Journal of the Korean Society of Marine Engineering, Vol.32, No.3, pp.413~420, 2008(ISSN 1226-9549)

Effect of Blade Angle on the Performance of a Cross-Flow Hydro Turbine

Young-Do Choi* · Jae-Ik Lim** · You-Taek Kim*** · Young-Ho Lee†
(Manuscript : Received March 20, 2008 ; Revised May 8, 2008)

Abstract : In order to improve the performance of cross-flow hydro turbine, detailed examination of the effect of the turbine configuration on the performance is needed necessarily. Therefore, this study is aimed to investigate the effect of blade angle on the performance of the cross-flow hydro turbine. Analysis of the turbine performance with the variation of the blade angle has been made by using a commercial CFD code. The results show that inlet and outlet angles of runner blade give considerable effect on the performance of the turbine. Pressure on the surface of the runner blade changes remarkably by the blade angle both at the Stages 1 and 2. Moreover, relatively small blade inlet angle is effective to produce higher value of output power. Recirculating flow in the runner passage causes remarkable hydraulic loss.

Key words : Cross-flow hydro turbine, Small hydropower, Blade angle, Performance, internal flow

Symbols

b : width of nozzle, runner and runner chamber [mm]
C_p : pressure coefficient($=(p-p_{ref})/\rho gH$)
d : diameter of runner [mm]
H : effective head [m]
n : rotational speed [min^{-1}]
N : unit rotational speed($=nd/\sqrt{H}$)
N_{bep} : unit rotational speed at the best efficiency point
p : static pressure [Pa]
p_{ref} : reference static pressure at draft tube outlet [Pa]
P : output power [W]
Q : volume flow rate [m^3/s]
r : radius from the center of runner axis [m]
r_2 : outer radius of runner [m]
u_1 : absolute velocity of runner inlet at Stage 1 [m/s]
Z : number of runner blade
α : blade inlet angleat Stage 1 or outlet angle at Stage 2 [°]
β : blade outlet angle at Stage 1 or inlet angle at Stage 2 [°]
v : fluid velocity [m/s]
η : efficiency

† Corresponding Author(Korea Maritime University, Division of Mechanical and Information Engineering, E-mail:lyh@hhu.ac.kr, Tel: 051)410-4293)
* Korea Maritime University, Industry-Academic Cooperation Foundation
** Korea Marine Equipment Research Institute
*** Korea Maritime University, Department of Marine System Engineering

θ_1^* : peripheral blade position at Stage 1
$\theta^*_{1\,passage\ average}$: averaged peripheral blade passage position at Stage 1
θ_2^* : peripheral blade position at Stage 2
$\theta^*_{2\,passage\ average}$: averaged peripheral blade passage position at Stage 2
ρ : density of working fluid [kg/m^3]

Subscripts

11 : inlet of runner at Stage 1
12 : outlet of runner at Stage 1
r : radial component of velocity
θ : tangential component of velocity

1. Introduction

Recently, small hydropower attracts attention because of its clean, renewable and abundant energy resources to develop. However, suitable turbine type is not determined yet in the range of small hydropower. Relatively high manufacturing cost by the complex structure of the turbine is the highest barrier for developing the small hydropower. Therefore, a cross-flow turbine is adopted to apply to the small hydropower.

Previous studies by researchers for the cross-flow turbine have tried to determine the optimum configuration of the turbine by experimental and numerical methods. Mockmore et al.[1]have used the methods of one-dimensional theoretical analysis and experiment. Khosrowpanah et al.[2], Abbas et al.[3] and Desai et al.[4] have tried experimentally to improve the turbine performance by modifying the shape of flow passages or by applying devices to the turbine. Moreover, Fukutomi et al.[5]-[7] have demonstrated the effect of component parts of the turbine on the turbine performance by series of studies using experimental and numerical methods. Even though the previous studies have found an approximate configuration of the turbine shape to achieve high performance, there exists still plenty of room for performance improvement.

Therefore, in this study, relations between runner blade angle and turbine performance are examined in detail using CFD analysis.

2. Turbine model and numerical methods

2.1 Cross-Flow Hydro Turbine Model

Fig. 1 shows the schematic view of cross-flow turbine model which had been used for experiment by Choi et al.[8] The number of runner blade is Z=26. The widths of nozzle, runner and runner chamber are all same, b=150mm. As the main purpose of this study is to confirm the effect of blade angle on the turbine performance, internal passage of the turbine model is simplified by eliminating guide vane at the inlet of nozzle and baffle plate in the runner chamber. Flow passage in the runner is divided into Stages 1 and 2 as shown in Fig. 2.

Five kinds of blade angle are selected by changing the inlet and outlet angles of the runner blade as shown in Table 1. Blade angle of Case B has the same configuration and size with that of the experiment[8].

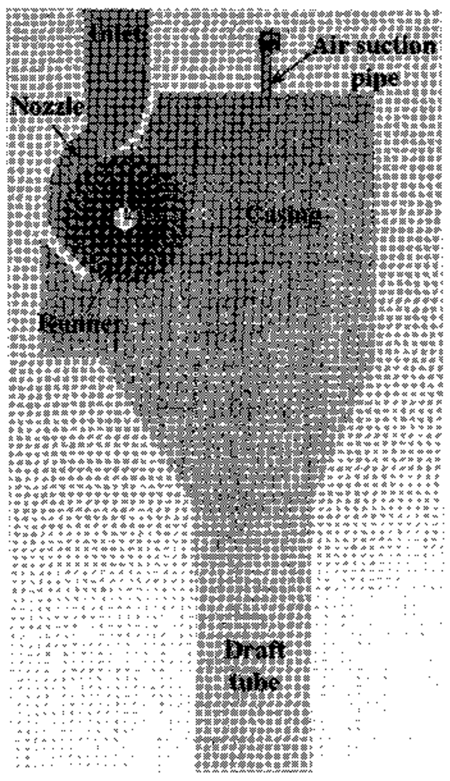


Fig. 1 Schematic view of cross-flow hydro turbine model

2.2 Numerical methods

For the numerical analysis of the turbine performance and internal flow characteristics by the blade angle at the runner inlet and outlet, a commercial CFD code ANSYS-CFX[9] is adopted. The grid number of about $2x10^6$ has been used as shown in Fig. 3. Fine hexahedral-grids are employed for runner passage to ensure relatively high accuracy of calculated results and tetrahedral-grids are applied to the other regions of the turbine.

As a turbulence model, *k-ω SST* turbulence model is used. Constant pressure at inlet and averaged outflow at outlet are the adopted boundary conditions. All the calculations for the test cases by the variations of blade angle and rotational speed are conducted under the condition of steady state.

In the case of working fluid, experimental condition included water supply from the turbine inlet and air supply from the air suction pipe. However, present CFD analysis is carried out under the condition of water supply only from the turbine inlet for the purpose of examining the effect of hydraulic loss by the blade angle in the runner passage on the turbine performance.

3. Results and discussion

3.1 Performance curves

Fig. 4 presents performance curves of the turbine model. The experimental result is referred to that of Choi et al.[8] and the result is achieved under the operational condition of water-air two-phase flow.

However, all the calculated results of Cases A to E are received from CFD analysis under the condition of single-phase flow of water.

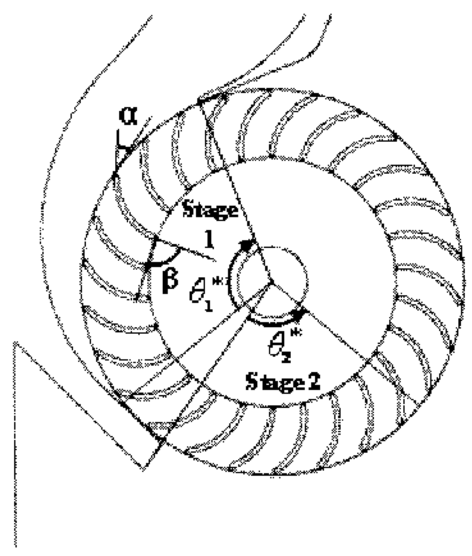


Fig. 2 Blade angles and stages

Table 1 Runner blade angles at the inlet and outlet of Stage 1

Case	Inlet angle α [°]	Outlet angle β [°]
A	25	87
B	30	87
C	35	87
D	30	80
E	30	100

Therefore, as shown in Fig. 4, there exists remarkable differences both of efficiency(η) and unit power($P/dbH^{3/2}$) by the working fluid conditions.

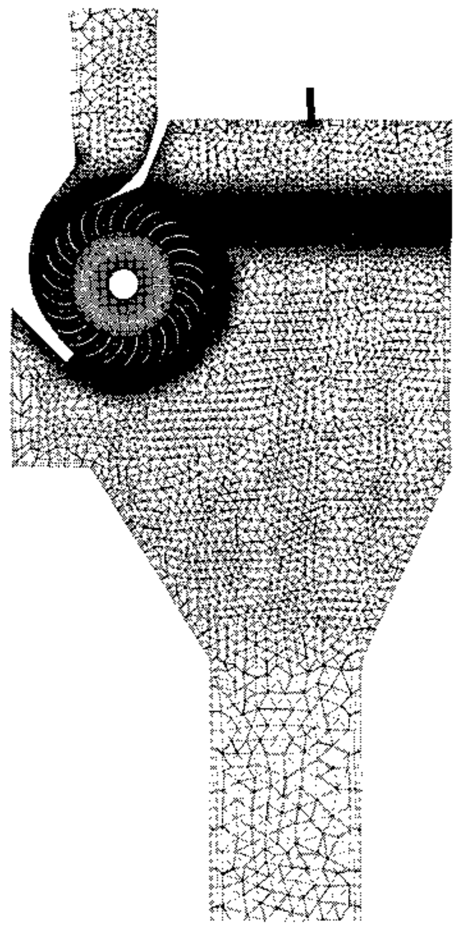


Fig. 3 Pressure pulsation at constant pressure

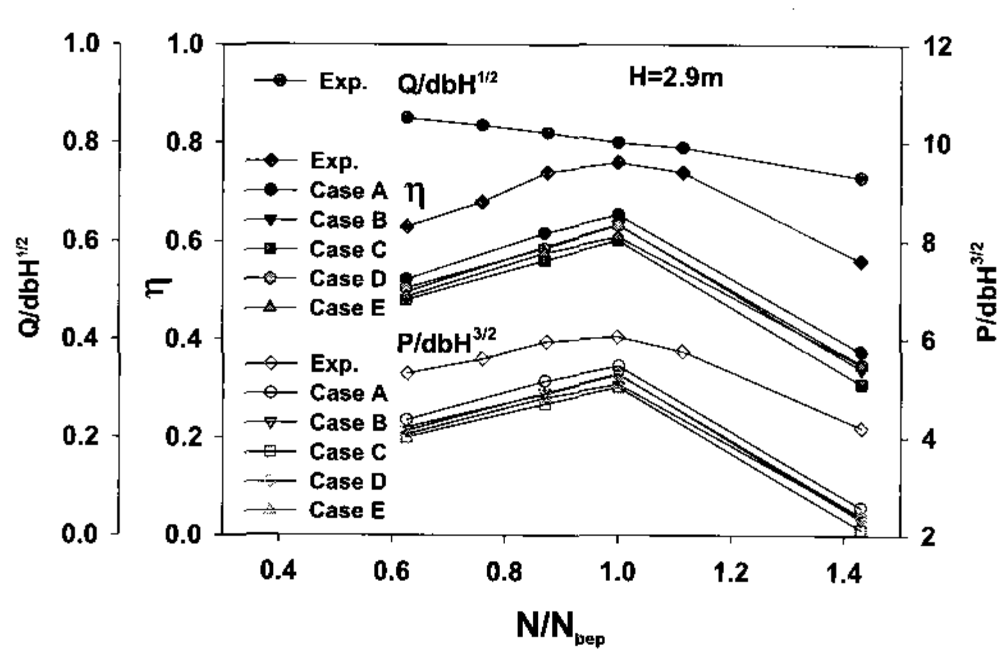


Fig. 4 Performance curves

In spite of the differences by the working fluid conditions, the tendency of performance curves with the variation of rotational speed indicates reasonable agreement with each other in the cases of efficiency and unit power.

The unit rotational speed at the best efficiency point locates at the same range of rotational speed (N/N_{bep}=1.0) both in the experimental and calculated results. Moreover, it is evident that efficiency and unit power vary with the variation of blade angle from the result of CFD analysis. As the inlet angle a of runner blade becomes smaller from the Cases C to A, the efficiency increases accordingly.

3.2 Output power analysis

In order to examine the effect of blade angle to the output power of the turbine in detail, total output powers calculated for the test cases are divided into each section of the Stages 1 and 2 and recirculation area as shown in Table 2.

Table 2 Change of output power by blade angle at each section of runner blades (working fluid : single-phase fluid of water)

	Output power P(kW)			
Case	Stage 1	Stage 2	Recirculation area	Total
A	0.321	0.234	-0.050	0.505
B	0.321	0.233	-0.065	0.490
C	0.355	0.149	-0.040	0.465
D	0.268	0.269	-0.049	0.489
E	0.457	0.081	-0.068	0.470

From the result of Table 2, it is clear that the blade angle gives large effect both on the total output power and the local output power. In order to increase the output power by optimum configuration, inlet blade angle should be relatively small but the blade angle should be determined to make the balance of generated value of output power at each local runner blade. As a whole, output power ratio at Stage 1 keeps almost over 60% of total output power. Therefore, configuration of blade angle should be designed to increase the output power effectively at Stage 1.

3.3 Velocity vectors and distributions

Fig. 5 shows velocity vectors in the internal flow field of the turbine model. As a whole, fluid velocity becomes accelerated along the contracted nozzle passage from the turbine inlet. After passing through the runner passage in the Stage 1, cross-flow within the runner gains accelerated velocity once more and then the flow enters to the inlet of the Stage 2. In the runner passage, there exists a large recirculating flow as indicated in the recirculation area. The recirculation flow generates considerable loss of output power as shown in Table 2. Therefore, the reason of large difference of efficiency and unit power by the working fluid conditions in the Fig. 4 can be explained by the recirculating flow.

As a usual method[8] of suppressing the recirculating flow, air layer is put into the runner passage by supplying air from the air suction pipe as shown in Fig. 1.

Fig. 6 indicates the velocity distribution at the inlet and outlet of Stages 1 and 2. Tangential velocity component v_θ at the inlets of Stages 1 and 2 decreases after passing through the blade passage and then, the tangential velocity at the outlet of Stage 2 becomes near 0.

However, radial velocity component v_r, which is related to the flow rate passing through the runner passage, maintains almost same velocity distributions at each stage. Therefore, the variation of tangential velocity distributions at each stage implies that the angular momentum of the fluid velocity is consumed by the runner blade and changed to output power.

Fig. 7 shows tangential velocity distributions by runner blade angle at Stages 1 and 2, respectively. Most high tangential velocity distribution at the blade inlet and outlet of Stage 1 is found at Case A among the cases. As the higher tangential velocity results in the more output power, the tangential velocity distribution have reasonable agreement with the output power in Fig. 4. Therefore, this result means that relatively small blade inlet angle is effective to produce higher value of output power.

3.4 Pressure distributions on the runner blade

Fig. 8 shows the static pressure contours in the turbine flow passage. Turbine inlet pressure decreases along the nozzle passage but the pressure at the nozzle outlet is distributed almost the same along the entry arc of runner inlet in the Stage 1. The fluid pressure passing through the passage of runner blades at Stage 1 drops rapidly.

From this result, it is obvious that the fluid pressure passing through the passage of runner blade is taken by the runner blades and the pressure is converted to output power.

In addition, very low pressure region is located in the recirculation area in which a large vortex is found as shown in Fig. 5.

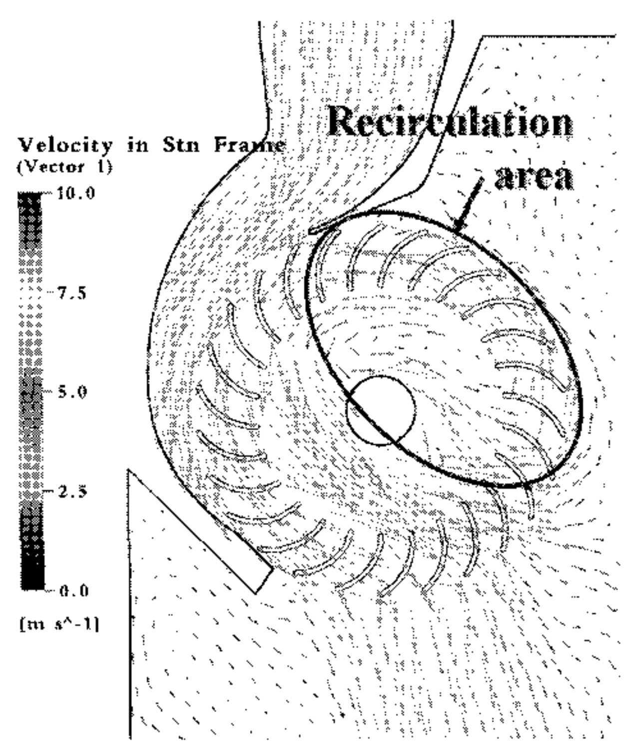


Fig. 5 Velocity vectors within the internal flow field (Case B, N/Nbep=1.0)

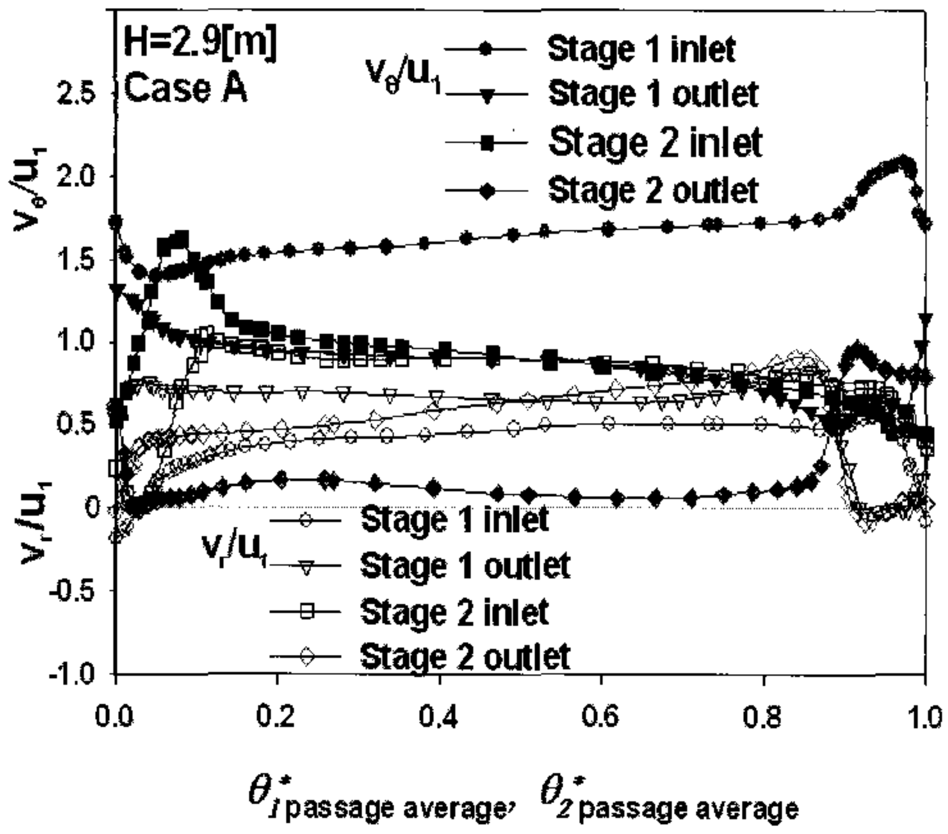


Fig. 6 Velocity distribution at the inlet and outlet of each stage

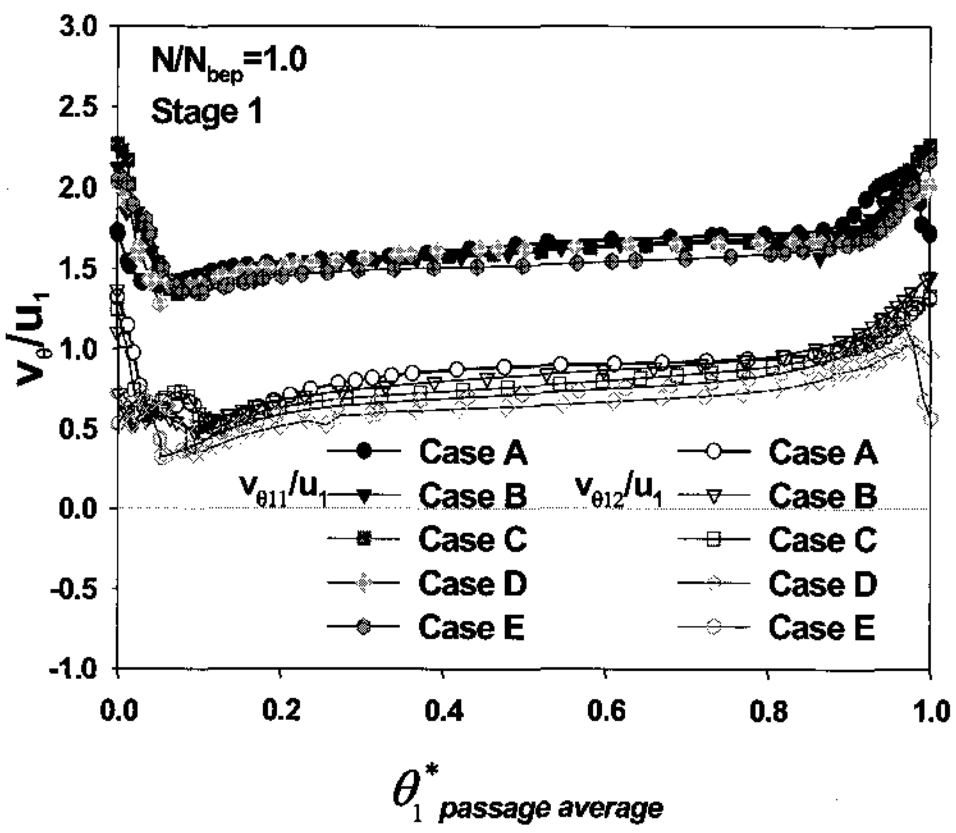


Fig. 7 Tangential velocity distributions at Stage1

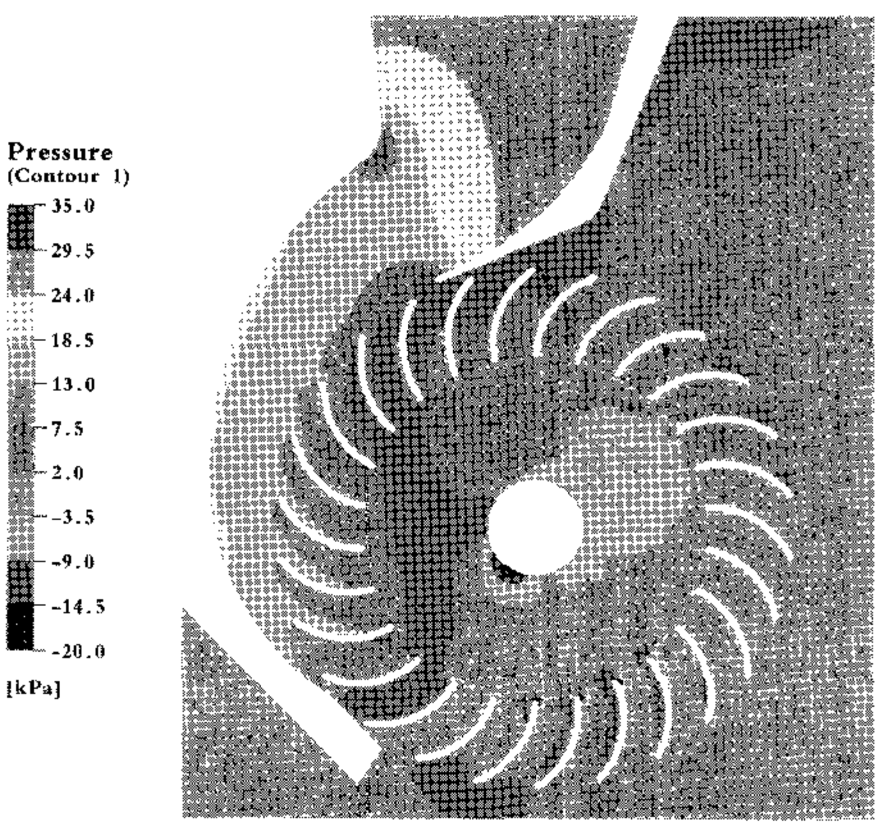


Fig. 8 Pressure contours within the flow field (Case B, N/Nbep=1.0)

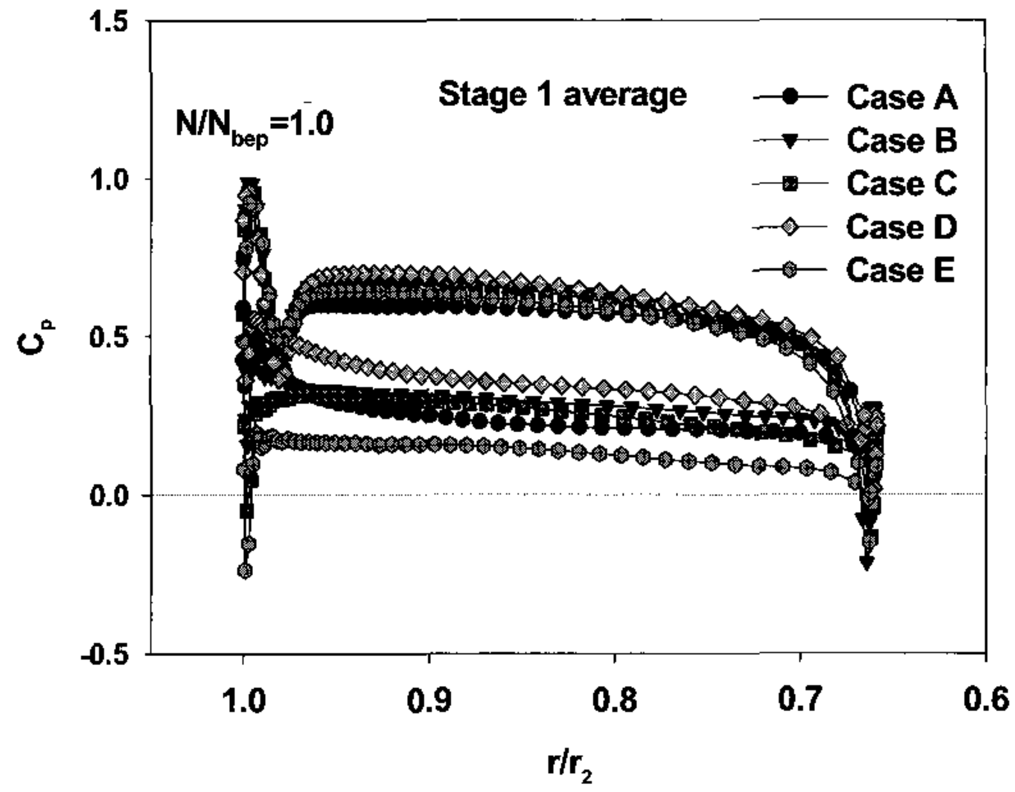


Fig. 9 Pressure distributions on the blade surface at Stage 1

Fig. 9 show averaged pressure distributions on the surface of runner blade by blade angle at Stage 1. As the closed area filled with the pressure curve is proportional to the output power of the turbine, the wider area by the pressure curve can contribute to the more output power.

Fig. 9 shows averaged pressure distributions on the surface of runner blades by blade angle at Stage 1. With the variation of blade angle, pressure distribu- tion changes remarkably.

Especially, the Case E shows the largest area which is filled with the pressure curve. Therefore, it can be thought that the output power generated by the highest pressure by Case E compensates the low output power generated by the relatively low tangential velocity as shown in Fig. 7 and then, the total output power keeps relatively low value among the test cases as shown in Fig. 4.

4. Conclusions

The effect of runner blade angle on the

performance of a cross-flow hydro turbine is examined in detail. From the results of the present study, the following conclusions have been obtained.

1. Runner blade angle gives large effect on the performance of a cross-flow hydro turbine. Especially, relatively small blade inlet angle is effective for the improvement of the turbine performance.

2. Relatively high tangential velocity distribution at the blade inlet and outlet of Stage 1 is found in the case of relatively small blade inlet angle. Therefore, relatively small blade inlet angle is effective to produce higher value of output power.

3. Pressure on the surface of the runner blade is sensitive considerably to the change of blade angle at Stage 1.

4. As recirculating flow in the runner passage causes considerable hydraulic loss, efficiency and output torque of the turbine decreases remarkably. Therefore, air layer in the runner passage can be a effective method to suppress the recirculating flow.

Acknowledgement

The authors express their appreciation to Korea Energy Management Corporation, Ministry of Commerce, Industry and Energy for their grant of research and development project of new & renewable energy Technology (Project No. : 2006-N-SH02-P-01-0).

Reference

[1] Mockmore, C. A. and Merryfield, F., "The Banki Water Turbine," No. 25, Engineering Experiment Station, Oregon State Colleg, Corvallis, Oregon, 1945.

[2] Khosrowpanah, S., Fiuzat, A. A. and Albertson, M. L., "Experimental Study of Cross-Flow Turbine," Journal of Hydraulic Engineering, Vol. 114, No. 3, pp. 299-314, 1988.

[3] Fiuzat, A. A. and Akerkar, B. P., "Power Outputs of Two Stages of Cross-Flow Turbine," Journal of Energy Engineering, Vol. 117, No. 2, pp. 57 70, 1991.

[4] Desai, V. R. and Aziz, N. M., "An Experimental Investigation of Cross-Flow Turbine Efficiency," Journal of Fluids Engineering, Vol. 116, pp. 545-550, 1944.

[5] Fukutomi, J., Nakase, Y. and Watanabe, T., "A Numerical Method of Free Jet from a Cross-flow Turbine Nozzle," Bulletin of JSME, Vol. 28, No. 241, pp. 1436-1440, 1985.

[6] Fukutomi, J., Senoo, Y. and Nakase, Y., "A Numerical Method of Flow through a Cross-Flow Runner," JSME International Journal, Series II, Vol. 34, No. 1, pp. 44-51, 1991.

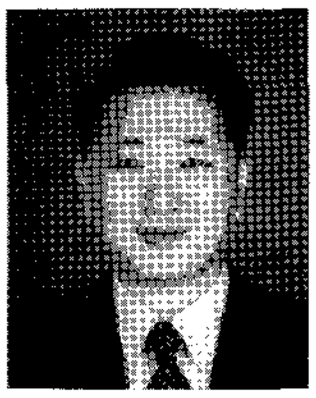
[7] Fukutomi, J., Nakase, Y., Ichimiya, M. and Ebisu, H., "Unsteady Fluid Forces on a Blade in a Cross-Flow Turbine," JSME International Journal, Series B, Vol. 38, No. 3, pp. 404-410, 1995.

[8] Choi, Y-D., Zhao, L. and Kurokawa, J., "A Study on the Optimal Configuration and Performance Improvement of a Micro Cross-Flow Hydraulic Turbine," Journal of the Korean Society of

Marine Engineering, Vol. 30, No. 2, pp. 296-303, 2006.

[9] ANSYS Inc., "ANSYS CFX Documentation," Ver. 11, http://www. ansys.com, 2007.

Author Profile

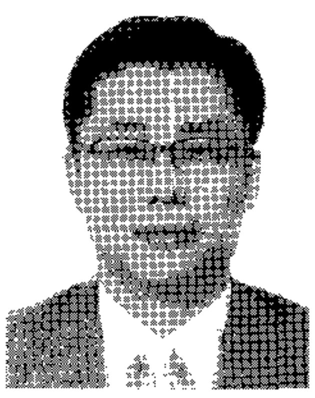


Young-Do Choi

He received his B.E. and M.Eng. degrees from Korea Maritime University, and his Dr.Eng. from Yokohama National University, Japan. He is currently a researcher in the Industry-Academic Cooperation Foundation at Korea Maritime University, Korea. His research interests include new & renewable energy, fluid machinery, PIV, CFD and flow visualization.

Jae-Ik Lim

He received his M.Eng. in 2008 from Korea Maritime University, Korea. He is currently a researcher in the Korea Marine Equipment Research Institute, Korea. His research interests include CFD and new & renewable energy.



You-Taek Kim

He received his B.E. and M.Eng. degrees from Korea Maritime University, and his Dr.Eng. from Kyushu Institute of Technology, Japan. He is currently an associate professor in the dept. of Marine System Engineering at Korea Maritime University, Korea. His research interests include new & renewable energy, fluid machinery, two-phase flow, cavitation, and CFD.

Young-Ho Lee

He received his B.E. and M.Eng. degrees from Korea Maritime University, and his Dr.Eng. from the University of Tokyo, Japan. He is currently a professor in the Division of Mechanical and Information Engineering at Korea Maritime University, Korea. His research interests include PIV, new & renewable energy, fluid machinery, CFD, FSI and underwater Ram-Jet.