Improving Low Head Bulb Turbine Design Through Simulation Technique

Noppong Sritrakul[1,*], Udomkiat Nontakeaw[1] and Yodchai Tiaple[2]

[1] Department of Mechanical and Aerospace Engineering, King Mongkut's University of Technology North Bangkok, 1518 Pracharat Sai 1 Road, Bang Sue, Bangkok 10800

[2] Naval Architecture and Marine Engineering, International Maritime College, Kasetsart University Si Racha Campus, 199 Moo 6, Sukhumvit Road, Tung Sukla, Si Racha, Chon Buri, 20230

*Corresponding Author: n_pp_ng@yahoo.com

Abstract

Design for better performance and efficiency of low head bulb turbine requires knowledge and understanding of the mechanism of flow that occurs within the turbine. In the past, Construction of bulb turbine started from the design and builds a hydro turbine model to test in laboratory. The analysis test result was obtained to update model. And then test it again. Repeat the process until the desired performance was satisfactory. After that, scale up the model to create a prototype turbine. Such methods were costly and time-consuming. Nowadays, the advance in computer science and fluid dynamics simulation techniques that can calculate faster and more accurately is used to simulate fluid flow behavior within turbine. The application of computational fluid dynamics (CFD) technique leading to speed up the design procedure and reduce the cost and time. However, still need to build and test the turbine model in turbine test rig to verify simulation result before being scaled up, and then the prototype turbine was created. This study represents the simulation result and compares with the experimental result from turbine test rig in MAE's Laboratory. The turbine test rig was designed to investigate the performance of low head bulb turbine with 330 mm. runner diameter. This study will be beneficial to develop a simulation technique that gives correction and accurate result in order to reduce both difficulty and cost of low head bulb turbine design.

Keywords: small hydro turbine, low head, bulb turbine, computational fluid dynamics, turbine test rig.

1. Introduction

Hydroelectric power generation, the use of flowing water to generate electricity, is currently the largest source of renewable energy as measured by electricity generation. Hydropower, the clean energy, still has lower cost per unit of energy generating than many kinds of renewable energy. Nowadays, the appropriated development of hydropower should focus to small size of water resources as there are lesser impacts on environment and less social opposition than big scale of the project. Most of existing small hydropower resources in Thailand is low water head level resources, which are called "low head" hydropower resources. Thus, the appropriated type of low head hydro turbine is the small axial turbine known as "Bulb turbine". Bulb turbine is properly used for low to moderate water head with high flow rate condition.

Construction of bulb turbine started from the design and builds a hydro turbine model to test in laboratory. The analysis test result was obtained to update model. Then test it again. Repeat the process until the desired performance was satisfactory. After that, scale up the model to create a prototype turbine. Such methods were costly and time-consuming. Nowadays, the advance in computer science and fluid dynamics simulation techniques that can calculate faster and more accurately is used to simulate fluid flow behavior within turbine. The application of computational fluid dynamics (CFD) technique leading to speed up the design procedure and reduce the cost and time. However, still need to build and test the turbine model in turbine test rig to verify simulation result before being scaled up, and then the prototype turbine was created.

This research work was aimed at using CFD to simulate flow in the bulb turbine and compared the result with the experimental result from turbine test rig in MAE's Laboratory. The obtained knowledge will be beneficial for the further design of the low head bulb turbine.

2. Background Concept

In a past decade, many researchers applied CFD for checking of flow pattern in several kind of hydro turbine, including the bulb turbine [1]. In ordinary simulation scheme, the shape of turbine which is drawn in CAD must be imported into commercial code of CFD (such as ANSYS FLUENT), then grids will be generated in order to create many cells with tiny volume for calculation. The equation of motion which is derived from the principle of conservation of mass and momentum will be solved by finite difference method or finite volume method. In real situation, the turbine works by some parts such as blade, hub, shaft, etc. are rotated with angular velocity and some parts such as shroud, guide vane, bulb surface, duct wall, etc. are stationary part. Thus, to achieve accuracy, flow of fluid has to analyze by multiple reference frames (MRF) capability.

2.1 Mathematical Models

To describe the turbulent phenomenon, the modification of Navier-Stokes equations (NSEs) was established by Reynolds averaging method, generally used to transform NSEs [1],[2].

By taking time average over the characteristic time of mean values, the instantaneous quantities of direct numerical simulation (DNS) equations were replaced by mean quantities and the additional unknown terms, describing the turbulence, were introduced.

2.1.1 Governing Equation

The modified NSEs, called Reynolds-Averaged Navier-Stokes (RANS) equations, can be written in the tensor form as

$$\frac{\partial \rho}{\partial t}+\frac{\partial(\rho \cdot u_i)}{\partial x_i}=0 \tag{1}$$

$$\frac{\partial(\rho \cdot u_i)}{\partial t}+\frac{\partial(\rho \cdot u_i \cdot u_j)}{\partial x_j} = -\frac{\partial p}{\partial x_i}+\frac{\partial}{\partial x_j}\left[\mu\cdot\left(\frac{\partial u_i}{\partial u_j}+\frac{\partial u_j}{\partial u_i}-\frac{2}{3}\cdot\delta_{ij}\cdot\frac{\partial u_l}{\partial x_l}\right)\right]+\tau_{ij} \tag{2}$$

The additional unknown terms were named Reynolds Stress and defined as

$$\tau_{ij}=\frac{\partial\left(\rho\cdot\overline{u_i'\cdot u_j'}\right)}{\partial x_j}$$

2.1.2 Boussinesq Hypothesis

The common fashion, used to model Reynolds Stress term, employs Boussinesq hypothesis, relating the Reynolds Stress with the velocity gradients:

$$\tau_{ij} = \mu_t \cdot \left(\frac{\partial u_i}{\partial x_j} + \frac{\partial u_j}{\partial x_i} \right) - \frac{2}{3} \cdot \left(\rho \cdot k + \mu_t \cdot \frac{\partial u_i}{\partial x_i} \right) \cdot \delta_{ij} \qquad (3)$$

2.1.3 Modeling turbulent eddy viscosity

For the calculation of turbulence flow, the effect of turbulent eddy motions was described and incorporated into the turbulent viscosity term (μ_t). In contrast to the molecular viscosity, the eddy viscosity depends strongly on the flow property. Therefore, selecting the turbulent model, accommodated the flow behavior of each application, is very important. The SST $k-\omega$ turbulent models for eddy viscosity were briefly described as follows.

2.1.4 SST *k-ω* Turbulent Model

The eddy viscosity for *k-ω* turbulent model is related as follows

$$\mu_t = \rho \cdot \frac{k}{\omega} \cdot \frac{1}{\max\left[\frac{1}{\alpha^*}, \frac{\Omega \cdot F_2}{a_1 \cdot \omega}\right]} \qquad (4)$$

The Ω presents mean rate of rotation and F_2 is the blending function. α^* is given by

$$\alpha^* = \alpha^* \cdot \left(\frac{\alpha_0^* + \mathrm{Re}_t / R_k}{1 + \mathrm{Re}_t / R_k} \right) \qquad (5)$$

where

$$\mathrm{Re}_t = \frac{\rho \cdot k}{\mu \cdot \omega}, \; R_k = 6, \; \alpha_0^* = \frac{\beta_t}{3} \text{ and } \beta_t = 0.072$$

The equations used to model the complex flow behaviors of the turbulent flow were k and ϖ equations and expressed as follows.

$$\frac{D(\rho \cdot k)}{Dt} = \frac{\partial}{\partial x_j}\left(\Gamma_k \cdot \frac{\partial k}{\partial x_j} \right) + G_k - Y_k + S_k \qquad (6)$$

and

$$\frac{D(\rho \cdot \omega)}{Dt} = \frac{\partial}{\partial x_j}\left(\Gamma_\omega \cdot \frac{\partial \omega}{\partial x_j} \right) + G_\omega - Y_\omega + D_\omega + S_\omega \qquad (7)$$

G_k and G_ω represent the production of k and ω, respectively, while Y_k and Y_ω present the dissipation of k and ω. Γ_k and Γ_ω are the effective diffusivity for k and ω.

2.2 Turbine Configurations and CFD Model

The commercial CFD code, ANSYS FLUENT was used to solve problem in steady state flow through 4 blades bulb turbine with 330 mm. runner diameters, 15 guide vanes (guide vane angle was 45 degree relative to turbine shaft axis line) as shown in Fig. 1. The hub and tip clearance of runner are ignore in this simulation.

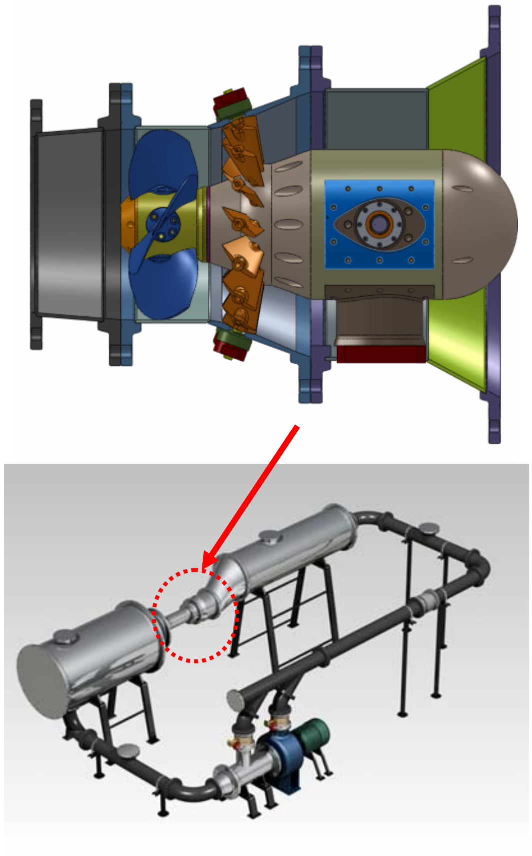

Fig. 1 Bulb turbine and turbine test-rig

The time-independent incompressible Navier-Stokes equations [3] [4] and the SST *k-ω* turbulent models was discretized using the finite volume method. QUICK and central differencing flow numerical schemes were applied for convective and diffusive terms, respectively. The discrete nonlinear equations were implemented implicitly. To evaluate the pressure field, the pressure-velocity coupling algorithm SIMPLE (Semi Implicit Method for Pressure-Linked Equations) was selected. The linearized equations were solved using multi grid method. Due to the geometrical complexity of the turbine and guide vane, the numerically approximated equations were performed on the collocated tetrahedral grid. The grid domain of guide vane, runner and outlet regime were constructed separately, comprising approximate total of 2,000,000 cells (approx. 700,000 cells of runner domain), shown in Fig. 2. Generating mesh at the interface between the domains is non-conformal.

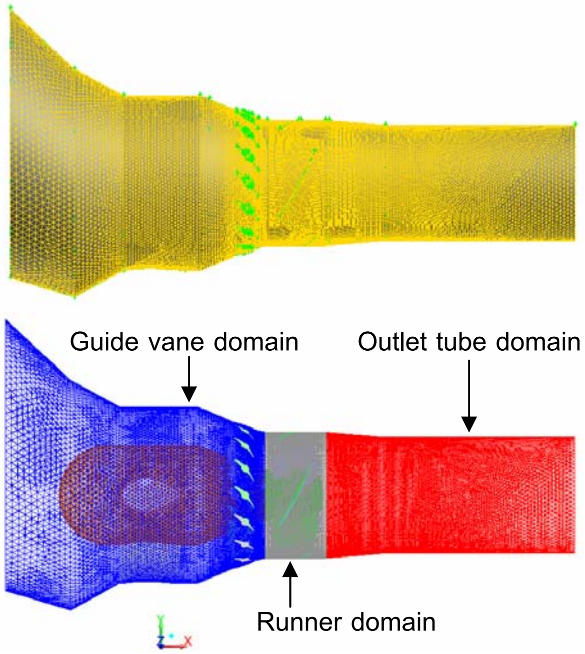

Fig. 2 Grid distribution of computational domain

This simulation was using mixing plane model. The mass flow rate (kg/s) condition was set for upstream inlet and the pressure outlet condition was set for outlet tube boundary. The boundary condition on mixing plane area was set to be the outlet guide vane which was the inlet runner. And outlet runner was the inlet of outlet tube. This simulation is set to be steady flow and SST k-ω was selected for turbulence model. The mass flow rate condition was 350 400 450 and 500 m^3/h on the inlet boundary.

3 Simulation Results

The solution was converged when the residual value less than $1x10^{-4}$. The static pressure distribution on Z-plane is revealed in Fig.3. It could be noticed at z=0, pressure distribution near hub of turbine is lowest. In Fig. 4, pressure contour on turbine blade is shown. It is noted that pressure value near tip leading edge is distributed quiet high, peak value is on pressure side and lowest value is on suction side. Cavitations possibly occurred on that suction side due to there was a lot of negative pressure.

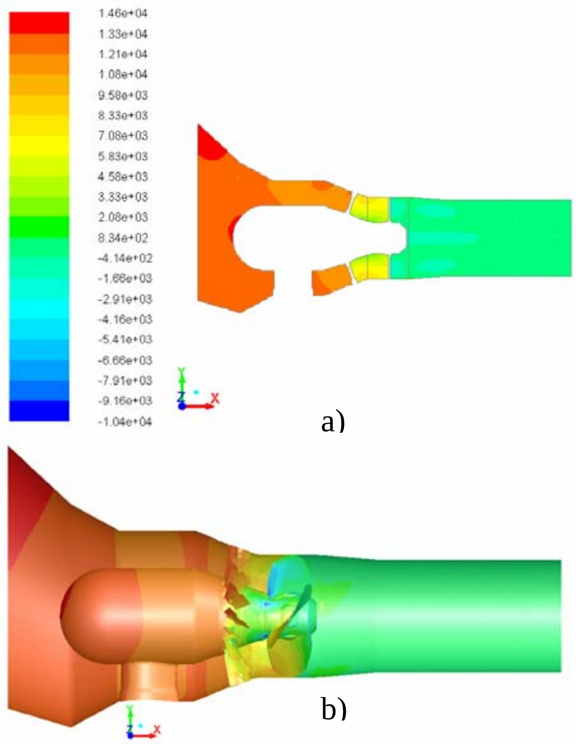

Fig. 3 The static pressure distribution

a) on Z-plane, b) on wall surface

(GV 45°, Flow rate 450 m^3/h, speed 250 rpm)

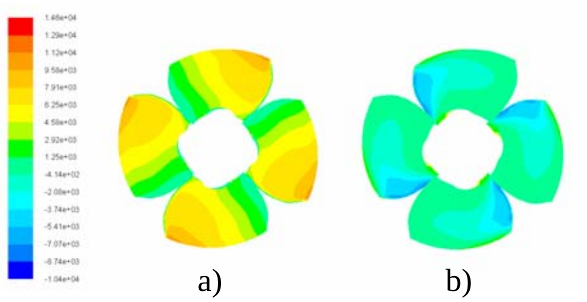

Fig. 4 pressure contour on turbine blade

a) Pressure side, b) Suction side

(GV 45°, Flow rate 450 m^3/h, speed 250 rpm)

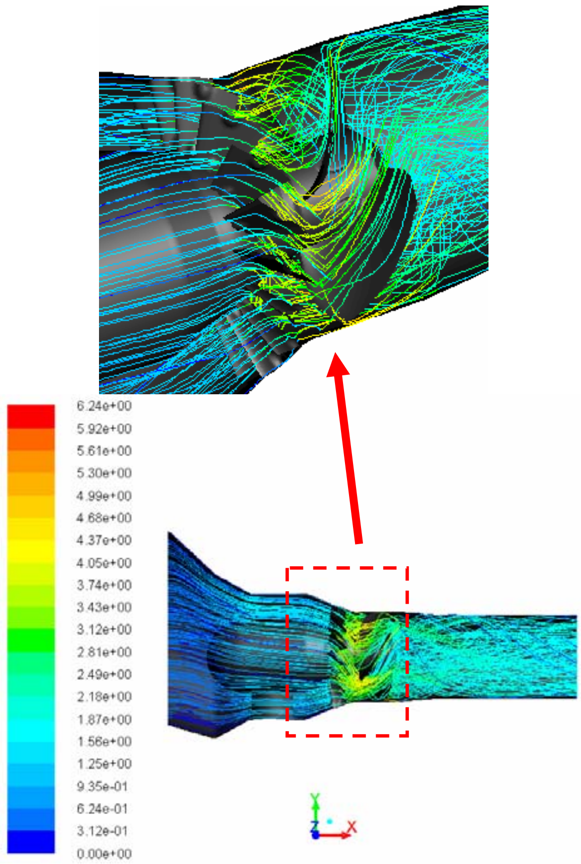

Fig. 5 pressure velocity field within the turbine

(GV 45°, Flow rate 450 m^3/h, speed 250 rpm)

The velocity field within the turbine was shown in Fig. 5. The velocity magnitude increased when passes through the guide vanes.

Turbine efficiency of varies flow (350 400 450 and 500 m^3/h) and speed were calculated and plot in Fig. 6.

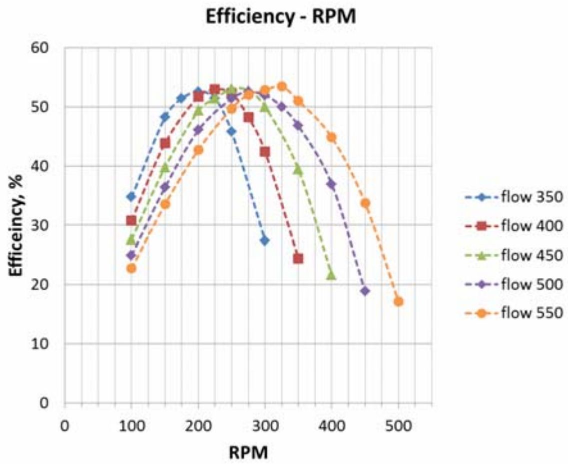

Fig. 6 Turbine efficiency result from simulation

(Guide vane 45 degree)

4. Experimental

4.1 Turbine test rig

The experiments were performed on a turbine test rig in MAE's Laboratory of KMUTNB as shown in Fig. 6. The test rig is a closed loop system design for Cavitation and Flow Velocity Investigation which consists of a small bulb turbine with 330 mm. runner diameter.

The model turbine was installed between high pressure upstream tank and downstream tank as shown in Fig. 1. The circulated water was provided by a centrifugal pump which was driven by a 1440 rpm, 100 HP, 3 phase motor. The volume flow rate was controlled by inverter unit. Flow measuring unit consists of an electro-magnetic flow meter and the other measurements consist of pressure transducers, thermometer and dynamometer.

Fig. 6 Turbine test rig in MAE's Laboratory of KMUTNB

4.2 Experimental Results

In the test, the guide vane angle set to 45 degree. First, Operate pump to the desired flow rate of 350, 400, 450 and 500 m^3/h. Then gradual apply break at dynamometer and maintain the flow rate by adjust the inverter. Record data of runner speed, upstream pressure, downstream pressure, and torque. The experimental result is shown in Fig. 7

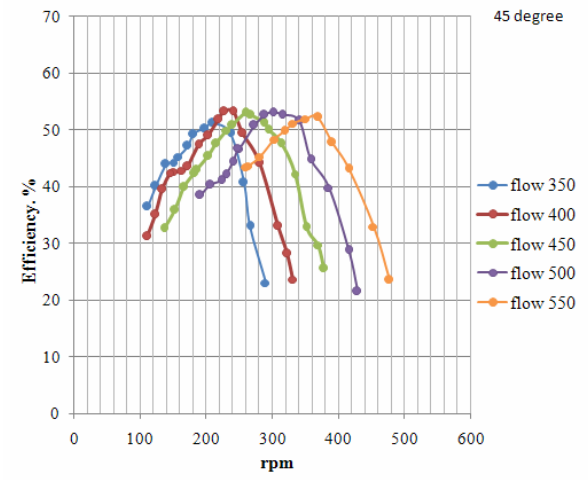

Fig. 7 Experimental results

4.3 Comparison with simulation results

The experimental results and simulation results were compared as shown in Fig. 8.

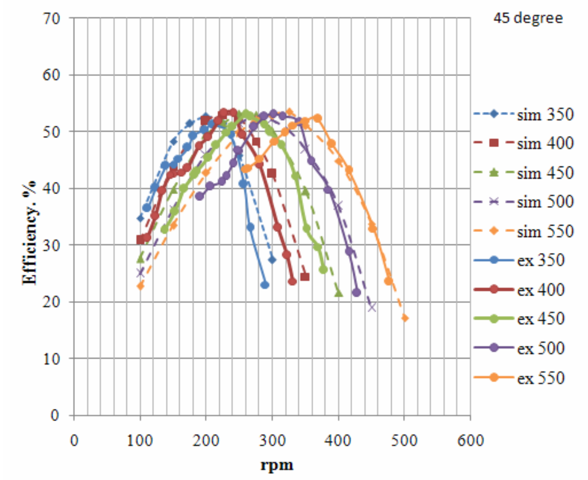

Fig. 8 Comparison results

The Comparison results in Fig. 8 showed the same trend of efficiency curves. The turbine efficiency results of the simulation were similar to the experimental results. The maximum efficiency point of each flow rate is slightly different speeds less than 30 rpm as shown in Table 1. There is a minimum different speed at the flow rate 400 m^3/h.

Table. 1 Maximum efficiency results

Flow (m^3/h)	Head (cm)		Speed (rpm)		Efficiency (%)	
	Exper.	CFD	Exper.	CFD	Exper.	CFD
350	69	80	210	200	38.23	52.56
400	97	104	227	225	39.80	52.95
450	117	134	259	250	39.61	53.07
500	136	167	301	275	39.66	52.52
550	163	189	350	325	38.80	53.45

The simulation results were consistent with the experimental results and had the same trend of turbine performance curves. The good agreement was accomplished by flow rate of 400 and 450 m^3/h.

5. Conclusion

The expected outcome from this research is self-know-how for supporting the design of the low head bulb turbine that is appropriate for working condition of water resources in Thailand. The CFD simulation can help design a low head bulb turbine. Using CFD to simulate fluid flow that occurs within the bulb turbine is the one of economic alternative. Simulation can save cost and time of design. Moreover, it is helpful and easily to modify turbine configuration and operated condition. Nowadays, an additional benefit of simulation technique is the simulation results can be transfer to calculate the strength in finite element procedure.

This research is the preliminary study for very low head turbine. The efficiency of model turbine is unsatisfied. The further study is planned to study cavitations effect and design optimized turbine blade shape for improved turbine performance. The author hopes that this research will be beneficial to the development of hydropower technology in Thailand.

6. Acknowledgement

The author would like to thank Department of Mechanical and Aerospace Engineering, King Mongkut's University of Technology North Bangkok, Department of Mechanical Engineering, Silpakorn University, Small hydro turbine research group of EGAT, and my work colleagues for all helpful suggestion and valuable assistance throughout the entire research.

7. References

[1] Anderson, J. D., et al. (1992). Introduction to computational fluid dynamics, Edited by Wendt, John F, Spriger-Verlag, New York

[2] Chung, T. J. (2002). Computational fluid dynamics,Cambridge, Cambridge University press.

[3] Tiaple Y. and Nontakaew U. (2005). The development of bulb turbine for low head storage using CFD simulation, *Asian Journal on Energy and Enviroment*, vol.6(03), 2005, pp. 186 – 192.

[4] Tiaple, Y., et al, 2005. Testing and numerical simulation of small hydro power, paper presented in ME-NETT National Conference 21th, Chonburi, Thailand.