Experimental and Numerical Studies on the Possibility of Duct Flow Low-power Generation Using a Butterfly Wind Turbine

Yutaka Hara¹, Shohei Kogo¹, Katsuhiro Takagaki¹, Makoto Kawanishi¹, Takahiro Sumi² and Shigeo Yoshida³

¹Department of Mechanical and Aerospace Engineering, Tottori University
4-101 Koyama-Minami, Tottori, 680-8552, Japan, hara@damp.tottori-u.ac.jp
²Department of Advanced Technology Fusion, Saga University
1 Honjo, Saga, 840-8502, Japan, sumi@me.saga-u.ac.jp
³Research Institute for Applied Mechanics, Kyushu University
6-1 Kasugakoen, Kasuga, Fukuoka, 816-8580, Japan, yoshidas@riam.kyushu-u.ac.jp

Abstract

An objective of this study is to demonstrate the validity of using a small wind turbine to recover the fluid energy flowing out of an exhaust duct for the generation of power. In these experiments, a butterfly wind turbine of a vertical axis type ($D = 0.4$ m) is used. The output performance is measured at various locations relative to the exit of a small wind tunnel ($W = 0.65$ m), representing the performance expected in an exhaust duct flow. Two-dimensional numerical analysis qualitatively agrees with the experimental results for the wind turbine power coefficient and rate of energy recovery. When the turbine is far from the duct exit (more than $2.5D$), an energy recovery rate of approximately $1.3\%$ is obtained.

Keywords: Vertical Axis Wind Turbine, Duct Flow Power Generation, Wind Energy, Butterfly Wind Turbine, Computational Fluid Dynamics

1. Introduction

A potential application for small wind turbines is in power generation from the flow of gases from exhaust ducts. This concept aims to recover the fluid energy flowing uselessly out of the ducts in commercial facilities and industrial factories as well as many other industries. Users need to be careful about the placement of turbine, because the installation of the turbine inside the duct or immediately outside the exit (e.g., [1]-[3]) will greatly increase the duct loss. Another option available to users is to place the wind turbine away from the duct exit (e.g., [4]-[6]), where effective energy recovery from the exhaust is still possible without influencing the flow inside the duct; these would, therefore, be useful in increasing the capacity factor of small wind turbines. The influence of the use of a wind turbine on duct flow has not been sufficiently investigated, however, there are a range of opinions as to the pros and cons of wind turbine usage for duct power generation. Experiments conducted by Sugata et al. [7] investigated the use of a propeller-type horizontal axis wind turbine (HAWT). They found no influence on the inside pressure of a duct if the length between the wind turbine and the duct exit was greater than the duct width. They also found that the electric power generated by the wind turbine could be increased by enlarging the diameter of the wind turbine to 1.75 to 2.0 times the width of the duct. However, the influence on the duct by the existence of the wind turbine was evaluated by the static pressure measured at just one point inside the duct. The investigation of the influence on the duct in detail is necessary to show the validity of duct flow power generation.

In the case of the power generation wherein the duct flow and/or natural wind are utilized to rotate the wind turbine according to the operational conditions of the exhaust system, a vertical axis wind turbine (VAWT) with non-directionality is thought to be a better selection than a HAWT, which needs a yaw control [8]. As far as the authors know, no examples exist of detailed study of the influence of VAWT on a duct given. However, it is not difficult to speculate that the influence of a VAWT on a duct should be asymmetrical against the centerline of the duct flow, since the rotational axis of the turbine is vertical to the main flow.

A main objective of this study is to demonstrate the validity of duct flow power generation, wherein a small VAWT is utilized to recover the fluid energy flowing out of the exit of an exhaust duct. In addition, the relationship between the output power and the location of the wind turbine, which is dependent upon the direction of turbine-rotor revolution, is investigated in this study.
For these experiments, a butterfly wind turbine (BWT) [9], which is an armless VAWT, is used as a test turbine and output performance is measured at various locations relative to the exit of a small wind tunnel, which acted as a model of exhaust duct. Using a BWT has the potential to reduce the costs of wind turbine manufacture and aftercare owing to the presence of fewer components and is therefore a desirable turbine for future power generating applications. The wind speed at the wind tunnel exit is also measured with and without the test turbine, and the energy recovery rate was obtained. However, a generator, or electric generating efficiency, is not considered in the present study. In addition, numerical analysis by two-dimensional computational fluid dynamics (CFD) has been conducted under conditions similar to experimental conditions to validate the experimental results and obtain detailed information about the influence of a wind turbine rotor on the duct. Properly speaking, three-dimensional simulation should have been performed; however, two-dimensional simulation was selected since the 3D-CFD needed very long calculation time and, in this study, many calculations regarding the conditions of the wind turbine arrangement were needed. Furthermore, although there may be possibility of usage of multiple wind turbines, the present study considers only the case of usage of single wind turbine per one exhaust duct due to the easiness of installation and the cost effectiveness.

2. Experimental System and Method

A schematic diagram of the experimental setup used in the present study is shown in Fig. 1. A small Eiffel-type wind tunnel with a square nozzle (contraction ratio: 1.515), which has a side length of \( W = 0.65 \) m, was used as an exhaust duct. The BWT was used in the experiments and comprised three cambered blades made of fiber-reinforced plastics consisting of vinylon fiber. The diameter of the rotor was \( D = 0.4 \) m, radius was \( R = 0.2 \) m, and height of the rotor was \( H = 0.3 \) m. The swept area of the wind turbine was \( A = 0.117 \) m\(^2\), and the nozzle exit area of the wind tunnel was \( A_0 = 0.4225 \) m\(^2\); thus, giving a blockage ratio of \( A_0/A \). Further details about the shape of the wind turbine can be found in reference [9]. As shown in Fig. 1, the rotational axis of the BWT was held vertically with two ball bearings. The rotational axis was connected to an induction motor (Mitsubishi Electric Corp., SF-JR 0.75 kW), which was controlled with an inverter (Mitsubishi Electric Corp., FR-S510W-0.75K) through a torque detector (Ono Sokki Co., Ltd., SS-005), and the speed of revolution was kept constant. A digital torque meter (Ono Sokki Co., Ltd., TS-3600B) was used as an indication of the measured torque and converted this to an analog voltage. As the output performance of the wind turbine was measured, wind speed at the center of the wind tunnel nozzle exit was simultaneously measured using a Pitot tube. The differential pressure obtained by the Pitot tube was converted to an analog electric current by a digital differential pressure gauge that was suitable for micro pressure (Okano Works Ltd., DMP201N). The output signals of the torque meter (rotor torque, and rotational speed values) and pressure gauge (wind speed values) were imported to a personal computer through an analog-to-digital converter (Turtle Industry Co., Ltd., TUSB-0412ADSM-SZ). The distance between the rotor center and the floor was 1.4 m, which was the same as the distance between the center of the nozzle exit and the floor. In this study, a base wind speed \( V_b \) was assumed to be 6 m/s, and the frequency of the inverter (Mitsubishi Electric Corp., FR-D720-1.5K) that controls the wind tunnel blower (Kamakura Seisakusyo Co., Ltd., GRL-6361) was fixed at 58.2 Hz throughout the experiments.

Figure 2 shows a coordinate system, the origin of which was set at the center of the nozzle exit, and the installation locations of the wind turbines, which are represented by black circles. The wind turbine output performance was measured at each location by increasing the rotational speed gradually under the base wind speed condition. The position of the turbine in the duct flow relative to the duct nozzle exit is described using the coordinates \( X \) and \( Y \). Measurements were carried out five times in each case with the turbine positioned at \( (X, Y) = (0.5 \) m, 0 m), \( (0.75 \) m, 0 m). The same measurements were conducted two times in each case with the turbine located at \( (X, Y) = (1.0 \) m, 0 m), \( (1.25 \) m, 0 m), and \( (1.5 \) m, 0 m). The reproducibility of the output performance obtained was very high, and the data scattering was small at all the locations. The averaged values obtained from the measurements regarding the output performance of the wind turbine were used in the following examinations. Hereafter, the
location of $X$ will be expressed in relation to the diameter ($X/D$), and the location of the $Y$ will be expressed in relation to the width of the nozzle exit ($Y/W$). Although the actual rotational direction of the wind turbine viewed from above is clockwise, all the figures and results regarding the experiments are depicted in this study as the counter-clockwise rotation to provide consistency with the CFD results described later.

3. Experimental Results

3.1 Location dependency of wind turbine output-performance

The output-performance of the wind turbine at varying $Y$ distances is shown in Figs. 3 (a), (b), and (c) for the cases where installation was at $X/D = 1.25$, $2.5$, and $3.75$, respectively. The vertical axes in Figs. 3 (a)-(c) show the rotor power coefficient, $C_p$, and the horizontal axes show the tip speed ratio, $\lambda$. In Fig. 3 (a), where $X/D = 1.25$, the maximum power coefficients at $Y/W = -0.69$ and $0.69$ are very low, while the maximum power coefficients measured at $Y/W = -0.46$ and $0.46$ are greater than 0.1. In Fig. 3 (b), where $X/D = 2.5$, the maximum power coefficient of $Y/W = 0.69$ is greater than that in Fig. 3 (a) and that measured at $Y/W = -0.46$ has decreased. And in Fig. 3 (c), where $X/D = 3.75$, it can be seen that the maximum power coefficient at $Y/W = 0.69$ has increased even further, and at $Y/W = -0.46$ it has again decreased. As shown in Figs. 3, the variation in $C_p$ in the $Y$ direction changes according to the length between the wind turbine and the nozzle exit. The maximum $C_p$ is obtained at approximately $\lambda = 1.05$ (300 rpm) in most cases where $Y/W$ is between $-0.46$ and $0.46$.

Figure 4 shows the variations regarding the relative location (only the cases of $Y/W = -0.46$, $0$, and $0.46$) of the maximum $C_p$ and the wind speed, $V_c$, that is measured at the center of the nozzle exit. The decrease of the $V_c$ at the closest location to the nozzle exit of $X/D = 1.25$ (corresponding to $X/W = 0.77$) indicates that the presence of the wind turbine has had a strong influence on the

![Fig. 3 Output-performance curves of butterfly wind turbine, which is located at (a) $X/D = 1.25$, (b) $X/D = 2.5$, and (c) $X/D = 3.75$, respectively. Color shows the location regarding the $Y$ direction.](image)

![Fig. 4 Variations regarding the relative location in both maximum power coefficient of wind turbine and wind speed at the center of wind tunnel nozzle exit.](image)

![Fig. 5 Wind speed distributions measured without wind turbine.](image)
when a wind turbine has not been installed is defined as the pressure at the nozzle exit should be almost the same as the ambient pressure. That is, the loss of pressure energy, the wind turbine. The static pressure distributions were not measured at the wind tunnel nozzle exit owing to the assumption that the location of $X/D = 1.25$. If Figs. 6 and 7 are considered, the torque distribution can be used to explain the relatively high power coefficient obtained and multiple streamtube model [11]. Large torque is generated at the highlighted portion where the azimuth angle varies from $20^\circ$ to $110^\circ$. If $\lambda = 1.05$ by the blade element momentum (BEM) theory [10] using the quadruple-multiple streamtube model [11]. Large torque is generated at the highlighted portion where the azimuth angle varies from $20^\circ$ to $110^\circ$. Although it remains a matter of speculation, the expansion of the exhaust flow might strengthen the effects of the unidirectional rotation of the rotor.

Another possible factor accounting for the behavior of $C_p$ is the effects that may rise from the rotation of the turbine. Figure 6 shows a schematic diagram showing the representative locations of the wind turbine, the rotational direction, and the expansion of the exhaust flow. The change in rotational torque acting upon a wind turbine blade in relation to the azimuth angle is shown in Fig. 7, which is calculated at the tip speed ratio $\lambda = 1.05$ by the blade element momentum (BEM) theory [10] using the quadruple-multiple streamtube model [11]. Large torque is generated at the highlighted portion where the azimuth angle varies from $20^\circ$ to $110^\circ$. If Figs. 6 and 7 are considered, the torque distribution can be used to explain the relatively high power coefficient obtained at the location of $X/D = 1.25$ and $Y/W = -0.46$. However, it is difficult to account for the downstream inversion of the magnitude in relation to $C_p$ at $Y/W = -0.46$ and $0.46$. Although it remains a matter of speculation, the expansion of the exhaust flow might strengthen the effects of the unidirectional rotation of the rotor.

3.2 Wind speed distribution at nozzle exit and energy recovery rate

To investigate the effect of a wind turbine on the wind tunnel or duct, the wind speed distribution at the nozzle exit was measured. A Pitot tube was placed at intervals of 0.1 m along the cross section of the nozzle exit; the locations of these are shown in Fig. 8. The wind speed distributions were measured without and with the wind turbine installed at the locations of $X/D = 1.25, 1.875, 2.5, 3.125, and 3.75$ for each of $Y/W = -0.46, 0.0, and 0.46$.

Figure 9 (a) shows an example of the wind speed distribution at the nozzle exit when a wind turbine was not installed, and Fig. 9 (b) shows the wind speed distribution when the wind turbine was installed at $X/D = 1.25$ and $Y/W = 0$. In these figures, the wind speed outside ($Y = \pm 0.4$ m and $Z = \pm 0.4$ m) the nozzle exit, represented by the dashed line in Fig. 9, is assumed to be $0$ m/s. The inside of the nozzle exit is divided in $104 \times 104$ cells, and the wind speed, $V_{j,k}$ ($j, k = 1, 2, \ldots, 104$) at each cell is obtained by linear interpolation using measurement data. The kinetic energy, $P_{KE}$, of the exhaust flow out of the nozzle was calculated by eq. (1):

$$P_{KE} = \frac{1}{2} \rho \Delta A \sum_{j=1}^{104} \sum_{k=1}^{104} V_{j,k}^2$$

(1)

where $\rho$ is the air density and $\Delta A$ is the area ($6.25 \times 6.25$ mm$^2$) of a cell. The fluid kinetic energy flowing out of the wind tunnel when a wind turbine has not been installed is defined as $P_{KE0}$. From eq. (1), the value of $P_{KE0}$ corresponding to the wind speed distribution of Fig. 9 (a) is calculated to be 64.4 W. Using a mean of three-time measurements, the kinetic energy of the exhaust flow in the case of Fig. 9 (b) is $P_{KE} = 60.9$ W. The loss of kinetic energy ($\Delta P_{KE}$) can be expressed as the difference between $P_{KE0}$ and $P_{KE}$, which in this case is equal to 3.5 W. This is thought to be the influence on the wind tunnel (or duct) by the existence of the wind turbine. The static pressure distributions were not measured at the wind tunnel nozzle exit owing to the assumption that the pressure at the nozzle exit should be almost the same as the ambient pressure. That is, the loss of pressure energy, $\Delta P_{PE}$, has
Fig. 8 Positions of the Pitot tube, i.e., gray circles, used when measuring the wind speed at the wind tunnel nozzle exit.

Fig. 9 Examples of wind speed distributions at wind tunnel nozzle exit when (a) no wind turbine is installed and (b) a wind turbine is installed at $X/D = 1.25$ and $Y/W = 0.0$.

not considered in the experiments presented in this study. Therefore, the energy recovery rate, $\eta$, is defined here by eq. (2):

$$\eta = \frac{P_{WT} - \Delta P_{KE}}{P_{KE0}}, \quad (2)$$

where $P_{WT}$ is the mechanical output power, $Q$ is the torque, and $\omega$ is the angular velocity of the wind turbine.

Figure 10 (a) shows $P_{WT}$, $\Delta P_{KE}$, and $\eta$ under the conditions where the wind turbine has been installed at the locations of $Y/W = 0.0$. In the closest installation case to the nozzle, $X/D = 1.25$, although the output power of the wind turbine is larger than other downstream cases, the energy recovery rate is a negative value because of the great loss of kinetic energy; or in other words, owing to the large influence that the turbine has on the wind tunnel. However, when the wind turbine is installed at $X/D = 1.875$ ($X/W = 1.154$) or more, $\eta$ is approximately 1.3%, because the influence of the turbine on the wind tunnel becomes small.

The measurement of $\eta$ was affected by the measurement error of the flow speed at the wind tunnel nozzle exit. To show the extent of this measurement error, an error bar is depicted for the energy recovery rates measured at $X/D = 1.25$ in Fig. 10 (a). In addition, note that the data shown in Fig. 10 (a) has been newly obtained by paying attention to the temperature stability and the previous data used in Fig. 10 in the conference study [12] was revised.

In Fig. 10(b), the energy recovery rates measured at $Y/W = \pm 0.46$ are compared with that of $Y/W = 0$. Recovery rates are much greater when the wind turbine is located away from the centerline of the exhaust flow; in this case, $Y/W = \pm 0.46$, than when the wind turbine is placed at the centerline of the wind tunnel ($Y/W = 0$). This is due to the large effect the presence of a wind turbine has when it is placed at the centerline of the exhaust flow.
Fig. 10 Experimental results on the recovery of fluid energy flowing out of an exhaust duct (small wind tunnel with the exit width of 0.65m (W)) with a butterfly wind turbine of 0.4m (D) × 0.3m (H) [9]. Left-hand side (a) shows the output power of wind turbine ($P_{WT}$), the decrease in kinetic energy at nozzle exit ($\Delta P_{KE}$), and energy recovery rate ($\eta$), which were obtained from experiments when wind turbine was installed at $Y/W = 0.0$. Right-hand side (b) shows the comparison of the energy recovery rate of $Y$ direction ($Y/W = -0.46, 0.0, and 0.46$).

4. Numerical Simulation

4.1 Simulation method

In order to verify the validity of the experimental results obtained in the previous sections, two-dimensional computational fluid dynamics (2D-CFD) analysis is conducted in this study. 2D-CFD analysis using a 2D-double-blade-rotor model as the computational object corresponding to the same experimental wind turbine (the cambered bladed BWT) had been carried out under the condition of uniform flow in the previous study [9]. The 2D-double-blade-rotor model, which is the same as the previous study [9], is used as the computational object in the present study in addition to a 2D-straight-duct model with a width,
The thickness of the duct wall is 0.004 m. Strictly speaking, in order to compare with the experimental rotor, 3-dimensional calculation was desired, however in order to save the computational cost (computation time), 2-dimensional calculation was adopted in this research.

Figure 11(a) shows the whole calculation domain, the dimensions of which are almost the same as the previous study. The width of the calculation domain is $24D$; the length between the center of the wind turbine model and the outlet is $40D$. Due to the ease of the model setting, the location of the double-blade-rotor model was fixed and the location of the straight-duct model was changed to adjust to the relative location between the rotor and the duct. Since the $L$ is held constant, the length between the center of the wind turbine model and the duct inlet varies from $21.25D$ to $23.75D$. Figure 11(b) is an enlarged view around the wind turbine model and the duct exit. As described above, the computational object is the double-blade-rotor model that is two-dimensionally approximated from the experimental wind turbine. The radius of the outer rotor is 0.2 m ($D = 0.4$ m), and that of the inner rotor is 0.12 m. The chord length of the outer blade is 0.11 m, and that of the inner blade is 0.14 m. The rotational axis was not considered in this 2D analysis. The radius of the rotating region shown in Fig. 11(b) is 0.3 m ($\varphi = 1.5D$). The direction of the rotor is counter-clockwise. A constant flow velocity of 6 m/s is set at the inlet inside the duct and a constant gage pressure of 0 Pa is given at the inlet outside the duct and at the outlet; these were used as the boundary conditions. The slip wall condition is given to the sides of the static rectangular region, while the no-slip wall condition is given to the surfaces of the blades and duct wall. The solver used in this study is STAR-CCM+ ver. 9.02. The CFD analysis is based on the Reynolds-averaged Navier–Stokes (RANS) equations under the conditions of two-dimensional, unsteady, incompressible, and viscous flow. The $k-\omega$ shear stress transport (SST) model [13] is adopted as the turbulence model.

Figure 12 shows the computational mesh generated by STAR-CCM+. The cell number of the static region including the inside of the duct is approximately 105,000 and that of the rotational region is about 150,000. Unstructured polyhedral mesh is adopted, except for near-wall regions where structured prism layer meshes are generated. The number of layers near the blade surfaces is 20 and the number of layers near the duct walls is 10. The minimum mesh size near the blade surfaces is approximately $1.5 \times 10^{-7}$ m ($y^+ = 0.17$) and that near the duct walls is approximately $4.3 \times 10^{-4}$ m ($y^+ = 3.3$). The rotational speed of the 2D-double-blade-rotor model is set at 286 rpm ($\lambda = 1$), which is almost corresponding to the maximum power condition obtained by the experiments except for the computations of the power curve (see Fig. 13). The time step, $\Delta t$, is decided so as to revolve the turbine rotor 1.0 degrees per step. For example, the time step in the case of $\lambda = 1$ is $\Delta t = 1/6/286 = 5.827505828 \times 10^{-5}$s. At least, fifty full rotations of the rotor were calculated in this study. For the case without the rotor, the calculation was continued until a pair of vortices, ejected from the duct exit, arrived at the outlet of the calculation domain. In other words, the calculation time corresponds to one hundred rotations of the rotor at $\lambda = 1$. The Reynolds number based on the duct inlet wind speed (i.e., the base wind speed, $V_b$) and the outer rotor diameter, $D$, is $Re = 1.6 \times 10^5$.

4.2 Simulation results

In Fig. 13, the power characteristics calculated for the 2D-wind turbine installed at $X/D = 2.5$ and $Y/W = 0.0$ in the exhaust flow from the duct are compared with the power characteristics obtained in the previous study [9] under uniform flow conditions and with the experimental results. The tip speed ratio, $\lambda$, was varied between 0.5 and 2.0 in the present CFD calculations shown in Fig. 13. Although the results of 2D-CFD in the uniform flow are very different from the experimental results, the results of 2D-CFD in the duct flow agree well with the experiments. In the previous study [9], it was speculated that the difference between the 2D-CFD and the experiments might be brought from the difference in the dimension or the object-rotor shape. However, according to Fig. 13, the difference in flow conditions, i.e., the difference in blockage ratio, was possibly responsible for the difference in the power characteristics between the 2D-CFD in the uniform flow and the experimental results.

![Fig. 13 Comparison of power coefficients obtained experimentally, black, and the present 2D-CFD in duct flow, red, and the previous 2D-CFD in uniform flow, blue, under the assumption that a wind turbine is located at $X/D = 2.5$ and $Y/W = 0.$](image-url)
(a) $X/D = 1.25, Y/W = -0.46$  
(b) $X/D = 1.25, Y/W = 0.0$  
(c) $X/D = 1.25, Y/W = 0.46$

Fig. 14 Pressure fields averaged during one rotor revolution.

Fig. 15 Distributions of kinetic energy ($K.E.$) and pressure energy ($P.E.$) at duct inlet and exit at the following conditions: (a) without wind turbine, (b) with wind turbine at $Y/W = 0$, (c) $Y/W = -0.46$, and (d) $Y/W = 0.46$. The styles of the line (solid, broken, and dotted) show the location of wind turbine in relation to the $X$ direction.
Figure 14 shows the examples of mean pressure fields, which are obtained by averaging the unsteady pressure fields calculated during the 52nd rotation under the condition of \( \phi = 1 \). Since the pressure in the rotating region is averaged on a moving coordinate system, there is discontinuity at the interface (circle of \( \phi \) 1.5\( D \)) between the rotational region and the static region. By focusing on the mean pressure field of the upstream adjacent portion of the rotating region, it is understood that the pressure when \( X/D = 1.25 \) and \( Y/W = 0.0 \) (Fig. 14 (b)) is highest and the duct flow is greatly affected by the existence of the rotor. The fact that the upstream pressure of the rotor when \( X/D = 1.25 \) and \( Y/W = -0.46 \) (Fig. 14 (a)) is somewhat higher than that when \( X/D = 1.25 \) and \( Y/W = 0.46 \) (Fig. 14 (c)) coincides with the trends seen experimentally, which were presented in Fig. 4.

Figure 15 shows the averaged distributions of kinetic energy, \( K.E. \), and pressure energy, \( P.E. \), at the duct inlet and exit, which are defined by eqs. (4) and (5) as the energies per unit area.

\[
K.E. = \frac{1}{2} \rho V^3
\]  

(4)

\[
P.E. = PV
\]  

(5)

Figure 15 (a) is the case where no wind turbine has been installed, and Figs. 15 (b), (c), and (d) are cases with wind turbine installed at \( Y/W = 0.0, -0.46, \) and 0.46, respectively. The \( K.E. \) at the duct inlet is constant, shown by the black line in Fig. 15, and is equal to 132.3 W for all the cases since the constant wind speed is given as the boundary condition. The lines in red are the \( K.E. \) at the duct exit, the lines in orange are the \( P.E. \) at the duct inlet, and the lines in blue are the \( P.E. \) at the duct exit. The styles of the lines (solid, broken, and dotted) show the location of the wind turbine in relation to the \( X \) direction. When the wind turbine is installed at the closest location to the duct exit, \( X/D = 1.25 \), the distributions in the \( K.E. \) and \( P.E. \) are greatly affected by the existence of the turbine. The distributions become similar to cases where no wind turbine is installed or when the wind turbine is installed much farther downstream.

5. Comparison Between Experimental and Simulation Results

Figure 16 shows the comparison between 2D-CFD results and experimental results for maximum power coefficient at a variety of different locations of the wind turbine in relation to the duct exit. Although there is a rather large difference between the 2D-CFD and the experiments, where \( Y/W = -0.46 \), both are in agreement in terms of the relative magnitude relation of \( C_p \) between \( Y/W = -0.46 \) and \( Y/W = 0.46 \). The tendency of the maximum \( C_p \), whose magnitude relation about the \( Y \) direction is changed between the near installation and the far installation of the turbine, might be strengthened by the asymmetry of the exhaust flow observed in these experiments.

To calculate the duct flow energies corresponding to the experimental conditions, the two-dimensional duct is assumed to have the length of 0.65 m in the \( Z \) direction. Figure 17 shows \( P_{KE} \) and \( P_{PE} \) at the duct exit at the cross-sectional area of 0.65 × 0.65 m², which are obtained by integrating eqs. (4) and (5). Figure 17 shows that the \( P_{KE} \) was 88.63 W, indicated by the one-dot chain line, and that the \( P_{PE} \) was 0.14 W, indicated by the two-dot chain line. It is demonstrated that the position of the wind turbine greatly affects the kinetic and pressure energies of the duct flow when the turbine is installed at \( X/D = 1.875 \) \( (X/W = 1.154) \) or less. On the other hand, there is almost no influence when the wind turbine is installed at \( X/D = 2.5 \) \( (X/W = 1.54) \) or more.

The loss of kinetic energy \( \Delta P_{KE} \) at the duct exit owing to the installation of the wind turbine is defined by \( P_{KE} \) − \( P_{KE0} \) and the increase in the pressure energy \( \Delta P_{PE} \) is defined by \( P_{PE} \) − \( P_{PE0} \). Considering the \( \Delta P_{KE} \) and \( \Delta P_{PE} \) as negative influences to the duct, the energy recovery rate, \( \eta \), in the 2D-CFD analysis is defined by the eq. (6) in this study.
This was calculated from the CFD results by assuming the 2D-rotor height to be the same as the height, 0.3 m, of the experimental turbine.

The energy recovery rates obtained in the 2D-CFD analysis are compared in Fig. 18 with those obtained experimentally (see Fig. 10 (b)). While the energy recovery rates calculated by the CFD have very large negative values at $X/D = 1.25$, effective energy recovery rates greater than 2.5% are obtained at $X/D = 2.5$ ($X/W = 1.54$) or more. Although the conditions assumed in the 2D-CFD differ from those in the experiments, the energy recovery rates of both agree well.

6. Conclusion

To demonstrate the validity of duct flow power generation, the effects of a vertical-axis-type wind turbine on the duct flow was examined experimentally using a butterfly wind turbine, with a diameter $D$, and a small wind tunnel, with a width $W$. Results were also calculated for similar conditions using 2D-CFD. The power characteristics calculated by 2D-CFD for situations where a wind turbine has been installed in the exhaust flow of a wind tunnel at the positions $X/D = 2.5$ and $Y/W = 0.0$ agreed well with the experimental results. The variations in the maximum power coefficient regarding the relative location of the wind turbine against the duct exit qualitatively agreed between the 2D-CFD and experimental results. Both the 2D-CFD results and experimental results showed the following facts: when the wind turbine was installed near the duct exit close to the center line ($X/D = 1.25$), the case ($Y/W = -0.46$) with the rotor half of the blade moving upstream higher power was generated than in the inverse location. On the other hand, when the wind turbine was installed far from the duct exit ($X/D = 3.75$) close to the center line of the exhaust flow, the case ($Y/W = 0.46$) wherein the rotor half of the blade moving downstream higher power was generated than in the inverse location. As the results of the experiments and the 2D-CFD analysis have demonstrated, it is possible to obtain an energy recovery rate of approximately 1.3% when the wind turbine is installed on the centerline of the exhaust flow at $X/D = 2.5$ ($X/W = 1.54$) or more away from the duct exit in the case of $D/W = 0.615$. The investigations about the validity of the duct flow power generation, in the cases of other ratios of the wind turbine diameter $D$ to the duct width $W$, are currently in progress.

Acknowledgments

This work was supported in part by the Tottori Prefecture Project for Promoting Environmental Research and the Collaborative Research Program of Research Institute for Applied Mechanics, Kyushu University.

### Nomenclature

- $A$: Swept area of wind turbine [m$^2$]
- $A_0$: Cross-section area of exit of wind tunnel (duct) [m$^2$]
- $\Delta A$: Cross-section area of a small cell on duct exit [m$^2$]
- $C_p$: Power coefficient of wind turbine
- $D$: Diameter of wind turbine [m]
- $H$: Height of wind turbine [m]
- $K.E.$: Kinetic energy of duct flow per unit area [W/m$^2$]
- $L$: Length of duct [m]
- $P_{KE}$: Kinetic energy of duct flow with wind turbine [W]
- $P_{KE0}$: Kinetic energy of duct flow without wind turbine [W]
- $P_{PE}$: Potential energy of wind turbine [W]
- $P_{PE0}$: Potential energy of duct flow without wind turbine [W]
- $R$: Radius of wind turbine [m]
- $Re$: Reynolds number ($=V_b D/\nu$)
- $\Delta t$: Time step [s]
- $V$: Wind speed [m/s]
- $V_b$: Base wind speed [m/s]
- $V_c$: Wind speed at the center of duct exit [m/s]
- $V_{j,k}$: Wind speed at a cell ($j, k = 1, \ldots, 104$) [m/s]
- $W$: Width of wind tunnel or duct [m]
- $X, Y, Z$: Coordinates [m]
\( P_{KE0} \): Kinetic energy of duct flow without wind turbine [W]
\( P_{PE} \): Pressure energy of duct flow with wind turbine [W]
\( P_{PE0} \): Pressure energy of duct flow without wind turbine [W]
\( P_{WT} \): Mechanical output power of wind turbine [W]
\( P.E. \): Pressure energy of duct flow per unit area [W/m²]
\( \Delta P_{KE} \): Decrease in kinetic energy (=\( P_{KE0} - P_{KE} \)) [W]
\( \Delta P_{PE} \): Increase in pressure energy (=\( P_{PE} - P_{PE0} \)) [W]
\( Q \): Rotational torque of wind turbine [Nm]

\( y^+ \): Wall distance normalized by viscous length
\( \eta \): Energy recovery rate
\( \lambda \): Tip speed ratio (=\( R \omega /V_b \))
\( \nu \): Kinematic viscosity [m²/s]
\( \rho \): Air density [kg/m³]
\( \psi \): Azimuth angle [deg]
\( \omega \): Angular velocity of wind turbine [rad/s]

References