Duct Design for Water Current Turbine Application

P. Khunthongjan and U.Teeboonma

Abstract— *The water turbine is the mechanical tool that changes the kinetic energy of water current to be useful with environmental influence slightly, since it's unnecessary to build a dam. Somehow, water current in rivers probably was not about to use as the energy resource because its velocity was quite slow. This research, so, studied and designed duct worked as Diffuser in order to be the augmentation factor and CFD Simulation for the water diffuser with Fluent commercial code program. The Field test resulted that they were consistent each other well and the designed diffuser was able to two times increase velocity of water.*

***Keywords*— Water duct, Diffuser, Duct, Augmentation factor, CFD Simulation.**

1. INTRODUCTION

The function of water turbine is similar to wind turbine that is it gets the kinetic energy of fluid and lower its velocity as said by Betz that if we want to maintain the energy capacity while the velocity is lower, we have to broaden space of the water duct (the limitation of kinetic energy usage from fluid is the specific energy per unit of fluid will be function of velocity^3). Anyway, if the velocity of fluid is lower, the more complicated of usages its energy. In order to maintain the power it's either need bigger size of turbine or add number of turbine or can be both. If we decide to increase size of the turbine the speed of blade tip may cause the Cavitations and also the location problem. Adding number of turbine was other alternative but it came together with high cost of the project in the inefficient way or the system would be more complicate if we tried to connect several sets of Impeller to the single shaft. Therefore, the possible way of kinetic energy usage from fluid with low velocity was to use Diffuser in order to widen Effective area of the duct; meanwhile, it's to speed up inlet water current. There are some diffuser application with windmill, for example**,** D.G. Phillips and others (1,2) from the University of Auckland, New Zealand studied and revised DAWT model called Vortec 7 that was built and designed by Foreman and Grumman. The External air jet was used to prevent fluid disperse and control the Boundary layer in Diffuser from building model by CFD method and Visualization test and found that the Power coefficient factor was four times increase. Toshio Matsushima et al.(3) studied the Frustum-shape Diffuser that was incentive capacity of small windmill from the model system by I-DEAS Frustum-shape Diffuser program and found Power coefficient factor was 1-7 times increase. Yuji Ohya et al.(4)studied and developed Diffuser-installed windmill. It's basically studied Nozzle type, Cylindrical type, and Diffuser type and found the latest one came with 1.8 times as the maximum speed. Not only the windmill, diffuser was applied with the water current turbine, David L.et al.(5) studied result of Diffuser of the energy from water turbine by the model system with the program of ANSYS CFX-5.7 Brian (6) installed 1.2*1.2 meters slot duct (2 sides Airfoil section slats) with the Cross flow turbine type. However, other additional details of diffuser were not mentioned. Karanja Kibicho and Anthony Sayers (7) studied the fluid flow in Wide angel diffuser since the disperse of pressure in Wide angel diffuser was irregular; one found it's flowing while another one not that called Stalling. It effected to overall flow in Diffuser and without efficiency. R. K. Sullerey et at. (8) had the comparative study the capacity between Straight and Curve and diffuser at Reynolds number $7.8x10^5$-$1.29x10^6$. The fluid flow in Diffuser depended on Divergence angle, length and width Throat and the vicissitudes of Inlet free stream.

This study aimed to design diffuser using Computational Fluid Dynamics (CFD) Fluent commercial code for horizontal axis water turbine application and also proposed the field test result at different circumstance.

2. MATERIAL AND METHOD

Water speed simulation in diffuser would use the program of Fluent commercial code. The simulation would regularly set the speed of fluid flow from the inlet boundary to the outlet (Fig.1) Domain was separated into quadrilateral cells total 30,194 cells as in Table 1.

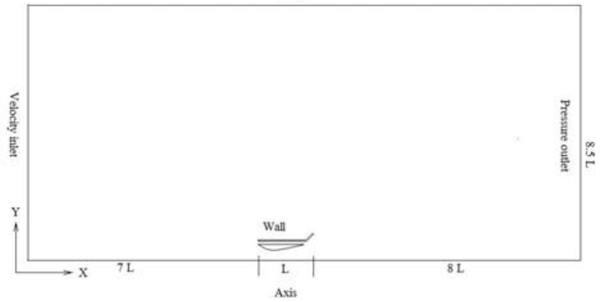

Fig.1 Boundary condition

P. Khunthongjan and U.Teeboonma are with Mechanical Engineering Department, Faculty of Engineering, Ubon Ratchathni University, Ubon Ratchathni Thailand. Email: palapum.k@egat.co.th.

Table. 1 Computational parameters

Solver	Axisymmetric, steady
Viscous model	K-epsilon RNG, Standard wall function
Turbulent kinetic energy k(m^2/s^2) Turbulent dissipation rate (m^2/s^3)	0.5 0.5
Pressure velocity coupling	simple
Under relaxation factor	0.3 for pressure, 0.7 for momentum
Discretization	
Pressure	standard
Convergence criteria	10^{-4}

Duct would be set with a rafting and was pulled with a small boat and test in a reservoir. The pressure of inlet boundary, nozzle, and external of Diffuser would be measured by Manometer principal, using 3 mm diameter transparent plastic pipe.

3. RESULT AND DISCUSSION

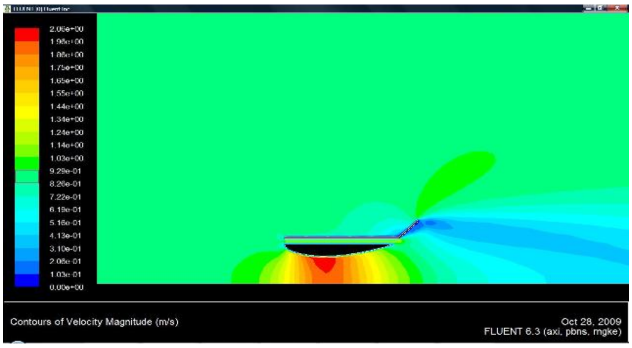

Fig.2 Velocity contour

Fig.2 showed Velocity contour of the simulation. Diffuser effected water speed if the inlet velocity was set at 0.9 m/s, the maximum velocity in diffuser was 1.8 (a double increase) that was happened around the nozzle and lower part, in particular, behind Flange and came about the whirlpool as in Fig.3.

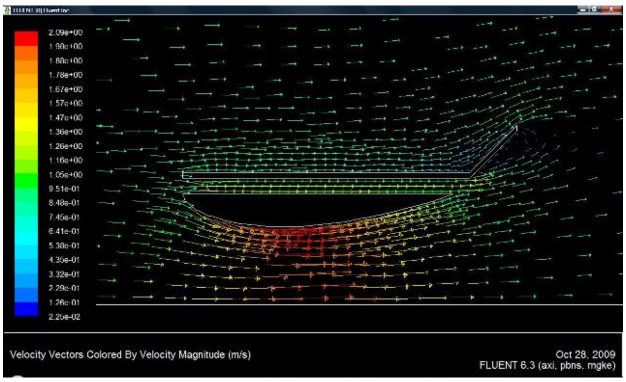

Fig.3 Velocity vector

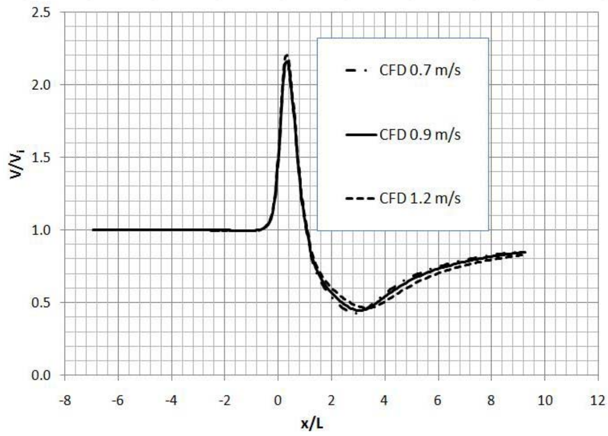

Fig.4 Velocity rate change

Fig.4 at the inlet velocity between 0.7 to1.2 found the velocity at in front of diffuser (x/L=-0.5) started to increase, at the entrance(x/L=0) velocity was 1.3 times and at the exit was 1.0 time of velocity at inlet boundary. At the nozzle of diffuser (x/L=0.3) maximum velocity increasing 2.2 times. Considering effect of velocity inlet to local velocity rate change, from inlet boundary(x/L=-7) to exit of diffuser, there were no effect. From exit (x/L=1) to pressure outlet boundary (x/L=9) inlet velocity was a little effect on the rate change of local velocity.

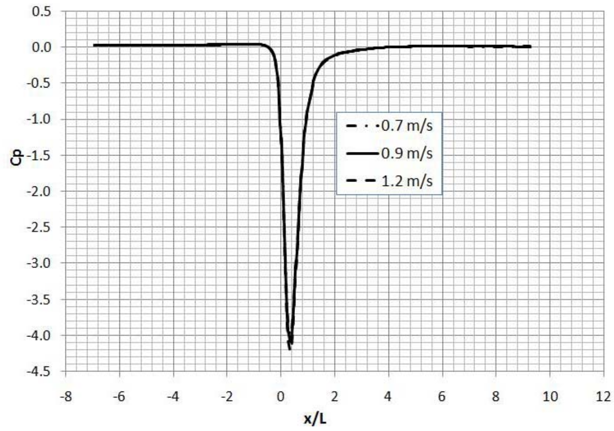

Fig.5 Pressure coefficient comparative

The fig.5 showed Pressure coefficient on the Diffuser axis. At the inlet (x/L=0) and outlet of Diffuser (x/L=1) the pressure coefficient was lower than zero, which the fluid flow through the Diffuser was higher and not differ even the inlet velocity change.

Fig.6 Velocity line graph from the water speed simulation. It shows three different speed experimental at three different points, external side, inlet and at the nozzle of the Diffuser. Bernoulli's equation was used to calculate the velocity. Experimental velocity at the inlet of diffuser (x/L=0) was faster than the external one (x/L=-7) same as simulation.

.

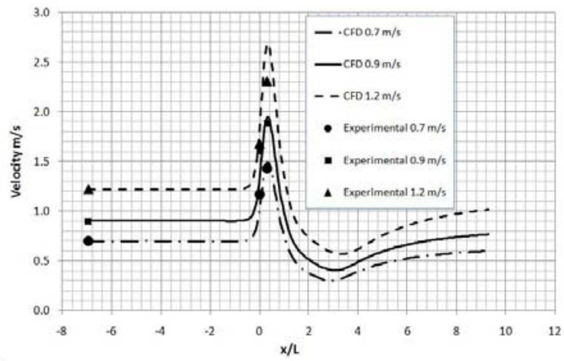

Fig.6 Simulation and Experimental results

4. CONCLUSIONS

1. The simulation result with Fluent commercial code program was consistent with Field test result, the diffuser designed doubled velocity of inlet water. Somehow, the case of the internal turbine installed in the Diffuser needs additional study.

2. The hypothesis of simulation was that the velocity was stable and regular. The test get done at deep water if compared to size of diffuser. If the diffuser designed was used or tested in different circumstance, such as shallow water, strong wind and wave etc., probably the results would be different.

3. The installment of duct with water turbine in order to increase velocity of fluid flow was another alternative of water turbine application, particularly in the area of low velocity of water e.g. rivers and canals in Thailand.

ACKNOWLEDGMENT

The authors would like to thank The EGAT (Electricity Generating Authority of Thailand) and Ubon ratchathani supported the fund.

REFERENCES

[1] Phillips, D.G., Richards, P.J., Flay, R.G.J., 2008, " Diffuser development for a diffuser augmented wind turbine using computational fluid dynamics", Department of Mechanical Engineering the University of Auckland, New Zealand.

[2] Phillips, D.G., Richards, P.J., Flay, R.G.J., 2002, "CFD modeling and the development of the diffuser augmented wind turbine", Wind and Structure, Vol.5, No.2-4, pp. 267-276.

[3] Toshio Matsushima, Shinya Takagi, Seiichi Muroyama, 2006, "Characteristics of a highly efficient propeller type small wind turbine with a diffuser", Renewable Energy, Vol. 31, pp. 1343–1354.

[4] Yuji Ohya, Takashi Karasudania, Akira Sakuraib, Ken-ichi Abeb, Masahiro Inouec, 2008, "Development of a shrouded wind turbine with a flanged diffuser", Journal of Wind Engineering and Industrial Aerodynamics, Vol. 96, pp. 524–539.

[5] David L. F. Gaden and Eric L. Bibeau, 2008, " Increasing Power Density of Kinetic Turbines for Cost-effective Distributed Power Generation", Department of Mechanical and Manufacturing Engineering, University of Manitoba, Canada.

[6] Brian Kirke, 2005, "Development in ducted water current turbines" available online at www. Cyberiad.net

[7] Karanja Kibicho and Anthony Sayers, 2008, "Experimental Measurements of the Mean Flow Field in Wide-Angled Diffusers: A Data Bank Contribution, Proceedings of world academy of science, engineering and technology, Vol.33, September 2008.

[8] Sullerey, R. K.,Chandra, B. and Muralidhar, V., 1983, "Performance Comparison of Straight and Curved Diffusers", Def Science Journal, Vol. 33, No 3, pp. 195-203.