Performance and internal flow characteristics of a cross-flow turbine by guide vane angle

Z M Chen¹, Y D Choi²

(61 Dorim-ri) 1666 Youngsan-Ro, Cheonggye-Myeon, Muan-Gun, Jeonnam, South Korea

E-mail: chenzhenmu@163.com

Abstract. This study attempts to investigate the performance and internal flow characteristics of a cross-flow turbine by guide vane angle. In order to improve the performance of a cross-flow turbine, the paper presents a numerical investigation of the turbine with air supply and discusses the influence of variable guide vane angle on the internal flow. A newly developed air supply from air suction Hole is adopted. To investigate the performance and internal flow of the cross-flow turbine, the CFD software based on the two-phase flow model is utilized. The numerical grids are made in two-dimensional geometry in order to shorten the time of two-phase calculations. Then a series of CFD analysis has been conducted in the range of different guide vane angle. Moreover, local output power is divided at different stages and the effect of air layer in each stage is examined.

1. Introduction
Recently, the cross-flow turbine has a great potential for small hydroelectric power development of both developed and undeveloped countries. Amongst renewable energy resources, the cross-flow turbine is the most reliable and cost effective with a very simple structure. The nozzle and the runner are made by processing steel plate and the blades are made by processing steel pipe giving it a simple design and manufacture. Therefore it can save more manufacturing cost and maintenance cost.

Among the present studies of cross-flow turbine, most studies have optimized the configuration of turbines to improve the efficiency by experiment. The turbine is designed based on Banki’s one-dimensional analysis method[1]. Abbas et al.[2] has carried out to identify the contribution of the cross-flow turbine stages to the power output generation. Khosrowpanah et al.[3], Fukutomi et al.[4] have tried to improve the performance of the turbine by changing the number of blades and the angles of water entry to the runner and the inner-to-outer diameter ratios to improve turbine performance. On the other hand, Choi et al. [5] has investigated the effect of air layer located in the turbine runner passage. The air layer plays a significant role to improve the turbine performance. However, the internal flow in the turbine passage is complex, especially with air layer. The flow with air layer has not been analyzed thoroughly until now.

Therefore, this study attempts to investigate the performance and internal flow characteristics of a cross-flow turbine with air layer by guide vane angle. A newly developed air supply method is adopted. To investigate the performance and internal flow of the cross-flow turbine, the CFD software based on the two-phase flow model is utilized. The numerical grids are made in two-dimensional geometry in order to shorten the time of two-phase calculations.
2. Turbine model and numerical method

2.1. Cross-flow turbine model
Figure 1 shows the schematic view of the cross-flow turbine model which is calculated by CFD software in this study. As shown in Fig. 1, there is a air suction hole. The air hole is installed on the top of the draft tube.

The number of the runner blades is \( Z = 30 \) and the diameter of the runner is \( d = 340 \text{mm} \). The inlet and outlet angles of the blade are \( \alpha = 34^\circ \) and \( \beta = 92.5^\circ \), respectively. The widths of the nozzle, runner and the draft tube are all same, the width is \( b = 500 \text{mm} \). The guide vane angle is from 0 degree to 32 degree, which means the turbine from closed to completely open states, respectively. The design point of the present test turbine at the test efficient point are \( H = 20 \text{m} \) for effective head, \( n = 530 \text{ min}^{-1} \) for rotational speed, \( Q = 0.516 \text{ m}^3/\text{s} \) for the flow rate.

![Figure 1. Schematic view of the numerical model.](image)

2.2. Numerical methods
For the numerical analysis of the turbine performance and internal flow by the variation of the Guide vane angle and different air flow rate, a commercial code of ANSYS CFX [6] is adopted. The grid element number of about \( 1.6 \times 10^6 \) for the whole flow field has been used. Fine hexahedra grids are employed to ensure relatively high accuracy of calculated results for the turbine model. Figure 2 shows the hexahedra numerical mesh for the turbine runner and one blade of runner with fine dense grids near the blade.

In order to shorten the time of the calculations, the model which is calculated for the CFD analysis is made in two-dimensional, even though the geometry of the turbine is three-dimensional. Because it can be assumed the flow in the turbine passage is uniform to the direction of main stream, which means no flow velocity in the direction to the runner depth.

The test model with air from air suction Hole is calculated by steady state. SST turbulence model is adopted as turbulence model because of its relatively good convergence in the complicated flow field of turbo-machinery in comparison with other models. Averaged mass flow rate at inlet and constant pressure at outlet of the calculation domain are used for boundary conditions. All of the calculations are conducted under the conditions of steady state.
Moreover, Table 1 shows the cases of CFD analysis with air supply from air suction Hole by the variation of guide vane angle.

![Numerical mesh for runner and one blade of runner.](image)

**Figure 2.** Numerical mesh for runner and one blade of runner.

| Analysis cases | Guide vane angle [°] | Water flow rate [m³/s] | Air flow rate [m³/s] |
|----------------|----------------------|-------------------------|----------------------|
| Case 1         | 15                   | 0.363                   | 0.363                |
| Case 2         | 18                   | 0.425                   | 0.425                |
| Case 3         | 20                   | 0.465                   | 0.465                |
| Case 4         | 22                   | 0.516                   | 0.516                |
| Case 5         | 25                   | 0.563                   | 0.563                |

### 3. Results and discussion

#### 3.1. Performance curves

Figure 3 shows the performance curves of the turbine model by variation of guide vane angle, including efficiency curves, water flow rate curve. The working fluid is water and air for experiment and CFD analysis. The efficiency of CFD is marked by circle symbol with hollow interior. The efficiency curve of experiment is marked by square symbol with hollow interior. Two efficiency curves by experiment and CFD analysis are almost coincident with small deviation in the ranges of the partial and excessive flow rates, which makes the methods and the computational results reasonable.

The performance curves show that the best efficiency point is at 20° (Case 3). The efficiency is lower for the partial and excessive guide vane angle in comparison to the 20° guide vane angle. From these results, it is concluded that the efficiency of the present cross-flow turbine can be improved by change to an appropriate guide vane angle.

In order to examine the performance variation of the guide vane angle, internal flow of the cross-flow turbine is investigated.
3.2. Velocity distribution
In order to investigate the internal flow, the average tangential velocity distribution in the runner blade channel is examined. Figure 4 shows the selected runner passages for the investigation of velocity distribution at Stage 1.

Figure 5 indicates the passage tangential velocity distribution at Stage 1 by guide vane angle of Case 1, Case 3 and Case 6 whose guide vane angle is 15°, 20° and 25°, respectively. The tangential velocity at the inlet of Case 3 is relatively higher compared with that of other cases. Furthermore, the velocity at the outlet of Case 3 is lowest. The area of the internal averaged velocity represents the velocity difference between inlet and outlet. The decreased velocity changes to output torque in the runner passage. Moreover, the result shows that the Case 2 has the highest velocity difference in three cases, which means it is a good condition for cross flow turbine operation when the guide vane angle is 20 degree.
3.3. Pressure distribution

Figure 14 shows the passage averaged static pressure distributions on the blades surface \((r/r_2)\) at Stage 1 by guide vane angle of Case 1, Case 3 and Case 6. The pressure distributions in the turbine by differential pressure coefficient:

\[
C_p = \frac{2(P_{in} - p)}{\rho(R^2\omega)^2}
\]  

(1)

Where \(P_{in}\) is the static pressure at the inlet of turbine and \(p\) is the local static pressure of the runner blade, \(R\) is the diameter of the runner, and \(\omega\) is the angular velocity of the runner.
From this result, it can be seen that the pressure difference in the upper and lower surfaces is quite small for Case 1 whose guide vane angle is 15 degree. As the closed area filled with the pressure curve is proportional to the output power of the turbine. Even though Case 6 has the largest area of the difference pressure coefficient, which means produces largest torque of the turbine, the efficiency is decrease in comparison with Case 3. Because of the water flow rate is increase accordingly. Therefore, this result implies that there is optimum guide vane angle to absorb static pressure efficiently from the water flow in the blade passage of turbine.

4. Conclusion
According to the investigation of the influence of guide vane angle, the guide vane angle gives considerable influence to the internal flow and performance of the cross flow turbine. From the results of the present study, the following conclusions have been obtained.
(1) The cross flow turbine of guide vane at 20 degree has a good condition for operation. There is highest efficiency when turbine operating at this guide vane angle.
(2) Pressure and velocity in the water flow passage of the turbine change to output torque in the runner passage.

References
[1] Mockmore C A and Merryfield F 1949 The banki water turbine, bulletin series no. 25 (Corvallis: Engineering Experiment Station, Oregon State System of Higher Education, Oregon State College)
[2] Fiuzat A A and Akerkar B P 1991 Journal of Energy Engineering 117(2) 57-70
[3] Khosrowpanah S, Fiuzat A A and Albertson M L 1988 Journal of Hydraulic Engineering 114(3) 229-314
[4] Fukutomi J, Nakase Y and Watanabe T 1985 The Japan Society of Mechanical Engineers 28(241) 1436-40
[5] Choi Y D, Kim J I and Lee Y H 2008 Journal of Fluid Science and Technology 3(3) 398-409.
[6] ANSYS Inc., “ANSYS CFX Documentation,” ver. 12, http://www.ansys.com, 2010