As. J. Energy Env. **2005**, 6(03), 186-192

Asian Journal on Energy and Environment

ISSN 1513-4121

Available online at www.asian-energy-journal.info

The Development of Bulb Turbine for Low Head Storage Using CFD Simulation

Yodchai Tiaple[1,*] and Udomkiat Nontakaew[2]

[1]Naval architecture and Marine Engineering, Kasetsart University Si Racha Campus, Thailand

[2]Department of Mechanical Engineering, King Mongkut's Institute of Technology North Bangkok, Bangkok, Thailand

*Author to whom correspondence should be addressed, email: yodchai_tp@yahoo.com

Abstract: Since the low head turbines are designed individually according to the local situation. That is required huge engineering efforts, times and money. In order to handle all requirements, the technical-economic analysis is unavoidable. Computational Fluid Dynamic (CFD) simulation can be a complement to model testing and help us to speed up the design procedure. The commercial software Fluent® is selected for determining flow pattern which is the result of flow path and blade geometry. The main objective of this study is to design the hydro turbine to suit for the specific site of Lower Mae Ping dam in Tak province. The design concerned several constrains i.e., the existing civil structure of the dam, the flow regulation for irrigation and the limitation of water level that can effect to the efficiency of hydro power plant at upper dam. The optimizations for all purposes are considered. These design procedures can be applied to most of hydro power resources in Thailand.

Keywords: Hydropower, Low Head, Bulb Turbine, CFD, Finite Volume

Introduction

Hydropower is the indigenous energy resource of our country. However, nowadays the large hydropower project is extremely difficult to develop as Thailand has been already resisted by those who are likely to be affected by planned projects. In the other hand the electricity consumption is increasing rapidly in every year as follow to the economic growth of the country. Thus, the necessity to find good reliability of power resources for electricity generation to supply consumer's demand adequately has so many constrains to overcome. Comprehensively, the Energy Ministry is promoting portfolio standard of renewable energy such as wind power, solar power, biomass power agriculture waste power, small hydro power, etc.The Renewable Portfolio Standard (RPS) is expected that it will be reached to 8% in the year 2011. With comparison of total cost among that renewable energy, the small hydropower project is still can be cheaper investment. However, by the reason that the imported small hydropower machine is very high cost,so it might not worth to invest in the import small hydropower technology. The answer to the above, it becomes seeking for self dependency on small hydro machines for new small hydro site and/or to increase the hydropower at existing site by small hydro machines.

Thus, the Lower Mae Ping Dam (see Fig. 1) was selected to be studied site as priority.General information, Lower Mae Ping Dam was basically proposed to be pump storage of hydro-generator unit eighth of Bhumibol Dam. In every year the quantity of water that pass through Lower Mae Ping dam is around 150 – 500 m^3/s with average head at 2.4 m.

Study Objectives

The objectives of this study are design and manufacturing small bulb turbine which is appropriate to use at very low head of hydropower resources. The execution of this research is taken under the collaboration between Electricity Generating Authority of Thailand (EGAT), the Joint Graduated School of Energy and Environment (JGSEE) and Research Centre for Advanced Computational Engineering (RACE). The set of turbine consists of 15 pieces of guide vane and a 4 propeller runner blade with diameter size 650 mm. The design criteria are upon basic data of existing hydraulic condition. The efficiency and flow pattern was studied by finite volume simulation in computer by using computational fluid dynamics (CFD) software.

Fig. 1 Lower Mae Ping dam.

Background Concept

Preliminary design

The study and development of very low head turbine, head lower than 3 m, is a very difficult work for engineer in formal time. It might cause a very expensive study and gain worst output. In present with the new engineering tools, every limitation can be overcome. The very low head turbine is possible and easy to design with small budget within shorter time. Thus the authors are sincerely confident that the development of low head turbine which is suitable for using inany region of country can be success.The first step of design, hydraulic data of Lower Mae Ping Dam was studied. The design has to minimize the impact that might decrease efficiency of Bhumibol Dam's hydro generators. Generally hydro machines of Bhumibol Dam can operate with normal efficiency when the head of water at Lower Mae Ping storage is ranging between 135 – 137 msl. If the head of water higher than that range, the efficiency of large dam turbine will be drop. Mostly the downstream head of Lower Mae Ping Dam is at 133 msl. Another limitation of design was come from dimension of the upstream stoplog of Lower Mae Ping Dam because the set of designed turbine will be installed inside it. Under all limitation the outcome from designed turbine is capped at 50 kW.

Using CFD to simulate flow in the bulb turbine

The design and development of hydro turbine in the past several decades mostly experiment by using test rig in laboratory. The outputs can figure out the number of flow rate, angular speed, torque of turbine, etc. However, that experiment still has many constrains and limitation, its outcomes are not enough for researcher to achieve the best design. Thus the higher efficiency hydro turbines are come from the longer experience of researcher in hydro company. Right now the gap of design is fulfilled by simulation method of computational fluid dynamics.

The evolution of simulation was initiated before 1950 [1] in the same time of evolution of computer. At that period the finite difference method (FDM) and the finite element method (FEM) are famous numerical method in solving the set of partial differential equations in calculation of CFD. In later decades, the FDM and FEM were coupling together [2] to solve the set governing equation and that new calculation is called finite volume method (FVM) which is very favorable method of CFD in present.

In a past decade, many researchers applied CFD for checking of flow pattern in several kind of hydro turbine, including the bulb turbine [3]. In ordinary simulation scheme, the shape of turbine (such as shape of turbine which is drawn in SolidWork) must be imported into commercial code of CFD (such as 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.

The Cfd Model

The algorithm structure of commercial CFD software generally divided into 3 steps as follow:

Pre-processing

Which is the preparing process. The problem will be identified in detail of concern parameters such as dimension and computational domain of the problem.Then appropriate mesh will be calculated by divide computational domain into many tiny pieces of volume. The next step of this step is choosing the best numerical model which is suitable to nature of problem and then set up all boundary conditions.

Solving
this process is solving the set of governing equations by using numerical method. All differential equations will be discritized. Then the approximated solution from numerical solving will be obtained

Post-processing
the solution in form of graphic will be shown in this process such as velocity vector of flow pass through turbine blade.

In this study FLUENT® V6.0 was selected to be the calculation tool for solving conservation of mass and momentum equation in steady state flow through 4 blades bulb turbine model size 650 mm diameter with 15 guide vanes as shown in Fig. 2 at net head = 3.0 m and flow rate = 1.88 m^3/s. The hub and tip clearance of runner are ignore in this simulation.

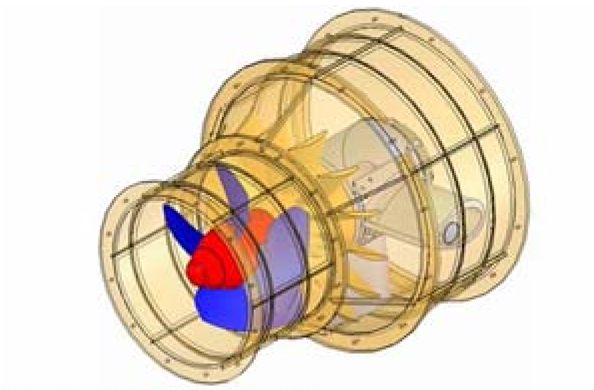

Fig. 2 Bulb turbine configuration.

In Fig. 3 hybrid grid of stator zone is shown, altogether hybrid grid of rotor zone is shown in Fig. 4. This simulation was using mixing plane model which was defined at the stator outlet and rotor inlet. The inlet condition of upstream boundary was set by using mass flow rate (kg/s) and/or velocity inlet (m/s) due to this study is incompressible flow problem and the outlet boundary condition was set by static pressure (N/m^2). The boundary condition in mixing plane area was set to be the outlet stator which was the inlet runner. This simulation is set to be steady flow study and k-ε was selected for turbulence model.

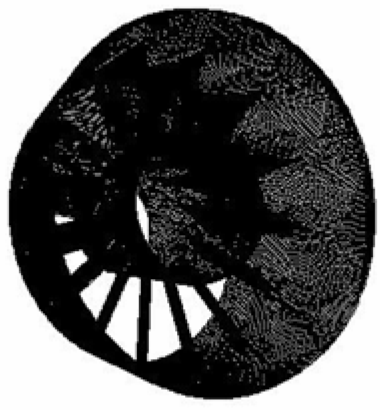

Fig. 3 Hybrid grid of stator zone.

Fig. 4 Hybrid grid of rotor zone.

Results

The simulation's output was set the residual value of continuity, x-velocity, y-velocity, z-velocity, k and ε at 0.001. The solution was converged at 450 iterations of calculation. The static pressure distribution along x-axis is revealed in Fig.5. It could be noticed at z=0, pressure distribution near hub of turbine is lowest. In Fig. 6, pressure contour on turbine blade is shown, the upper picture is the view of pressure side and the lower picture is the view of suction side. 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. It is possible that on that suction side will be occurred cavitations due to there is a lot of negative pressure, this outcome is in good agreement with phenomena of flow in Fig.7 which is shown separated flow on tip leading edge at suction side. The velocity magnitude of flow along x-axis since upstream position to downstream position is appeared in Fig 8. The highest speed of flow was found in front of blades, then the speed is slightly slow down after pass through runner blades.

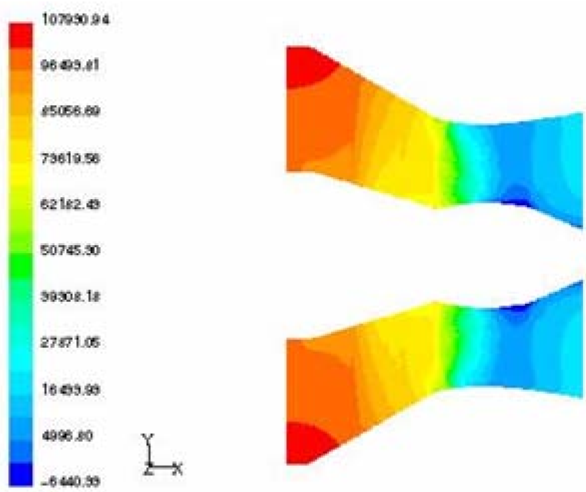

Fig. 5 Static pressure distribution along x-axis.

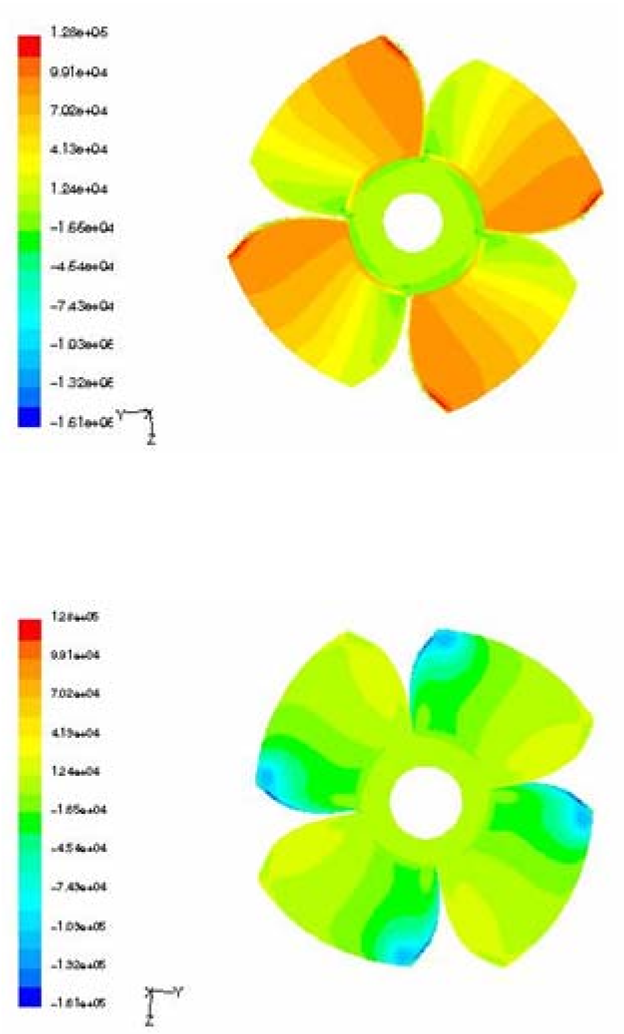

Fig. 6 Static pressure distribution on runner blade for pressure side (upper) and suction side (lower).

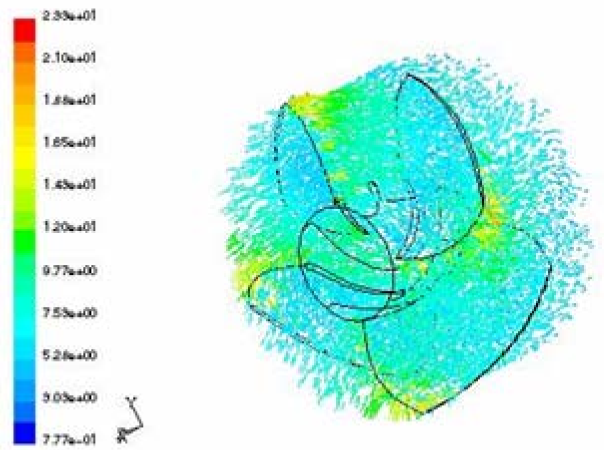

Fig. 7 Velocity vector pass through runner blade.

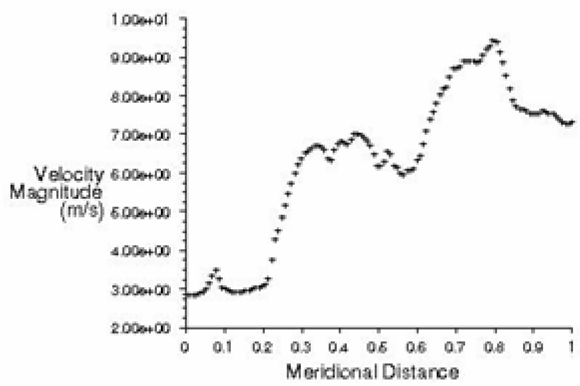

Fig. 8 Velocity magnitude along x-axis.

Conclusion

The study to design low head bulb turbine by using mathematical model of computational fluid dynamics is the one of economic alternative. This study can save cost and time of design. Altogether, it is easily to modify turbine shape and hydraulic condition which is the other methods cannot do. Thus the highest efficiency turbine for very low head condition is possible to do with low cost and short time. Another useful outcome is the simulation results can be used for co-analyze in finite element procedure.

This study is the preliminary study for very low head turbine. The efficiency of this design is approximated at $\eta = 7\omega/\rho gHQ = 82\%$ which is not high enough to satisfy. However, the further study is planned to design optimization based on numerical flow simulation in which to achieve the better efficiency. The authors sincerely hope that this study will be the big and important step of self-dependency of hydropower technology for Thailand.

Acknowledgements

The authors would like to express sincere thank all members of small hydro turbine research group. Financial support from Electricity Generating Authority of Thailand (EGAT) and the Joint Graduated School of Energy and Environment (JGSEE) is gratefully acknowledged.

References

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

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

[3] Gehrer, A., Egger, A. and Rainer, J. (2003) ANDRITZ Hydro power news, Issue 4.