International Journal of Mechanical Engineering and Robotics Research

ISSN 2278 – 0149 www.ijmerr.com
Vol. 3, No. 3, July, 2014

Research Paper

ANALYSE THE EFFECT OF MASS FLOW RATE ON THE EFFICIENCY OF PICO TURBINE & VALIDATE WITH EXPERIMENTAL RESULTS

Urvin Rameshbhai Patel[1]*, Devendra A Patel[1] and Manish S Maisuria[1]

*Corresponding Author: **Urvin Rameshbhai Patel** ✉ urvin.patel@utu.ac.in

Pico hydro is a green energy that consumes small streams to generate electricity without depends on any sources of non-renewable energy. This green scheme offers a cheap, efficient, reliable and cost effective of alternative energy and hence, there is no need to worries about the fuel source, capital cost, pollution and life expectancy. Even though the power generated is less than 5kW, but the benefit gain from this energy is the ability to raise the standard living of residents in remote areas. In this paper, model of a propeller turbine is made based on data of thesis "Design of a Low Head Pico Hydro Turbine for Rural Electrification in Cameroon" (Patrick Ho-Yan, 2012) and experimental work for pico turbine done by (Patrick Ho-Yan, 2012), is taken for the reference for simulation work. This CFD result will be compared with experimental results for validation. After doing simulation work an experimental results and CFD results are seems to be same. Their nature of curve from the results are approximately matching. This variation is about 31.21% with respect to experimental results.

Keywords: Pico turbine, Computational fluid dynamics (CFD), Flow analysis

INTRODUCTION

Pico hydro is water power output capacity up to 5 kW. It was given the name "pico" by Nigel Smith because it needs some different ways of thinking to micro, mini and larger hydropower. There are thousands of sites where people have a source of falling water but do not have electricity. For these rural communities, pico hydro is the lowest-cost technology for generating electricity. Analysis of pico turbine is done through CFD software.

Computational fluid dynamics, usually known as CFD, is a branch of fluid mechanics that uses numerical methods and algorithms to solve and analyse problems that involve fluid flows. Computers are used to perform the calculations required to simulate the interaction of liquids and gases with surfaces defined by boundary conditions. With high-speed supercomputers, better solutions can be achieved.

[1] Mechanical Engineering Department, CGPIT, Bardoli India.

MATERIALS AND METHODS

Basically for flow analysis, computational fluid dynamics (CFD) tool is used which solves the fluid property parameters by finite volume method. The software selected for CFD analysis is ANSYS 14.5. This includes following steps,

CFD Analysis Stages

The computer-based analysis process consists of the following stages:

- **Initial Thinking**

It is very important to understand as much as possible about the problem being simulated. This helps in choosing the correct settings in order to accurately describe the problem, and also in analysing the results.

- **Mesh Generation**

In this stage the flow domain is divided into sufficiently small cells, the distribution of which determines the positions where the flow variables are calculated.

- **Flow Specification**

In this stage the physical properties of the fluid, the flow parameters, and the boundary conditions are specified.

- **Calculation of the Numerical Solution**

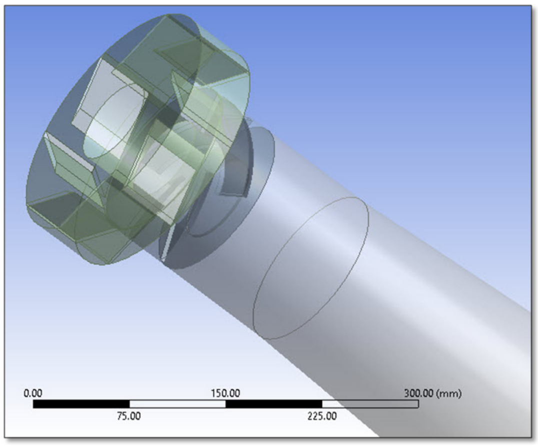
The CFD software is run to calculate the numerical solution to the flow problem. The user must provide the information that will control the numerical solution.

Ansys CFX tool discretises the problem geometry by finite volume method which is explained below:

Flow Simulation of Pico Turbine

- Modelling Detail (Patrick Ho-Yan, 2012)

Figure 1: Model of Pico Turbine



- **Runner Dimension**

Thickness of blade = 4 mm,

Number of Blades = 4

Inside straight distance = 63.80 mm

Hub diameter = 70 mm

- **Stator Tube Dimensions**

Cylinder inside diameter = 132.50 mm,
Cylinder Length = 120.00 mm

Cylinder outside diameter = 141.30 mm

- **Inlet Guide Vanes**

Height of vane = 52 mm
Width of vane = 60.60 mm

Thickness of vane = 4 mm

- **Draft Tube Dimensions**

Length = 400 mm

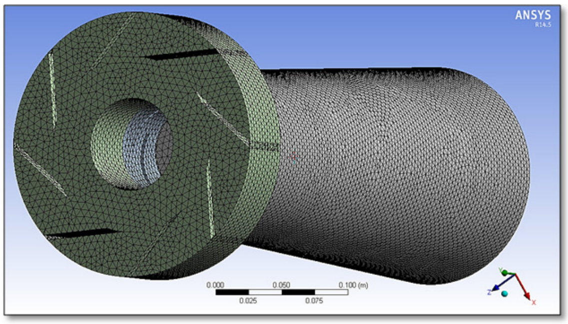
Inlet Radius = 71 mm

Outlet Radius = 150 mm

- **Meshing**

Meshing details are shown in Figure 2 in which the number of nodes are 111116 and elements are 603072 created of tetrahedral shape.

Figure 2: Meshing of Turbine Assembly



On completing the meshing, the process will move forward to CFX-Pre tool where boundary conditions are applied i.e. inlet, outlet, wall boundary conditions are supplied.

Inlet boundary Condition

Boundary condition at inlet of the turbine is taken as head (2 m) available at the inlet of the turbine in terms of relative pressure of 19620 Pascal.

Outlet Boundary Condition

Outlet boundary condition is taken as mass flow rate of water. Four types of flow rates are considered i.e. 5.61 L/s, 7.3 L/s, 11.1 L/s, 12.9 L/s at different speeds.

After completing CFX-Pre-process, solver tool will solve the problem in terms of numerical quantity. This quantity is showed by the CFX-Post in which results are in the form of vector profile, contour profile and streamlines. These results are discussed below.

RESULTS AND DISCUSSION

Characteristics of fluid flow will displays in CFX-Post tool. Results which are obtained by solver will take into account to understand the flow region.

Both results are compared to check nearness of values. CFD results which are obtained by simulation is explained below:

Grid Independent Test

Simulation work begins with mesh generation. So different mesh sizes can be generated in turbine geometry. This test is done to know about variation of results with respect to change of mesh size, i.e. number of elements and number of nodes. When the mesh cell size is lesser, efficiency of turbine is also low. Gradually increase the mesh size, it will show the efficiency of turbine will increase up to one stage. After that further increase the size of mesh but no effect on efficiency. It will become constant shown in Figure 3 and Table 1. So, that value of mesh size is considered for analysis.

Figure 3: Grid Independent Test

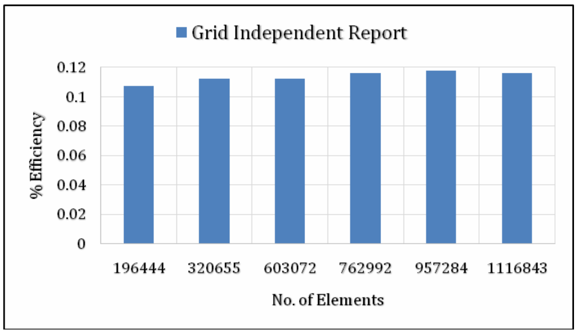


Table 1: Comparison of No. of Nodes and No. of Elements Verses Efficiency

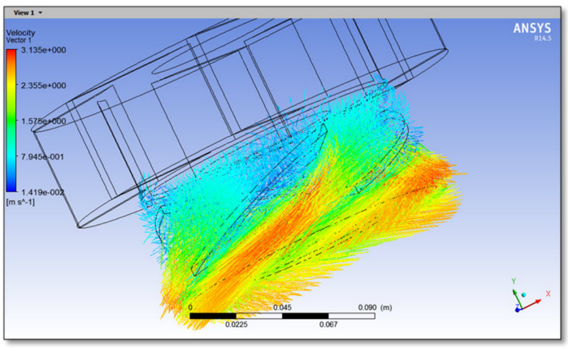
Sr. No.	Nodes	Elements	Efficiency %
1	40188	196444	0.1072
2	64032	320655	0.1121
3	111116	603072	0.1123
4	143322	762992	0.1161
5	180911	957284	0.1176
6	215217	1116843	0.1161

Results shows very minor change in efficiency with respect to mesh size as shown in Figure3. So, 603072 elements of mesh size is selected for analysis.

Vector Profile in Default Runner Domain and Runner Blade

In runner domain the velocity vectors are entered at minimum velocity of which is shaded with sky-blue colour and exit with reddish colour. This indicates the flow is transferring energy to runner blade. Magnitude of vectors are in the form of velocity potential, shown in Figure 4. Vectors show how the direction of flow occur, how it strikes on blade etc.

Figure 4: Velocity Vector Profile in Runner Domain



It is clearly seen that the velocity vectors are firmly passed through the runner domain. Colour is indicating the fair result of fluid flow.

Streamlines in Turbine Assembly

Flow of fluid particles are described by streamlines. Different flow rates and at different speeds fluid flow is identified. Streamlines are in the form of velocity profile. Snapshot of streamline is shown in Figure 5.

Turbine Power Versus Rotation Speed Under Constant Flow Rates

Table 2 describes the turbine shaft power and head versus rotational speed and efficiency versus speed under constant flow conditions which are graphed in Figures 6 to 9.

Figure 5: Streamlines in Turbine Assembly

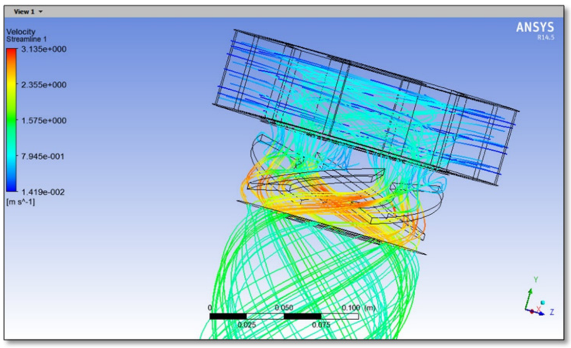
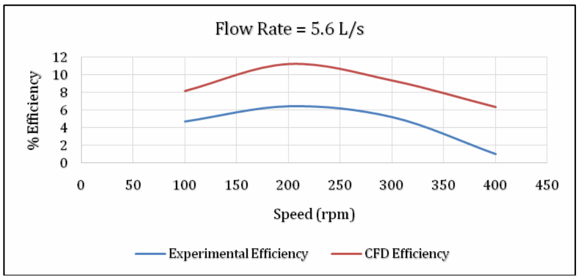


Table 1: Comparison of No. of Nodes and No. of Elements Verses Efficiency

Flow Rate	Head	Speed	Torque	Experimental	CFD Efficiency Efficiency
kg/s	m	rpm	Nm	%	%
5.6	2	100	0.858	4.713	8.177
	2	200	0.591	6.440	11.265
	2	300	0.328	5.215	9.378
	2	400	0.166	1.003	6.336
7.3	2	300	0.867	12.836	19.0100
	2	400	0.562	12.614	16.4270
	2	500	0.312	10.18	11.4056
	2	600	0.100	5.385	4.37229
11.1	2	730	0.850	21.311	29.8356
	2	800	0.614	17.213	23.6184
	2	900	0.270	9.528	11.6842
	2	960	0.068	3.688	3.1296
12.9	2	800	1.350	12.857	44.6838
	2	900	0.949	14.898	35.3374
	2	1000	0.552	10.7256	22.8218
	2	1100	0.157	5.5102	7.15893
	2	1130	0.041	2.653	1.89989

Figure 6: Efficiency vs Speed at Constant Flow rate 5.61 L/s



A Figure 7 shows a good agreement of CFD results with experimental results. Nature of curve nearly same. Difference is created due to some assumptions made in CFD calculations.

Figure 7: Efficiency vs Speed at Constant Flow Rate 7.3 L/s

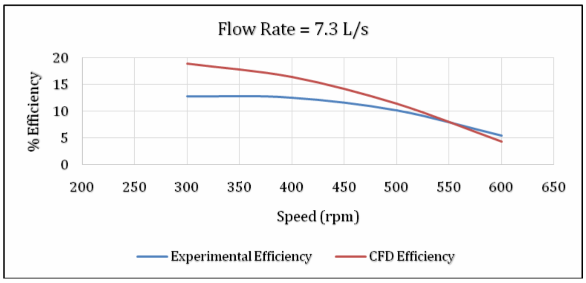


Figure 8: Efficiency vs Speed at Constant Flow Rate 11.1 L/s

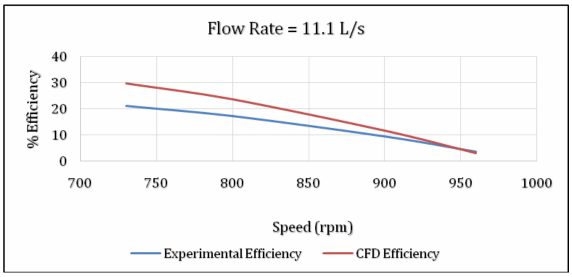
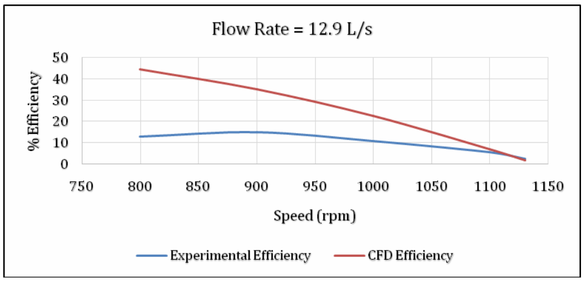


Figure 9: Efficiency vs Speed at Constant Flow Rate 12.9 L/s



Above results shows that the nature of curve are nearly same and the values differs due to neglecting friction at walls, smooth walls, no slip conditions at surface contacts with fluids.

CONCLUSION

- In flow simulation result shown in Figure 9, the highest measured shaft power was 113.094W with an efficiency of 44.68% at 12.9 L/s. Maximum power is produced up to some extent after that it is reducing in nature.
- Observing Figures 6 to 9, by increasing the flow rate and speed of runner, the efficiency is increased up to some extent after that it is gradually decreased.
- After doing simulation work an experimental resultsand CFD results (shown in Figures 6 to 9)are seems to be same. Only the difference is that the CFD results do not consider friction at walls, slip condition, roughness of walls of blades, casing etc. Their nature of curve from the results are approximately matching. This variation is about 31.21% with respect to experimental results.

- Low head propeller turbine technology was determined to be the most suitable option based on several merits including improved access, ease of manufacture, portability, low cost, and reduced system complexity. A propeller turbine with a runner with constant thickness, curved, twisted, and variable chord length blades was designed to enable more power generation due to minimising the losses with an operating head of 2 m and 25 L/s flow rate.

REFERENCES

1. Bansal R K (1983), "A Textbook of Fluid Mechanics and Hydraulic Machines", *1st Edition, Laxmi Publications*.
2. Patrick Ho-Yan (2012), "Design of a Low Head Pico Hydro Turbine for Rural Electrification in Cameroon", *University of Guelph*.
3. Rajput R K (1998), "A Textbook of Hydraulic Machines", *1st Edition, S. Chand & Company*.
4. Singal R K (2009), "Hydraulic Machines Fluid Machinery", *International Publishing House Pvt. Ltd*.
5. Singh and Nestmann (2009), "Experimental Optimization of a Free Vortex Propeller Runner for Micro Hydro Application", *Elsevier Journal of Experimental Thermal and Fluid Science*, Vol. 33, pp. 991-1002.
6. Singh and Nestmann (2011), "Experimental Investigation of the Influence of Blade Height and Blade Number on the Performance of Low Head Axial Flow Turbines", *Elsevier Journal of Renewable Energy*, Vol. 36, pp. 272-281.
7. Subramanya K (2010), "Fluid Mechanics & Hydraulic Machines", *1st edition, Tata McGrawHill Publications*.