Numerical Analysis of Kaplan Turbine Runner for Micro-hydropower

Soe Pyae Aung[a], B. SHIN[b]

Abstract

The row of renewable energy has been considered as the first priority for global environmental concerns. Hydropower is a good way of reducing environmental impacts. Low head power plants are expected to be implemented increasingly in the future for economic, geographical and environmental purposes. Axial flow turbines are well suited for these types of applications. The conventional method of turbine performance become costly and time-consuming for several designs testing for design optimization of hydro turbines. The advance in numerical methods for simulation, Computational Fluid Dynamics (CFD) has become an effective tool for prediction detailed flow information between the space of the turbine runner to obtain high-efficiency turbine design. In the present paper, the detailed design calculation of the runner of the Kaplan turbine and computational results of the turbine such as the runner exerted power, pressure coefficient, flow rate, and turbine characteristics curves are presented.

Keywords: Computational Fluid Dynamics, Kaplan turbine, Blade design, Turbine characteristics

1. INTRODUCTION

Hydraulic turbines convert hydraulic energy of water into mechanical energy which is further converted into electrical energy. This energy obtained is known as hydro-electric power which is one of the cheapest forms of energy generation. The amount of production of the electricity from a hydropower installation depends on the quantity of water passing through a turbine and head of water available. The choice of turbines depends mainly on the head and the flow rate. There are basically two types of hydro turbines in hydropower systems such as impulse and reaction turbines. In an Impulse turbine, water is stored in a reservoir at a high place as potential energy. This energy is changed into kinetic energy in water jet and struck the buckets to get power from water. For reaction turbine, combined the action of pressure and water flow are changed into mechanical energy. Between them, reaction turbines such as axial flow turbines are more suitable for low head operation [1].

Axial flow turbines are available with both fixed blades called propeller turbines and variable pitch blades called Kaplan turbines which may be mounted either horizontally or vertically. It allows the fluid to enter the runner axially and discharge the fluid axially. In axial turbines, the runner is the most important component of the turbine. In the performance test, the experimental test of turbine models does not give detailed information about the flow behaviors and variation of local design parameters like circulation around the runner, lift force, the degree of reaction, the coefficient of pressure and so on. And it also involves a high cost due to construct test rigs and time-consuming.

On the other hands, the development of the numerical methods and computational power lead to computational fluid dynamics. CFD can be used to understand flow phenomena and to check the turbine characteristics of alternate designs of turbines for optimization before final experimental testing. Also, numerical simulation by CFD can minimize the cost and reduce the time spent on optimizing the Kaplan turbine design. In addition, CFD becomes a cost-effective tool to provide detailed flow information inside the complete turbine space as a whole so that interaction between different components could also be considered. Therefore, CFD has been widely used by designer and researchers to optimize its design. Sometimes, however, there are differences between the two results such as in the maximum value of the efficiency[2] even though the validation of numerical simulation with experimental results are equally the same efficiency variation pattern.

Ma Htoo Htoo Hlaing designed a 3kW Kaplan turbine for micro hydropower[3], and constructed it and tested its performance. According to the test result, the runner was modified and the efficiency of the turbine was more improved than the existing one. The overall efficiency and mechanical efficiency was 56% and 64%. Win Pa Pa Aye[4] analyzed velocity variation on a runner by a numerical method in solid work software. In her paper, it was checked numerical and experimental data of velocity distribution on the runner. But she did not consider the effect of flows on turbine runner output.

a) Master Student, Dept. of Mechanical Design Systems Engineering
b) Professor, Dept. of Mechanical Design Systems Engineering

In this paper, the detailed design of 3kW turbine runner and 2-D simulation of a mid-span of the runner blade using CFX commercial software is presented. It also includes that computational results checked with calculated ones.

2. DETAILED DESIGN CALCULATION OF 3KW KAPLAN RUNNER

There are many essential parts in axial flow turbine such as guide vane, runner, casing and draft tube. Kaplan turbine was designed to have a minimum number of blades. The runner is the most important component of the turbine and its blade profile is designed at the different section from the hub to the casing to get the best performance and efficiency. Runner blades have a slight curvature and cause relatively low flow losses. The water enters the blades in an axial direction from one side and leaves through the other side so that a large quantity of water flows through the runner.

In this paper, the 1-D flow method[4] was chosen for designing the runner of propeller turbine and calculations were done as below: The design power was 3kW and available head on the turbine was 2m.

The power developed by a turbine and the other parameters were calculated by the following formula.

Power
- $P = \rho g Q H \eta_o$ (1)

Specific speed of the turbine
- $Ns = 885.5/H^{0.25}$ (2)

Speed of the turbine runner
- $N = NsH^{1.25}/P^{0.5}$ (3)

Periphery coefficient
- $\emptyset = 0.0233N_s^{2/3}$ (4)

Diameter of the runner
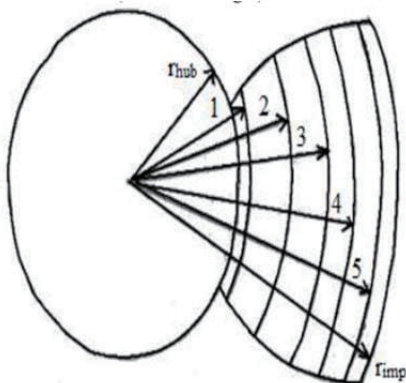- $D = (84.5\emptyset\sqrt{H})/N$ (5)

Fig. 1. Blade divides into five sections

Based on specific speed, values of flow coefficient, speed coefficient and hub to shroud diameter ratio (d/D) were selected.

2.1 Radius for each blade section

In the space of the runner, it can be divided into five sections as shown in Fig.1.

Section I,

$$r_1 = \frac{d}{2} + 0.015d \tag{6}$$

Section III,

$$r_3 = \frac{D}{2}\sqrt{\frac{1 + (d/D)^2}{2}} \tag{7}$$

Section II,

$$r_2 = r_1 + \frac{r_3 - r_1}{2} \tag{8}$$

Section IV,

$$r_4 = r_3 + \frac{r_5 - r_3}{2} \tag{9}$$

Section V,

$$r_5 = \frac{D}{2} - 0.015D \tag{10}$$

2.2 Geometric selection of airfoil

After having detailed parameters of the turbine runner, it is necessary to get the blade shape for the various chord length of each blade section. Therefore, the blade shapes of five cross sections were obtained by multiplying chord lengths into x, y coordinates of the unit chord length of the NACA wing. Figure 2 shows an example of the third cross-section of the turbine blade.

Equations for x, y coordinates of the unit chord length of the NACA wing are as follows.

$$y_t = \frac{t}{0.2}(0.2969\sqrt{x} - 0.126x - 0.3516x^2 + 0.2843x^3 - 0.1015x^4)$$

$$y_c = \frac{m}{p^2}(2px - x^2) \qquad \text{from x=0 to x=p}$$

$$y_c = \frac{m}{(1-p)^2}((1 - 2p) + 2px - x^2) \qquad \text{from x=p to x=c}$$

$$\tan\theta = \frac{dy_c}{dx}$$

$$y_u = y_c + y_t\cos\theta$$

$$y_l = y_c - y_t\cos\theta$$

2.3 Flow parameters of the turbine

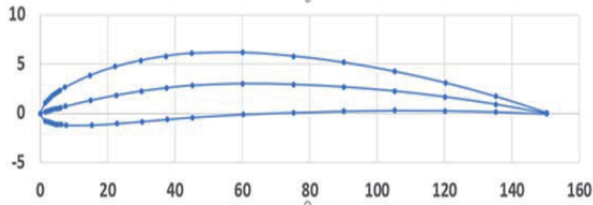The inlet and outlet velocity diagram of the Kaplan turbine runner is shown Fig.3.

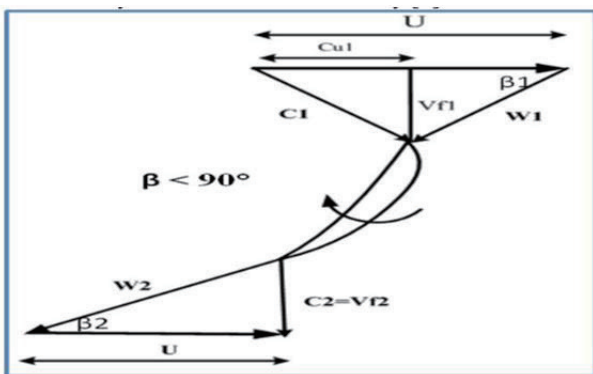Fig. 2. Blade shape of the third cross-section

Fig. 3. Inlet and outlet velocity triangles of Kaplan turbine

Tangential velocity,

- $U = 2\pi N r / 60$ (11)

Whirl velocity,

- $V_u = \tau_h g H / U$ (12)

Blade angles at inlet and outlet

- $\tan \beta 1 = V_{f1} / (U - V_{f1})$ (13)

- $\tan \beta 2 = V_{f2} / U$ (14)

Blade spacing

- $t_s = 2 r \pi / Z$ (15)

Parameters of the runner are shown in Table1. Detailed parameters for five cross sections of the Kaplan Turbine runner were obtained by the above equations.

3. BOUNDARY CONDITION

This shape was obtained by drawing in Inventor software[5]. And then triangle meshes were used for computation. The number of nodes and elements are 100,118 and 99,943. Inlet boundary was front of the leading edge and outlet boundary was back of trailing

Table 1. Detailed parameters of runner for 3 kW

	Section				
	1	2	3	4	5
Radius	0.056	0.08	0.103	0.118	0.132
attack angle(deg)	14.89	5.305	2.4	1.83	1.62
latic angle(deg)	44	51	57	61	64
blade spacing(m)	0.088	0.125	0.162	0.184	0.207
chord length(m)	0.097	0.128	0.15	0.155	0.153
blade mean angle(deg	61	45	35	31	28
U(m/s)	4.9	7.03	9.13	10.38	11.63
β1	76	52	39	34	30
β2	49	39	32	29	26

edge. Wall boundary was created at two NACA wings. Another two edges were periodic boundaries conditions.
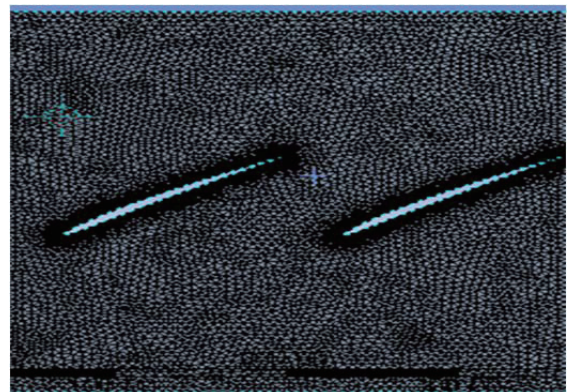

Fig. 4. Two dimensional geometry meshing

4. NUMERICAL SIMULATIONS

The water flow rate and atmospheric pressure were imposed at the inlet and the outlet boundaries, respectively. No slip wall boundary condition was used on the blade. k-omega SST turbulence model which is suitable for this kind of flow was used. Numerical simulation was done by using CFX software[6] with various flow rates at 843rpm and 2 m head of water.

The average pressure at the blade inlet and outlet was calculated for the given flow rates. Pressure contour is presented in Fig.5. Maximum

pressure was 58.95kPa at leading edges and minimum pressure occurred on the suction side of -17.57kPa.
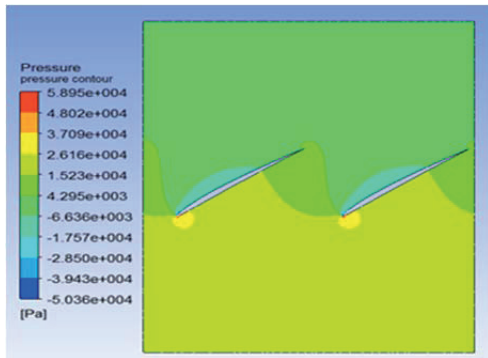

Fig. 5. Computational result of pressure contour for Q=0.287(m³/s)

The pressure difference between the suction side and pressure side leads to the lift force at the blade and produces mechanical power. The pressure coefficient on the runner blade surfaces from its leading edge to trailing edge at its mid-span at a discharge of 0.287 kg/s and speed of 843rpm is shown in Fig. 6. The pressure difference between pressure and suction surfaces firstly increases from leading edge as water strikes on the runner and after that, it decreases toward the trailing edge.
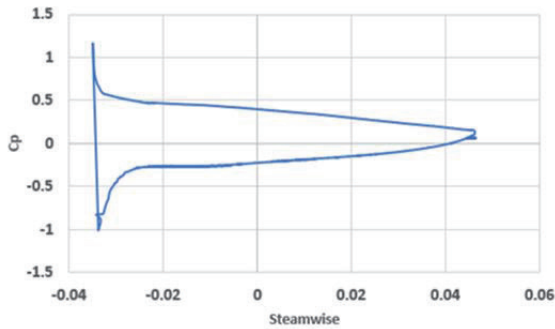

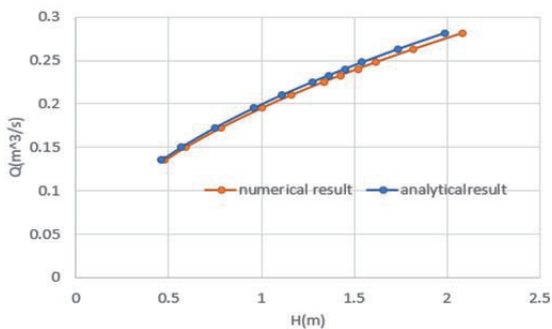Fig. 6. Pressure coefficient of the blade at midspan

Fig 7. Mass flow rate vs head

An analytical solution was done to check the simulation results. The analytical solutions are obtained by turbine scaling laws.

Figure 7 shows that the relation of mass flow rate and head of the turbine. Power and head relation
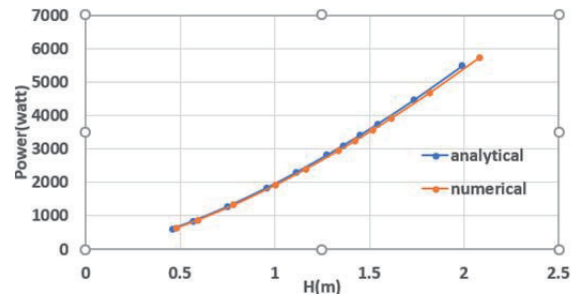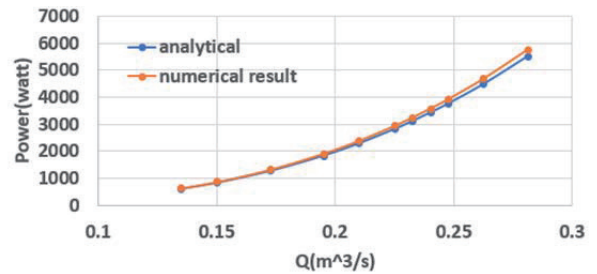

Fig. 8. Power and head

Fig. 9. Power and flow rates

is shown in Fig. 8 for two methods. Both results have the same trends except numerical results are slightly lower than theoretical ones. The power exerted by the runner is changed with variation in input flow rate as shown in Fig.9. According to Fig.9, although numerical analysis results are a bit greater than theoretical ones, they have the same trends also.

The turbulence kinetic energy around the airfoil is shown in Fig.10. The maximum turbulence kinetic
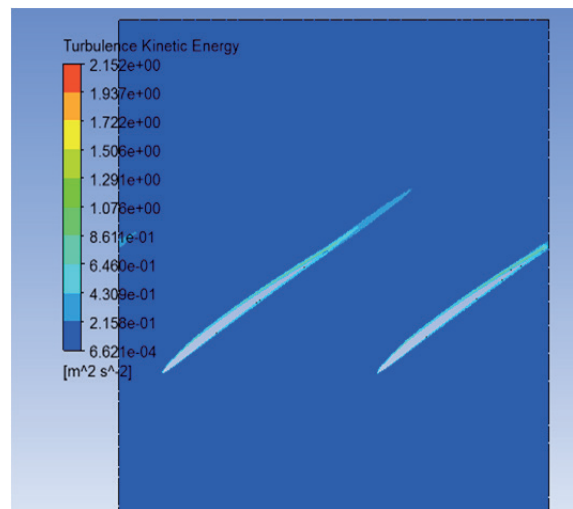

Fig. 10. Computational result of turbulence kinetic energy contour for Q=0.278(m³/s)

energy occurs at the trailing edge of the airfoil with the value of 1.58828 (J/ kg).

5. CONCLUSION

It has been observed from a comparison of the numerical simulations and theoretical results of the turbine at various flows rates. All the simulation results have the same trends with theoretical ones. Hence, it is concluded that CFD approach can be used to study the flow pattern inside the turbine space and to optimize the design by different combinations of the design parameters and geometry at low cost in lesser time. The result of optimized design may need to be verified through model testing. This procedure will minimize time and the amount spent in the development and optimization of hydraulic turbines. It was confirmed that the higher the flow rate, the more output power can be obtained.

Nomenclature

P=output power of turbine(kW)
U = tangential velocity (m/s)
V_u = whirl velocity (m/s)
D = diameter of the turbine runner (m)
r = radius of the turbine runner(m).
g = gravitational acceleration (m/s^2)
H = net head (m)
p = static pressure at any point on blade surface profile (Pa)
Q = discharge through turbine (m^3/s)
W = relative velocity at any point of blade surface (m/s)

β = relative flow angle (°)
t_s= spacing between two runners (m)
ρ = mass density of water (kg/m^3)
z = total number of blade
η_o= over all efficiency of the turbine
Ns = specific speed of the turbine
N = speed of the turbine (rpm)
$V_{f1,2}$ = flow velocity (m/s)
τ_h = hydraulic efficiency
$\emptyset$ = Periphery coefficient
Subscripts 1 and 2 denote values of parameters at inlet and outlet of runner respectively.

REFERENCES

1) Edy E. Jiménez: "The final study report, achievable renewable energy targets for Puerto Rico's renewable energy portfolio standard Puerto Rico," Puerto Rico's energy affairs administration, 2009.
  (http://www.uprm.edu/aret/) .

2) V. Prasad, V. K. Gahlot, and P. Krishnamachar: "CFD approach for design optimization and validation for axial flow hydraulic turbine," Indian J. Eng. Mater. Sci., Vol. 16, pp. 229-236, 2009.

3) Htoo Hlaing: "Design, Construction and Performance Testing of 3kW Kaplan Turbine for Micro Hydropower," Ph.D. Thesis at Yangon Technology University, June 2010.

4) Win Pa Pa Aye, Zaw Moe Htet, and Htay Htay Win: "Design and Velocity Distribution of Runner Blade for Axial Flow Turbine," Int'l J. Sci. and Eng. and Tech. Research, Vol 3, pp. 2205-2210, 2014.

5) https://www.autodesk.com/products/inventor/overview

6) https://www.ansys.com/products/fluids/ansys-cfx