Comparative Numerical Study of Conventional and Hydraulic Wells Turbine for Ocean-Wave Energy Conversion

Zhafan Ghafara Gunawan¹,²*, and Sutardi¹

¹Mechanical Engineering Department, Institut Teknologi Sepuluh Nopember, Surabaya 60111, Indonesia
²Education and Training Centre, PT. PLN (Persero), Jakarta 12550, Indonesia

Abstract. The Wells turbine is an ocean wave energy converter that has been implemented in several countries around the world. The hysteresis phenomenon, which causes a torque reduction when the rotor acquires air flow acceleration, is one of the Wells turbine's drawbacks. One enhancement strategy is converting the conventional Wells turbine, which operates in air, into a hydraulic Wells turbine, which operates in ocean water. This method aims to obtain a higher density of working fluid, which can increase the momentum acting on the Wells turbine blades. This research compared the performance and hysteresis phenomenon of conventional and hydraulic Wells turbines. A CFD method based on Reynolds Averaged Navier-Stokes (RANS) and k-ω SST turbulence model is used in this study. Under transient conditions, 3D modeling with periodic boundaries is applied to model the hysteresis phenomenon. This study demonstrated that the hydraulic Wells turbine outperforms the conventional Wells turbine. According to the simulation result, the hydraulic turbine achieves a significant improvement in maximum torque, approximately 124% of the conventional air turbine torque. The concept of immersing the Wells turbine in ocean water improves its efficiency as well. Furthermore, unlike the conventional one, the hydraulic Wells turbine does not exhibit hysteresis.

1 Introduction

The Wells turbine is an ocean wave energy converter that has been implemented in several countries around the world. The Wells turbine [1] is an axial flow turbine that is self-rectifying and is installed in an oscillating water column (OWC) system [2]. Such a turbine is essential since this technology has a minimal environmental impact. As a result, a number of researchers have conducted extensive research on the Wells turbine in order to improve its performance. There are several experiments [3–5] and numerical simulations [6–9] of the Wells turbine to investigate its performance and characteristics. The hysteresis phenomenon, which causes a torque reduction when the rotor acquires air flow acceleration, is one of the Wells turbine's drawbacks.

* Corresponding author: zhafan.ghafara@pln.co.id
Setoguchi et al. [3] presented the first experimental investigation into this hysteresis phenomenon. To replicate the bi-directional airflow in the OWC system, they used a moving piston inside a cylinder chamber connected to the turbine duct. This research investigated the turbine operational behavior with rotor geometry variations, but the result from all geometries shows a similar hysteresis with a lower torque coefficient during piston acceleration versus deceleration. Moreover, Ghisu et al. [10] carried out a numerical study about the hysteresis phenomenon of the Wells turbine. The simulation results indicated that hysteresis was caused by the compressibility effect of the air inside the OWC chamber.

To enhance the performance of the Wells turbine, Bocotti [11] proposed a new concept of a hydraulic Wells turbine by designing an OWC immersed in seawater. This patent aims to obtain a higher density of working fluid, which can increase the momentum acting on the turbine blades. Furthermore, Hamed et al. [13] investigated the increasing performance of the hydraulic Wells turbine, which uses water as a working fluid. A performance comparison of conventional and hydraulic Wells turbines also numerically simulated by Hashem et al. [12], the result showed that the maximum efficiency of hydraulic Wells increased about 400%. Nawar et al. [13] also carried out a numerical analysis of hydraulic Wells turbine with casing groove variations.

However, all of the previous hydraulic Wells turbine simulations were done in a steady state. Because the hysteresis phenomenon only appears in transient model, none of the prior research examined the hysteresis phenomenon when the Wells turbine operates in seawater. Therefore, the purpose of the present study is to examine the performance and hysteresis phenomenon of the hydraulic Wells turbine numerically in a transient simulation.

2 Methodology

2.1 System Layout

In the system of conventional Wells turbine (Fig. 1a), the oscillating motion of the seawater surface will create compression and suction to the air above the seawater. The increasing pressure from the compression will force the air to flow into the Wells turbine and cause the turbine to rotate. When the seawater surface lowers, a relative vacuum pressure will force the air to flow back to the turbine. Even if the airflow is bidirectional inside the turbine casing, the self-rectifying Wells turbine will only rotate in a single direction. On the other hand, the hydraulic Wells turbine (Fig. 1b) has a different system where there is no air in the OWC chamber but only seawater that oscillates inside the chamber. When the seawater flows through the turbine bidirectionally, the turbine will rotate in one direction.

![Fig. 1. The layout of conventional Wells turbine (a) and hydraulic Wells turbine (b)](image-url)
As illustrated in Fig. 1b, unlike the conventional Wells turbine, the entire hydraulic Wells turbine is immersed in the seawater. Thus, the working fluid has a higher density that will increase the turbine blade's momentum and the turbine's torque [11–13]. Besides that, due to the incompressible characteristic of the seawater, there will be no compressibility effect inside the OWC chamber. As a result, the hysteresis phenomenon of the Wells turbine is predicted to be eliminated.

2.2 Computational Methodology

The computational methodology of this research refers to the experimental setup from Setoguchi [3]. The equipment consists of a reciprocating piston inside a chamber with a 1.4m diameter and connected to a turbine casing with a 0.3m diameter. An electric motor moves the reciprocating piston to model the wave motion. In this research, the sinusoidal piston motion represented the oscillating seawater surface with the position equation:

\[ x = A \cdot -\cos \left( \frac{2\pi}{T_Z} t \right) \]  

The velocity motion is modeled as the differential of the position equation with the following sinusoidal equation:

\[ v = \frac{2\pi}{T_Z} A \cdot \sin \left( \frac{2\pi}{T_Z} t \right) \]  

where:

- \( x \) : seawater surface position
- \( v \) : seawater surface velocity
- \( A \) : wave amplitude
- \( T_Z \) : wave period

The dimensions of the Wells turbine and the fluid specifications in the current simulation are shown in Table 1. The turbine geometries referred to the rotor and chamber dimensions in Setoguchi’s experiment [3] and Ghisu’s simulation [10]. This numerical simulation is carried out with two fluid variations: air and seawater.

<table>
<thead>
<tr>
<th>Parameter</th>
<th>Description</th>
</tr>
</thead>
<tbody>
<tr>
<td>Airfoil type</td>
<td>NACA 0020</td>
</tr>
<tr>
<td>Chord length (c)</td>
<td>90 mm</td>
</tr>
<tr>
<td>Number of blades</td>
<td>6</td>
</tr>
<tr>
<td>Chamber radius</td>
<td>700 mm</td>
</tr>
<tr>
<td>Chamber length</td>
<td>1200 mm</td>
</tr>
<tr>
<td>Rotor casing radius</td>
<td>150 mm</td>
</tr>
<tr>
<td>Hub radius</td>
<td>105 mm</td>
</tr>
<tr>
<td>Tip gap</td>
<td>1 mm</td>
</tr>
<tr>
<td>Fluid</td>
<td>Air (compressible, ideal gas)</td>
</tr>
<tr>
<td></td>
<td>Seawater (incompressible, ( \rho = 1023 \text{ kg/m}^3 ))</td>
</tr>
<tr>
<td>Rotor speed</td>
<td>2500 rpm</td>
</tr>
<tr>
<td>Blade solidity</td>
<td>0.67</td>
</tr>
<tr>
<td>Wave amplitude (A)</td>
<td>0.2 m</td>
</tr>
<tr>
<td>Wave period (T)</td>
<td>6 s</td>
</tr>
</tbody>
</table>
The CFD (computational fluid dynamics) simulation was carried out using Ansys Fluent 2023 R2. CFD techniques are based on the solution of the Navier-Stokes equations, which govern any fluid flow completely, with the continuity equation as follows:

\[
\frac{\partial \rho}{\partial t} + \nabla \cdot (\rho \vec{V}) = 0 \tag{3}
\]

For incompressible flow, the density (\(\rho\)) is constant and the equation becomes:

\[
\nabla \cdot \vec{V} = \frac{\partial \mu}{\partial x} + \frac{\partial v}{\partial y} + \frac{\partial w}{\partial z} = 0 \tag{4}
\]

To demonstrate the hysteresis phenomenon, the unsteady Reynolds-Averaged Navier Stokes (RANS) equations were applied to the simulation. This study used the k-\(\omega\) SST turbulence model for better accuracy because of its proper prediction near the wall. The transport equation of this turbulence model is as follows:

\[
\frac{\partial (\rho \omega)}{\partial t} + \nabla \cdot (\rho \omega \vec{U}) = \nabla \cdot \left( \left( \mu + \frac{\mu_t}{\sigma_{\omega,k}} \right) \nabla \omega \right) + \gamma_2 \left( 2\rho S_{ij} S_{ij} - \frac{2}{3} \rho \omega \frac{\partial \vec{U}_i}{\partial x_j} \delta_{ij} \right) - \beta_2 \rho \omega^2 + \frac{2 \rho \sigma_{\omega,2} \omega}{\sigma_{\omega,k}} \frac{\partial k}{\partial x_k} \frac{\partial \omega}{\partial x_k} \tag{5}
\]

As shown in Fig 2, the domain is designed with an inlet located at 5 times of the turbine chord length (c) and an outlet at 7 times of the turbine chord length (c). The design of the domain (Fig. 2) consists of a velocity inlet with 3% turbulence intensity [10], a pressure outlet, a hub wall, and a shroud wall. Two periodic boundaries were applied to simplify the domain. The sinusoidal velocity equation (Eq. 2) is set to the inlet boundary using the expression feature provided in Ansys Fluent. The rotating zone, consisting of the turbine blade, is set to rotate at 2500rpm using the mesh motion method.

---

Fig. 2. Computational domain of the Wells turbine
Table 2 provides the detailed setup of the current numerical simulation compared to the previous simulation performed by Ghisu et al. [10] and Halder et al. [14]. Both previous simulations used a 3D transient case to examine the hysteresis over time. Still, Halder et al. [14] only performed the turbine simulation in the incompressible fluid, so the hysteresis was not visible under the normal un-stalled condition. While Ghisu et al. [10] simulated the turbine operation in the compressible air only, the hysteresis was successfully detected. Therefore, this research provided two simulations with different working fluids: air as incompressible fluid and seawater as incompressible fluid.

In this numerical method, once the maximum residual values for the continuity, momentum, and turbulence equations reached $10^{-5}$, the solutions were considered to be converged. The time step size is determined at $1.25 \times 10^{-3}$ s. According to Ghisu et al. [15], this time step size is already adequate to model the hysteresis phenomenon precisely. The Wells turbine was simulated under 1.5 waves in 9 seconds of operation time. As a result, there were 7200 time-steps in this model with a maximum of 30 iterations per time-step.

### Table 2. Numerical simulation setup

<table>
<thead>
<tr>
<th></th>
<th></th>
<th></th>
<th></th>
</tr>
</thead>
<tbody>
<tr>
<td>Solver</td>
<td>Pressure based, 3D transient</td>
<td>Pressure based, 3D transient</td>
<td>Pressure based, 3D transient</td>
</tr>
<tr>
<td>Model</td>
<td>Viscous, K-ω SST</td>
<td>Viscous, K-ω SST</td>
<td>Viscous, K-ω SST</td>
</tr>
<tr>
<td>Material</td>
<td>Air (ideal gas)</td>
<td>Air (incompressible)</td>
<td>Air (ideal gas) Seawater (incompressible)</td>
</tr>
<tr>
<td>Residual</td>
<td>$10^{-5}$</td>
<td>$10^{-5}$</td>
<td>$10^{-5}$</td>
</tr>
<tr>
<td>Time step size</td>
<td>$1.25 \times 10^{-3}$ s</td>
<td>$1 \times 10^{-3}$ s</td>
<td>$1.25 \times 10^{-3}$ s</td>
</tr>
<tr>
<td>Number of waves</td>
<td>3</td>
<td>1.5</td>
<td>1.5</td>
</tr>
<tr>
<td>Wave period (T)</td>
<td>6 s</td>
<td>10 s</td>
<td>6 s</td>
</tr>
<tr>
<td>Simulation time (t)</td>
<td>18 s</td>
<td>15 s</td>
<td>9 s</td>
</tr>
<tr>
<td>Number of time step</td>
<td>14400</td>
<td>15000</td>
<td>7200</td>
</tr>
<tr>
<td>Angular velocity</td>
<td>2500 rpm (41.67 rps)</td>
<td>2000 rpm (33.33 rps)</td>
<td>2500 rpm (41.67 rps)</td>
</tr>
<tr>
<td>Rotation step size</td>
<td>0.052 rotation</td>
<td>0.033 rotation</td>
<td>0.052 rotation</td>
</tr>
<tr>
<td>Max iteration/time step</td>
<td>20</td>
<td>30</td>
<td>30</td>
</tr>
</tbody>
</table>

### 3 Results and Discussion

To obtain an acceptable numerical simulation result, the GIT (grid independency test) and model validation were carried out. The GIT process was started by creating several grid variations in the meshing process. To validate the simulation model in this work, the performance of the current model was compared to the performance of the previous experiment and simulation with the same configuration.

#### 3.1 Grid Independence Study

In this process, there were four models with different grid amounts to ensure that the grid variation had a minimum influence on the simulation result. The number of mesh was raised from 381,792 elements to 2,312,720 elements (Fig. 3). As a result, the mesh with 1,276,120 elements of the grid was selected in this study due to its minimum relative error, as presented in Fig. 3.
The meshing process in this research aimed to reach a good quality mesh, represented by the maximum skewness value of 0.74 and the minimum orthogonal quality of 0.33. The mesh is modeled in a fully structured grid of hexahedral elements (Fig 4). A mesh inflation method was used to model the grid near the blade wall (Fig. 4d) with the distance of the first grid to the wall controlled to achieve the $y^+ < 5$ at the maximum flow coefficients (Fig. 5). This model used complex auxiliary lines along the domain to achieve a fully hexahedral mesh by dividing all the domain surfaces into a quadrilateral shape, as shown in Fig. 6.

**Fig. 3.** The result of grid independency test

**Fig. 4.** The computational mesh : (a) mesh of the domain, (b) mesh around the blade, (c) mesh at the shroud, (d) mesh at the hub
Fig. 5. The y+ contour at the hub, shroud, pressure side and suction side

Fig. 6. Auxiliary lines to achieve quadrilateral shape
3.2 Validation

In the validation process, the simulation was set at the same configuration as the experiment of Setoguchi et al. [3] and the simulation of Ghisu et al. [10]. The Wells turbine simulation was carried out only in the air. The comparison of the torque coefficient ($T^*$) over the flow coefficient ($\phi$) is plotted in a graph (Fig. 7). The validation result showed that the current simulation has an identical trend with the previous research. The following Eq. 3 and Eq. 4 were used to gain the non-dimensional parameter torque coefficient ($T^*$) and the flow coefficient ($\phi$)

$$T^* = \frac{T}{\rho \omega^2 R^5} \quad (3)$$

$$\phi = \frac{V_a}{U_t} \quad (4)$$

where:

- $T$: turbine torque
- $\omega$: rotational speed
- $\rho$: fluid density
- $R$: rotor radius
- $V_a$: axial velocity
- $U_t$: tangential velocity

The torque coefficient graph in Fig. 6 showed that when the Wells turbine acquired an acceleration air flow, the turbine produced a lower performance. But the performance was higher when it was receiving the deceleration airflow. The overall performance from 3 different research studies shows a similar trend, except at $\phi = 0.125$ and at $\phi = 0.06$, where the experiment result has a higher torque at the deceleration phase.

The hysteresis phenomenon is exhibited in all cases, with some differences at specific flow coefficients. For instance, at $\phi = 0.06$, the experiment showed the most significant value of hysteresis, followed by the current simulation, and Ghisu’s simulation exhibited the narrowest hysteresis. But mainly, the maximum performance of all turbines at $\phi = 0.225$ reached approximately 0.11. The result of the current study has a more stable performance compared to Setouguchi’s experiment. It also performs similarly to Ghisu’s simulation due to the exact numerical setup.
3.3 Quantitative Comparison

This research determined several quantitative data that consist of torque coefficient \( T^* \), pressure drop coefficient \( P^* \), and efficiency \( \eta \) to compare the performance as well as the hysteresis phenomenon of the conventional and hydraulic Wells turbine. The torque coefficient \( T^* \) is a non-dimensional parameter that represents the moment generated by the rotor and has been formulated as shown in Eq. 3. The other non-dimensional parameter that is analyzed in this study is pressure drop coefficient \( P^* \) which related to the amount of energy extracted by the Wells turbine, while the efficiency \( \eta \) of the Wells turbine is formulated as the comparison of the output energy and the differential energy between the turbine inlet and outlet. The formula of \( P^* \) and \( \eta \) is represented in Eq. 5 and Eq. 6 below.

\[
P^* = \frac{\Delta P}{\rho \omega^2 R^2}
\]

\[
\eta = \frac{T \omega}{\Delta P Q}
\]

where:
\( \Delta P \): differential pressure  \( \omega \): rotational speed  \( T \): turbine torque  \( \rho \): fluid density  \( R \): rotor radius  \( Q \): flowrate

All three non-dimensional parameters were represented as a graph versus the flow coefficient \( \phi \). The conventional and hydraulic Wells turbines were simulated using sinusoidal inlet flow until they reached the maximum \( \phi = 0.225 \). In Fig. 8, the \( T^* \) of both turbines is illustrated as a function of \( \phi \) with a higher overall performance of the hydraulic Wells turbine. At \( \phi = 0.15 \), the \( T^* \) of the hydraulic Wells turbine began to exceed the \( T^* \) of the conventional Wells turbine. At the maximum \( \phi \), the hydraulic Wells turbine produced a higher torque coefficient, about 126% of the conventional one. The other significant result is the hydraulic Wells turbine did not generate the hysteresis phenomenon as in the conventional Wells turbine, which has a lower \( T^* \) during the acceleration phase.

![Fig. 8. The torque comparison of conventional dan hydraulic Wells turbines](image-url)
One indicator that determines the energy extraction capability of the Wells turbine is the pressure drop coefficient ($P^*$). The $P^*$ is measured using the differential pressure across the rotor inlet and outlet. In Fig. 9, the hydraulic Wells turbine has a more significant $P^*$ than the conventional Wells turbine. From the $\phi = 0.08$, the $P^*$ of the hydraulic Wells turbine started to surpass the $P^*$ of the conventional Wells turbine. Thus, on the $\phi < 0.08$, the hydraulic Wells turbine has a lower $P^*$. The maximum $P^*$ achieved by the hydraulic turbine is 115% times that of the conventional Wells turbine. Moreover, the hysteresis of the $P^*$ value only appeared on the hydraulic Wells turbine.

![Fig. 9. The pressure drop comparison of conventional dan hydraulic Wells turbine](image)

The efficiency of both turbines was also measured to compare the energy conversion performance of the Wells turbine in the air and seawater. The efficiency is shown in Fig. 10, with a similar phenomenon to previous hysteresis. The hydraulic Wells turbine exhibits a better efficiency at almost the entire flow coefficient than the conventional one.

![Fig. 10. The efficiency comparison of conventional dan hydraulic Wells turbine](image)
3.4 Qualitative Comparison

Besides comparing the quantitative result, this study also examined the comparison of the qualitative data, consisting of tangential velocity contour and pressure contour around the turbine blade. The tangential velocity streamline across the blade is shown in Fig. 11, where both of the Wells turbines acquired a similar tangential velocity streamline, except near the trailing edge. At $\phi = 0.16$ and $\phi = 0.23$, the flow separation area of the conventional Wells turbine near the trailing edge was more significant than the flow separation area of the hydraulic Wells turbine. That larger separation area could reduce the aerodynamic performance of the turbine; thus, at $\phi = 0.16$ and $\phi = 0.23$, the conventional Wells turbine has a lower torque coefficient ($T^*$) and pressure drop coefficient ($P^*$).

Fig. 11. The tangential velocity streamline of hydraulic and conventional Wells turbines
Pressure contour around the Wells turbine blade is also essential regarding the differential pressure between the suction side and pressure side of the airfoil. The pressure contour in Fig. 12 showed that the hydraulic Wells turbine has much higher pressure at the order of $10^6$, while the pressure of the conventional Wells turbine was at the order of $10^3$. In addition, the high-pressure area of the hydraulic Wells turbine is much higher at $\phi = 0.16$ and $\phi = 0.23$. As a result, the significant high pressure allows the hydraulic Wells turbine to reach higher $P^*$ at $\phi = 0.16$ and $\phi = 0.23$.

**Fig. 12.** The pressure contour of hydraulic and conventional Wells turbines

---

**4 Conclusion**

Based on the numerical simulation result of the hydraulic and conventional Wells turbine, the performance of the hydraulic Wells turbine is higher than that of the Wells turbine operated in the air. From the quantitative data, the maximum torque coefficient ($T^*$) of the hydraulic Wells turbine is 124% of the maximum $T^*$ of the conventional Wells turbine. The
pressure drop coefficient ($P^*$) of the hydraulic Wells turbine also outperformed the conventional Wells turbine at $\phi > 0.8$, with the maximum $P^*$ achieved at 114% of the conventional Wells turbine. The overall efficiency ($\eta$) of the hydraulic Wells turbine is also superior to the conventional Wells turbine.

Furthermore, the hysteresis phenomenon is eliminated in almost the entire performance of the hydraulic Wells turbine because the function of $T^*$, $P^*$, and $\eta$ versus $\phi$ for the hydraulic Wells turbine has a similar trend during the acceleration and deceleration airflow. The qualitative data demonstrated that the Wells turbine that operated in seawater has a considerable enhancement, with a minor separation area around the trailing edge and a larger high-pressure area near the pressure side of the turbine airfoil.

Acknowledgment

The authors would like to extend their appreciation to the Mechanical Engineering Department of ITS for providing the devices for computing simulations. The author also would like to thank PT. PLN (Persero) for providing financial support during the study.

References

1. A. A. Wells, Fluid Driven Rotary Transducer - BR. Pat. 1595700 (1976)
2. A. F. de O. Falcão, Renewable and Sustainable Energy Reviews 14, 899 (2010)