Simulation and Flow Analysis of 3 kW Propeller Turbine

[1]Ma Su Hlaing Thein, [2]Dr. Nan Aye Myint

[1]Master Candidate, [2]Lecturer
Department of Mechanical Engineering
West Yangon Technological University, Yangon, Myanmar
Email – [1]suhlaingthein88@gmail.com, [2]cherrypan082@gmail.com,

Abstract: *Most of the electrical power is produced by hydro turbines. The most effective point for propeller turbine is the cheapest to get energy. This paper is used computational fluid dynamics software package (CFX) to study and analyse the flow condition in propeller turbine. I have chosen site location from Ma Mya Dam project. It is located in Ye' Ywar, Kha Paung, Paung long and Zaw Gyi plants. The propeller turbine components are runner, spiral casing, guide vane, draft tube and drive shaft. The propeller turbine flow rate and head are 0.2816m^3/s and 2m. The runner diameter is 238mm and hub diameter is 73 mm. There are four blades in this propeller turbine. The three dimensional blade profiles and assembly of propeller turbines is draw by Inventor software. Flow analysis of propeller turbine is conducted by numerical simulation (CFX).*

Key Words: *Propeller Turbine, Inventor software, CFD, Flow rate, ANSYS CFX.*

1. INTRODUCTION:

Today's global renewable energy is the most important a role of hydro-power plant. Higher usage of renewable energy could not solve the problems overnight, it is very important move in the right direction [1]. Hydraulic turbines change the energy of flowing water into mechanical energy and then convert changes as electrical energy. Hydro-electric power is get energy and lowest price. A hydropower build depends on the quantity of water flow through a turbine and head of water available [2]. Propeller turbine's rotation speed is faster rate and low rate, thus which are more solid than other types of turbines. Components of propeller turbines are spiral casing, runner, guide vane, draft tube and drive shaft.

In computational fluid dynamics (CFD) technique is gained high level of safety and develops a practical tool made another type of fluid flow machinery. Propeller turbine can be stable test with computational fluid dynamic (CFD). Numerical simulation methods are costly and time consuming and then to provide detail flow of inside turbine. CFD could be considered between different components [2].

Diaelhag Khalifa, M.Sc [2013] is designed blade system of axial flow turbine using CFD. Numerical simulation result shows velocity and pressure variation by average circumference area (ACA) and distribution between hub and shroud, efficiency and power output affected by the rotational speed of the runner [3].

Ruchi Khare, [2015] is compute effect of solidity on flow pattern in Kaplan turbine runner. In this paper, the runner with different number of blades and solidities are shown. The decrease in absolute velocity and increase in whirl velocity from hub to tip at inlet and outlet at all solidities confirms characteristics of axial flow reaction turbine [4].

Aadilahemad Momin, [2017] is design and development of Kaplan turbine runner blade. Kaplan turbine is selected depend on the main characteristics and then each section, velocity and blades angles from velocity diagram are calculated. Using of simulation on the blade in Solid Works software know velocity distribution on the blade [5].

2. PROPELLER TURBINE:

Propeller turbine is axial flow reaction turbine, generally used for low head. The basic propeller turbine consists of a propeller turbine, similar to a ship's propeller, fitted inside a continuation of the penstock tube. The fluid in the propeller turbine converts through a right angle into the axial direction between the guide vanes and the runners and then work on the runners. The propeller runner has always four or six blades similar to a ship's propeller.

The flow approaching the runner blades is a free vortex that is whirl velocity inversely proportional to radius, whereas the velocity of the blades is directly proportional to radius. To create for the varying between the fluid velocity and the blade velocity as the radius increases, the blade are twisted as the angle with the axis of the rotor (or) shaft being more at the tip than at the hub [6] .

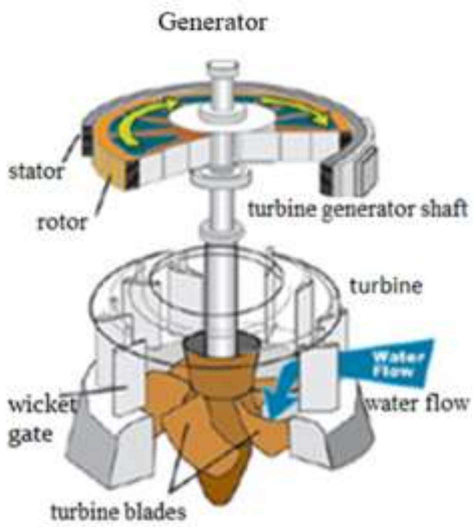

Figure 2.1 Propeller turbine

2.1 Calculated Results for 3kW Propeller Turbine

The calculated results data for propeller turbine are shown in the following table (2.1).

Table 2.1. RESULTS DATA FOR PROPELLER TURBINE

Parameters	**Symbol**	**Value**	**Unit**
Power	P	3	kW
Gross head	H	2	m
Net head	H_n	1.88	m
Flow rate	Q	0.2816	m^3/s
Hydraulic efficiency	η_h	0.94411	-
Runner diameter	D	0.2383	m
Hub diameter	d	0.073	m
Number of blade	z	4	-
Specific speed	n_{QE}	1.6869	-
Rotational speed	n	28.3152	s^{-1}
Flow velocity	V_f	6.4712	m/s
Guide vane outer diameter	D′	0.321	m
Number of guide vane	z_1	8	-
Guide vane angle	α	83.07°	$^{\circ}$
Casing inlet velocity	v_f	4.4791	m/s
Casing diameter	D″	0.3813	m
Shaft diameter	d_s	20	mm

The selection of the angle of attack is depending on the NACA series Air foil. I have chosen for blade profiles GOE 430 air foil. Propeller turbine is draw by Inventor software.

3. NUMERICAL SIMULATION:

The results can be obtained by testing to predict the flow behaviour with Naiver-Stocks equations and turbulence models in CFD. Fig. 2 shows various steps to make a CFD simulation. All the components of the hydraulic geometry are prepared, domain discretization i.e., meshing and physics definition can consider doing for the system. Using pre-processing can visualize the result which can be computed solver.

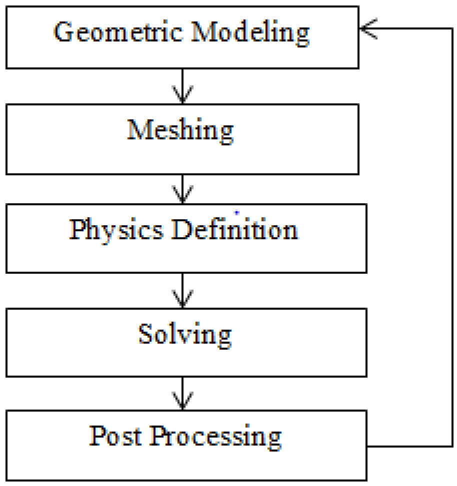

Figure 3.1 Flow chart of ANSYS software

Turbo and ANSYS CFX 16 software were used performing by the computational investigations. In CFX-Pre defined the physics of the simulation domain.The mesh files, specification of flow physics, boundary condition, initial values and solver parameters are included the processing module. Using the boundary conditions specified in the definition file get by The CFX-solver module solver the governing equation from the post-processor.

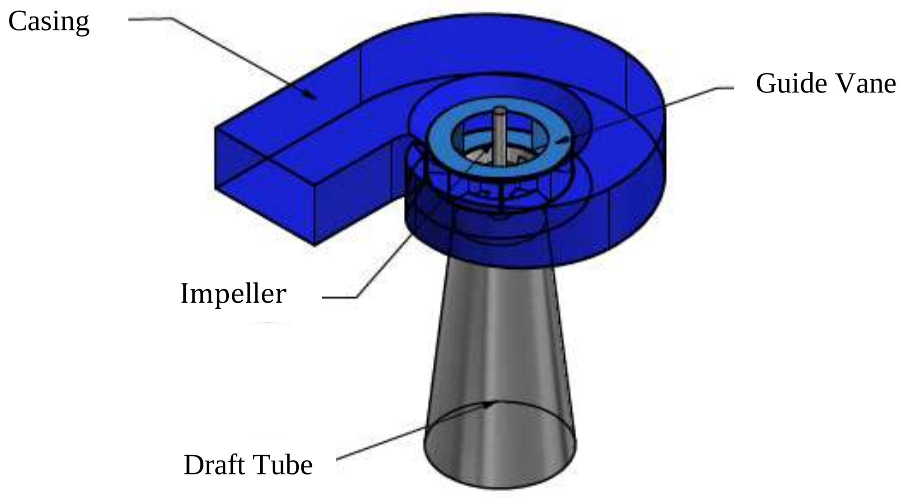

Figure 3.2 Propeller turbines by using Inventor software

Using CFX-posits a flexible state of the art post-processor observed by the result of the CFD analysis. The CFX-posits allows easy visualization of the result of CFD simulations.

3.1 Meshing

For the CFD model, volumetric meshing with unstructured tetra meshing option was form for grid generation in spiral casing, guide vane, draft tube etc. Moreover, meshing of propeller turbine was taking away in Turbo-grid software using hexa meshing option.

Based on the understanding of behaviour of flow, it is necessary more concentration more mesh element near wall is having large velocity gradients. Away from wall, the mesh size can be made coarser to reduce overall mesh numbers.

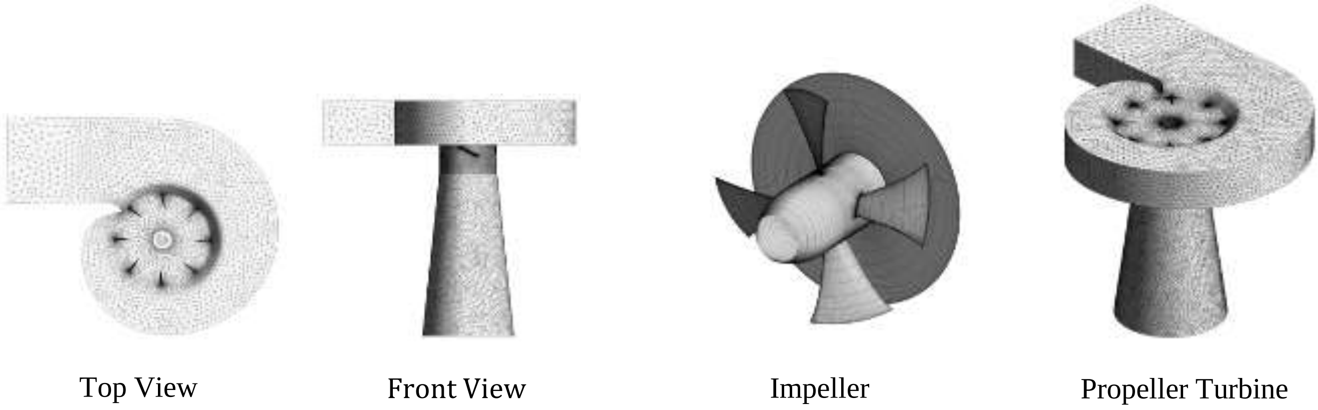

Figure 3.3 Meshing for propeller turbine

Table 3.1 DETAILS OF MESHING VARIOUS COMPONENTS

Domain	Nodes	Elements
R1	551080	517264
S1	251531	936449
S2	62451	176185
All Domain	865062	1629898

Estimating of grid type has been selected depending upon the geometric complexity, flow field and the cell and element types supported by the solver. The mesh density is high enough to store all relevant flow feathers. Good quality mesh is important to get the accurate results.

3.2 Physics Definition

In numerical flow simulation, boundary condition is the most important role. This boundary condition is made correct fill in the definition file. The domain was specified as a Buoyant, Steady state analysis with working fluid as water at 25°C and reference pressure as 0 atm. The turbulence model was selected as (SST) model for analysis. Domain of angular velocity is 1700 rev/s and gravity is 9.81 m/s^2.

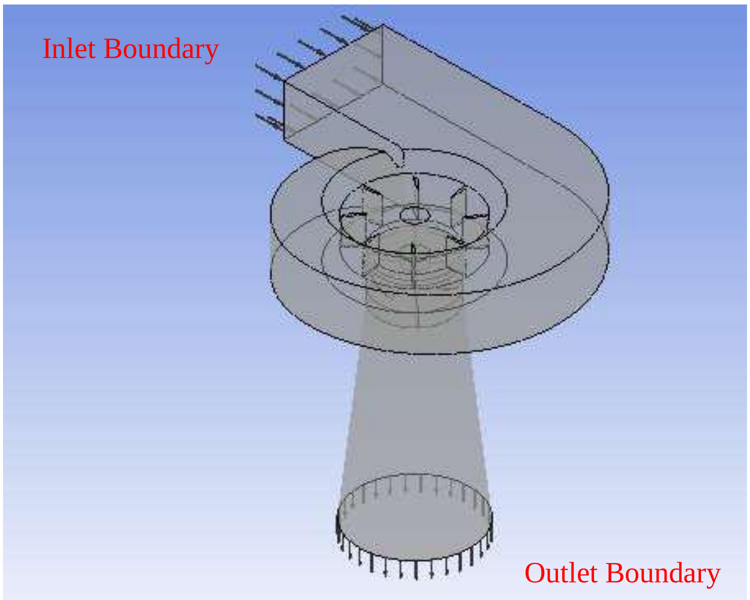

Figure 3.4 Locations of inlet and outlet boundary condition

At the inlet of the casing, the velocity condition is effected on the runner, at the outlet of the draft tube, static pressure is using as outlet boundary condition as shown in Fig.5. The flow regime at both inlet and outlet boundaries were specified as subsonic.

Table 3.2 INLET BOUNDARY CONDITION AND OUTLET BOUNDARY

Type	Name	Face to apply	Parameters value
Inlet volume flow	Inlet volume flow 1	The inner face of the casing	4.4791 m3/s
Static pressure	Static pressure	The outer face of the draft tube	101300 [Pa]

All the internal and outer walls of the flow region were specified as the boundary type wall with flow condition as no slip wall. Solution scheme and convergence criteria specified by the solver control parameters. High resolution scheme was specified for the solution while for convergence the residual target for Max Residual values was specified as 10^{-4} [8].

4. RESULTS AND DISCUSSION:

Design and off-design operation conditions are depends on the low head vertical propeller turbine. The CFD simulations are assumed converged when all the residuals are less than 10^{-4}, which is sufficient for most engineering problems. The numerical flow simulation of propeller turbine under unsteady flow condition is made by using ANSYS CFX 16 software.

Fig.4.1 is show the flow velocity inside the propeller turbine. It shows the water flow from the inlet penstock, passing through the spiral casing and guide vane and then rotate the runner leaft the draft tube outlet.The velocity increase the runner tip and then velocity decrease the flow rate through the draft tube.

Fig. 4.2 is about the variation of pressure contour in propeller turbine. The pressure decrease over the runer blades and then increase alomg the draft tube at almost an atmosphereric pressure.

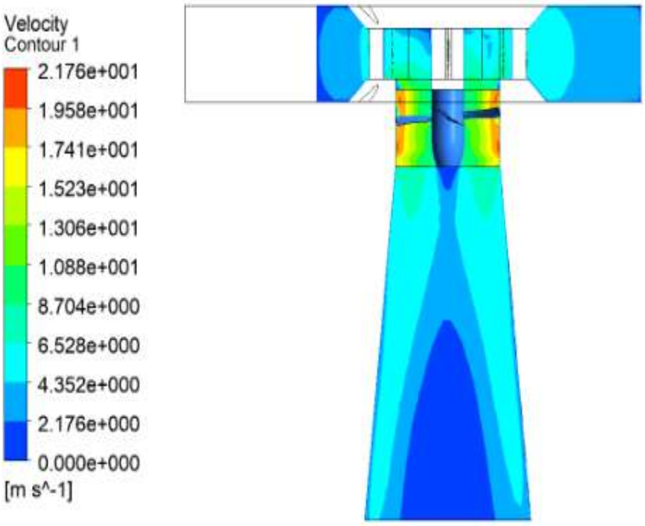

Figure 4.1 Variations of velocity components in propeller turbine

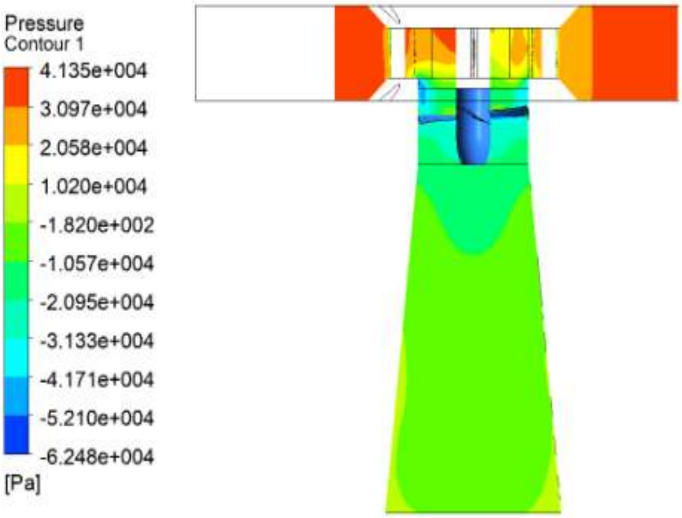

Figure 4.2 Variations of pressure components in propeller turbine

Fig .4.3 is about the pressure on blade to blade at mid span (50%). In this figure pressure contours it is seen that pressure the inside suction of runner decreases from its inlet to outlet which is just because the pressure energy is being converted into mechanical energy.

Fig.4.4 is about the pressure difference between pressure and surface. Firstly increase from leading edge as water strikes on the blade and after that decreases smoothly towards the trailing edge. Apart from design condition the pressure difference enough to create the required torque on blades but is not get optimum efficiency.

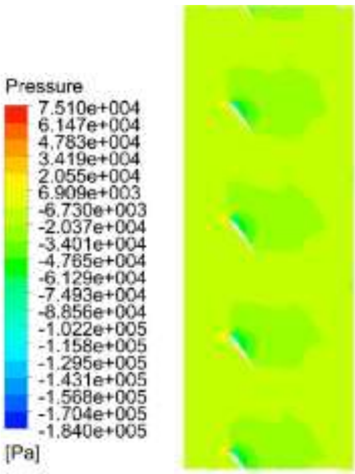

Figure 4.3 Pressure contours on blade to blade view at mid span (50%)

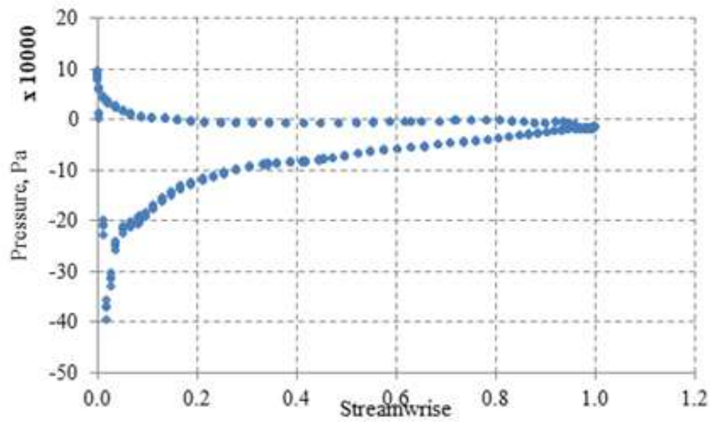

Figure 4.4 Blade loading at mid span (50%)

Fig .4.5, Torque and flow rate are simulated for five different percentages. This simulation input data of speed is 1700 rev/s and gravity 9.81 m^2/s and to get the simulation result of flow rate and torque. If the flow rate is increase, torque is also increased.

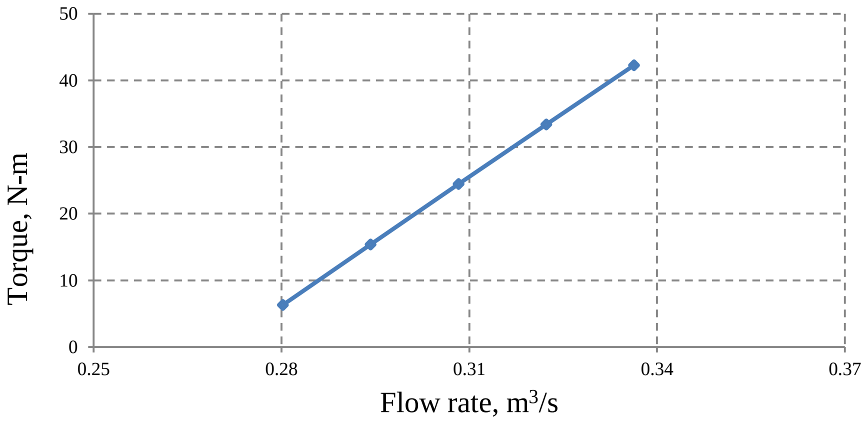

Figure 4.5 Variation of torque and flow rate

The amount of power and flow rate as shown in fig 4.6. So, according to Fig.4.6, the amount of power is directly proportional to the flow rate.

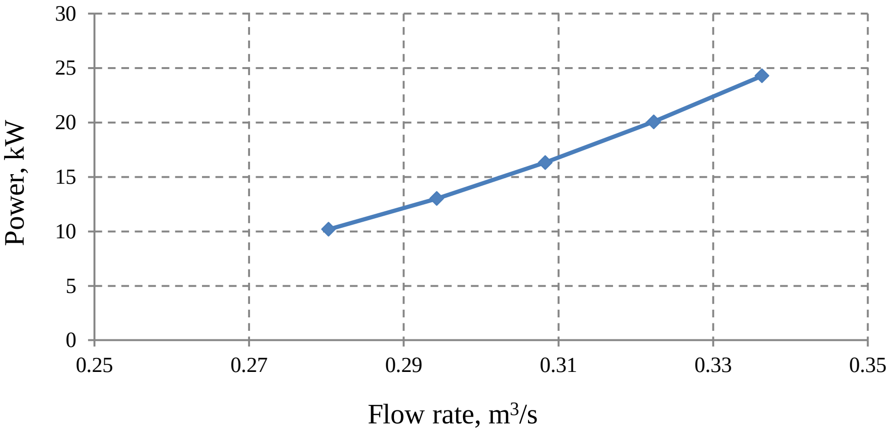

Figure 4.6 Variation of power and flow rate

5. CONCLUSION:

Hydropower plant is one of the most important parts to generate electricity. The output power of turbine depends on the head and flow rate. Propeller turbine is draw by Inventor software. The simulation input data of speed is 1700 rev/s, casing inlet velocity 4.4791 m/s and gravity 9.81m^2/s and then to get the simulation result of flow rate and torque. The velocity and pressure distributions acting on the propeller turbine are simulated by using ANSYS software. From this result, it can be said that the required output power is mainly depends on the flow rate. So, in order to get the maximum flow rate, the blade inlet and outlet angle should be adjusted.

6. RECOMMENDATIONS:

According to the simulation result, ANSYS-16 is able to produce good computational results in an efficient way. This numerical simulation result should be compared the real situation of same design propeller turbine and redesign the real propeller condition to obtain the maximum efficiency.

REFERENCES:

1. Ajaz Bashir Janjua, Muhammad Shahid Khalil and Muhammad Saeed, (2013): Blade Profile Optimization of Kaplan Turbine Using CFD Analysis, *Mehran University Research Journal of Engineering & Technology, 32*(4).
2. Ravi Shankar Shukla, (2017):Design of Propeller Turbine for Micro Hydro Power Station Using CFD, *International Journal of Scientific Engineering and Science*, *1*(7), Pp.37-41.
3. Diaelhag Khalifa, M.Sc, (2013): Simulation of an Axial Flow Turbine Runner's Blades Using CFD, *1st Annual International Interdisciplinary Conference, Azores, Portugal*.
4. Ruchi Khare and Vishnu Prasad, (2015): *Effect of Solidity on Flow Pattern in Kaplan Turbine Runner, 6*(2).
5. Aadilahemad Momin, Nairutya Daveand and Parth Patel, (2017): Design and development of Kaplan Turbine Runner Blade, *International Journal of Innovative Research in Science, Engineering and Technology,* 6(8).
6. Ruchi Khare and Vishnu Prasad, (2015): Effect of Solidity on Flow Pattern in Kaplan Turbine Runner, *6*(2),
7. https://www.indiamart.com/proddetail/ propeller.
8. Shahil S Charania, Vishal Soil, (2012): Evaluation of Vertical Kaplan Turbine Using CFD, *Proceedings of the Thirty Ninth Conferences on Fluid Mechanics and Fluid Power*.