Performance of an Open Ducted Type Very Low Head Cross-Flow Turbine

Zhenmu Chen, Van Thanh Tien Nguyen, Morihito Inagaki, and Young-Do Choi*

Abstract— *Cross Flow Turbine (CFT) known as a Banki turbine or an Ossberger turbine is usually used in the small hydropower, because of its economical and simple structure. This study develops a new kind of CFT suitable for very low head and remote rural region which turbine researched barely before. The new design of the turbine is with open duct inlet channel, without turbine guide vane and nozzle for a more simple structure. The open duct inlet channel can be also suitable in the remote rural region where there are some materials of sediment such as sand and pebble come with flow from upstream that can cause break down. However, the CFT with open duct inlet channel and low head show relatively low efficiency. Therefore, the purpose of this study is developing a new CFT and modifying the turbine inlet open duct bottom line (IODBL) location and angle to improve the performance. The internal flow is investigated to examine the influence of turbine shapes on the performance. The results show that an appropriate turbine IODBL location and angle play an important role on improving the turbine performance, and there is significantly efficiency improvement by optimizing turbine IODBL location.*

***Keywords*— Cross flow turbine, very low head, open duct, turbine open angle.**

1. INTRODUCTION

Cross Flow Turbine (CFT) also known as Banki turbine or Ossberger turbine is ideal turbine for the small hydropower, because of its economical and simple structure. This study develops a new kind of CFT with free flow inlet channel and low head, without guide vane and nozzle for a more simple structure. The free flow inlet channel can also be suitable for the remote rural region where there are some sediment such as sand and pebble that come from upstream and enter the turbine structure. However, the CFT with free flow inlet channel and low head show relatively low efficiency. Therefore, the purpose of this study is developing a new CFT and improving the performance of turbine.

In order to increase the efficiency of turbine, some present studies suggest a method of installing an internal deflector into the runner centre which can guide the water flow to a correct angle to improve the performance by CFD analysis and experiment. The authors investigate the influence of different shapes on the performance. The results show that the CFT efficiency could be improved by use of a well-designed internal deflector [1]-[6].

There is also new method to improve performance of the traditional cross flow turbine by supplying air into the chamber to suppress the negative torque where there is hydraulic loss [7], [8]. On the other hand, Choi et. al. [9] performed the experiment and CFD analysis to study the influence of nozzle shape, runner blade angle, and runner blade number on the turbine performance, and also examined the effect of air layer on the performance.

For the very low head CFT, the way of flow entry into the runner is very important. Therefore, the location of IODBL affects the performance of turbine significantly. In this study, a set of turbine IODBL locations are determined to investigate the performance and the internal flow of the very low head CFT.

2. METHODOLOGY

CFT Model

Figure 1 shows the schematic view of the new type very low head CFT model. The structure of the turbine consists of open ducted inlet water channel, runner and draft tube, but guide vane and nozzle, which are a simple structure for turbine. Turbine inlet channel is open duct, which means that there is a free flow in the turbine inlet channel. The draft tube plays a role of reducing the pressure of draft tube to suck the water into turbine chamber. The water flow from turbine inlet channel is separated into two flows: one part of the water flows into the draft tube through the runner passage, which converts the kinetic and pressure energy into the output power of runner, and the other overflows into the downstream with the some materials of sediment, preventing damage to the runner structure.

The turbine height is $H_{turbine}$ = 5m, which also is the draft tube length. The turbine head between the water level of upstream and downstream is H = 4.3m, which is very low in contrast to other typical cases of the hydro turbine.

The number of the blade is *Z = 26*. The diameter of the

Zhenmu Chen is with the Graduate School, Department of Mechanical Engineering, Mokpo National University, South Korea. E-mail: chenzhenmu@163.com

Van Thanh Tien Nguyen is with the Graduate School, Department of Mechanical Engineering, Mokpo National University, South Korea. E-mail:thanhtienck@gmail.com

Morihito Inagaki is with Tokyo Electric Power Company, Japan. E-mail: inachanzura@ybb.ne.jp

Young-Do Choi (corresponding author) is with Department of Mechanical Engineering, Institute of New and Renewable Energy Technology Research, Mokpo National University, South Korea. phone: +82-61-4502419; fax: +82-61452-6376; E-mail: ydchoi@mokpo.ac.kr

outer runner is D_1=372mm; the inner diameter is D_2= 250mm; $D_1/D_2 \approx 0.67$. The runner blade inlet angle and outlet angle are β_{b1}=33° and β_{b2}=83°, respectively.

Figure 2 shows the cases and parameters of different turbine IODBL location. To investigate the effect of the turbine IODBL location on the performance, 5 Cases of different turbine IODBL location are conducted. By changing the turbine IODBL location along with changing the turbine open angle, the turbine open angle is changed from 70° to 117°, and the case is named from Cases 1 to 5, respectively. These 5 cases are conducted at the same rotational speed.

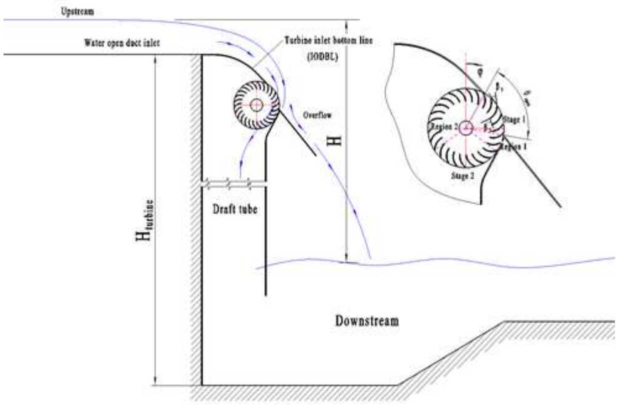

Fig.1. Schematic View of the CFT.

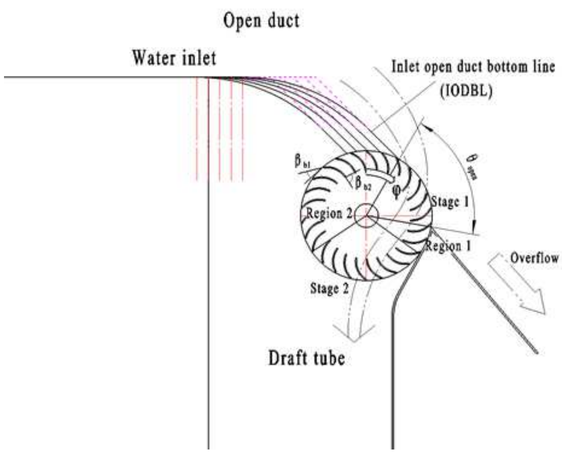

Fig.2. Variation of turbine IODBL location.

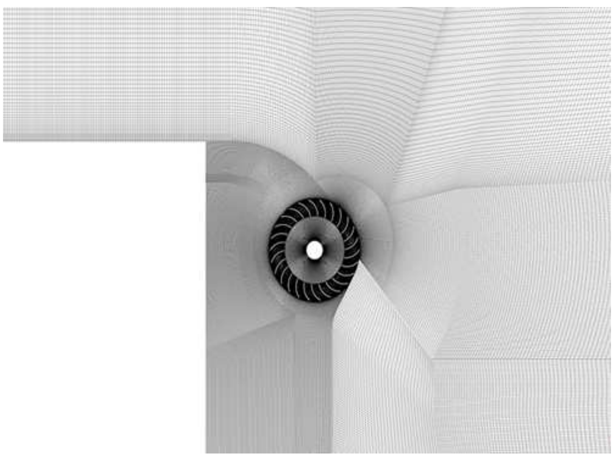

Fig.3. Final Hexahedral Numerical Mesh of Near Area of the Runner.

Numerical Methods

CFD analysis is a useful tool for predicting hydraulic machinery performance at various operating conditions. This study employs a commercial CFD code of ANSYS CFX to conduct CFD analysis. As the direction of gravity is considered, the transient analysis with two phase flow (water and air) is conducted. Two dimensional geometry and mesh are applied in order to decrease the calculation time because the flow in the turbine can be assumed to be uniform of main stream direction which means no flow velocity component in the direction of the thickness.

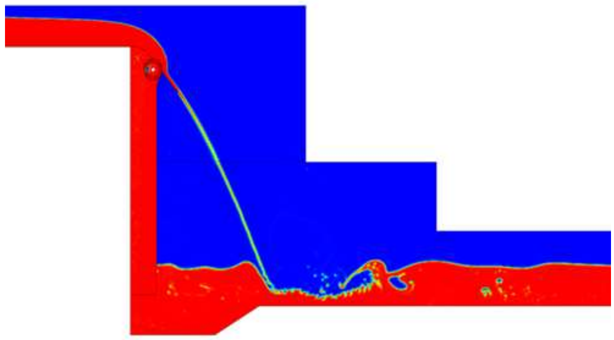

Fig.4. Water volume fraction of CFT by CFD analysis (Case 3)

Table 1. Cases of different turbine IODBL location

Cases	Open angle θ_{open} [°]
Case 1	70
Case 2	88
Case 3	100
Case 4	109
Case 5	117

The boundary condition for normal speed is set for the water flow at the inlet and outlet, and the velocity of 0 m/s is set for the air flow. The boundary condition of opening is set at the top of open duct and downstream domain.

The final hexahedral numerical mesh of near area of the runner and the water volume fraction of CFT are shown in Figure 3 and Figure 4.

Phase change, heat transfer and mass transfer effects are neglected. The circumferential location (θ) direction is clockwise starting from the top of the runner. For convenience, the area of runner passage is divided into four regions: Stages 1 and 2, Regions 1 and 2. Stage 1 obtains first output power and Stage 2 takes the second output power. However, Regions 1 and 2 consume output power by hydraulic loss.

3. RESULTS

Performance Curves of the Investigated Cross Flow Turbine

Figure 4 shows the water volume fraction of CFT in

Case 3 by CFD analysis**.** In this figure, the red part is water, and the blue part is air. Most water from upstream is sucked into the draft tube, and also some water overflows to downstream directly by the effect of gravity.

The draft tube is almost full with water. The upstream and the downstream is free flow and there is a free surface between the air and water. In order to examine the effect of IODBL location on the performance, the performance curves and internal flow are investigated. The efficiency is determined based on the potential energy difference between upstream and downstream by the following equation:

$$\eta = \frac{T\omega}{\rho g H Q} \tag{1}$$

where η is the CFT efficiency; T is the output torque; ω is the angular velocity of runner; H is the turbine head and also is the water level difference between upstream and downstream. Q is the water flow rate in through the draft tube.

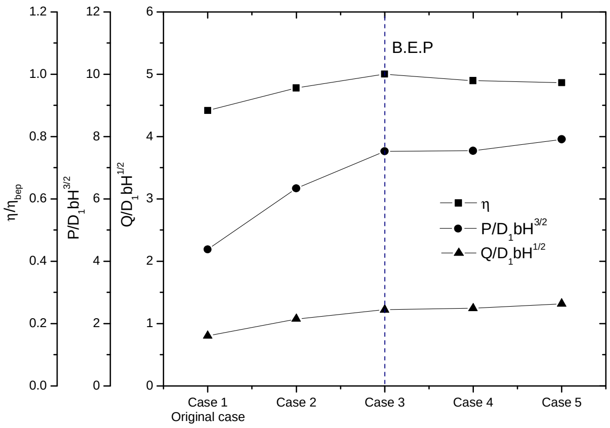

Fig. 5. Performance Curves of the Investigated Cross Flow Turbine.

Figure 5 shows the performance curves by the turbine IODBL location. All the data are normalized as shown in Fig. 5. Case 1 is the original case. From Cases 1 to 5, the output power and flow rate increase along with augmenting the turbine open angle θ_{open}. The present CFD analysis [10] result shows that efficiency drops at small and larger turbine open angle θ_{open}, and turbine has peak efficiency at the Case3. Larger turbine open angle θ_{open} is good for water flowing into the draft tube, and contributes to generating more output power for turbine. A suitable location of turbine IODBL at periphery of runner gives a significant effect on the performance improvement of this turbine. In order to examine the influence factor for the performance variation of the very low head cross flow turbine model, internal flow is investigated.

Velocity Vector and Pressure Contours

The tangential velocity in the blade flow passage is very important because it directly affects the angular momentum of the runner and the output power of the cross flow turbine. In this study, the velocity vector on the runner passage is investigated. Corresponding to the IODBL angle, Cases 1, 3 and 5 are chosen as typical cases to investigate the internal flow influenced by IODBL location on the periphery of runner. Figure 6 shows velocity vectors at Stage 1 and 2, and the pressure contours at runner passage by the three typical cases. From the figure of the velocity vectors, it can be observed that there is uniform flow at the Stage 1 of runner passage in Case 1, but there are large recirculating flow at Stage 1 in Case 3. Larger turbine open angle θ_{open} causes the hydraulic loss at Stage 1 by recirculating flow that occurs to a large extent. However, at Stage 2 of runner passage, the velocity vectors show uniform outflow in Case3, but the flow in the runner passage separate with the blade in Case 1, and the change in fluid flow quite drastically between the three cases. It is conjectured that this change at Stages 1 and 2 discussed aforementioned affects the turbine performance as seen by the performance curve.

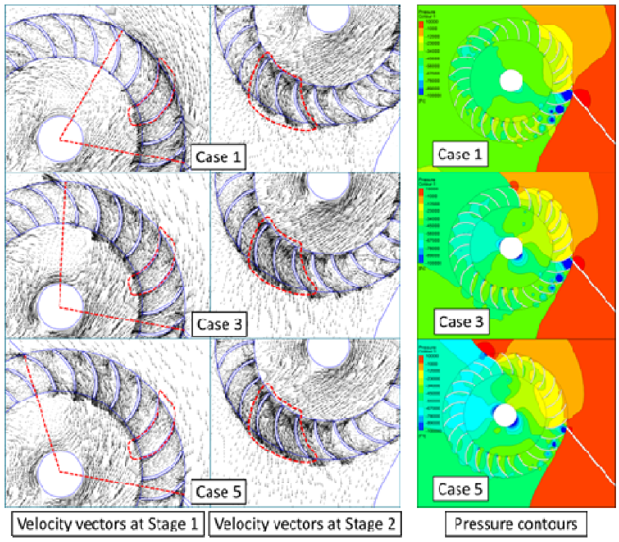

Fig. 6. Velocity Vectors at Stages 1 and 2, and Pressure Contours at Runner Passage.

From the pressure contours at runner passage shown in Figure 6, there exists a high pressure region on the pressure side of runner blades at the leading position of the flow direction. The high pressure on the blade pressure side means the power to the rotation of the runner. The more pressure difference between the blade pressure and suction side, the more output power is generated. The pressure distribution is relatively even at blade pressure and suction side at Stage 1 in Case 5, but there exits obviously pressure difference at blade pressure and suction side in Case 1. However, there is contrary trend of the pressure distribution at Stage 2. The effect of increasing the turbine open angle gives obviously rising pressure difference at Stage 2, but the effect at the Stage 1 is opposite.

Velocity Triangle and Pressure Contours

For turbo-machines, the water flow velocity triangle on the entrance and exit blade passage is a very important factor that affects the performance of the cross flow turbine. In order to investigate the effect of turbine IODBL on the internal flow and performance in detail, the velocity triangle on the runner periphery is measured. Figure 7 presents the flow velocity triangle at the entrance of Stage 1 and the exit of Stage 2, and the velocity triangle on a blade of the cross flow turbine. α is the absolute angle that is between absolute velocity (V) and the runner tip velocity (u). β is the relative angle that is between relative velocity (w) and the runner tip velocity (u). β_{b1} is the inlet angle of blade. The power delivered from the fluid is thus [11]-[13] This is the Euler turbo-machine equation, showing that power is functions of runner tip velocities ($u_{1,2}$) and the absolute water flow tangential velocities ($V_{u1,2}$). This equation is assumed by neglecting the energy loss at runner center passage between exit of Stage 1 and entrance of Stage 2.

$$p_w = \rho Q(u_1 V_{u1} - u_2 V_{u2}) \qquad (2)$$

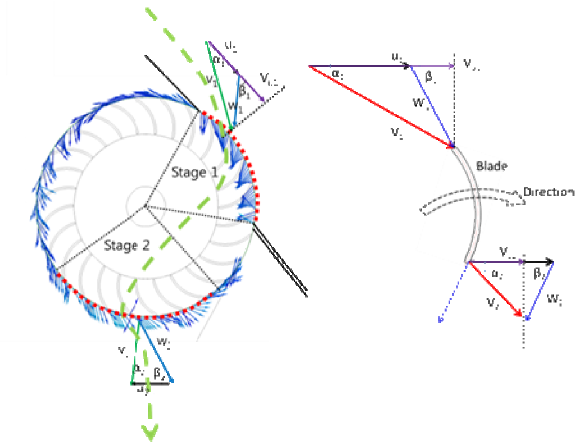

Fig.7. Diagram View of Velocity Triangle.

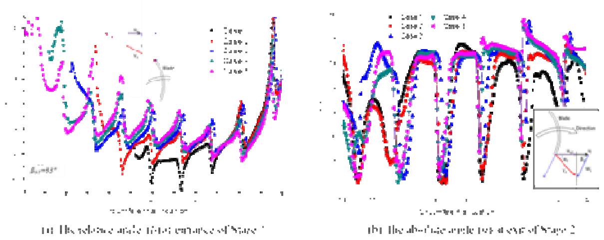

Fig. 8. Velocity Triangle Distribution Curves.

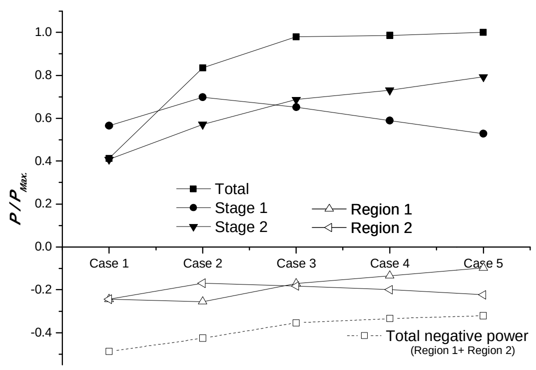

Fig. 9. Averaged Output Power Distribution Curve on the Circumferential.

In Equation (2) it can be seen that for maximum output power, the absolute water flow tangential velocity at outlet of Stage 2 should $V_{u2}=0$, in which case the absolute angle $\alpha = 90°$.Moreover, for the water flow into the runner passage, the relative angle β should be close to the blade inlet angle β_{b1}. Figure 8 reveals the velocity triangle distribution curves at the entrance of Stage 1 and the exit of Stage 2, respectively. From the relative angle (β) distribution at entrance of Stage 1, it can be seen that the β of Case 1 is the closest to β_{b1}, there is no separation flow between water flow and the blade surface in the runner passage, this phenomenon is also proved in Figure 7. From Cases 2 to 5, the relative angle β is farther away from blade inlet angle β_{b1}, which means that larger turbine open angle causes relative angle farther away the best flow inlet angle. From the absolute angle at exit of Stage 2, the absolute angle of flow has almost no change from the circumferential location of 180º-220º, which has most of the output power generated. The effect of the turbine IODBL location is poor on the absolute angle at flow exit of Stage 2.

Output Power Distribution

The runner passage area is divided by four regions to check the output power distribution on the circumferential. Figure 9 shows the averaged output power distribution on the circumferential by the four regions (Stages 1, 2 and Regions 1, 2). The total negative power is the sum of the output powers at Regions 1 and 2, which are hydraulic loss by recirculating flow. The total negative power keeps stable after Case 3, and that reduces rapidly from Cases 1 to 3. By opening the open angle, the hydraulic energy loss at Regions 1 and 2 is suppressed effectively, but that impact becomes small if the turbine with too large open angle. Therefore, even though the relative angle of Case 1 is the closest to β_{b1}, here exists large extent of loss output at Regions 1 and 2. Thus, the efficiency of Case 1 shows lower than that of Case 3. Moreover, the output power rises at Stage 2 for larger open angle, but it reduces at Stage 1 except for Case 1. The output power at Stage 1 in Case 1 is small, so it is conjectured that the circumferential range of Case 1 at Stage 1 is the smallest one that contributes the least output power.

4. CONCLUSIONS

This paper presents the characteristic of a very low head cross flow turbine with open ducted inlet channel, the performance and internal flow improvement by optimizing the IODBL location. The performance of the turbine is improved significantly by a suitable location of turbine IODBL at the periphery of the runner.

The larger turbine open angle gives larger pressure difference at Stage 1 and reduces the recirculating flow at Stage 2. However, there is a large recirculating flow at Stage 1 when the turbine open angle and the water flow increase separately with the blade surface, which exists hydraulic energy loss.

The effect of the increase of the turbine open angle on the relative angle at flow entrance of Stage 1 is farther way from the blade inlet angle, but there exists a large extent of loss output at Regions 1 and 2 with larger turbine open angle. Thus, the efficiency of Case 1 shows lower than that of Case 3.

REFERENCES

[1] Kokubu, K. Kanemoto, T. Son, S.W and Choi Y.D. 2012. Performance improvement of a micro eco cross-flow hydro turbine. *Journal of the Korean Society of Marine Engineer* 36(7) 902-909.

[2] Kokubu, K. Yamasaki, K. Honda, H. and Kanemoto, T. 2012. Effect of Inner Guide on Performances of Cross Flow Turbine. *26th IAHR Symposium on Hydraulic Machinery and Systems* (doi:10.1088/1755-1315/15/4/042035).

[3] Kokubu, K. Son, S W. Kanemoto, T. and Choi, Y. D. 2011. Internal flow analysis on a micro cross-flow type hydro turbine at very low specific speed range. In *Proceedings of the 11th Asian International Conference on Fluid Machinery*, 195.

[4] Croquer, S. D. Andrade, J. D. Clarembaux, J. Jeanty, F. and Asuaje, M. 2012. Numerical investigation of a banki turbine in transient satate reaction ratio determination. In *Proceeding of ASME Turbo Expo 2012 GT2012-68809.*

[5] Croquer, S. D. Andrade, J. D. Clarembaux, J. Jeanty, F. and Asuaje, 2012. Use of CFD tools in internal deflector design for cross flow turbine efficiency improvement. In *Proceeding of the ASME 2012 Fluids Engineering Summer Meeting FEDSM2012-72017*.

[6] Hayati, O. 2000. Effect of interior guide tubes in cross flow turbine runner on turbine perform. *Internal Journal Energy*. Res 24 953-964.

[7] Chen, Z. and Choi, Y. D. 2013. Performance and internal flow characteristics of a cross flow turbine by guide vane angle. *In Proceedings of the 6th International Conference on Pumps and Fans with Compressors and Wind Turbines IOP Conf. Ser*. Mater. Sci. Eng. 52 052031.

[8] Choi, Y. D. Yoon, H. Y. Inagaki, M. Ooike, S. Kim, Y. J. and Lee, Y. H. 2010. Performance improvement of a cross-flow hydro turbine by air layer effect. *Journal of Fluid Machinery* 13(4) 37-44.

[9] Choi Y D, Lim J I, Kim Y T and Lee Y H 2008. Performance and internal flow characteristics of a cross-flow hydro turbine by the shapes of nozzle and runner blade. *Journal of Fluid Science and Technology* 3(3) 398-409.

[10] ANSYS Inc. 2013 “ANSYS CFX Documentation,” ver. 14, from the World Wide Web: http://www.ansys.com.

[11] White, F. M. 1999. Fluid Mechanics, New York: WCB/McGraw-Hill.

[12] Gorla, R. S. R. and Khan, A. A. 2003. Turbomachinery, Design and Theory, New York: Marcel Dekker.

[13] Sammartano, V. Arico, C. Carravetta, A. Fecarotta, O. and Tucciarelli, T. 2013. Banki-michell optimal design by computational fluid dynamics testing and hydrodynamic analysis. (Energies 6(5) 2362-2385).