IOP Conference Series: Earth and Environmental Science

PAPER • OPEN ACCESS

A study on performance prediction of Darrieus-type hydroturbine operated in open channels using CFD with actuator disk method

To cite this article: Y Suzuki *et al* 2019 *IOP Conf. Ser.: Earth Environ. Sci.* **240** 042020

A study on performance prediction of Darrieus-type hydroturbine operated in open channels using CFD with actuator disk method

Y Suzuki[1], Y Katayama[2], S Watanabe[2], S Tsuda[2] and A Furukawa[3]

[1]Graduate School of Engineering, Kyushu University, 744 Motooka, Nishi-ku, 819-0395 Fukuoka, Japan
[2]Department of Mechanical Engineering, Kyushu University, 744 Motooka, Nishi-ku, 819-0395 Fukuoka, Japan
[3]Oita National College of Technology, 1666 Maki, 870-0152 Oita, Japan

katayama@mech.kyushu-u.ac.jp

Abstract. A performance of Darrieus-type hydroturbine is strongly influenced by the channel and flow condition. These flow conditions are different from place to place and also dependent upon the seasons, therefore it is difficult to study these influences only by experiments. On the other hand, numerical simulation can be adopted for various flow conditions. However, calculation costs are very expensive since fully unsteady simulation taking account of free surface of water should be conducted for this turbine as a cross-flow type. Then, in this paper, a simple numerical model is developed. In this model, instead of solving the complex flow field around the turbine, it is modeled by an actuator disk which imposes the total pressure difference consumed by the rotating turbine. Our previous study suggested that the head coefficient defined as the total pressure difference across the runner normalized by the dynamic pressure with area averaged flow velocity into the turbine seemed to well represent the specific performance of Darrieus-type hydroturbine. In this paper, the specific performance is determined from the experiment in one channel, and the corresponding total pressure change is locally applied to the actuator disk as a function of local inflow velocity. The predicted overall head coefficient, which is defined as the total pressure difference between far upstream and downstream normalized by area averaged velocity downstream of the turbine, is compared with experiment. As a result, when the flow velocity or depth decreases, the overall head coefficient increases. The proposed model can qualitatively reflect this influence of flow velocity and depth on the turbine performance in most cases, while quantitatively the predicted overall head coefficient is different from that in the experiments, indicating the necessity of further modification of the model for quantitative prediction.

1. Introduction

Hydropower is one of the renewable energies and its effective utilization is highly expected in order to realize sustainable low carbon society. Recently, it is difficult to construct new hydropower station with large power because of few available locations and conservation of environment. On the other hand, the extra-low head hydropower less than 2m is almost undeveloped. There are a lot of available sites in the extra-low head condition such as agricultural waterways and small rivers. The extra-low head hydropower is expected to be utilized as local energy source because a lot of these sites can be found near urban area. However, the generated power in extra-low head sites is generally very small. So, more output power of hydroturbine with lowered production cost should be achieved to realize the effective utilization of the extra-low head hydropower.

The Darrieus-type hydroturbine is one of the high speed cross-flow turbines. It has a simple structure, which consists of simple two-dimensional blades aligned along a pitch circle around the rotating shaft. The Darrieus runner is driven by lift force generated by the blades, therefore it yields high efficiency with high rotational speed against the flow velocity [1]. In our previous study, the optimum design of the Darrieus-type runner, the blade profiles, the number of blades and the blade attitude against the pitch circle, have been clarified for confined configuration [2]. The performance of Darrieus-type hydroturbine is strongly influenced by the channel and flow condition, for example the channel width, the depth of water flow, the flow speed and so on. These flow conditions are different from place to place and also dependent upon the seasons, therefore it is difficult to study these influences by only experiments. On the other hand, numerical simulation (CFD) can be adopted for various flow conditions. However, calculation costs are very expensive since fully unsteady simulation taking account of free surface of water should be made for this turbine as a cross-flow type. Then, in this paper, a simple numerical model is developed. In this model, instead of solving the complex flow field around the turbine, it is modeled by an actuator disk which imposes the total pressure difference consumed by the rotating turbine in a certain operating condition. There are several studies which express turbine as actuator disk. Whelan et al. theoretically predicts the performance of a horizontal axis turbine placed in the tidal current flow [3]. Sun et al. investigate the effect of free surface close to the wake profile of actuator disk modeled for a horizontal axis turbine placed in the open channel [4]. Our previous study suggested that the head coefficient defined as the total pressure difference across the runner normalized by the dynamic pressure with area-averaged flow velocity into the turbine seemed to well represent the specific performance of Darrieus-type hydroturbine [5]. In this paper, the specific performance is determined from the experiment in one channel, and the corresponding total pressure change is locally applied to the actuator disk as a function of local inflow velocity. The predicted overall head coefficient, which is defined as the total pressure difference between far upstream and downstream normalized by area-averaged velocity downstream of the turbine, is compared with experiment.

2. Experimental and numerical procedures

2.1. Experimental apparatus

Figure 1 shows the geometry of a tested Darrieus-type runner. The runner has five blades (Z=5) of NACA0018 profile whose chord length is l=45mm and span length is B=300mm. The blades are tangentially set along the runner pitch circle with the diameter of D=300mm. The runner is installed in a rectangular open water channel as shown in Figure 2, whose width is W=1200mm (=4D). In experiments, the upstream and downstream water levels h_u, and h_d, the angular velocity of the runner ω and the generated torque T are measured with the both flow rate Q and downstream water level h_d kept constant. The area-averaged velocity in the channel V_d is separately used as a reference velocity. From these measured values, the total head difference between upstream and downstream H=H_u-H_d is calculated from the deference of water level with that of the dynamic head based on the area-averaged flow velocity (H_u=h_u+$V_u^2/2g$, H_d=h_d+$V_d^2/2g$) where the cross-sectional averaged flow velocities are V_u (=Q/h_uW) and V_d (=Q/h_dW). The downstream water level is set to be higher than the blade height by a weir installed downstream. The turbine performance is expressed by the dimensionless numbers such as the head coefficients C_h, the power coefficients C_p, the turbine efficiency η defined as follows.

$$C_h = \frac{H}{V_d^2/2g} \quad (1), \quad C_p = \frac{T\omega}{\rho B D V_d^3/2} \quad (2), \quad \eta = \frac{T\omega}{\rho g Q H} \quad (3)$$

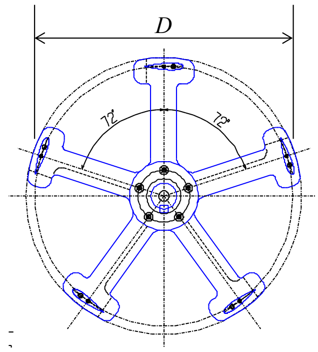

Figure 1. Geometry of Darrieus-type runner.

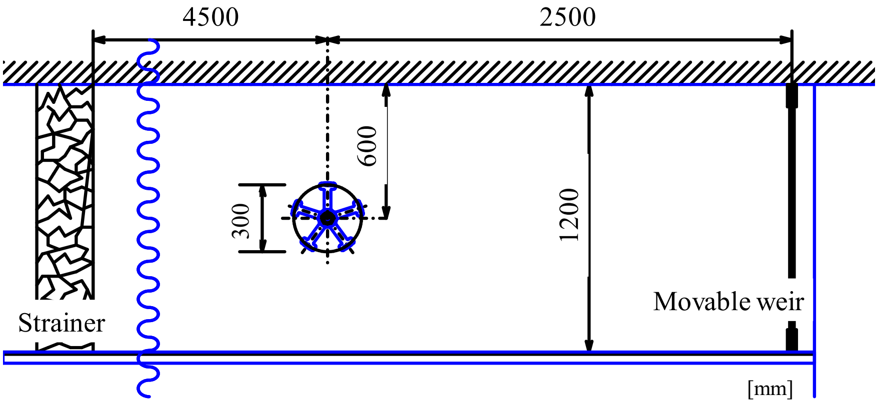

Figure 2. Schematic view of tested open water channel.

2.2. Numerical simulation setup

Since the open channel flow has free surface, analysis of air-water two-phase flow is necessary. The numerical simulations are carried out by using a commercial CFD code, ANSYS-CFX 14.0, 15.0 or 16.2. Figure 3 shows the numerical model. The model size is determined based on the experimental equipment as mentioned before. Total grid number of the model is approximately 1.87 million. The standard k-ε model is employed for the turbulence model. In this study, Darrieus runner is expressed by an actuator disk to simplify the calculation. The position of the actuator disk is set at the center of the runner shaft in the actual installation, and the inlet and outlet boundary are set 4.5m upstream and 2.5m downstream from the actuator disk respectively. These distances are in accordance with the experimental water channel. There is a clearance of 20mm between the actuator disk and the bottom of the water channel, which is also the same as in the experiment. The boundary conditions applied in this analysis are summarized in Table 1.

In this analysis, steady state analysis is performed under all conditions. As an initial condition, the hydrostatic pressure distribution with the uniform velocity is given in the water region of the entire model. The artificial time step is set to 1/10 to 1/100 of the solver estimate. The analysis is carried out for a sufficiently long time until the residual of RMS becomes less than 10^{-3} and the water level is fixed.

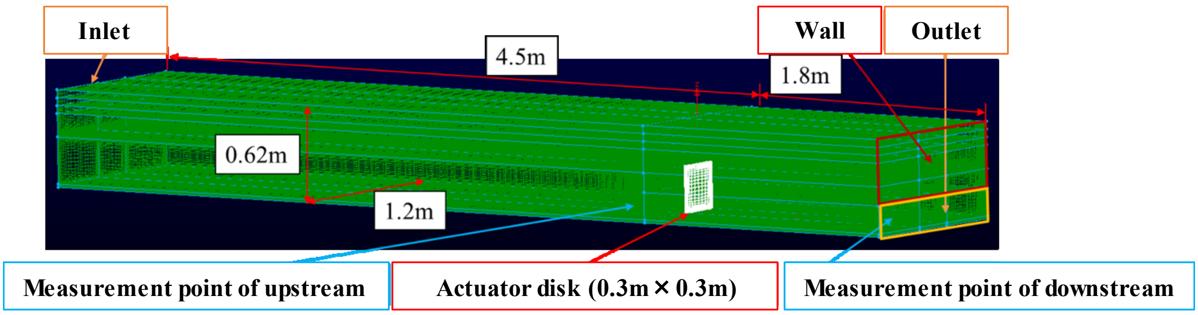

Figure 3. Calculation model.

Table 1. CFD boundary condition.

Inlet	Velocity
Side wall, Bottom wall, Weir	Non-slip
Upper surface of calculation model	Opening condition
Outlet	Hydrostatic pressure distribution

3. Actuator disk

The actuator disk is square-shaped with the width of D=0.3 (equal to the runner diameter) and the height of B=0.3m (the blade span length). At the actuator disk, the pressure difference across the Darrieus runner is imposed. The pressure difference across the runner is given through the head coefficient C_h^* defined as follows;

$$C_h{}^* = \frac{H_u - H_{out}}{V^{*2}/2g} \tag{4}$$

where H_u is the total head upstream of the runner, H_{out} is the total head just exit of the runner, and g is the gravitational acceleration. V^* is the averaged flow velocity into the runner, that is value obtained from calculating the flow rate flowing into the runner Q^* by 2D CFD and dividing by the runner area. This C_h^* is considered to be unaffected by the channel shape if the runner shape is the same. In our previous study [5], C_h^* has been obtained by applying one-dimensional streamtube model to the experimental result, which is shown in Figure 4. From the regression analysis, C_h^* is obtained as a linear function of speed ratio U/V^* as follows;

$$C_h{}^* = A(U/V^*) + B = 3.5(U/V^*) - 3.2 \tag{5}$$

where U is the peripheral speed of runner. From equations (4) and (5), the pressure difference ΔP given by the actuator disk is expressed by the following equation.

$$\Delta P = \rho V_x^{*2}\{A(U/V_x^*) + B\}/2 \tag{6}$$

Although the head coefficient $C_h{}^*$ has been obtained as an area-averaged value, it is applied locally at the actuator disk by referring the local velocity V_x^*. Since the values of A=3.5 and B=-3.2 are found to end in in-sufficient prediction as shown later, the sensitivity of prediction against A and B is also examined.

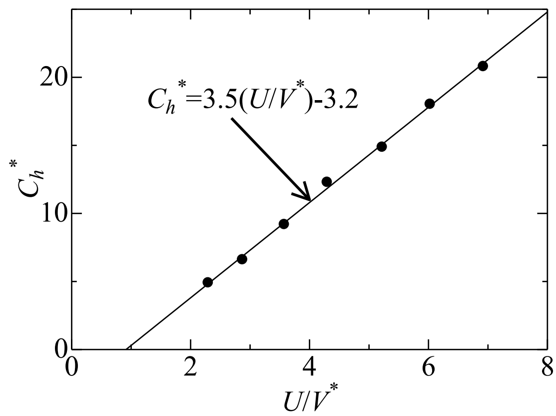

Figure 4. Head coefficient C_h^* [5].

4. Numerical simulation and experimental results

Figure 5 shows a typical example of the CFD result of the flow field by using the actuator disk. The grey surface shows the free surface. As for the upstream and downstream water level measurement positions in the CFD, the upstream side is taken at 1.5D and the downstream side is taken at 4.5D from the actuator disk as similar to the experiment. Experimental and predicted results are compared by the head coefficient C_h. In CFD, the water levels h_u (=$P_u/\rho g$) and h_d (=$P_d/\rho g$) are calculated from the static pressures P_u and P_d at the bottom of water level measurement points upstream and downstream. The total head difference $H=H_u-H_d$ is calculated by considering the dynamic head difference of cross-sectional area-averaged velocity into the water level difference, and C_h is calculated by equation (1). Figure 6 shows the comparison of C_h between the experiment and the prediction. The predicted results show the similar tendency to the experimental ones in terms of dependency on U/V_d, but the predicted C_h shows apparently larger value. The reason of this quantitative discrepancy is supposed to (1) assumption of uniform flow velocity at the outlet of the Darrieus runner in the derivation of C_h* (i.e. A and B) using one-dimensional stream-tube model and (2) thin actuator disk assumption against the rotating runner. However, since the qualitative agreement is achieved, it is conceivable that there exists an appropriate combination of A and B giving better quantitative agreement. Therefore, the influence of A and B on predicted C_h is investigated as shown in Figure 7. It is seen that for the water level of h_d/D=1.33 to 1.43, A=2.0 and B=-1.9 gives a good prediction, while for h_d/D=1.23, A=2.0 and B=-0.9 gives better. We do not know the exact reason for the dependency of appropriate combination of A and B on the water level, but one possible reason is the existence of free surface closer to the runner for h_d/D=1.23, with which an interaction between free surface and individual rotating blades is in-negligible. This is not considered in the prediction model.

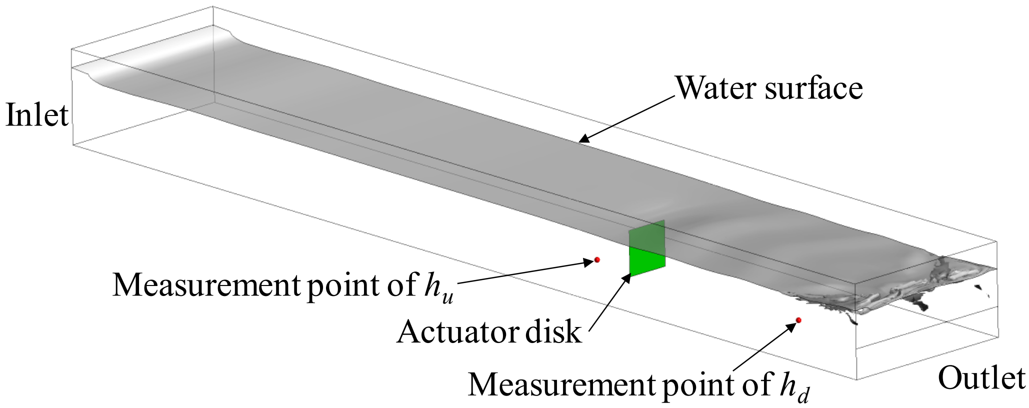

Figure 5. Typical example of simulated free surface (A=2.0, B=-1.9, h_d/D=1.40).

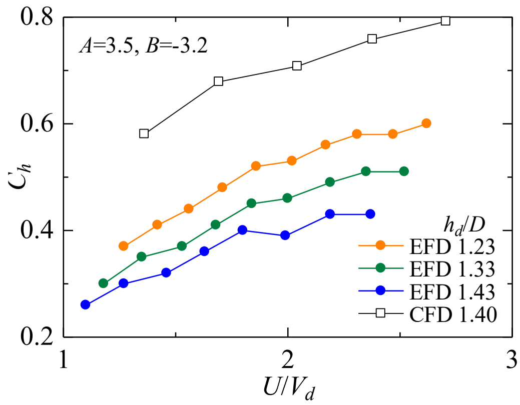

Figure 6. Comparison of experimental and simulated values of head coefficients C_h.

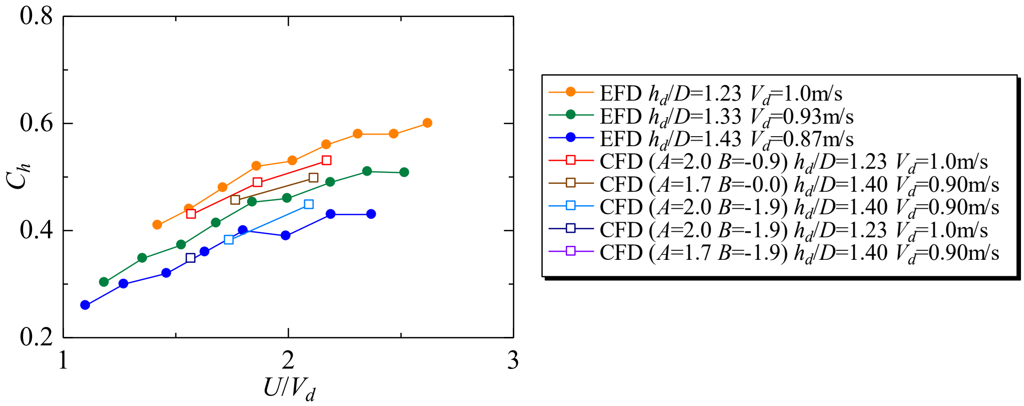

Figure 7. Comparison of head coefficient with various combinations of A and B in equation (6).

4.1. *Influence of water levels on experimental and predicted performances*

The predicted and experimental results of C_h for the different water levels h_d with a constant downstream velocity V_d (V_d=0.81m/s for prediction and V_d=0.87m/s for experiment) are shown in Figures 8 and 9 respectively. The values of A and B used in the prediction are 2.0 and -0.9 respectively. In the both results, C_h decreases as the water level becomes higher, indicating that this analysis could qualitatively reproduce the experiment result.

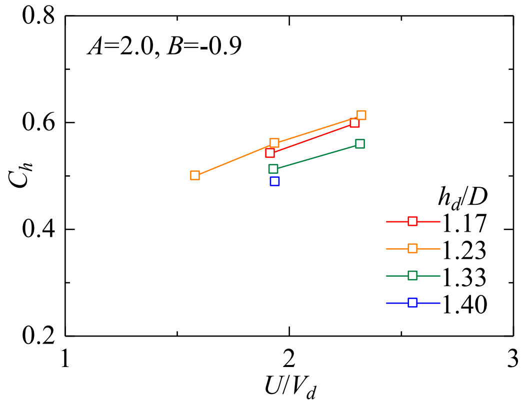

Figure 8. Relationship between speed ratio U/V_d and head coefficient C_h by CFD (V_d=0.81m/s).

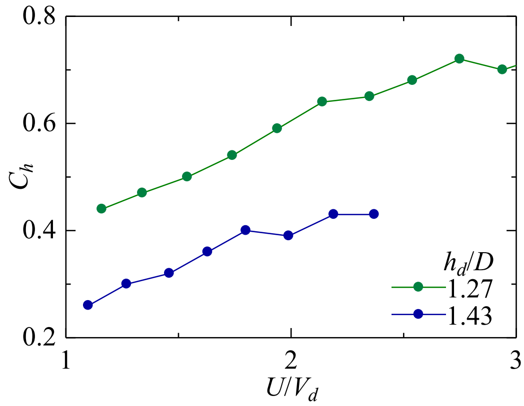

Figure 9. Relationship between speed ratio U/V_d and head coefficient C_h by experiment (V_d=0.87m/s).

4.2. *Influence of downstream velocity on experimental and predicted performances*

Figures 10 and 11 respectively show the predicted and experimental results under the different downstream velocity V_d with a constant water level (h_d/D=1.23 for prediction and h_d/D=1.33 for experiment). The values of A and B used in the prediction are 2.0 and -0.9. In Figure 10, the head coefficient C_h monotonously decreases as V_d increases. On the other hand, V_d=0.81m/s gives the highest value of C_h in the prediction as shown in Figure 9. The CFD result for velocity range of V_d>0.81m/s shows good agreement with that by experiment.

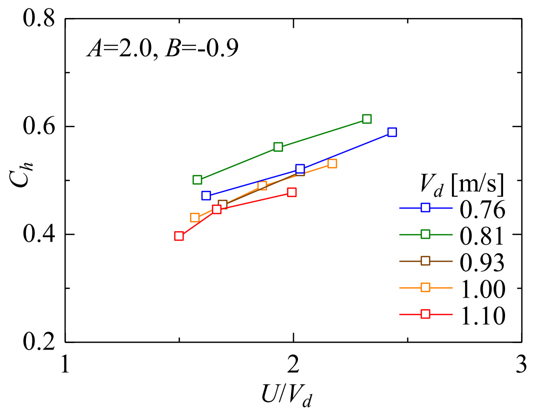

Figure 10. Relationship between water level h_d/D and head coefficient C_h by CFD (h_d/D=1.23).

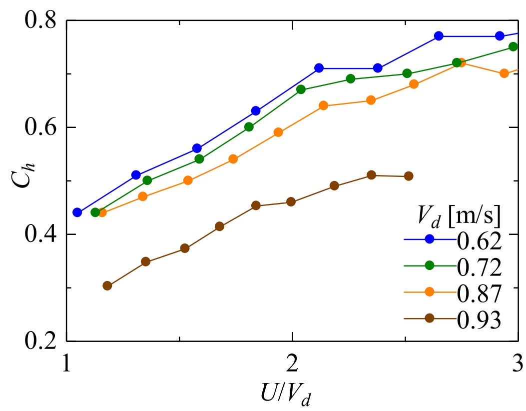

Figure 11. Relationship between water level h_d/D and head coefficient C_h by experiment (h_d/D=1.33).

5. Conclusion

In this study, the performance prediction method of the Darrieus-type hydroturbine installed in the open channel is proposed by combining a three dimensional CFD with actuator disk method. The turbine part is replaced with the actuator disk, which enables a simpler CFD with less computational cost. The results are compared with the experimental ones. The main results are summarized as follows.

(1) With the head coefficient of runner C_h^* assumed to be a function of speed ratio, qualitative agreement of performance with the experiment can be achieved. However, for quantitative agreement, tuning of parameters for C_h^* should be conducted.
(2) In the case of the same flow velocity, the head coefficient between upstream and downstream C_h becomes lower as the water level becomes higher. This qualitative tendency is reproduced by the proposed prediction method.
(3) In the case of the same water level, C_h becomes lower as the flow velocity increases. This tendency is partially reproduced by the proposed prediction method, while in the low flow velocity case, the discrepancy between the prediction and experiment is observed.

Since the proposed prediction method qualitatively reproduces the experimental results, the way of giving the pressure difference across the actuator disk by C_h^* seems to be appropriate. However, in order to realize quantitative prediction, it is considered necessary to add some elements of flow conditions to the model equation of C_h^*.

References

[1] Paraschivoiu I 2002 *Wind Turbine Design with Emphasis on Darrieus Concept* (Polytechnic International Press) pp 26-29
[2] Furukawa A, Watanabe S, Shimokawa K, Okuma K, Takenouchi K 2010 *Proc. 5th Int'l Conf. of Cooling and Heating Tech.* Bandung Indonnesia pp 1-8
[3] Whelan J I, Graham J M R, Peiro J 2009 *Journal of Fluid Mechanics* **64** pp 281-291
[4] Sun X, Chick J P, Bryden I G 2008 *Renewable Energy* **33** pp 1267-1274
[5] Nakashima K, Watanabe S, Matsushita D, Tsuda S, Furukawa A 2016 *Proc. the IAHR Conf. 2016*, IAHR2016-229