Performance prediction of loop-type wind turbine

Wonyoung Jeon1, Jeanho Park2, Seungro Lee3, Youngguan Jung1, Yeesock Kim4 and Youngjin Seo1

Abstract
An experimental and analytical method to evaluate the performance of a loop-type wind turbine generator is presented. The loop-type wind turbine is a horizontal axis wind turbine with a different shaped blade. A computational fluid dynamics analysis and experimental studies were conducted in this study to validate the performance of the computational fluid dynamics method, when compared with the experimental results obtained for a 1/15 scale model of a 3 kW wind turbine. Furthermore, the performance of a full sized wind turbine is predicted. The computational fluid dynamics analysis revealed a sufficiently large magnitude of external flow field, indicating that no factor influences the flow other than the turbine. However, the experimental results indicated that the wall surface of the wind tunnel significantly affects the flow, due to the limited cross-sectional size of the wind tunnel used in the tunnel test. The turbine power is overestimated when the blockage ratio is high; thus, the results must be corrected by defining the appropriate blockage factor (the factor that corrects the blockage ratio). The turbine performance was corrected using the Bahaj method. The simulation results showed good agreement with the experimental results. The performance of an actual 3 kW wind turbine was also predicted by computational fluid dynamics.

Keywords
Loop-type wind turbine, horizontal axis wind turbine, computational fluid dynamics, blade design, wind energy

Date received: 30 September 2018; accepted: 25 February 2019

Handling Editor: Ka-Veng Yuen

Introduction
Renewable energy sources are gaining increasing interest, as they play a significant role in satisfying growing energy demands. Companies involved in the production of commercial wind power generators are already producing large megawatt (MW) class wind turbines, with rotor diameters exceeding 100 m. On the other hand, no significant attention has been paid to the production of small wind turbines until recently, when they started to attract attention due to the global investments and research in smart grid projects. Smart grids can maximize energy efficiency by integrating information and communication technologies into electric power grids, and monitoring electricity usage and supply in real time. Based on these information and communication technologies, small-sized power generation systems have emerged in the smart grid context, because prosumers produce electricity at the same time as existing users consume electricity. Wind power generators

1Department of Mechanical Engineering, Kumoh National Institute of Technology, Gumi, South Korea
2Research Institute of Industrial Technology Convergence, Korea Institute of Industrial Technology, Ansan, South Korea
3Department of Mechanical Engineering, Jeonbuk National University, Jeonju, South Korea
4Department of Civil Engineering, California Baptist University, Riverside, CA, USA

Corresponding author:
Youngjin Seo, Department of Mechanical Engineering, Kumoh National Institute of Technology, 61 Daehakro, T315, Gumi 39177, Gyeongangbuk-do, South Korea.
Email: yjseo@kumoh.ac.kr

Creative Commons CC BY: This article is distributed under the terms of the Creative Commons Attribution 4.0 License (http://www.creativecommons.org/licenses/by/4.0/) which permits any use, reproduction and distribution of the work without further permission provided the original work is attributed as specified on the SAGE and Open Access pages (https://us.sagepub.com/en-us/nam/open-access-at-sage).
(together with solar energy sources) are used for small power generation systems, because solar cells cannot solely generate sufficient power to meet the demand. Unlike large wind power generators, small wind power systems that are connected to such self-sufficient power supply systems are typically located near residential complexes. However, installing wind turbines near to residential areas inevitably causes noise problems due to the aerodynamic low-frequency wind turbine noise, which may lead to both physical and psychological detrimental effects such as nervous breakdowns and insomnia.\(^1,2\) The main origin of this aerodynamic noise is the tip vortex that occurs at the blade tip when the blade rotates at high speed. As the loop-type wind turbines examined in this study are characterized by their ability to generate high torque at low speed, the rotational speed of the turbine is half of that of a conventional turbine, and the blade tips are closed. Therefore, much less aerodynamic noise is caused by the tip vortex of wind turbines.\(^3\) Such turbines are structurally stable, because each two blades form one blade with small blade deformation, leading to low vibration and a long lifespan. This wind turbine capability to generate high torque at low rotation speeds can be attributed to the fact that it has optimal tip speed ratio (TSR) at low rotation speed, because of its larger solidity compared to the blade type turbine, as shown in Figure 1. The main origin of this aerodynamic noise is the tip vortex that occurs at the blade tip when the blade rotates at high speed. As the loop-type wind turbines examined in this study are characterized by their ability to generate high torque at low speed, the rotational speed of the turbine is half of that of a conventional turbine, and the blade tips are closed. Therefore, much less aerodynamic noise is caused by the tip vortex of wind turbines.\(^3\) Such turbines are structurally stable, because each two blades form one blade with small blade deformation, leading to low vibration and a long lifespan. This wind turbine capability to generate high torque at low rotation speeds can be attributed to the fact that it has optimal tip speed ratio (TSR) at low rotation speed, because of its larger solidity compared to the blade type turbine, as shown in Figure 1. \(\text{TSR} = \lambda = \frac{\omega R}{U} \) where \(\omega\) is angular velocity, \(R\) is radius of turbine, and \(U\) is inflow wind velocity. Optimal TSR is the TSR that the wind turbine generates maximum power. The TSR when the time that the wind passes from the front blade to the rear blade and the time when neighboring blades rotate by \(1/n\) is called optimal TSR. The optimal TSR can be obtained from equation (2)

\[ \lambda_{\text{optimal}} = \frac{2\pi}{n} \left( \frac{R}{D} \right) \]

where \(n\) is number of blades and \(D\) is wind travel distance through the turbine. And the solidity \(\sigma\) is defined as the ratio of the projected area of the turbine \((A_P)\) to the swept area of wind turbine \((A_S)\) as shown in equation (3)

\[ \sigma = \frac{A_P}{A_S} \]

Many parameters such as inflow angle \((\phi)\), angle of attack \((\text{AOA, } \alpha)\), twisted angle \((\theta)\), chord length \((C)\), thickness of airfoil \((t)\), TSR \((\lambda)\), swept area \((A_S)\) which are dependent on each other are considered for the appropriate design of wind turbines. Numerous studies have been conducted on the design of turbines. Thumthae and Chitsomboon\(^4\) conducted an analytical study to determine the AOA of non-twisted blades. Guntur and Rensen\(^5\) compared four methods to determine the AOA such as inverse blade element momentum (BEM) method, two computational fluid dynamics (CFD) methods (computing the local induced average velocities and computing the local induced velocities at the blade), comparison of two-dimensional (2D) and three-dimensional (3D) \(C_P\) curves method. The overall design of wind turbine blades is based on the BEM theory. Glauert\(^6\) developed Froude’s BEM theory, a model that can be applied to a wind turbine to predict the performance of a wind turbine. Lanzafame and Messina modified the Glauert propeller theory model.

\[ \text{Figure 1. Comparison of solidity between conventional wind turbine blade and loop-type wind turbine blade (left: conventional wind turbine blade; right: loop-type wind turbine blade).} \]
to fit wind turbines and developed a mathematical model for their aerodynamic design. The accuracy of the developed mathematical model was evaluated and verified, and then the power coefficient at low wind speeds was maximized. Lanzafame and Messina also used the BEM theory to predict the performance of a small wind turbine used for wind tunnel testing and adjusted the twist angle to achieve maximum efficiency using stall control. Many studies have confirmed the effectiveness of employing CFD to optimize blade efficiency. Abe and Ohya utilize CFD to verify design concepts of the flange diffuser wind turbine. Bazilevs et al. applied CFD to the early stages of blade design and proposed simulation and performance evaluation methods.

As the purpose of this study is to investigate the performance of 3 kW small wind turbines, studies conducted on small wind turbines have been reviewed. Mayer et al. examined the change in the initial motion pattern of a wind turbine according to the pitch angle of a small 5 kW wind turbine and found that the idling time of the wind turbine decreased as the pitch angle increased. Wright and Wood discussed the torque generation and power characteristics of small wind turbines. Howell et al. conducted experiments using CFD (2D and 3D) to evaluate the performance of a small vertical axis wind turbine (VAWT). Neal and Clark proved that using a soft stall (electrical loading control) on a 900 W small wind turbine improves its power generation.

Even if the wind turbine is small, because the blade length exceeds 1 m it is necessary to conduct experiments in wind tunnel using a reduced scale models. Choo conducted an experiment by mounting a scale model on a vehicle to analyze the aerodynamic of a wind turbine using a 1/48 scale model of a 5 MW offshore wind turbine. Zaghi et al. studied the blockage effect in wind tunnel tests through simulations and experiments. Wang et al. analyzed the performance of a cross-axis wind turbine (CAWT) using a wind tunnel test. The performance of the tested CAWT was compared with that of a typical VAWT. As no research has ever been conducted on evaluating the performance of loop-type wind turbines, this study is aimed to examine the performance of a miniature model of a loop-type wind turbine and predict the performance of an actual wind turbine.

A 1/15 scale model of a previously designed 3 kW loop-type wind turbine (manufactured using a 3D printer) is used in this study. The obtained results are compared to the experimental and flow analysis results, to verify the validity of the CFD analysis. Then, 3D simulations of the tested 3 kW wind turbine are performed, to predict the performance of a full sized wind turbine.

**Figure 2. Schematic of loop-type wind turbine blade.**

### Design of loop-type wind turbine

The basic form and components of the loop-type wind turbine blade are shown in Figure 2. The blade is divided into two portions, a front end and a rear end. The spine line forms the overall blade shape, and the airfoil forms the shape of the blade along the spine line. The shape of the airfoil shown in Figure 2 is not the commonly used airfoil developed by the National Advisory Committee for Aeronautics (NACA) or National Renewable Energy Laboratory (NREL). It is an arc-shaped airfoil, similar to that used in a VAWT. The front and rear blades of the loop-type turbine have slightly different roles. The front blades can achieve high efficiency at low speeds, and the rear blades are designed to achieve high efficiency at high speeds.

The design of the 3 kW loop-type wind turbine tested in this study is well documented in a study by Kim et al. The overall design process is shown in Figure 3. First, the diameter of the wind turbine was set by reversing Betz’s law to determine the size of the wind turbine corresponding to the power. Meanwhile, the power coefficient \( C_P \) of the turbine was conservatively set to 0.2. The power coefficient \( (C_P) \) of the turbine is defined as equation (4)

\[
C_P = \frac{P_T}{\frac{1}{2} \rho A_3 U^3} \quad (4)
\]

where \( \rho \) is density of air and \( P_T \) is the power from wind turbine. The optimal position of each airfoil is predicted using the diameter of the wind turbine and number of blades. The derived TSR \( \lambda_{optimal} \) is 2.32. The inflow angle, the AOA, and the twisted angle are determined in the 2D airfoil configuration based on the TSR. Finally, the chord length is determined based on the inflow angle and lift coefficient, which is assumed to be 0.8 to determine the chord length. However, such
a blade design results in an abnormally long chord. Thus, the chord length is linearized in the \( r/R \) range of 0–0.75, where \( r \) is the local radius at which chord is located and \( R \) is the radius of the turbine. Since tip of the blade has significant effect on the turbine performance, the \( r/R \) range was limited to 0.75 while shape of the tip remains the same.19

In the abovementioned design study, qualitative comparisons of the scaled models were carried out. In this study, the previously designed A model turbine is quantitatively evaluated.

A proper similarity law should be applied for the quantitative evaluation of the tested wind turbine. As a 1/15 scale experimental model is used for the wind tunnel test, two similarity laws can be applied to the model wind turbine, Reynolds number similarity, and TSR similarity. The Reynolds number range of the miniature model is \( 8 \times 10^3 \)–\( 2.66 \times 10^4 \) and the actual wind turbine is \( 1.5 \times 10^5 \)–\( 6.49 \times 10^5 \). In the case of Reynolds number similarity, it is necessary to use a fluid with high flow velocity or low kinetic viscosity as the model scale is reduced. However, many difficulties are associated with using Reynolds number similarity for wind tunnel experiments. On the other hand, TSR similarity law is most widely used method in most wind tunnel experiments because it is necessary to increase the angular velocity by reducing the radius of wind turbine. Hence, the similarity is obtained by equation (5), as follows

\[
TSR = \lambda_1 = \lambda_2 = \frac{\omega_1 R_1}{U_1} = \frac{\omega_2 R_2}{U_2}
\] (5)

**Numerical study**

FLUENT (version 18.2; ANSYS Inc., Canonsburg, PA, USA) was utilized to perform the 3D numerical analysis. The time domain is classified into two types: steady state and transient state. The spatial domain (flow field) is commonly used for steady and transient analysis. The 1/15 scale model is located at the center of the flow field when viewed from the front. The cross section of the flow field is 1500 mm × 1500 mm and the length is 2600 mm. The steady-state analysis is a simulation that changes the rotational velocity at the same constant flow rate as in the wind tunnel test (TSR controlled state). The angular velocity was controlled to 250–1000 r/min. The transient state (no-load state) analysis can confirm the movement from the stop of the turbine to the steady state. All movements except the axial rotation of the turbine were limited (1-degree-of-freedom movement). The angular velocity of the turbine is measured using transient simulations when the rotational velocity of the turbine no longer increases (i.e. in the state of aerodynamic equilibrium). This confirms the maximum rotational speed of the blade under ideal conditions. The numerical analysis allows an appropriate setting of the turbine rotation speed during the steady-state simulation.

**Geometry and grid generation**

The domain of analysis is shown in Figure 4. The entire area is divided into a stop area and a rotation area, and an interface is created to interpolate data between these areas. The width and height of the flow field were set to five times the radius of rotation to avoid wall effect. The length of the flow field was 10 times the radius of rotation to sufficiently simulate the wake region.

Mesh is generated using ANSYS Meshing (version 18.2; ANSYS Inc.). The elements started with a fine mesh on the blade surface to gradually form a rough mesh that expresses the complex shape of the blade. The growth rate of the mesh is set to 1.1; thus, the mesh gradually grows to avoid excessive deterioration of mesh quality. The mesh used in the analysis was approximately 3.3 million polyhedral. Because of the complexity of the curve forming the blade surface, skewness and orthogonal quality metrics were used to determine the mesh quality. The skewness was set at less than 0.75 and the orthogonal quality was more than 0.38, except for some blade surface grids. Using mesh quality metrics is considered suitable for numerical simulations due to the improved grid quality provided in FLUENT.
The blade surface utilized the inflation function to create a layer of prismatic meshes. Five layers are used with growth rate of 1.2. The height of the first grid was 0.12 mm. The value of \( y^+ \) was set to 3–6 to capture the boundary layer flow at the blade surface. The viscous sublayer wall function was used. \( y^+ \) is that defined by equation (6) is known as the dimensionless wall distance. The value of \( y^+ \) changes for each case because the rotational velocity of turbine changes even if the inflow velocity is the same.

\[
y^+ = \frac{y_p u_t}{v} \quad (6)
\]

\[
y_p = \frac{y^+ y}{u_t} \quad (7)
\]

**Solver setting**

Transient analysis that employs dynamic mesh and turbine mass and moment of inertia is performed to determine the ideal maximum rotational velocity of the turbine during no-load operation. In order to appropriately perform the analysis, small time steps must be used because of the fast rotation velocity of turbine. Although this method involves time-consuming computations, it can properly express the turbine motion, which cannot be shown by a steady-state analysis using a moving reference frame (MRF). The time step is set to 0.005 s. The time step of the transient turbo machine is recommended as shown in equation (8)

\[
\Delta t = \frac{1}{\text{number of blades}} \times \frac{10}{\text{rotational velocity}} \quad (8)
\]

Steady-state analysis using the MRF method was performed to confirm the force applied to the turbine at specific wind velocity and rotational velocity of turbine. This can be used to verify the torque and power of turbine.

Parallel computing is used to minimize the computation time. The flow field is divided into 16 compartments, which can be allocated to each core (Intel Xeon E5-2630v3 × 2) to utilize all the computer resources. A pressure-based solver was used for incompressible flow. The viscous model employed for this CFD analysis is shear stress transport (SST) \( k-\omega \) model, which is generally used for rotational bodies. This model shows reliable results in modeling backpressure gradient and separation flow. The analysis can be divided into two types of time domain, 6-degree-of-freedom (6-DOF) analysis under dynamic mesh module, which is a transient state analysis, and MRF analysis, which is a steady-state analysis. The dynamic mesh 6-DOF analysis is used to obtain the maximum rotational velocity of turbine according to the wind velocity at no-load. When using a dynamic mesh, the turbine mass and moment of inertia must be inserted as a user-defined function (UDF), because the turbine accelerates from a stationary state. When the MRF method is employed, the turbine power is obtained by varying the turbine rotation speed according to the specific flow rate. The TSR is set within the range of 0.5–2.5; thus, the optimum TSR can be included. Pressure velocity coupling was achieved using the semi-implicit method for pressure linked equation (SIMPLE) scheme.

**Experimental study**

**Wind tunnel**

To test the performance of the 3 kW loop-type wind turbine without directly producing a prototype, a 1/15 scale model was used to conduct the wind tunnel experiment. A schematic diagram of the experiment is shown in Figure 5. The wind tunnel used in the experiment is an open type with a maximum wind speed of 12 m/s and a turbulence intensity of under 2.5%. It has a 1000 × 1000 mm rectangular test section. The wind is stabilized by the flow straightener installed at the inlet and then flows into the test section. The wind velocity inside the wind tunnel is monitored in real time using 16 hot-wire anemometers (UAS 1500, °C Port 3600;
Degree Controls Inc., Milford, NH, USA) placed in front of the test section. If the diameter of the turbine is denoted as $D$, the position at which the flow velocity is measured must be $2D$. When the reference wind velocity measurement position is located close to the turbine, the air flow is stagnated by the turbine and the wind speed is measured to be low. This causes the turbine power coefficient to be overestimated. If the flow velocity is measured from too far away, the wind speed will not yet be stabilized, and the correlation between the reference wind speed and the power becomes low. According to the International Electrotechnical Commission (IEC 61400) standard, the flow velocity of horizontal axis wind turbine (HAWT) should be measured from a distance 2 to 4 times the rotor diameter.\(^2\(^\text{1}\)\) Obstacles in the wind tunnel disturb the flow in the test section. The inner wall of the wind tunnel can be regarded as an obstacle. Therefore, the cross-sectional area of the wind tunnel and the area occupied by the test subject should be considered in order to prevent any change in the flow velocity caused by the wall and test subject when the flow passes around the test subject. In wind tunnel tests, the blockage ratio (BR), $\beta$, is defined as the ratio of the area occupied by the specimen to the flow area

$$\beta = \frac{\text{swept area}(A_s)}{\text{wind tunnel cross-sectional area}(A_T)} \quad (9)$$

BR must be less than 5% in a typical wind tunnel test. Chen and Liou\(^2\(^\text{2}\)\) reported that no blockage effect correction is required for small HAWT experiments when the BR is less than 10%. With a wind turbine radius of 0.141 m and swept area of the reduced model of 0.063 m\(^2\), the BR is approximately 6%. However, a BR of approximately 18% is obtained when the area occupied by the support device for centering the turbine is considered. Therefore, it is necessary to calibrate the power coefficient after the experiment.\(^2\(^\text{3}\)\)

**Small-scale model**

A scale model was prepared using a 3D printer for the wind tunnel experiments. The nozzle thickness was 0.4 mm, and the layer thickness was 0.2 mm. The shape of the wind turbine curved surface can be expressed well using nylon filament. The turbine was printed in seven parts (turbine hub $\times$ 1, front blade $\times$ 3, rear blade $\times$ 3) and then assembled. The printed and computer-aided design (CAD) models are compared in Figure 6. The model was placed inside the wind tunnel as shown in Figure 5. The turbine was placed in the center of the wind tunnel section, and a photoelectric sensor (BS5-T1M; AUTONICS Inc., Busan, Republic of Korea) was installed on the rotating shaft to measure the rotational speed. The sensor resolution is 2000 Hz. The pulse can be measured every 90°. The photoelectric
sensor was connected to a pulse meter (MP5Y-45; AUTONICS Inc.), which converted the pulse signal to angular velocity. A gear box was installed on the rotating shaft to operate the generator at an appropriate rotating velocity. The generator was connected to a multimeter (DT4252; HIOKI, Ueda, Nagano, Japan) to measure the power generated by the turbine.

Based on the experimental results, the power coefficient is determined by the TSR. The actual turbine flow cannot be determined by a miniature model experiment. However, the wind turbine rotational velocity and power coefficient are measured and compared to the results obtained by the CFD simulation to verify the reliability of the 3D analysis. Then, the actual turbine performance is predicted using the CFD analysis performed for the 3 kW turbine.

**Results and discussion**

**Unloaded wind turbine analysis**

The transient analysis was used to obtain the angular velocity of the blade over time, and the wind velocity ranged from 1.5 to 4 m/s. Figure 7 exhibits the computation results, including the angular velocity over time and the saturation time for each wind velocity. It can be seen that as the wind speed increases, the turbine rotational speed increases, and the saturation time decreases.

The results of the above CFD analysis compared to the experimental results are shown in Figure 8. The range of angular velocity was 332–868 r/min in the CFD analysis and 231–641 r/min in the experiment. The maximum angular velocity obtained by the experiment is lower than the simulation value. This can be attributed to the loss of mechanical energy in the actual experimental apparatus. Such mechanical losses (bearing friction, loss due to vibration, etc.) were confirmed by comparing the experimental and numerical rotational kinetic energy of the turbine. The mechanical loss is defined as the difference between the rotational kinetic energy of the wind turbine through the experiment and the maximum rotational kinetic energy through CFD. The rotational kinetic energy of turbine can be expressed by equation (10), as follows

\[
KE_T = \frac{1}{2}I\omega^2
\]  

where \(KE_T\) is the kinetic energy of the turbine, \(\omega\) is the rotational speed of the turbine, and \(I\) is the inertia moment of the turbine, \(I = 1.87E-04\, \text{kg}\,\text{m}^2\). Figure 9 shows the rotational kinetic energy and mechanical loss of turbine plotted against wind speed. As shown in the
The rotational kinetic energy was proportional to the square of the wind speed. The experimental results show that the kinetic energy does not sharply increase compared to CFD results. This can be attributed to the fact that the mechanical loss by the experimental apparatus is linearly proportional to the wind speed; thus, the loss rate is large at low wind velocities similar to the current experiment range. The average loss rate in the current range is 51.5%; however, the loss rate is expected to decrease with higher wind velocities. Therefore, the power test of the wind power generator should be performed when the wind velocity is over 4 m/s.

**Performance analysis**

Figure 10 shows the power coefficient obtained by the wind tunnel test and CFD simulation. The CFD analysis results confirmed that the highest efficiency is achieved at $\lambda_{optimal} = 2.31$. The power coefficient at the
design point is 0.34, which is higher than the designed power coefficient of 0.2. The wind tunnel test results indicate that higher efficiency level is obtained by the experiment compared to the simulation results. This shows that the turbine power coefficient is overestimated because of the 18% BR of the wind tunnel. Bahaj et al.’s\textsuperscript{24} method was used to calibrate the TSR and $C_P$, as follows

\begin{equation}
C_{PF} = C_{PT} \left( \frac{U_T}{U_F} \right)^3
\end{equation}

\begin{equation}
TSR_F = TSR_T \left( \frac{U_T}{U_F} \right)
\end{equation}

where $U_T$ is the wind speed when the wind turbine is installed, $U_F$ is the wind speed when no turbine is placed in the wind tunnel, $C_{PF}$ is the power coefficient after calibration, and $C_{PT}$ is the power before calibration. As the power coefficient is proportional to the wind speed by a factor of 3, the cube of the reference wind speed ratio is used as the correction factor. The Bahaj method is based on an actuator disk model that passes through a turbine in a stream tube surrounding the generator. Figure 11 shows the calibration result using the above method.

The calibration results revealed that mostly lower power coefficient values were obtained by the experiment compared to those obtained by CFD. This is attributed to fact that the electrical power is measured in the experiment. The typical mechanical efficiency is 0.9, electrical efficiency is 0.9, and the expected power coefficient is shown during actual turbine operation. The calibrated experimental results were compared to expected power coefficient. Even though the experimental TSR of $< 1.1$, which is much different from the expected value, the difference between the simulated and experimental TSR values is less than 10%. The power coefficient curve fit indicates that the CFD results can be considered reliable. Therefore, the performance of actual size turbines can be predicted using the simulation method applied to the small model.

**Performance prediction**

The CFD analysis method was used to analyze the actual scale turbine flow as the reliability of this method is confirmed as discussed above. The model was scaled up 15 times to analyze the performance of actual size turbines. Similar settings to those of the reduced scale model simulation are used. The flow velocity was fixed at 12 m/s. The results of CFD analysis are shown in Figure 12, which compares the power coefficients of the reduced scale model and the actual size turbine.

When the actual turbine operates at 12 m/s, 125 r/min ($\lambda = 2.31$), it is expected power coefficient about 0.43. The power coefficient of the actual turbine is about 30% higher than that of the miniature model. The performance difference between two models is due to the Reynolds number. Low Reynolds number model shows a lower power coefficient than the actual model in the experiments.\textsuperscript{25,26} The Reynolds number affects the maximum power coefficient of the turbine because it has large effect on the lift-to-drag ratio.

**Conclusion**

This study aimed to evaluate the performance of a loop-type wind turbine that has never been investigated. This HAWT has a low optimal TSR to reduce aerodynamic noise. The 3 kW wind turbine is designed to deliver full power under a wind speed of 12 m/s with
a TSR of 2.32. To evaluate the performance of this new turbine, a 1/15 scale 3D printed model was investigated by simultaneously using a wind tunnel test and 3D CFD analysis. The TSR similarity was applied to the reduced scale model. As the wind tunnel has a closed structure, the BR, which is the ratio of the obstacles inside the wind tunnel to the cross section, is considered. Bahaj method was employed to calibrate the BR. The CFD analysis results indicate that the point at which the reduced scale model reached the maximum power was similar to that of the optimal TSR, and the power coefficient at this point was 0.34. The experimental results revealed that the corrected power coefficient is very close to the numerical results when the electrical and mechanical losses are considered. The performance of the actual 3 kW loop-type wind turbine was predicted based on the study conducted on the reduced scale model. The power coefficient of the actual turbine was about 0.43 (when $\lambda = 2.31$), which was larger than the reduced scale model. This is due to the difference in the Reynolds number of the two models. The actual turbine performance is expected to be approximately 5 kW when the TSR is 2.31. The method developed in this study to predict the performance of a reduced scale turbine model can potentially be employed to evaluate the performance of other types of wind turbines. The future studies in this research domain will focus on validating the performance of the method developed in this study using a conventional turbine as a reference, to evaluate the performance of a loop-type wind turbine.

Declaration of conflicting interests
The author(s) declared no potential conflicts of interest with respect to the research, authorship, and/or publication of this article.

Funding
The author(s) disclosed receipt of the following financial support for the research, authorship, and/or publication of this article: This paper was supported by Kumoh National Institute of Technology.

ORCID iD
Youngjin Seo https://orcid.org/0000-0001-9850-1310

References
1. Hanning C. Wind turbine noise, sleep and health. The Northumberland County Council Core Issues and Options Report Consultations, Northumberland & Newcastle Society, Newcastle upon Tyne, July 2012.
2. Rogers AL, Manwell J and Wright S. Wind turbine acoustic noise. A White Paper prepared by the Renewable Energy Research Laboratory, Department of Mechanical and Industrial Engineering, University of Massachusetts Amherst, Amherst, MA, 7 January 2006.
3. Shirou R. Wind energy. J JWEA 2010; 34: 26–28.
4. Thumlaeh C and Chitsomboon T. Optimal angle of attack for untwisted blade wind turbine. Renew Energ 2009; 34: 1279–1284.
5. Guntur S and Rensen NNS. An evaluation of several methods of determining the local angle of attack on wind turbine blades. J Phys: Conf Ser 555: 012045.
6. Glaubert H. Aerodynamic theory. In: Durand WF and Div L (eds) Airplane propellers, Vol. 6. Berlin: Springer, 1935, pp.324–330.
7. Lanzafame R and Messina M. Fluid dynamics wind turbine design: critical analysis, optimization and application of BEM theory. Renew Energ 2007; 32: 2291–2305.
8. Lanzafame R and Messina M. Power curve control in micro wind turbine design. Energy 2010; 35: 556–561.
9. Abe K and Ohya Y. An investigation of flow fields around flanged diffusers using CFD. J Wind Eng Ind Aerod 2004; 92: 315–330.
10. Bazilevs Y, Hsu MC, Akkerman I, et al. 3D simulation of wind turbine rotors at full scale. Part I: geometry modeling and aerodynamics. Int J Numer Meth Fl 2011; 65: 207–235.
11. Mayer C, Bechly ME, Hampsey M, et al. The starting behaviour of a small horizontal-axis wind turbine. Renew Energ 2001; 22: 411–417.
12. Wright AK and Wood DH. The starting and low wind speed behavior of a small horizontal axis wind turbine. J Wind Eng Ind Aerod 2004; 92: 1265–1279.
13. Howell R, Qin N, Edwards J, et al. Wind tunnel and numerical study of a small vertical axis wind turbine. Renew Energ 2010; 35: 412–422.
14. Neal BA and Clark RN. Speed control of a small wind turbine using electrical loading. Bushland, TX: USDA-Agricultural Research Service, 2007.
15. Choo H. Experimental test and analysis of a scaled 5 MW offshore wind-turbine with blade pitch angles. J Wind Energy 2014; 5: 12–21.
16. Zaghi S, Muscari R and Di Mascio A. Assessment of blockage effects in wind tunnel testing of wind turbines. J Wind Eng Ind Aerod 2016; 154: 1–9.
17. Wang WC, Chong WT and Chao TH. Performance analysis of a cross-axis wind turbine from wind tunnel experiments. J Wind Eng Ind Aerod 2018; 174: 312–329.
18. Kim Y, Park J, Lee NK, et al. Profile design of loop-type blade for small wind turbine. Int J Precis Eng Man 2017; 4: 387–392.
19. Liu X, Wang L and Tang X. Optimized linearization of chord and twist angle profiles for fixed-pitch fixed-speed wind turbine blades. Renew Energ 2013; 57: 111–119.
20. Hartwanger D and Horvat A. 3D modelling of a wind turbine using CFD. In: NAFEMS UK conference, Cheltenham, 10–11 June 2008.
21. IEC 61400-12-1:2005. Standard: wind turbines—part 12-1: power performance measurements of electricity producing wind turbines.
22. Chen TY and Liou LR. Blockage corrections in wind tunnel tests of small horizontal-axis wind turbines. Exp Therm Fluid Sci 2011; 35: 565–569.
23. Gab JH, Seung-Ho L and Soon-Duck K. Effect of blockage ratio on wind tunnel testing of small vertical-axis wind turbines. *J Korean Solar Energy Soc* 2014; 34: 98–106.
24. Bahaj AS, Molland AF, Chaplin JR, et al. Power and thrust measurements of marine current turbines under various hydrodynamic flow conditions in a cavitation tunnel and a towing tank. *Renew Energ* 2007; 32: 407–426.
25. McTavish S, Feszty D and Nitzsche F. Evaluating Reynolds number effects in small-scale wind turbine experiments. *J Wind Eng Ind Aerod* 2013; 120: 81–90.
26. Ge M, Tian D and Deng Y. Reynolds number effect on the optimization of a wind turbine blade for maximum aerodynamic efficiency. *J Energ Eng* 2016; 142: 04014056.