Performance Prediction of Darrieus-Type Hydroturbine with Inlet Nozzle Operated in Open Water Channels

K. Nakashima\(^a\), S. Watanabe\(^b\), D. Matsushita\(^c\), S. Tsuda\(^b\) and A. Furukawa\(^d\)

\(^a\)Graduate School of Engineering, Kyushu University, 744 Motooka, Nishi-ku, 819-0395 Fukuoka, Japan
\(^b\)Department of Mechanical Engineering, Kyushu University, 744 Motooka, Nishi-ku, 819-0395 Fukuoka, Japan
\(^c\)Department of Mechanical Engineering, Kyushu Sangyo University, 2-3-1 Matsukadai, Higashi-ku, 813-8503 Fukuoka, Japan
\(^d\)Oita National College of Technology, 1666 Maki, 870-0152 Oita, Japan

fmnabe@mech.kyushu-u.ac.jp

Abstract. Small hydropower is one of the renewable energies and is expected to be effectively used for local supply of electricity. We have developed Darrieus-type hydro-turbine systems, and among them, the Darrieus-turbine with a weir and a nozzle installed upstream of turbine is, so far, in success to obtain more output power by gathering all water into the turbine. However, there can several cases exist, in which installing the weir covering all the flow channel width is unrealistic, and in such cases, the turbine should be put alone in open channels without upstream weir. Since the output power is very small in such a utilization of small hydropower, it is important to derive more power for the cost reduction. In the present study, we parametrically investigate the preferable shape of the inlet nozzle for the Darrieus-type hydroturbine operated in an open flow channel. Experimental investigation is carried out in the open channel in our lab. Tested inlet nozzles are composed of two flat plates with the various nozzle converging angles and nozzle outlet (runner inlet) widths with the nozzle inlet width kept constant. As a result, the turbine with the nozzles having large converging angle and wide outlet width generates higher power. Two-dimensional unsteady numerical simulation is also carried out to qualitatively understand the flow mechanism leading to the better performance of turbine. Since the depth, the width and the flow rate in the real open flow channels are different from place to place and, in some cases from time to time, it is also important to predict the onsite performance of the hydroturbine from the lab experiment at planning stage. One-dimensional stream-tube model is developed for this purpose, in which the Darrieus-type hydroturbine with the inlet nozzle is considered as an actuator-disk modelled based on our experimental and numerical results.

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 exploit hydropower station for large power because of few available locations and conservation of environment. However, the extra-low head hydropower less than 2m is almost undevloped. 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. The Darrieus turbine 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 [2]. In addition, it is found that the generated torque can be increased by putting the inlet nozzle in front of the Darrieus runner [3, 4], and we have developed the Darrieus-turbine with a weir and a nozzle installed upstream of turbine. However, in cases of rivers and water channels with wide width, installing the weir covering all over the flow channel width is unrealistic. Then, the performance of Darrieus turbine putted alone in an open but still narrow channel flow have been investigated [5], and it has been found that the generated shaft power can be significantly increased by installing the inlet nozzle. The installation of the nozzle is not aiming at increasing the net flow rate into the runner as realized by several channeling devices proposed by past literatures [6]. It is intended to concentrate, by such a structurally simple device, the flow toward the central part of runner inlet, where the torque generated by the blade is large. It seems that the power can be further increased with the more appropriate design of the inlet nozzle.

In the present study, to improve the performance of Darrieus-type hydroturbine operated in open water channels, the preferable shape of inlet nozzle is experimentally investigated in a parametric manner. The flow mechanism leading to the performance improvement is studied by two-dimensional numerical simulations. Then, since there are various flow conditions such as water depth, channel width and flow rate in real open channel flows, it is very important to predict the onsite performance of the hydroturbine from lab experiment at the planning stage. Therefore, a simple one-dimensional stream-tube model is proposed for this purpose, in which the Darrieus-type hydroturbine with the inlet nozzle is considered as an actuator-disk model. Finally, based on our experimental and numerical results obtained in our open flow channel, the prediction of hydroturbine performance in other channels with the wide range of channel width is attempted.

2. Experimental and numerical procedures

2.1. Experimental apparatus

Figure 1 shows geometries of a tested Darrieus-type runner and an inlet nozzle. The runner has five blades ($Z=5$) of NACA0018 profile considering the self-stating characteristics and the reduction of torque ripples. The chord and span lengths are $l=45$mm and $B=300$mm respectively. The blades are tangentially set along the runner pitch circle with the diameter of $D=300$mm. The runner is installed in a rectangular open water channel as shown Fig. 2, whose width is $W=1.200$mm ($=4D$). In experiments, the upstream and downstream water levels $h_{up}$, and $h_{down}$, the angular velocity of the runner $\omega$ and the generated torque $T$ are measured with the flow rate $Q$ and downstream water level $h_{down}$ kept constant. From these measured values, the total head difference between upstream and downstream $H$ is calculated on the specific energy basis considering the dynamic head based on the area-averaged flow velocity. The downstream water level is set to be sufficiently higher than the blade height by a weir. The turbine performance is expressed by the dimensionless numbers such as the head coefficients $C_h$ and $C'_h$, the power coefficients $C_p$ and $C'_p$, the turbine efficiency $\eta$ defined as follows

\[ C_h = \frac{H}{V_d^2/2g} \]
\[ C'_h = \frac{H}{V_u^2/2g} \]
\[ C_p = \frac{T\omega}{\rho BW_{in} V_d^3/2} \]
\[ C'_p = \frac{T\omega}{\rho BS_{in} V_u^3/2} \]
\[ \eta = \frac{T\omega}{\rho g Q H} \]
where $V_d = Q/Wh_d$, $V_n = Q_n/S_{in}B$, $g$ and $\rho$ respectively denote the area-averaged downstream velocity, the area-averaged velocity at outlet of the inlet nozzle, the gravitational acceleration and the water density. $W_{in}$ and $S_{in}$ are the nozzle inlet and outlet widths respectively. The averaged velocities $V_d$ and $V_n$ (hereafter called downstream and nozzle velocities) are separately used as the reference velocities for $C_h$ and $C_p$, $C'_h$ and $C'_p$ respectively. Since $Q_n$, that is the flow rate through the nozzle into the runner, cannot be measured directly, it is estimated from the measured local velocity $V_u$ at the point located 225mm (=0.75$D$) upstream from the center of the runner (see Fig. 2) using a portable propeller type velocity meter, as described later.

In the present study, the inlet nozzles called straight line nozzle (SL nozzle) which consists of two flat plates are tested. In order to derive more generated power, the effects of nozzle geometry on the turbine performance are investigated parametrically by experiments and numerical simulations. At first, the effects of the nozzle converging angle $\phi$ on the turbine performance is investigated. Three SL nozzles are tested, which have different nozzle converging angles ($\phi=30$, 45 and 60deg s.) with the identical nozzle inlet and outlet widths of $W_{in}=1.8D$ and $S_{in}=0.8D$. After that the effect of the nozzle outlet width $S_{in}$ on the turbine performance is also investigated using three SL nozzles which have different nozzle outlet widths ($S_{in}=0.7D$, 0.8$D$ and 0.9$D$) with the identical inlet width and converging angle of $W_{in}=1.8D$ and $\phi=45$deg ees.

Through this study, the total water flow rate $Q$ and the downstream depth of water $h_{down}$ are set as $Q =450L/s$ and $h_{down}=430mm$, and the runner is operated under the water surface. The Froude number is calculated as $Fr = V_d/(g h_{down})^{1/2} =0.46$, indicating that the flow is kept in the sub-critical flow regime.

2.2. Numerical simulation setup

Since the Darrieus runner is a crossflow type, the unsteady simulation is necessary. For simplicity, two-dimensional unsteady numerical simulations are carried out by using a commercial CFD code, ANSYS-CFX 14.0 or 15.0. Figure 3 shows the two-dimensional numerical analytical model. It is difficult to understand the flow mechanism around the runner only using open channel model (Fig. 3(a)) because the flow bypassing out of the nozzle differs every moment [7]. Then in the present study, the numerical
simulations using a closed channel model, in which the upstream weir is installed to prevent the bypassing flow as shown in Fig.3 (b), are also carried out. Total grid numbers of the model are approximately 380K for the open channel model and 450K for the closed channel model, while, in some case, the dense grid with 1,350K nodes is used for the open channel model. The standard k-ε model is employed for the turbulence model. The uniform velocity of 1m/s for the open channel model while 0.2m/s for the closed channel model is given at the inlet boundary. The opening boundary condition with constant static pressure is applied for the outlet boundary. The wall boundaries, runner and nozzle surface are treated as non-slip walls. In the present study, the unsteady calculations are carried out until almost the stable periodic behaviors of the torque performance are obtained. The time step is set equal to the time duration of runner rotation of 1 deg. The turbine performance is evaluated in dimensionless forms such as the head coefficients $C_{hc}$ and $C'_{hc}$, the power coefficients $C_{pc}$ and $C'_{pc}$, and the turbine efficiency $\eta$, which are defined as

$$C_{hc} = \frac{(P_{in} - P_{out})}{\rho V_n^2} / 2$$  \hspace{1cm} (6)$$

$$C_{pc} = \frac{T\omega}{\rho BW_n V_n^3} / 2$$  \hspace{1cm} (7)$$

$$C'_{hc} = \frac{(P_{in} - P_{out})}{\rho V_n^2} / 2$$  \hspace{1cm} (8)$$

$$C'_{pc} = \frac{T\omega}{\rho BS_n V_n^3} / 2$$  \hspace{1cm} (9)$$

where $P_{in}$ and $P_{out}$ are mass-flow-averaged total pressures at inlet and outlet boundaries respectively. In the present simulations, the flow rate $Q_n$ can be calculated easily at the section indicated in Fig. 3 (c), and then, the averaged velocity at the outlet of inlet nozzle $V_n$ referred as nozzle velocity is calculated as $V_n=Q_n/BS_{in}$.

3. Parametrical investigation of nozzle shape

3.1. Experimental and numerical simulation results

3.1.1. Effect of nozzle converging angle. Figure 4 shows the turbine performances with SL nozzles having the converging angle $\phi$ of 30, 45 and 60 degs. The horizontal axis is the speed ratio $U/V_d$, where $U$ is the peripheral speed of runner. The vertical axes are the head and power coefficients per blade, $C_b/Z$ and $C_p/Z$, and the turbine efficiency $\eta$. In the experiments, although $C_b$ takes similar values regardless of the nozzle converging angle of $\phi$, $C_p$ with larger angles of $\phi=45$ and 60 degs is slightly higher than that with $\phi=30$ degs. Then, the turbine efficiency $\eta$ is higher for $\phi=45$ and 60 degs than that for $\phi=30$ degs. From these results, larger nozzle converging angle is effective to increase the generated power.

Results of numerical simulation using the open channel model represented by double symbols show the similar tendency to the experimental results.

![Figure 4](image_url)

**Figure 4.** Turbine performance with SL nozzles for various nozzle converging angles.
3.1.2. Effect of nozzle outlet width. The results with SL nozzles having various nozzle outlet widths $S_{in}$ of 0.7$D$, 0.8$D$ and 0.9$D$ and constant converging angle of $\phi=45$ degress are shown in Figure 5. In experiments, the runner with $S_{in}=0.9D$ shows slightly lower head coefficient $C_h$ than the other two nozzles. The power coefficient $C_p$ increases with widening $S_{in}$. As a result, the turbine with SL nozzle having $S_{in}=0.9D$ yields higher efficiency $\eta$ than the others. The results of numerical simulations roughly show the similar tendency to the experiments.

![Figure 5. Turbine performance with SL nozzles for various nozzle outlet width.](image)

3.2. Discussions

Figure 6 shows the torque coefficient variation generated by one of the five blades with SL nozzle ($\phi=30$ degress, $S_{in}=0.8D$) obtained by numerical simulation at $U/V_d=2.827$, where the torque coefficient $C_{tt}$ is defined by $C_{tt}=T/t/(pQV_dD/2)$. $T_t$ is the torque generated by one of the five blades. A red curve represents the averaged curve of the last 10 runner revolutions and ten black curves the instantaneous torque variations. Firstly, it is noticed that the majority of the torque is generated when a blade locates within the nozzle passage region ($\theta=42$–138 degress). This large torque is an essential one for the torque characteristics of the Darrieus hyroturbine. Focusing on the instantaneous torque, it is found that the generated torque differs every runner revolution. It is seemed that the unsteadiness of the flow outside the runner and the fluctuation of the nozzle flow rate $Q_n$ cause the fluctuation of the generated torque $T_t$ [7]. This fluctuation prevents us from precisely understanding the flow mechanism around runner by observing the instantaneous flow field.

![Figure 6. Instantaneous and averaged torque variation obtained by numerical simulation.](image)
Now, it can be noticed that the power coefficient $C_p$ is reduced to the product of three terms as:

$$C_p = \frac{T\omega}{\rho BW_n V_d^3/2} = C'_p \left( \frac{V_n}{V_d} \right)^3 \left( \frac{S_{in}}{W_{in}} \right)$$  \hspace{1cm} (11)

The first term is the power coefficient $C'_p$, based on the nozzle velocity $V_n$, which is considered to represent the pure power characteristic of the turbine. The second term is the nozzle velocity ratio $V_n/V_d$, which represents the water collection ability of the nozzle. The third term is the ratio of the nozzle outlet width to inlet width $S_{in}/W_{in}$. Figure 7 shows (a) $C'_p$, the first term of (11), and (b) the product of the second and third terms of (11) calculated from results of numerical simulation for the open channel model. The products of the second and third term of (11) take similar values regardless of the nozzle shapes, meaning that the water collection and the acceleration of water are almost the same with each SL nozzles. This result indicates that the increase of power coefficient $C_p$ with the appropriate nozzle design is achieved by the improvement of the pure performance of hydroturbine expressed by the power coefficient $C'_p$. As mentioned above, the torque performance is characterized by the generated torque within the nozzle passage region ($\theta=42\sim138$ degs). In order to understand the flow mechanism within the nozzle excluding the effect of the outside flow, numerical simulations using the closed channel model with the weir installed upstream are carried out. Figure 8 shows (a) the torque coefficient $C^\ast=T/(\rho B S_{in} V_n^2 D/2)$ and (b) the power coefficient $C'_p$ per blade obtained by numerical simulations using the closed channel model. It can be seen that $C'_p$ shows similar tendency to that in the open channel model (Fig. 7(a)), indicating the validity of this approach. The pure power coefficient $C'_p$ increases with increasing the nozzle converging angle $\phi$ up to 60 degs.

![Figures 7 and 8](https://example.com/figures.png)

**Figure 7.** Power coefficient $C_p$

**Figure 8.** Torque coefficient and power coefficient in numerical simulation with closed channel.

![Figure 9](https://example.com/figure9.png)

**Figure 9.** Instantaneous torque variation obtained by the closed channel model.
Figure 9 shows the instantaneous torque coefficient generated by one of the five blades $C'_t$ at the speed ratio of $U/V_n=4.0$ with the nozzle converging angles of $\phi=30$, 45 and 60degs. Focusing on the generated torque within the nozzle region ($\theta=42$–138degs), three major differences (i)-(iii) can be found. From $\theta=40$–80degs denoted as (i), the instantaneous torque with $\phi=60$degs is larger than those with $\phi=30$ and 45degs. On the other hand, from $\theta=80$–120degrees denoted as (ii), the instantaneous torque with $\phi=30$degs is larger than those with $\phi=45$ and 60degs. Figures 10(a), (b) and (c) show the instantaneous pressure and streamline (absolute in nozzle but relative in runner regions) distributions when a blade locates at $\theta=45$, 115 and 130degs respectively. Looking at the absolute streamlines in Figs. 10(a) and (b), it can be found that the flow is well guided by the nozzle walls in each nozzle. Then, it seems that the attack angle to the blade is larger with the converging angle of $\phi=60$degs at the region (i) while it is larger with $\phi=30$degs at the region (ii), leading to the different tendency of the torque characteristics at regions (i) and (ii). At $\theta=130$degs denoted by (iii), the large torque peak appears in the case with the converging angles of $\phi=45$ and 60degs. In Fig. 10(c), we can easily notice that the high pressure region is wide especially in those two cases, resulting in the large lift force, i.e. the large torque at $\theta=130$degs. Based on the above observations, the preferable flow direction at the region (i) and the pressure rise

Figure 10. Instantaneous pressure distribution and streamlines around blade at $U/V_n=4.0$.  
owing to the interaction between a blade and nozzle edge at the region (iii) are responsible for the high turbine performance with the nozzle converging angle of $\phi=60\,\text{degs}$.

In terms of the effect of nozzle outlet width $S_{in}$, the pure power coefficient $C'_{p}$ is larger with the large outlet width of $S_{in}=0.9D$ as shown in Fig.8, which is responsible for the larger power coefficient $C_p$ as can be found from Eq.(11). Looking at the torque coefficient $C'_{t}$, it can be seen that the maximum generated torque is similar for $S_{in}=0.8D$ and $0.9D$, but the turbine with $S_{in}=0.9D$ shows the maximum torque coefficient at the higher speed ratio region than that with $S_{in}=0.8D$. Since the shaft power is calculated by the torque multiplied by the runner angular speed, the power coefficients $C'_{p}$ and $C_p$ become larger for the larger nozzle outlet width.

4. Performance prediction of Darrieus turbine installed in open water channel

In the planning stage of the installation of hydroturbine, it is important to predict the onsite output power generated by the turbine. If the power coefficient $C_p$ is known for the site, the output power can be calculated using the area-averaged velocity at the site as the downstream velocity $V_d$. The power coefficient $C_p$ is related to the pure performance of the turbine $C'_{p}$ through Eq.(11), but $C'_{p}$ cannot be easily obtained by experiments because of the difficulty in the measurement of the nozzle velocity $V_n$. Then, we are now proposing the estimate method of the velocity $V_n$ from the local velocity $V_u$ and that of $C'_{p}$ (the first term in Eq.(11)), next the prediction method of velocity ratio $V_n/V_u$, that is the second term of Eq. (11) is to be derived, and finally the prediction method of the performance of Darrieus turbine from the one operated in an open water channel to that in another is to be proposed.

4.1. Estimation of pure power coefficient of turbine $C'_{p}$

In order to calculate the power coefficient $C'_{p}$, the nozzle velocity $V_n$ (area-averaged velocity at nozzle exit) has to be specified. Since $V_n$ is hardly measured by the experiment, it is now estimated by the combined method of experiment and numerical simulation. In experiments, the local velocity $V_u$ is measured at $0.75D$ upstream of the runner center (see Fig.2). In numerical simulation, both $V_n$ and $V_u$ can be calculated, then the ratio of the nozzle velocity to the local velocity $V_n/V_u$ can be obtained and shown in Fig. 11. By using this velocity ratio $V_n/V_u=1.348$, we can estimate the nozzle velocity $V_n$ in experiments by multiplying it to the measured local velocity $V_u$. Figure 12 shows the pure power coefficient $C'_{p}$, that is the power normalized by the dynamic pressure based on the estimated nozzle velocity as well as the power coefficient $C_p$ for two different flow rates $Q$ and two different downstream water depths $h_d$. From Fig. 12, the large discrepancy between experiments and numerical simulations can be seen for $C_p$, whereas the both experimental and numerical results roughly collapse onto one curve for $C'_{p}$. Then, $C'_{p}$ seems to be suitable to represent the pure performance as we expected, which can be used for the prediction of onsite performance of turbine.
4.2. Velocity ratio $V_d/V_t$

The velocity ratio $V_d/V_t$ depends on the channel width, since the flow avoids to enter the turbine more easily as the channel width becomes wider. Therefore, $V_d/V_t$ must be accurately predicted for the prediction of the onsite turbine performance. For this purpose, we propose a one-dimensional stream-tube model shown in Fig. 13. The Darrieus hydroturbine is modelled by an actuator disk with the head coefficient defined by $C_h=2g(H_{in}-H_{out})/V_d^2$, where $H_{in}$ is the total head at outlet of hydroturbine. Therefore, the head consumed only by the turbine is considered in this head coefficient. The water flows into the hydroturbine with the area of $sBD$ and discharges from the turbine through the area of $k_1BD$, where $s$ and $k_1$ mean the contraction ratio of nozzle (i.e. $S_{in}=sD$) and the effective outlet area factor of the turbine respectively. The flow which does not enter the turbine but flows outside of the turbine is contracted due to the blockage of the nozzle, then the effective area for outside flow is calculated as $Wh^*-k_2BW_{in}$, where $k_2$ means the blockage factor of nozzle. Along the streamline (a) from upstream (water depth: $h_{up}$, total head: $H_{up}$) to the outside of the turbine and streamline (b) from upstream to the turbine outlet, the Bernoulli equation is separately applied. The consumed head by the turbine is considered for the streamline (b). In the control volume (c), the momentum conservation law is applied. Combining these three equations, the velocity ratio $V_d/V_t$ can be calculated. In the present study, the effective nozzle outlet factor $k_1$ is fixed as 1.4, which is determined by referring to the downstream velocity distribution measured in the experiments. The calculated speed ratio $V_d/V_t$ for the various blockage factors $k_2$ is shown in Fig. 14. The velocity ratio varies significantly with the change of $k_2$. This result indicates that the appropriate estimation of the blockage factor $k_2$ is essential for more accurate prediction of the onsite performance of the Darrieus hydroturbine.

![Figure 13 Schematic view of our analysis model.](image)

![Figure 14 Calculated velocity ratio $V_d/V_t$ for various blockage expansion factor $k_2$.](image)

4.3. Prediction of power coefficient $C_p$ in various channel widths

Here, the prediction of the power coefficient $C_p$ in various channel widths is attempted by the one-dimensional stream-tube model. The blockage factor $k_2$ has been decided to meet the good agreement between the calculated velocity ratio and the experimental one as shown by the red broken curve in Fig. 14. The pure power coefficient $C_p'$ and the head coefficient $C_h'$ used for the prediction are shown in Fig. 15. Figure 16 shows the predicted power coefficient $C_p$ for various channel width $W/D$. From this figure, it can be found that the slight increase of the width from $W/D=4$ to 5 makes a significant deterioration of $C_p$. This is because the velocity ratio $V_d/V_t$ significantly decreases with the slight increase of the channel width. Wider the channel width, $C_p$ curve seems to converge gradually to the one curve. Since the same values for $k_1$ and $k_2$ are used through this prediction regardless of the channel width, validation of the prediction method is still necessary to be made.
5. Conclusion
In this study, the effects of inlet nozzle geometry on the performance of Darrieus hydroturbine in an open channel have been investigated experimentally and numerically. Then the prediction method of onsite performance of Darrieus hydroturbine has been proposed based on the one-dimensional stream-tube model. The main results are summarized as follows,

1) The large nozzle converging angle is effective for increasing the power generated by the Darrieus runner. It seems that the preferable flow direction and the interaction between the blade and the nozzle outlet edge are possibly responsible for the increase of the power.

2) The nozzle outlet width of 0.9D (D: pitch circle diameter) seems to be the best among 0.7D-0.9D for the open channel flow for larger output power.

3) The proposed one-dimensional stream-tube model can well predict the turbine performance, provided the flow parameters such as the blockage factor due to the turbine are properly given.

4) According to the proposed prediction method, the power coefficient is significantly deteriorated if the turbine is installed in wider open flow channel.

Some of the present simulations have been carried out using the computer facilities at Research Institute for Information Technology, Kyushu University, Japan. We would like to express our sincere gratitude for it.

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 Indonesia pp 1-8
[3] Matsushita D, Okuma K, Watanabe S, Furukawa A 2008 J. Fluid Sci. and Tech. Vol 3 No 3 pp 387-397
[4] Shimokawa K, Furukawa A, Okuma K, Matsushita D, Watanabe S 2012 Renewable Energy Vol 41, pp 376-381
[5] Matsushita D, Moriyama R, Nakashima K, Watanabe S, Okuma K, Furukawa A 2014 IOP Conf. Ser.: Earth Environ. Sci. 22 062007
[6] Khan, M. J., Iqbal, M. T. and Quaicoe, J. E., 2006, Proc. 2006 Canadian Conference on Electrical and Computer Engineering, Ottawa, Ont., pp. 2288-2293.
[7] Nakashima K, Moriyama R, Matsushita D, Watanabe S, Tsuda S, Furukawa A 2015 Proc. the ASME-JSME-KSME Joint Fluids Eng. Conf. 2015, AJKFluids-22491.