Research on the flow field of undershot cross-flow water turbines using experiments and numerical analysis

Y Nishi¹, T Inagaki ¹, Y Li¹, R Omiya² and K Hatano²

¹. Department of Mechanical Engineering, Ibaraki University, 4-12-1 Nakanarusawacho, Hitachi-shi, Ibaraki, 316-8511, JAPAN

². Graduate School of Science and Engineering, Ibaraki University, 4-12-1 Nakanarusawa-cho, Hitachi-shi, Ibaraki, 316-8511, JAPAN

E-mail: y-nishi@mx.ibaraki.ac.jp

Abstract. The purpose of this research is to develop a water turbine appropriate for low-head open channels in order to effectively utilize the unused hydropower energy of rivers and agricultural waterways. The application of the cross-flow runner to open channels as an undershot water turbine has come under consideration and, to this end, a significant simplification was attained by removing the casings. However, the flow field of undershot cross-flow water turbines possesses free surfaces. This means that with the variation in the rotational speed, the water depth around the runner will change and flow field itself is significantly altered. Thus it is necessary to clearly understand the flow fields with free surfaces in order to improve the performance of this turbine. In this research, the performance of this turbine and the flow field were studied through experiments and numerical analysis. The experimental results on the performance of this turbine and the flow field were consistent with the numerical analysis. In addition, the inlet and outlet regions at the first and second stages of this water turbine were clarified.

1. Introduction

Water turbines used in hydroelectric power generation can be broadly classified as water turbines that employ enclosed conduits (penstocks) to conduct water at high and low heads, and those for use in open channels [1-3] with low and ultra-low heads. Water turbines employing penstocks are the mainstream in hydroelectric power generation and are used in large-scale centralized power generation involving water storage reservoirs, balancing reservoirs, and pipelines. However, it is currently difficult to carry out new construction due to a decline in suitable construction locations and concerns regarding the effects on the surrounding ecosystem. In contrast, water turbines for use in open channels are directly installed in locations such as agricultural water channels and small-scale rivers, and these turbines require almost no auxiliary equipment. This approach minimizes the burden on the environment while facilitating easy serviceability. However, these turbines are characterized by low turbine efficiency and excessive outer diameters relative to the water depth, and have thus been mainly used as a source of motive force until now. Present design methods are far from established. Therefore, it is necessary to clarify the complicated flow fields involving a free surface in order to design water turbines of high efficiency and high rotational speed for use in open channels. With the development of computational fluid
dynamics technology, the flow fields involved with various water turbine designs, such as a cross-flow water turbine [4,5] for use with penstocks and a spiral water turbine [6,7] for use in open channels, have been investigated by numerical analysis. However, the treatment of a flow field involving a free surface must capture the interface between the fluids of water and air efficiently and correctly. In addition, the flow fields involving a rotating runner are very complicated and numerical analyses remain problematic. Therefore, few such studies of these complex analyses have been reported. Under such a background, we focused our attention on runners in cross-flow turbines [8,9] used with penstocks under intermediate and low head conditions with the aim of developing a water turbine for use in open channels, which is also suitable for ultra-low heads. Substantial simplification was attained in the analysis by eliminating the guide vane, casing, etc., and a cross-flow turbine was applied to an open channel as an undershot water turbine [10]. The performance of the cross-flow turbine (undershot cross-flow turbine) installed in an open channel was clarified experimentally [10]. In this study, a combination of particle image velocimetry (PIV) measurements and numerical analysis is used to investigate the flow fields involving the free surface of an undershot cross-flow water turbine.

2. Experimental Apparatus and Method

The test runner [10] used in this study is illustrated in Fig. 1, and its specifications are presented in Table 1. The runner has the following dimensions: outer diameter \(D_1 = 0.18\) m, width \(b = 0.24\) m, clearance between runner and side walls \(\gamma = 5\) mm, and clearance between runner and floor \(\delta = 5\) mm. Other specifications for this runner were determined in reference to a typical cross-flow runner [9] used in conjunction with a penstock. The circumferential angle \(\theta\) defined the negative direction of the X-axis as \(\theta = 0^\circ\) and the counterclockwise rotation as positive. On the outer circumference of the runner, the measuring points of the flow were prepared at a position shifted 9 mm toward the path direction on the outer side. Moreover, on the inner circumference of the runner, the measuring points of the flow were prepared at a position shifted 9 mm toward the path direction on the inner side.

A summarized illustration of the experimental apparatus is shown in Fig. 2. The equipment used in this experiment consisted of an open-air circulation water tank that was used to simulate an open channel. The experiment was conducted under a condition setting with the flow rate set to \(Q = 0.003\) \(m^3/s\). The flow rate \(Q\) was measured using an electromagnetic flow meter. The load on the water turbine was altered using a load machine, and the rotational speed \(n\) and torque \(T\) were measured using an electromagnetic pickup and a torque meter, respectively, from which we obtained the output power \(P\). Water depth was measured at 2 points, upstream and downstream from the runner, providing for an upper stream water depth \(h_3\) and a lower stream water depth \(h_4\). These depths were derived through measurements obtained at a distance of \(2D_1\) from the center of the runner in both the upstream and downstream directions. Measurements at each location were obtained from a point on the wall surface and from the center point of the channel having a width \(B = 0.25\) m. In addition, both the upstream and downstream flow velocities, \(v_3\) and \(v_4\), respectively, were obtained using the measured upper stream water depth \(h_3\), the measured lower stream water depth \(h_4\), and the flow rate \(Q\) via the following equations.

\[
v_3 = \frac{Q}{(Bh_3)} \\
v_4 = \frac{Q}{(Bh_4)}
\]

In this study, the test runner and side walls of the open channel were made from transparent acrylic. A green diode-pumped solid-state high-power laser (CVI Melles Griot, 85-GHS-309, 3 W, 532 nm) was used as a light source to visualize the flow field around the test runner. Nylon 12 with a diameter of around 100 \(\mu\)m and a specific gravity of 1.02 was used as tracer particles. The runner width central was illuminated by the laser sheet. Original images were acquired at a spatial resolution of \(800 \times 600\) pixels. The region around the test runner was divided into three parts and time series images were
recorded at a recording speed of 200 or 500 fps depending on the local velocity. Based on the recorded images, the flow field around the test runner was post-processed using the direct cross-correlation method provided in the Flow Expert software package (Katokoken Co. Ltd., ver. 1.1.2.0).

| Specifications of the runner |
|-----------------------------|
| Outer Diameter: $D_1$        | 0.18 m |
| Inner Diameter: $D_2$        | 0.12 m |
| Inlet Angle: $\beta_{b1}$    | 30°    |
| Outlet Angle: $\beta_{b2}$   | 90°    |
| Runner Width: $b$            | 0.24 m |
| Side Clearance: $\gamma$    | 5 mm   |
| Bottom Clearance: $\delta$  | 5 mm   |
| Number of Blades: $Z$        | 24     |

3. Method and Conditions for the Numerical Analysis

In this study, a three-dimensional unsteady flow analysis was conducted using the ANSYS CFX 13.0 general-purpose analysis code. The working fluids were water and air. In order to simulate the flow field involving a free surface, the uniform model [11] of an Euler–Euler simulation was applied as a multiphase flow model. The basic equations used in the model are based upon mass, momentum, and volume conservation [11]. The standard k–$\varepsilon$ model was adopted as the turbulent flow model, and the standard wall function was used to handle regions near wall surfaces.

The computational domain of the simulated system and the computational grid of the test runner are shown in Figs. 3 and 4, respectively. The computational domain is composed of the runner and the upstream and downstream domains. The length of the upstream and downstream domains to the center of the runner is $9D_1$ and $10D_1$, respectively. Control surfaces are established at a distance of $2D_1$ in the
upstream and downstream directions from the runner center. The velocity and depth of the water are monitored at these control surfaces as they would be in an experiment. The computational grid includes 462,000 elements in the runner domain, and about 420,000 combined elements in the upstream and downstream domains. Therefore, the computational grid includes a total of about 882,000 elements. For boundary conditions, the mass flow rate was applied to the inlet boundary, free outflow (with a relative air pressure of 0 Pa) was applied to the outlet boundary, and rotational speed was applied to the runner domain. In addition, the top surface of the computational domain was permeable to the atmosphere in order to allow air to move freely in and out of the computational domain, while non-slip conditions were applied to all the other walls. The boundary of the rotational and static domains was joined by the transient rotor–stator method. For initial conditions, the experimental value of the upstream flow velocity $v_3$ was used as the flow velocity in the simulations. The volume fraction $VF_{ih}$ of water was defined in accordance with the following formula using the step function.

$$VF_{ih} = \text{step}(h - y)$$ (3)

Here, $y$ is the coordinate of the height direction in the computational domain, and the water depth $h$ uses the experimental value of the upper stream water depth $h_3$ in the simulations. Therefore, the position $y \leq h_3$ is the domain of water and the position $y > h_3$ is the domain of air. In addition, the time step was adjusted to ensure that the runner would undergo one rotation every 180 steps, and the time step was recalibrated until fluctuations in the flow became negligible.

4. Experimental and Analytical Results and Discussion

4.1. Comparison of Water Turbine Performance

Comparisons of the experimental and calculated values of water turbine performance are shown in Fig. 5. Both the experimental and calculated values of the torque $T$ decrease with increasing rotational speed $n$. However, the calculated value of $T$ exhibits a higher value than the experimental value over the entire range of rotational speeds. Hence, the calculated value of the output power $P$ is also higher than the experimental value for all rotational speeds. In order to investigate this difference, the torque was measured when there was no runner, and the value obtained was nearly in agreement with the torque difference between the calculated and experimental values. Therefore, although it has an influence of a multiphase and a turbulent flow model, the torque reduction by mechanical friction loss in the experiment is not considered in the simulation. This would seem to account for the observed
differences between the calculated and experimental values for $T$. As mentioned previously, the water turbine performance is considered to have been captured to some extent in the numerical analysis.

4.2. Comparison of the Flow Fields

The undershot cross-flow water turbine used by this study differs from the cross-flow water turbine [4,5,8,9] for use with penstocks. Since there is no casing covering the runner, the interface between the water and air changes, particularly at the inlet and outlet regions of the runner. In general, since the flow field of a cross-flow runner is two-dimensional, it is examined below by considering the flow field in the runner width central at a rotational speed of $n = 22 \text{ min}^{-1}$.

The absolute velocity vectors of the circumference of the runner, as determined by the experiment and numerical analysis, are shown in Figs. 6 (a) and (b), respectively. However, the region near the bottom of the channel cannot be experimentally visualized on account of the experimental apparatus. The experimental and calculation results of the flow fields containing water depth are qualitatively in agreement. Moreover, at the low $\theta$ region of the first stage outlet, the phenomenon in which water flows backwards between the back blades is captured by the calculations in a similar manner as observed experimentally.

In order to determine the water and air interface, the time average values of the volume fractions, $VF_1$ and $VF_2$, of the water at the runner’s outer and inner circumferences, respectively, as determined by numerical analysis are shown in Fig. 7. Here, $VF = 1$ is water, $VF = 0$ is air, and $VF = 0.5$ is an interface of water and air. From Fig. 7, the regions of water at the outer and inner circumferences of the runner are determined as $\theta = 21^\circ$–$129^\circ$ and $\theta = 45^\circ$–$132^\circ$, respectively.

The velocity triangles of the undershot cross-flow water turbine are shown in Fig. 8. Here, $v_r$ defines the inside of the radial direction as positive. Moreover, the absolute flow angle $\alpha$ and the relative flow angle $\beta$ define the rotational direction (counterclockwise rotation) of the runner as positive.

The time average values of the radial component $v_{r1}$ and the circumferential component $v_{\theta1}$ of the absolute velocity at the outer circumference of the runner are shown in Figs. 9 (a) and (b), respectively, as determined by both experiment and numerical analysis. The experimental values of $v_{r1}$ and $v_{\theta1}$ are obtained only for the region of water evaluated by the time series images (i.e., only points lying outside $75^\circ < \theta < 105^\circ$). Moreover, the calculated values of $v_{r1}$ and $v_{\theta1}$ are exhibited only for the region of water determined from the abovementioned volume fraction. However, on account of the experimental apparatus, $75^\circ < \theta < 105^\circ$ of the outer circumference of the runner could not be measured. From Fig. 9 (a), the calculated value of $v_{r1}$ gradually decreases with increasing $\theta$, eventually reaching negative values. At the outer circumference of the runner, $v_{r1}$ is positive in the first stage inlet region, and it is negative in the second stage outlet region. In the first stage inlet region within the measured limits, the experimental and calculated values of $v_{r1}$ are mostly in agreement. At the second stage outlet region, although the tendency for the calculated values of $v_{r1}$ to decrease with increasing $\theta$ is in agreement with the experimental values; the calculated values of $v_{r1}$ are slightly larger than the experimental values.

From Fig. 9 (b), the calculated value of $v_{\theta1}$ increases with increasing $\theta$, and, after obtaining a maximum, it decreases slightly. At the first stage outlet region, the experimental and calculated values of $v_{\theta1}$ are mostly in agreement. At the second stage outlet region, although the tendency for the calculated values of $v_{\theta1}$ to decrease with increasing $\theta$ is in agreement with the experimental values, the calculated values are slightly smaller than the experimental values. The second stage outlet region is where the reversal flow containing the leakage flow of the bottom and the cross-flow join. In addition, the water depth is shallow and the depth undergoes substantial periodical fluctuations. Thus, because the phenomenon represents a complicated and unsteady flow field, it is not thought to have been fully covered in this analysis.

The time average values of the radial component $v_{r2}$ and the circumferential component $v_{\theta2}$ of the absolute velocities at the inner circumference of the runner are shown in Fig. 10 (a) and (b), respectively, as determined by both experimental and numerical analyses. The experimental and calculated values of $v_{r2}$ increase with increasing $\theta$ and after obtaining a maximum, they decrease and
become negative. At the inner circumference of the runner, $v_r$ is positive in the first stage outlet region and is negative in the second stage inlet region. At the first stage outlet and second stage inlet regions, within the limits which were measured, the experimental and calculated values of $v_r$ and $v_u$ are mostly in agreement.

![Torque and power performance curves](image)

Fig. 5 Torque and power performance curves

![Absolute velocity vectors](image)

(a) Experiment

(b) Calculation

Fig. 6 Absolute velocity vectors ($n = 22\text{min}^{-1}$)
Fig. 7 Volume fraction of water at the outer and inner circumferences ($n = 22 \text{min}^{-1}$, Cal.)

Fig. 8 Velocity triangles

Fig. 9 Absolute velocities at the outer circumference ($n = 22 \text{min}^{-1}$)

(a) Radial component

(b) Circumferential component
5. Conclusion

The performance of an undershot cross-flow turbine and the flow fields involving a free surface were investigated by experiment and numerical analysis. As a result, the experimental and calculated results of the performance and flow fields of this water turbine agreed satisfactorily. Moreover, the flow fields of this water turbine in the first stage inlet and outlet regions and the second stage inlet and outlet regions were clarified. Within the low $\theta$ region of the first stage outlet, the water is observed to be flowing backwards between the back blades.

**Nomenclature**

- $B$: Channel width [m]
- $b$: Runner width [m]
- $D$: Runner diameter [m]
- $g$: Gravitational acceleration [m/s²]
- $H$: Effective head [m] = $h_3 + \frac{v_3^2}{2g} - h_4 - \frac{v_4^2}{2g}$
- $h$: Water depth [m]
- $n$: Rotational speed [min⁻¹]
- $P$: Output power [W] = $2\pi n T/60$
- $Q$: Flow rate [m³/s]
- $T$: Torque [N·m]
- $u$: Circumferential velocity [m/s]
- $V_F$: Volume fraction of water
- $v$: Absolute velocity [m/s]
- $w$: Relative velocity [m/s]
- $Z$: Number of blades
- $\alpha$: Absolute flow angle [°]
- $\beta$: Relative flow angle [°]
- $\beta_b$: Blade angle [°]
- $\gamma$: Clearance between runner and side walls [m]
- $\delta$: Clearance between runner and floor [m]
- $\eta$: Turbine efficiency ($= P/\rho g Q H$)
- $\theta$: Circumferential angle [°]
- $\rho$: Fluid density [kg/m³]

**Subscripts**

1. Inner circumference of runner
2. Outer circumference of runner
3. Upper stream
4. Lower stream
5. Radial component
6. Circumferential component

**References**

[1] T. S. Reynolds, 1983, “Stronger Than A Hundred Men: A History of the Vertical Water Wheel”, The Johns Hopkins University Press.

[2] Y. Fujiwara, H. Kado and Y. Hosokawa, 1987, “History and Performance of Traditional Water wheel in Western Europe”, Transactions of the Japan Society of Mechanical Engineers, Vol.
A. Furukawa, S. Watanabe, D. Matsushita and K. Okuma, 2010, “Development of ducted Darrieus turbine for low head hydropower utilization”, Current Applied Physics, Vol. 10, pp. 128-132.

Y. Choi, J. Lim, Y. Kim and Y. Lee, 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, Vol. 3, No. 3, pp. 398-409.

J. D. Andrade, C. Curiel, F. Kenyery, O. Aguillón, A. Vásquez and M. Asuaje, 2011, “Numerical Investigation of the Internal Flow in a Banki Turbine”, International Journal of Rotating Machinery, pp. 1-12.

J. Matsui, 2010, “Internal Flow and Performance of the Spiral Water Turbine”, Turbomachinery, Vol. 38, No. 6, pp. 358-364. (in Japanese)

J. Matsui, 2012, “Internal Flow and Performance of the Spiral Water Turbine : 2nd Report : Model Experiment and CFD under Inclined Axis Operation”, Turbomachinery, Vol. 40, No. 2, pp. 91-96. (in Japanese)

C. A. Mockmore and F. Merryfield, 1949, “The Banki Water Turbine”, Bulletin Series, 25.

J. Fukutomi, 2000, “Characteristics and Internal Flow of Cross-Flow Turbine for Micro Hydro Power”, Turbomachinery, Vol. 28, No. 3, pp. 156-163. (in Japanese)

Y. Nishi, T. Inagaki, R. Omiya, T. Tachikawa, M. Kotera and J. Fukutomi, 2011, “Study on a Cross-Flow Water Turbine Installed in an Open Channel”, Turbomachinery, Vol. 39, No. 8, pp. 467-474. (in Japanese)

ANSYS, Inc., 2010, “ANSYS CFX-Solver Theoretical guide”, pp. 137-139. (in Japanese)