Flow Simulations on Blades of Hydro Turbine

Weerapon Nuantong, Sirivit Taechajedcadarungsri*

Mechanical Engineering Department, Faculty of Engineering, Khon Kaen University, Khon Kaen 40002, Thailand
Tel: +66-43-202845, Fax: +66-43-202849, *Email: sirtae@kku.ac.th

Abstract

One of the renewable energy is hydroelectric power generation. Hydro turbine is a rotary engine that extracts energy from a fluid flow by transferring the potential energy to electricity generation. Depending on head and water flow rate, variation of pressure and momentum cause the runner blades to rotate. This research studied the effects of pressure and velocity of fluid flow on blades which help in improving the hydro turbine efficiency. The computational Fluid Dynamics (CFD) was used to simulate the pressure and velocity distributions on blades of hydro bulb turbine, which consists of runner with five blades and rotating at 980 rpm, by using Fluent Software. The LES model of turbulence flow, under the practical condition of unsteady and incompressible fluid flow, was conducted in order to study the effects of blade angles on hydro turbine installed at Huai Kum Dam drainage pipeline. At the average head of 21 m, blade twist angle of 25^o and the blade camber angle of 32^o, the simulation was applied on varying guide vane angle at different angles of 60^o, 65^o and 70^o respectively for comparing the maximum and minimum pressure on blades. The simulation showed that, at guide vane angle of 60^o, 65^o and 70^o, the maximum pressures at leading pressure side are 213 kPa, 217 kPa and 207 kPa and the minimum pressures at leading suction side are -473 kPa, -465 kPa and -581 kPa, respectively. By adjusting the guide vane angle, it clearly affects the pressure distribution and the efficiency of the hydro turbine. This case study will serves as the guideline for blade design of hydro bulb turbine for the improvement of turbine efficiency.

Keywords: Blade angle, Hydro Turbine, LES model

1. Introduction

Nowadays, computational Fluid Dynamics (CFD) is a useful tool to optimize the design of hydro turbines and improve in their efficiencies; CFD is used to analyze the fluid flow helping the design part to be cost and time saving.

The CFD simulation of fluid flow pass hydro turbine is to analyze the effect of blade angle, inlet guide vanes and runner blades, on pressure and velocity distributions of hydro bulb turbine. And the results would be useful as the guideline for blade design of hydro bulb turbine.

2. Description

2.1 Hydro turbine

Hydro turbine is a rotary engine that extracts energy from a fluid flow by transferring the potential energy to electricity generation [1].

The hydro turbine has two broad categories: Impulse turbine and Reaction turbine. One of the reaction turbines is the hydro bulb turbine, which is an axial flow machine suitable for using at medium to low head and high volume flow rate. The number of blades of hydro bulb turbine is suggested to be four, five or six blades on the runner and Pitch/ chord length ratio is about 1-1.5. [2]. Blade diameter depends on available head and flow rate. Hub/blade diameter ratio is about 0.35-0.60 [3].

2.2 Governing equation for CFD

Continuity equation of fluid flow can be written as:

$$\frac{\partial \rho}{\partial t} + div(\rho \mathbf{u}) = 0 \tag{1}$$

Conservation of momentum equation can be given as:

$$\frac{\partial \bar{u}_i}{\partial t} + \frac{\partial \bar{u}_i \bar{u}_j}{\partial x_j} = -\frac{1}{\rho}\frac{\partial \bar{P}}{\partial x_i} + v\frac{\partial}{\partial x_j}(\frac{\partial \bar{u}_i}{\partial x_j}) + \frac{\partial \tau_{ij}}{\partial x_j} \quad (2)$$

Conservation of energy equation can be written as [4]:

$$\rho\frac{DE}{Dt} = -div(p\mathbf{u}) + \left[\frac{\partial(u\tau_{xx})}{\partial x} + \frac{\partial(u\tau_{yx})}{\partial y} + \frac{\partial(u\tau_{zx})}{\partial z} + \frac{\partial(v\tau_{xy})}{\partial x} + \frac{\partial(v\tau_{yy})}{\partial y} + \frac{\partial(v\tau_{zy})}{\partial z} + \frac{\partial(w\tau_{xz})}{\partial x} + \frac{\partial(w\tau_{yz})}{\partial y} + \frac{\partial(w\tau_{zz})}{\partial z}\right] + div(k \quad grad \quad T) + S_E \quad (3)$$

Eddy – Viscosity Model of Subgrid Scale Models is given by [5]:

$$\tau_{ij} = v_t(\frac{\partial \bar{u}_i}{\partial x_j} + \frac{\partial \bar{u}_j}{\partial x_j}) + \frac{1}{3}\tau_{kk}\,\delta_{ij} \quad (4)$$

Eddy – Viscosity Model of Smagorinsky-Lilly can be written as:

$$v_t = \rho L_s^2 \left|\bar{S}\right| \quad (5)$$

L_s can be written as:

$$L_s = \min(Kd, C_s V^{\frac{1}{3}}) \quad (6)$$

3. Simulation

The simulation is on the hydro bulb turbine which consists of five-blade runner and rotates at 980 rpm. With the head of 21 m, blade twist angle of 25^o and blade angle of 32^o, the inlet guide vane angle is set to be 60^o, 65^o and 70^o respectively, for comparing maximum and minimum pressure on blades. By using the CFD code (Fluent), the LES model is set to be unsteady and incompressible flow. The tetrahedral mesh of guide vanes, hub and blades is constructed (Fig 1).

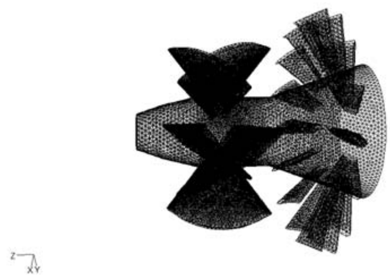

Fig 1. Mesh of Guide Vane, Hub and Blade

After meshing and specification of boundary conditions, the models are exported into Fluent-Mesh program. These files are imported into file case-mesh of Fluent program. The following steps had been followed:

- Set up the scale of model (Grid Mode).
- Set up and define mode.

- Viscous model
- Materials
- Boundary condition
- Grid interfaces (outlet bulb and inlet Guide Vane, outlet Guide Vane and inlet rotor, outlet rotor and inlet Draft tube)

• Simulate for the results

4. Results of simulation

Work is done by the fluid rotating the runner at 980 rpm. Static pressure distributions on runner blades are shown in Fig 2, 3 and 4, at different guide vane angles. The maximum and minimum pressures on the blades are shown in Table 1 and 2.

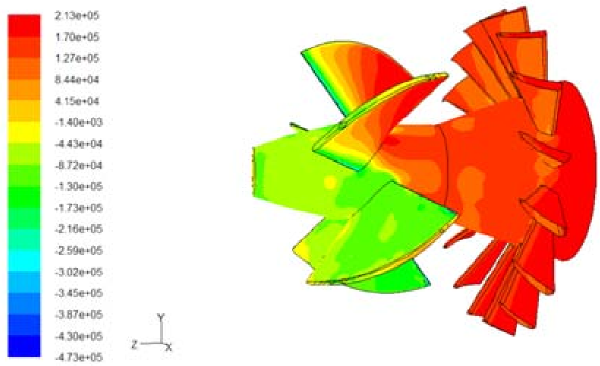

Fig 2. The pressure distribution on runner blades at guide vane angle of 60^{o}

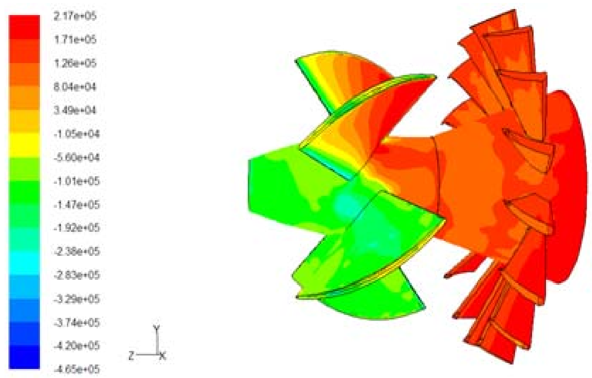

Fig 3. The pressure distribution on runner blades at guide vane angle of 65^{o}

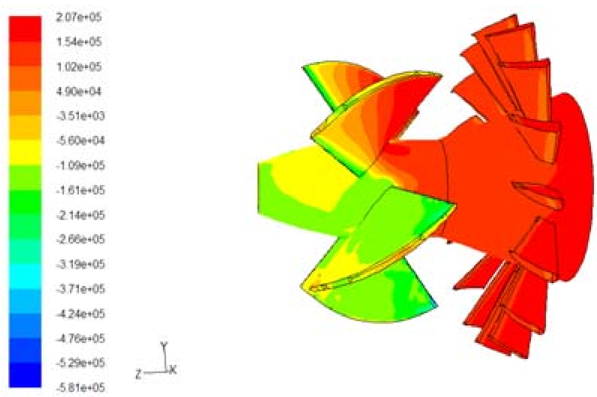

Fig 4. The pressure distribution on runner blades at guide vane angle of 70^{o}

Table 1 Maximum static pressure on blade angle at 32^o

Guide Vane angle	Static Pressure (kPa)
60^o	213
65^o	217
70^o	207

Table 2 Minimum static pressure on blade angle at 32^o

Guide Vane angle	Static Pressure (kPa)
60^o	-473
65^o	-465
70^o	-581

The static pressure and velocity distribution on blade angle at 32^o with guide vane angle at 70^o of pressure side are shown in Fig 5 and 6 respectively; the maximum pressure is located at the leading edge of the blade.

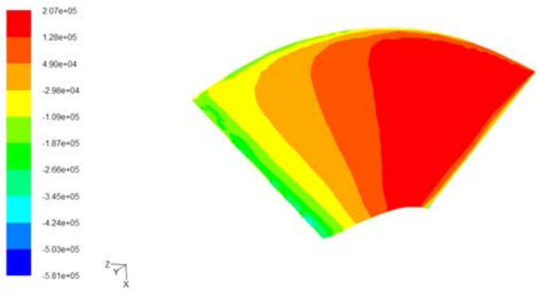

Fig 5. The pressure distribution on blades of Pressure side

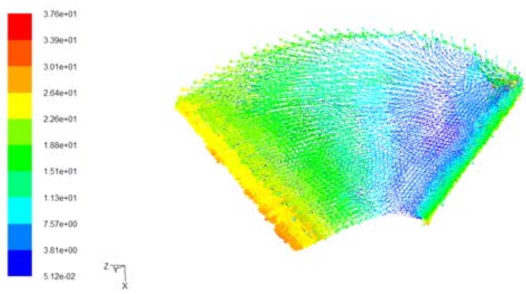

Fig 6. The velocity distribution on blades of Pressure side

The static pressure and velocity distribution on blade angle at 32^o with guide vane angle at 70^o of Suction side are shown in Fig 7 and 8 respectively; the maximum pressure is located at the tailing edge of the blade.

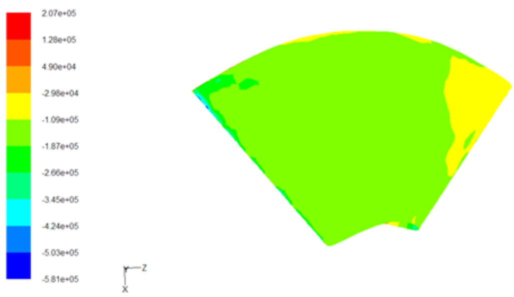

Fig 7. The pressure distribution on blades of Suction side

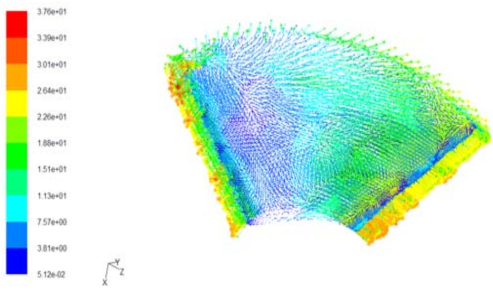

Fig 8. The velocity distribution on blades of Suction side

As shown in Fig 9, the pressure flow pass blade angle at 32^{o}, with guide vane angle at 70^{o}, the pressure difference before and after entering the runner causes the blade to rotate.

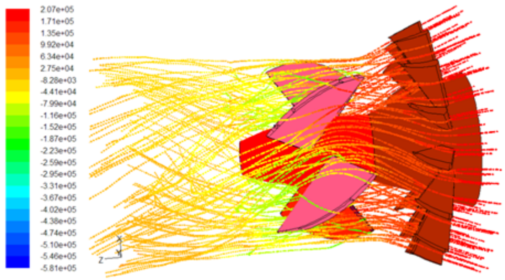

Fig 9. Pressure flow pass blades runner

Fig 10 shows lines of velocity flow pass blade angle at 32^{o}, with guide vane angle at 70^{o}.

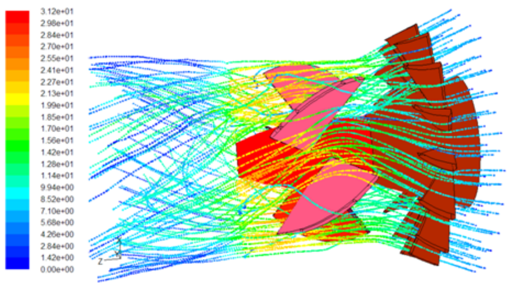

Fig 10. Lines of velocity flow pass blades runner

Fig 11 shows the streamlines of fluid flow pass blade angle at 32^{o}, with guide vane angle at 70^{o}.

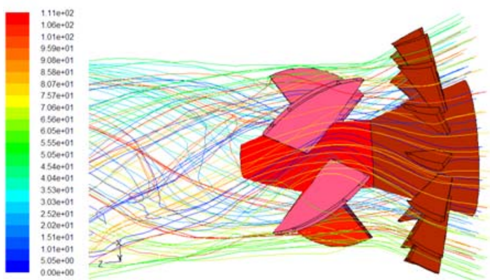

Fig 11. Streamlines of fluid flow pass blades runner

5. Conclusion

The CFD simulation results in this paper show pressure values on runner blades at specific head and fluid flow. The difference in maximum pressure on blade is from the change of flow area and fluid velocity due to the change in blade angle at 32^o and guide vane at 60^o, 65^o and 70^o respectively. The LES model of turbulence flow, under the practical condition of unsteady and incompressible fluid flow, was conducted in order to study the effects of blade angles on hydro turbine installed at Huai Kum Dam drainage pipeline. At the average head of 21 m, blade twist angle of 25^o and the blade camber angle of 32^o, the simulation was investigate on varying guide vane angle at different angle of 60^o, 65^o and 70^o respectively for comparing the maximum and minimum pressure on blades. The simulation showed that at guide vane angles of 60^o, 65^o and 70^o, the maximum pressures at leading pressure side are 213 kPa, 217 kPa and 207 kPa and the minimum pressures at leading suction side are -473 kPa, -465 kPa and -581 kPa respectively. By adjusting the guide vane angle, it clearly affects the pressure distribution and the efficiency of the hydro turbine. This case study serves as the guideline for blade design of hydro bulb turbine for the improvement of turbine efficiency.

References

[1] Taechajedcadarungsri, S. (2008). *Fluid Machinery*. Khon Kaen University.

[2] Sayers, A. T. (1990). *Hydraulic and compressible flow turbomachines*. New York: McGraw-Hill.

[3] Niyomvat, S. and Niyomvat, B. (2006). *Fluid Machinery*. June Publishing Co.

[4] Versteeg, H.K. and Malalasekera, W. (1995). *An introduction to computational fluid dynamics: The finite volume method.* New York: Longman Group.

[5] FLUENT 6.0 User's Guide. (2001). Chapter 10: *Turbulence Model*. New Hampshire: Fluent, Incorporated.