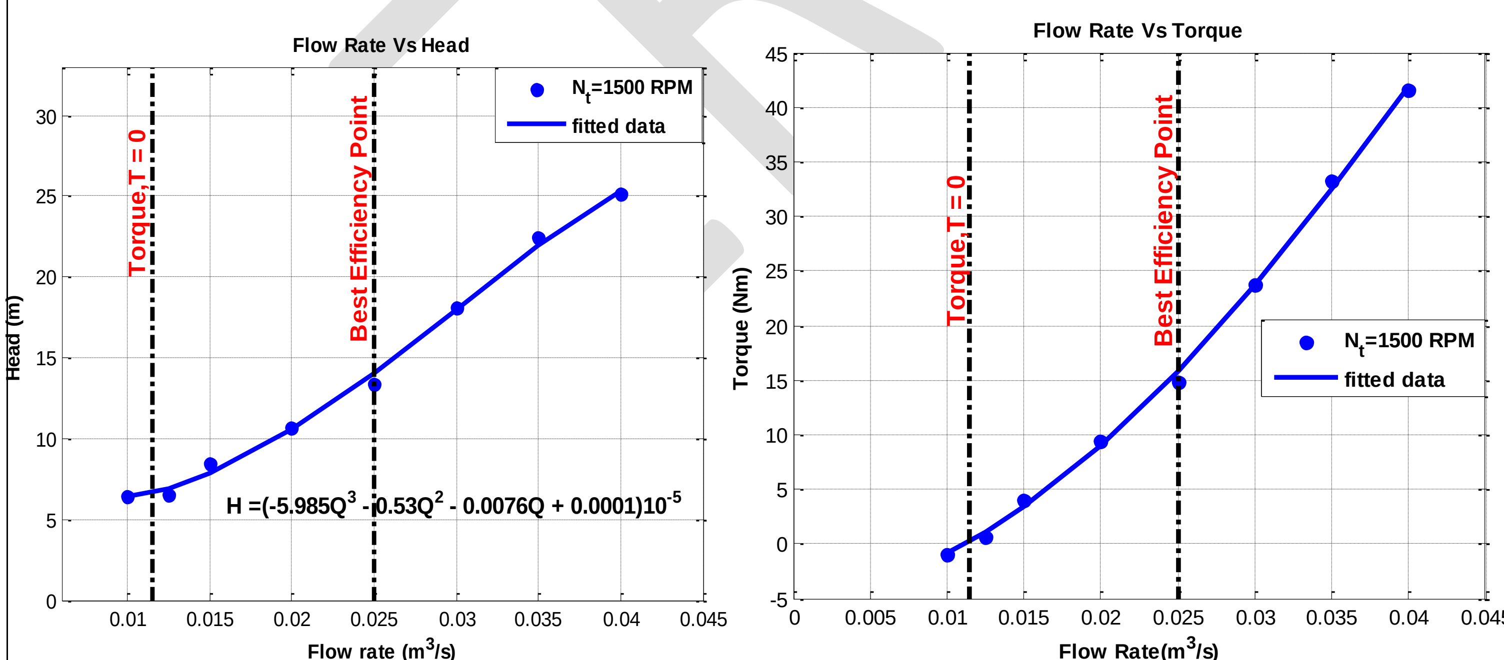

Fig.18: Predicted performance characteristics curves for selected PAT model

ACKNOWLEDGMENT

We thank all the extension and GIZ staff who provided us with information about micro hydro potential site with description of the site.

CONCLUSION

Performance analysis of PAT using CFD code Ansys CFX is made on a centrifugal pump. A 3D geometric model for the selected centrifugal pump is used to compute the steady state solution in turbine mode. The results are presented using different graphical displays of pressure and velocity distribution of the PAT model. Numerical results are also obtained from the simulation to evaluate the performance characteristics of various models under consideration.

To check the consistency of the selected centrifugal pump with the model, the first simulation is made considering the original pump running in normal operating mode. The numerical result of the simulation in direct operating pump mode shows a good agreement with the experimental tested results. The turbine mode characteristic curves at angular velocity 1500 RPM are also prepared using the simulation result when the centrifugal pump operates in reverse mode. The PAT simulation results showed that when the PAT is made to operate from low to high head condition, the required flow rate at the site also increases. And the power output vs. flow rate characteristic curve of the PAT shows that when the PAT is installed at different hydro site with flow rate matching the available head based on the head – flow rate characteristic curve, the power output of the PAT increases. The maximum efficiency of the PAT is obtained when it operate sat 0.025 m^3/sec flow rate and 13.378 m head. The head value at BEP, predicted by **Stephanoff [15]** method, has closer result with the CFD value.

The effect of round trailing edge is also simulated and compared with the original trimmed impeller tip. The results show that round trailing edge has caused lower head and higher efficiency value compared to the trimmed edge.PAT speed at constant head and flow rate is also considered to evaluate the effect on the PAT performance. The numerical result shows that as the impeller speed increases, both the power output and efficiency of the PAT increase until 1400 RPM. And the result also showed that the impeller torque decreases as the result of increasing the speed. Lastly, The effect of draft tube when added PAT is also analyzed and the result shows the draft tube causes the head to increase while the overall system efficiency get lower.

REFERENCES:

[1]. “Water Sector Development Program”, *Main Report*, vol. 1, 2002.

[2]. AkliluDalelo, “Rural Electrification in Ethiopia: Opportunities and Bottlenecks”.

[3].Meder K., “Application of Environment Assessment Related to GIZ ECO Micro Hydropower Plants in the Sidama Zone/Ethiopia”, *MSc Thesis, Heidelberg University*, 2011.

[4].“EEPCo Report”, 2001.

[5]. Rama S. R. Gorla and Aijaz A. Khan, “Turbo Machinery Design and Theory”, 2003.

[6]. Chapallaz J.M., Eichenberger P., Fischer G., “Manual on Pumps Used as Turbines”, *Vieweg, Braunschweig*, 1992.

[7].Singh P., "Optimization of Internal Hydraulics and of System Design for Pumpsas Turbines with Field Implementation and Evaluation", *Ph.D. Thesis,University of Karlsruhe,*2005.

[8].AbebeTilahun,"Assessmentof Micro Hydro Power Potential of Selected Ethiopian Rivers- A Case Studyin the North-West Part of the Country",*MSc thesis*, *Addis Ababa Institute of Technology*, 2011.

[9]. Sonia Rawal& J.T Kshirsagar, "Numerical Simulation on a Pump Operating in a Turbine", *Proceedings of the Twenty – Third International Pump Users Symposium*, 2007, pp. 21-27.

[10].DerakhshanS. &Nourbakhsh A., "Experimental Study of Characteristic Curves of centrifugal pumps working as turbines in different specific speeds", *Experimental Thermal and Fluid Science*,vol. 32, 2008, pp. 800–807.

[11]. DerakhshanS. &Nourbakhsh A., "Theoretical, Numerical and Experimental Investigation of Centrifugal Pumps in Reverse Operation", *Experimental Thermal and Fluid Science*,vol. 32, 2008, pp.1620 –1627.

[12]. Barrio R., Fernández J., ParrondoJ. and BlancoE., "Performance Prediction of a Centrifugal Pump Working in Direct and Reverse Mode Using Computational Fluid Dynamics",*International Conference on Renewable Energies and Power Quality*, 2010.

[13].Hyoung Woo Oh,"Applied Computational Fluid Dynamics", 2012.

[14]. "Reference Guide for Ansys CFX 14.5", 2012.

[15].Claudio Alatorre-Frenk, "Cost Minimization in Micro-Hydro Systems Using Pumps as Turbines",*Ph.D. Thesis*, *University of Warwick*, 1994.

[16]. Nautiya H., Varun, Kumar A. and Yadav S., "Experimental Investigation of Centrifugal Pump Working as Turbine for Small Hydropower Systems", *Energy Science and Technology*, vol. 1, no. 1, 2011, pp. 79-86.

[17].Williams A., "Pumps as Turbine: A User's Guide", 1997.

[18]. Agarwal T.,"Review of Pump as Turbine (PAT) for Micro-Hydropower", *International Journal of Emerging Technology and Advanced Engineering*, vol. 2, 2012, pp. 163-169.

[19]. "Laboratory Manual and Test Data", *Addis Ababa University*, *Mechanical Engineering Department*