Flow and performance characteristics of a direct drive turbine for wave power generation

Deepak Divashkar Prasad a,1, M. Rafiuddin Ahmed a,*, Young-Ho Lee b

a Division of Mechanical Engineering, The University of the South Pacific, Private Mail Bag, Lautala Campus, Suva, Fiji Islands
b Division of Mechanical and Energy System Engineering, Korea Maritime and Ocean University, 1 Dongsan-dong Youngdo-ku, Busan, 606-791, South Korea

A R T I C L E   I N F O
Article history:
Received 21 September 2013
Accepted 19 February 2014
Available online 7 March 2014

Keywords:
Wave power
Flow characteristics
Direct drive turbine
Primary energy conversion
Turbine power

A B S T R A C T
A cross-flow turbine, also known as Banki turbine, is a hydraulic turbine that may be classified as an impulse turbine. It has gained interest in small and low head establishments because of its simple structure, cost effectiveness and low maintenance. The present work expands on this idea and aims to implement a cross-flow turbine as Direct Drive Turbine (DDT) for wave power generation. Waves have enormous amount of energy which is environment-friendly, renewable and can be exploited to satisfy the energy needs. A Numerical Wave-tank (NWT) was used to simulate the waves using the commercial CFD code ANSYS-CFX. The base model was firstly studied at five different wave periods without the turbine. The highest water power (PWP) of 32.01 W was recorded at T = 3 s. A cross-flow turbine was then incorporated and the simulation was validated at T = 2 s. In addition to this, the performance of the turbine at T = 2.5 s and T = 3 s at different turbine speeds was also studied. The highest turbine output power of 14 W was recorded at a turbine speed of 30 rpm at the wave period of 3 s, giving a turbine efficiency of 55%.

© 2014 The Authors. Published by Elsevier Ltd. This is an open access article under the CC BY-NC-ND license (http://creativecommons.org/licenses/by-nc-nd/3.0/).

1. Introduction

Power generation utilizing renewable sources has become a common practice recently, reflecting the major threats of climates change due to pollution, exhaustion of fossil fuels, and the environmental, social and political risks of fossil fuels. Fortunately, renewable energy sources are available in many countries and this can be exploited to satisfy energy needs with little or no impact on the environment. Hydro-power has always been an important energy resource and wind power has its share of success. However, there exists another source which contains vast amount of energy – the ocean energy. Ocean contains energy in the forms of thermal energy and mechanical energy: thermal energy from solar radiation and mechanical energy from the waves and tides. The generation of power with ocean waves is presented in this paper.

Ocean waves arise from the transfer of energy from the sun to wind and then water. Solar energy creates wind which blows over the ocean, converting wind energy to wave energy. This wave energy can travel thousands of miles with little energy loss. Most importantly, waves are a regular source of power with an intensity that can be accurately predicted several days before their arrival (NOAA Central Library, 2011). Wave is available 90% of the time compared to wind and solar resources which are available 30% of the time. In addition to this, wave energy provides somewhat 15–20 times more energy per square meter than wind or solar (Wavemill Energy Corp., 2011). There is approximately 8000–80,000 TWh/year or 1–10 TW of wave energy in the entire ocean, and on average, each wave crest transmits 20–50 kW/m.

Wave power refers to the energy of ocean surface waves and the capture of that energy to do useful work. There are many energy devices or energy converters available that can be used to extract power from ocean surface waves. The interest in wave energy extraction started way back and a number of devices were proposed and studied by Isaacs et al. (1976), McCormick (2007), Falnes and Budal (1978), Falnes (2002) and Stahl (1892). Japanese wave-power pioneer Masuda (1985), Salter (1974, 1989), Budal and Falnes (1977) and McCormick (1974) were leading pioneers and have made significant contribution to the field of wave energy conversion. Wave energy conversion devices have stimulated the imagination of designers such as Drew et al., 2009; Falnes, 2007; Thorpe, 2000; Bedard, 2007a; Bedard et al., 2010; Meisen and Loiseau, 2009 and given birth to a lot of new concepts. Wave power devices are generally categorized by the method used to capture the energy of the waves. They can also be categorized by the location and power take-off system. Few of the best known
device concepts are point absorbers, overtopping terminators, attenuator and Oscillating Water Columns (OWC).

Point absorber utilizes wave energy from all directions at a single point by using the vertical motion of waves (Bedard, 2007b). The length (along the direction of wave propagation) and width of a point absorber are small compared to the usual wave length. The majority of wave energy converter designs are point absorbers for instance the AquaBuoy by Finavera Renewables Inc. (Global Greenhouse Warming.Com, 2011). Wave energy devices oriented perpendicular to the direction of the wave are known as terminators. In overtopping terminators, the wave is first concentrated by wings and then focused towards a central reservoir. The amplified waves surge up a ramp and fill a reservoir at a level above sea level. The potential energy of the water trapped in the reservoir is then converted to electrical energy through a low head turbine which is connected to a generator. Perhaps the best known overtopping device today is the Wave Dragon (Wave Dragon, 2011). Attenuator, sometimes called linear absorbers are long multi-segment floating structures oriented parallel to the direction of the waves. The differing heights of waves along the length of the device causes flexing where the segments connect, and this flexing is connected to hydraulic pumps or other converters (Union of Concerned Scientists, 2011) for instance the Pelamis (Pelamis Wave Power, 2011). Another such device is the Irish McCabe Wave Pump (U.S. Department of Interior, 2006).

Oscillating Water Column (OWC) – is a partially submerged, hollow structure positioned, either vertically or at an angle, normally in shallow water or onshore. OWC uses the same principle as a piston in an engine. It generates electricity in a two-step process. As a wave enters the column, there is an increase in the pressure of entrained air which is held over the column of water; this air is then forced past a turbine. As the wave retreats, the air is drawn back past the same turbine due to the reduced air pressure on the ocean side of the turbine. Most commonly, a Wells turbine is used in OWC because it has the advantage of rotating in the direction irrespective of the airflow direction. However, Savonius rotors are also proposed and tested for OWCs (Ram et al., 2010). Onshore OWC is relatively cheap because there is no need for sub-sea grid connection, easier to maintain and has easy accessibility. However, onshore OWC devices capture less wave energy due to the loss of energy to seabed friction when compared to its near-shore and offshore counterparts.

Literature review shows there are varieties of wave energy devices in existence which can be employed to extract power form ocean surface waves. There is a vast amount of knowledge and it can be further used to develop new devices or even improve on the existing devices. Oscillating Water Column (OWC) is one of the best designed concepts to extract wave energy. However, all the existing OWC use air turbines to convert the pneumatic energy (compressed air) to mechanical and then electrical energy. The turbines that use the oscillating flow of air have problems such as relatively high rotational speed variation and aerodynamic losses due to high noise coming from the turbine passage at extreme sea conditions. To address this problem, Fukutomi and Nakase (1990) and Choi et al. (2007, 2008) have proposed a Direct Drive Turbine (DDT) which uses water as the working fluid. Prasad et al. (2010) presented the results from a detailed study of the effect of front guide nozzle shape on energy conversion in DDT for wave power generation. The turbine is fully submerged in water and under the action of incoming waves generates power bi-directionally. Therefore, the present study aims to use a DDT of the cross-flow type (Banki Turbine) to generate power from ocean surface waves. The cross-flow turbine is widely used for hydro-power applications and it possesses many advantages; as stated by Olgun (1998), apart from cost-effectiveness and ease of construction; it is self-cleaning, there is no problem of cavitation and its efficiency does not depend much on the flow rate compared to other types of turbines.

A Numerical Wave-tank (NWT) is used in the present work and the waves in the numerical wave-tank were generated by a piston type wave maker which was located at the wave-tank inlet. The paper is divided into two parts. The first part looks at the flow characteristics and primary energy conversion in the base model at different wave periods without the turbine. More specifically, the flow in the front guide nozzle and the augmentation channel is studied. The second part involves simulation including the cross-flow turbine. The model was first validated with experimental data at a wave period of 2 s. Upon this, the model was further tested at wave periods of 2.5 s and 3 s at different turbine speeds. The entire model is solved in a commercial CFD code ANSYS-CFX.

2. Methodology

2.1. Experimental setup

To test the accuracy of numerical method used to generate waves in NWT the code was validated against experimental data. The experiments were conducted in a 2D wave channel having a length of 35 m, width of 1 m and depth of 1 m as shown in Fig. 1. The turbine test section was located 15 m downstream of the
wave-maker. The wave channel was installed with a piston type wave-maker. By controlling the displacement and velocity of the wave-maker desired waves of various heights and periods was obtained. The torque generated by the turbine was measured using a torque meter. Pulley was attached on the runner shaft and via a timing belt the torque was transferred to the torque meter for data logging. The rotational speed (N) of the turbine was measured using a revolution counter attached to the torque meter.

A capacitance type wave gauge was installed 3.65 m upstream from the turbine centre. This gage was used to measure the incoming wave properties such as wave height (H) and wave period (T). Another wave gauge was installed in the rear chamber to record the oscillation of the water level in the chamber which was then used to calculate the volume flow rate (Q). Two pressure transducers one each in the front nozzle and rear nozzle were attached to measure the pressure and later the reading was analyzed to obtain the head loss across the turbine (ΔH). The data was handled using a data logger. All the digital signal measurements were logged simultaneously and data acquisition was done at 20 ms intervals. Measurement uncertainties for turbine performance under a loaded condition were estimated to be Q = ± 1.39%, ΔH = ± 1.0%, T = ± 1.4%, P_f = ± 1.5% and η = ± 2.23% respectively. Here P_f and η are turbine power and turbine efficiency respectively.

2.2. Modeling

Three-dimensional modeling was carried out using commercial software, UniGraphics NX 4. Fig. 2 shows the test model with the turbine. The total length of the augmentation channel was 700 mm. The width of the front guide nozzle, the augmentation channel and the rear chamber was also 700 mm. The augmentation channel consists of front nozzle, rear nozzle and the turbine. Fig. 3 shows the schematic diagram for the augmentation channel and front guide nozzle. The front guide divergence angle, α, was 14° and the front guide nozzle inlet width, W_f, was 823 mm. The length, height and width of Numerical Wave-tank (NWT) were 15 m, 1.5 m and 1 m respectively and the height of the rear chamber was 1.5 m.

Schematic of the runner of the cross-flow turbine is shown in Fig. 4. There are a total of 30 blades, the length of the runner, L, is 700 mm, the outer diameter D_o is 260 mm and the inner diameter D_i of the runner is 165 mm. The blade entry and exit angles are 30° and 90° respectively. These dimensions are from the actual runner used in the experiments.

2.3. Numerical method

Computational grid is generated using ANSYS ICEM – CFD. The computational domain is discretized with hexahedral grid. The hexahedral grids are used to ensure that the obtained results are of highest quality that is, high accuracy. The total number of nodes for all the models was 500,000. Fig. 5 shows grid generation for the various parts.

The individual components were exported to ANSYS CFX Pre. The physical models that are to be included in the simulation are selected, and the fluid properties and the boundary conditions are specified. The waves in the numerical wave-tank (NWT) were generated by the piston type wave maker which was located at one end of the NWT. The wave maker plate was assigned a sinusoidal motion with the general formula given in Eq. (1).

\[ x_{dis} = A \sin \omega_0 t \]  

where \( x_{dis} \) is displacement of the wave maker plate in x-direction, \( A \) is the amplitude, \( \omega_0 \) is the frequency and \( t \) is the simulation time-step. Fig. 6 shows the schematic of the numerical wave-tank. This is a multi-phase simulation where there are two phases present – namely water and air. To capture the air–water interface, Volume of Fluid (VOF) method similar to the one used by Lui et al. (2008) was used.

An unsteady simulation (transient simulation) was performed based on Reynolds averaged Navier–Stokes (RANS) equations with \( k-\epsilon \) turbulence model. The time discretization of the equations was achieved with the implicit second order Backward Euler scheme (Lais et al., 2009). The computational grid was divided into five domains; moving mesh section, NWT, front guide nozzle, augmentation channel (houses the turbine) and the rear chamber as shown in Fig. 7.
The right hand boundary is the wave maker plate which moved sinusoidally with a specified displacement. The side walls and the bottom wall of the moving mesh section were modeled as walls with unspecified mesh motion. The top wall of the moving mesh section, NWT and the rear chamber was open to the atmosphere hence; the boundary condition was set as opening with relative pressure set to 0 Pa. To prevent the influence from this boundary on the formation of the surface waves, the distance between the free surface and the upper boundary has to be sufficient (Clauss et al., 2005). For this reason, the influence of the wave-tank height on the flow was first studied in detail. The instantaneous velocity profiles at the inlet and outlet of the front guide nozzle for wave-tank heights of 1 m and 1.5 m are shown in Fig. 8. The results show very little to no difference in the velocity and hence the wave-tank height of 1.5 m was chosen for the detailed study. The rest of the outside walls of the computational domain were modeled as solid walls with no-slip boundary condition. The no-slip condition ensures that the fluid moving over the solid surface does not have a velocity relative to the surface at the point of contact. Lastly, appropriate interface regions were created. For interface, the mesh connection method was automatic.

3. Results and discussion

3.1. Flow characteristics at different wave periods

A total of five different wave periods were chosen. It was between 2 s and 3 s with increments of 0.25 s. The objective was to see how different wave conditions affect the water power and hence the primary energy conversion. In Fig. 9 the superficial velocity contours in the numerical wave-tank are shown for the time instant when the wave maker is pushing the water towards the back wall that is, it has moved to its maximum position in negative x-direction. High energy flow is observed as the wave period increases from 2 s to 2.5 s. However, for \( T = 2.75 \) s, the kinetic energy is lower than that recorded for the wave period of 2.5 s. As for \( T = 3 \) s, it recorded the highest velocity.

The effect of wave period on the wave height for constant movement of the wave-maker plate is shown in Fig. 10. The wave height was monitored in the middle of the NWT. The wave height was calculated from the data just before when the wave had traveled to the back wall. This duration was chosen to avoid the reflected waves from affecting the result. Period corresponding to 2.5 s recorded the maximum wave height of 0.225 m and afterwards there is a significant drop in the wave height at lower wave periods. This result gives an important insight that maximum wave height is possible at a particular period by fixing other parameters. For the current study, the water depth and the wave-maker plate movement were kept constant. Similar observations were made by Lal and Elangovan (2008). There is an increase in the wave height as the period decreases from 3 s to 2.5 s. From 2.5 s to 2 s the wave height decreases significantly. This decrease in the wave height is because at intermediate depths, there is a transitional behavior of the wave velocity. If the water is very shallow (\( d \approx \lambda /7 \)), the velocity of the crest of the wave is too fast compared to that of the trough and the wave breaks (Rosa, 2005).

The velocity vectors at the same instants when the water is flowing in the front guide nozzle are shown in Fig. 11. It is clear from Fig. 11 that higher velocity is recorded for higher wave period. At \( T = 3 \) s the flow has more energy when compared to \( T = 2 \) s and \( T = 2.5 \) s and this is quantified in Fig. 12.
Fig. 12 shows the average velocities recorded at section 1 to section 3 in the front guide nozzle in the XY plane at \( z = 0 \) for the wave periods of 2 s, 2.5 s and 3 s. The averaging was done over 10 s period from 20 s to 30 s. This range was chosen because the water oscillation in the rear chamber and the head loss across the turbine stabilizes after time of 20 s. Taking average for 10 s ensures that the result captures the changing flow direction eight times. This provides good estimate of the average conditions. The point on the lower wall is denoted as \( y/H_{oi} = 0 \) while that on the upper wall as \( y/H_{oi} = 1 \). The cross sectional height at section \( i \) that is at sections 1–3 is represented by \( H_{oi} \). The turbine was not included in the computational domain. The reason for this was to study the flow pattern without turbine first because of the flow complexities that arise when turbine is included and this makes the analysis difficult. It was important to study the flow in the front guide nozzle because its performance significantly affects the performance of the turbine. Since the flow is oscillating, water constantly flows in and out of the front guide. It must be designed in such a way that there is a gradual increase in the velocity as the water flows from the inlet to the exit of the front guide. In addition to this, the design should be such that it improves the flow characteristics in the attachment downstream to it, mainly the augmentation channel.

Looking at the velocities at sections 1 and 2, the velocity recorded near the upper wall is higher than that recorded near the lower wall. For sections 1 and 2, the velocity changes dramatically between \( y/H_{oi} = 0.15 \) and \( y/H_{oi} = 0.75 \). At the front guide nozzle exit, that is at section 3, the velocity almost at the middle, \( y/H_{oi} = 0.45 \) is lower than that recorded at the outer walls. There is a sharp decrease which is due to the re-circulation region which is present when water either enters or flows out of the front guide nozzle. However, higher velocity is again recorded near the upper wall than the lower wall. At all the sections, velocity increases significantly close to the upper wall due to convergence effect (higher convergence angle). At every section higher velocity is recorded at \( T = 3 \) s and lowest velocity is recorded at \( T = 2 \) s.

Velocity vectors in the augmentation channel are shown in Fig. 13. It is shown at the instant when water is flowing into the augmentation channel. When water is advancing into the augmentation channel, re-circulating flow is observed near regions A and B.
and B. On the other hand when the water flows out, re-circulating flow is observed near regions C and D. The size of the re-circulating region gets smaller as the wave period increases from 2 s to 3 s. From Fig. 12, it is clear that the highest velocity in the augmentation channel was recorded at \( T = 3 \) s. The average velocity at the turbine section at the front nozzle exit was also studied and is shown in Fig. 14. There is a dramatic increase in the average velocity for \( T = 2.5 \) s and \( T = 3 \) s compared to \( T = 2 \) s. This increase is directly due to better flow characteristics in the front guide nozzle at higher wave periods. The result suggests that if the flow in the front guide nozzle can be improved, better flow with high energy can be achieved in the augmentation channel. This in turn directly improves the performance of the turbine which will be discussed later.

Using the water depth and the wave length, it was determined using the criteria that the wave propagation was in intermediate water depths, \( (0.05 \lambda < d < 0.5 \lambda) \) and the power in the incoming waves was calculated respectively using the intermediate depth wave equations.

\[
c = \sqrt{\frac{g \lambda}{2 \pi} \tanh \left(\frac{2 \pi d}{\lambda}\right)} \tag{2}
\]

\[
\varepsilon_g = \frac{1}{8} \rho \left(1 + \frac{4 \pi d}{\lambda} \cdot \frac{1}{\sinh(4 \pi d/\lambda)}\right) \tag{3}
\]

\[
E = \frac{1}{8} g H^2 \tag{4}
\]

\[
P_{\text{Wave}} = E \varepsilon_g \tag{5}
\]
where $C_p$ is the phase velocity, $C_g$ is the group velocity, $g$ is acceleration due to gravity, $\rho$ is the water density, $E$ is the energy density per unit area and $P_{Wave}$ is the wave energy flux or wave power.

The available Water power ($P_{WP}$) is given by Eq. 7:

$$Q = \frac{V}{T} = \frac{AC_s \times (2\Delta Y)}{T} = \frac{2AC_s \Delta Y}{T}$$

(6)

$$P_{WP} = \rho g Q \Delta H$$

(7)

$$P_{Avail} = P_{Wave} \times W_G$$

(8)

$$C_f = \frac{P_{WP}}{P_{Avail}}$$

(9)

Fig. 10. Wave height in the NWT for different wave period.

Fig. 11. Velocity vector in the front guide nozzle at different wave periods.

Fig. 12. Average velocity in the front guide nozzle at different wave periods.
For the given period $T$, there are two oscillations in the rear chamber that is, the water level rises to a maximum and then falls to a minimum so displacing twice the volume and that is why $\Delta Y$ is multiplied by 2 in Eq. (6). $A_{SC}$ is the rear chamber cross sectional area which was 0.175 $m^2$. Primary energy conversion $C_f$ was obtained by non-dimensionalizing water power, $P_{WP}$ with the power available at the front guide nozzle inlet, $P_{Avail}$.

Water power and primary energy conversion for different wave periods are presented in Table 1. It is apparent from Table 1 that at

<table>
<thead>
<tr>
<th>Period (s)</th>
<th>$P_{Wave}$ (W m$^{-1}$)</th>
<th>$P_{Avail}$ (W)</th>
<th>$P_{WP}$ (W)</th>
<th>$C_f$</th>
</tr>
</thead>
<tbody>
<tr>
<td>2</td>
<td>82.46</td>
<td>67.86</td>
<td>20.36</td>
<td>0.30</td>
</tr>
<tr>
<td>2.25</td>
<td>98.75</td>
<td>81.27</td>
<td>21.10</td>
<td>0.26</td>
</tr>
<tr>
<td>2.5</td>
<td>131.68</td>
<td>108.37</td>
<td>28.95</td>
<td>0.27</td>
</tr>
<tr>
<td>2.75</td>
<td>114.57</td>
<td>94.29</td>
<td>26.91</td>
<td>0.29</td>
</tr>
<tr>
<td>3</td>
<td>107.35</td>
<td>88.35</td>
<td>32.01</td>
<td>0.36</td>
</tr>
</tbody>
</table>

Fig. 13. Velocity vector in the augmentation channel for different wave periods.

Fig. 14. Average velocity at the turbine section at front nozzle exit for different wave.
The turbine was 20.36 W and with turbine was 14.95 W which corresponds to a decrease of 27%. For $T=2.5\, s$ a significant reduction of about 37% was recorded. On the other hand, the reduction in the water power for $T=3\, s$ was 20% indicating that the turbine did not offer that much of flow resistance. Table 2 reveals an interesting observation, even though at $T=2.5\, s$ the wave power is higher than that at 3 s but the power available to the turbine (water power) is more at $T=3\, s$. In simple words, higher the water power, higher will be the turbine output power.

Table 3 shows the turbine power while the turbine efficiency is given in Fig. 16 for the different wave periods and turbine speed respectively. The turbine power for a fixed turbine speed increases with increasing wave period. There is a significant increase in the turbine power at 2.5 s and a dramatic increase in the turbine power at wave period of 3 s. This is because of higher water power as highlighted in Table 2 hence the turbine is able to extract more energy from the incoming and outgoing flow through the augmentation channel. The results indicate that for this device, higher power is produced from incoming waves with longer wavelengths.

The efficiency increases with increasing rotational speed, reaches a maximum and decreases from here onwards as shown in Fig. 16. In the present study, the number of blades was fixed at 30. The only variables were the wave period and the turbine speed. Under these varying conditions, there has to be a point where the turbine has the highest efficiency. The flow is generally constant at a given wave period and if the turbine is rotating too fast, looking at an instant, water passing through the turbine blade

### Table 2
Water power at different wave periods with and without turbine.

<table>
<thead>
<tr>
<th>Period (s)</th>
<th>$P_{\text{wave}}$ (W m$^{-1}$)</th>
<th>Water power, $P_{\text{WP}}$ (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>Without turbine</td>
<td>With turbine</td>
</tr>
<tr>
<td>2</td>
<td>41.23</td>
<td>20.36</td>
</tr>
<tr>
<td>2.5</td>
<td>65.84</td>
<td>28.95</td>
</tr>
<tr>
<td>3</td>
<td>53.67</td>
<td>32.01</td>
</tr>
</tbody>
</table>

### Table 3
Turbine power at different wave periods.

<table>
<thead>
<tr>
<th>rpm</th>
<th>Turbine power (W)</th>
</tr>
</thead>
<tbody>
<tr>
<td></td>
<td>$T=2, s$</td>
</tr>
<tr>
<td>20</td>
<td>5.07</td>
</tr>
<tr>
<td>25</td>
<td>5.95</td>
</tr>
<tr>
<td>30</td>
<td>6.54</td>
</tr>
<tr>
<td>35</td>
<td>6.71</td>
</tr>
<tr>
<td>40</td>
<td>6.33</td>
</tr>
</tbody>
</table>

3.1.1. Part 2 Turbine performance

The turbine is now included in the calculation domain for simulations for the wave periods of 2 s, 2.5 s and 3 s. In addition to this, the turbine speed was varied from 20 rpm to 40 rpm. Firstly, the CFD result was validated with experimental data at $T=2\, s$ as shown in Fig. 15. The result shows very good agreement between CFD and the experimental data. The difference between CFD and experimental result is within 3%. Once the code was validated simulation at $T=2.5\, s$ and $T=3\, s$ was performed. The turbine power, $P_T$ and turbine efficiency, $\eta_T$ were calculated using Eqs. (10) and (11).

$$P_T = \frac{P_{\text{ave}} \times \omega}{10^5}$$

$$\eta_T = \frac{P_T}{P_{\text{WP}}}$$

There is a significant drop in the water power when the turbine is present in the augmentation channel due to further flow resistance offered by the turbine. For $T=2\, s$, water power without turbine was 20.36 W and with turbine was 14.95 W which corresponds to a decrease of 27%. For $T=2.5\, s$ a significant reduction of about 37% was recorded. On the other hand, the reduction in the water power for $T=3\, s$ was 20% indicating that the turbine did not offer that much of flow resistance. Table 2 reveals an interesting observation, even though at $T=2.5\, s$ the wave power is higher than that at 3 s but the power available to the turbine (water power) is more at $T=3\, s$. In simple words, higher the water power, higher will be the turbine output power.

Table 3 shows the turbine power while the turbine efficiency is given in Fig. 16 for the different wave periods and turbine speed respectively. The turbine power for a fixed turbine speed increases with increasing wave period. There is a significant increase in the turbine power at 2.5 s and a dramatic increase in the turbine power at wave period of 3 s. This is because of higher water power as highlighted in Table 2 hence the turbine is able to extract more energy from the incoming and outgoing flow through the augmentation channel. The results indicate that for this device, higher power is produced from incoming waves with longer wavelengths.

The efficiency increases with increasing rotational speed, reaches a maximum and decreases from here onwards as shown in Fig. 16. In the present study, the number of blades was fixed at 30. The only variables were the wave period and the turbine speed. Under these varying conditions, there has to be a point where the turbine has the highest efficiency. The flow is generally constant at a given wave period and if the turbine is rotating too fast, looking at an instant, water passing through the turbine blade

![Fig. 15. Comparison between CFD and experimental result at $T=2\, s$.](image1)

![Fig. 16. Turbine efficiency at different wave periods and turbine speeds.](image2)
is unable to impart energy effectively because the time between two successive blades to come in contact with the fluid is very short. On the other hand if the turbine rotates too slowly, the water passes quickly through the blade passage and again imparts very little energy. So it is critical to obtain the speed at which the water passes through the blade passage and again imparts energy. Even when water is rotating too slowly, the water passes through the blade passage still has some energy and interestingly the energy imparted to the blade is maximum within the same region at stage 1.

The advantage of using cross-flow turbine in this device is that the flow passes through the runner twice hence imparting more energy which ultimately produces more power. From Fig. 16 it is seen that as just before the water enters the turbine the flow accelerates. The flow loses some of the energy as it passes through the passage of blades at stage 1. Due to the reduction in the effective flow area, the flow again accelerates just before entering the blade passage at stage 2.

When water is flowing into the augmentation channel, it flows into the front nozzle passes through the turbine at stage 1 and 2. It flows into the rear nozzle and into the rear chamber where water rises up. The water rises to a maximum and then falls, as it falls, it passes through the rear nozzle, turbine and the front nozzle. Under the action of the incoming waves, the flow in the augmentation channel changes direction. However, the orientation of the front and rear nozzle is such that the turbine will rotate in the same direction irrespective of the flow direction.

The instantaneous velocity at the turbine section of the front nozzle at the exit is shown in Fig. 18 for the wave period of 3 s and turbine speed of 30 rpm. As expected, the velocity drops for the case when the turbine is present. The difference represents the amount of energy extracted by the turbine from the flow. The result also shows that high energy flow at stage 1 is present between 0° and 50° and most of the energy is extracted from this region. The energy imparted to the blades from 50° onwards is very little. Even when water is flowing out of the augmentation channel, energy imparted to the blade is maximum within the same region at stage 1.

Flow field between the blade passage is shown with the help of velocity vectors in Fig. 19. The cross-flow turbine is generally considered an impulse turbine which converts the kinetic energy of the incoming flow to rotational energy (mechanical energy of turbine). Flow in region A at the lower surface of the blade decelerates. Water directly hits the lower surface of the blade and imparts kinetic energy to the blade. This causes the blade to move up and rotate the turbine counter clockwise. On the other hand, flow on the upper surface of the blade accelerates as shown in region B. The fast moving water creates slightly lower pressure on the upper surface when compared to the lower surface of the blade which further causes the turbine to rotate counter clockwise. Therefore, it is interesting to observe that under the action of waves, the cross-flow turbine behaves like an impulse and reaction turbine.

4. Conclusions

Flow and performance characteristics of a direct drive turbine were studied in a numerical wave tank. The wave period and the
Fig. 19. Velocity vector between the blade passage at stage 1 for T= 3 s and 30 rpm.

rotational speed of the turbine were varied. The maximum power in the waves, $P_{\text{Wave}} = 131.68$ W/m was obtained at a wave period 2.5 s which corresponded to primary energy conversion of 0.27. On the other hand, water power increased as the wave period increased. Water power was the deciding factor which determined at which wave period the performance was the best. The results indicated that higher energy was available in both the front guide nozzle and the augmentation channel at the wave period of 3 s. The water power was 32.01 W and the primary energy conversion decreases. The peak in efficiency basically indicated that the interaction between the turbine and flow was maximized at this optimum rotational speed. At this speed maximum energy was extracted hence higher turbine power and efficiency. Maximum turbine power of 14 W which corresponds to an efficiency of 55% was obtained at the wave period of 3 s.

References


Bedard, R., 2007b. Power and energy from the ocean energy waves and tides-A review. Electric Power Research Institute


