A study on the flow characteristics of a direct drive turbine for energy conversion generation by experiment and CFD

2012 IOP Conf. Ser.: Earth Environ. Sci. 15 042040

(http://iopscience.iop.org/1755-1315/15/4/042040)

A study on the flow characteristics of a direct drive turbine for energy conversion generation by experiment and CFD

Y J Cho[1], M A Zullah[2], M Faizal[3], Y D Choi[4] and Y H Lee[3]

[1] Hyosung Co. Technique Development Team Changwon, 641-290, Korea
[2] Green and Industrial Technology Center, Korean Register of Shipping, Yuseong-gu Daejeon, Republic of Korea
[3] Division of Mechanical and Energy Systems Engineering, Korea Maritime University Busan, 606-791, Korea
[4] Mokpo National University. Mokpo, 534-729, Korea

E-mail: yj2@hyosung.com

Abstract. A variety of technologies has been proposed to capture the energy from waves. Some of the more promising designs are undergoing demonstration testing at commercial scales. Due to the complexity of most offshore wave energy devices and their motion response in different sea states, physical tank tests are common practice for WEC design. Full scale tests are also necessary, but are expensive and only considered once the design has been optimized. Computational Fluid Dynamics (CFD) is now recognized as an important complement to traditional physical testing techniques in offshore engineering. Once properly calibrated and validated to the problem, CFD offers a high density of test data and results in a reasonable timescale to assist with design changes and improvements to the device. The purpose of this study is to investigate the performance of a newly developed direct drive hydro turbine (DDT), which will be built in a caisson for extraction of wave energy. Experiments and CFD analysis are conducted to clarify the turbine performance and internal flow characteristics. The results show that commercial CFD code can be applied successfully to the simulation of the wave motion in the water tank. The performance of the turbine for wave energy converter is studied continuously for a ongoing project.

1. Introduction

The need for utilizing renewable energy resources is important since fossil fuels contribute to pollution and are also depleting [1]. Renewable energy resources are available in many countries which can be exploited to satisfy energy needs with little or no impact on the environment. The ocean contains energy in the form of thermal and mechanical energy: thermal energy from solar radiation and mechanical energy from the waves and tides.

Ocean surface waves are a regular source of power with an intensity that can be accurately predicted several days before their arrival. Waves are available 90% of the time compared to wind and solar resources which are available 30% of the time, at a given location. In addition to this, wave energy provides about 15 to 20 times more energy per square meter than wind or solar [2]. There are many devices used to convert ocean wave energy [3,4]. McCormick [4] has given some basic methods for converting wave energy: heaving and pitching bodies make use of the surface displacement of water as an energy source, pressure devices utilize the hydrostatic and dynamic pressure changes beneath the water waves, surging wave energy converters capture wave energy as waves enter the surf zone, cavity resonators make use of the displacement of water in a water column, and particle motion converters obtain energy from the moving water particles [5].

This study shows the use of a cross flow type Direct Drive Turbine (DDT) for wave energy extraction. A DDT has many advantages; apart from cost effectiveness and ease of construction, it is also self cleaning, has less cavitation problems, and its efficiency is much less dependent on the flow rate compared to other types of turbines [6]. The performance and internal flow characteristics of a cross-flow hydro turbine was studied by Choi et al. [7]. They presented performance and internal flow characteristics results against the nozzle shape, runner blade angle, and the runner blade number. Experimental studies were conducted by Khosrowpanah et al [8], and Desai et al. [9], to improve the turbine performance by modifying the shape of the flow passages or applying devices to the turbine.

The present work first focuses on conducting CFD tests without the turbine to find the optimum angle which gives the highest primary wave energy conversion efficiency. Together with optimum angle results, an experimental validation to the CFD work for only a 7° inlet angle is given. The 7° angle is, however, not the optimum angle.

2. Numerical and experimental procedure

2.1. Numerical procedure

Figure 1 shows numerical wave tank simulated in Ansys CFD. The length, height and the width of numerical wave tank are 15 m, 1.5 m and 1 m. The height of the rear chamber is also 1.5 m. The bold plate in figure 1A shows the moving mesh section that generated sinusoidal waves. Four cases of the inlet channel were tested. These cases are summarized in table 1.

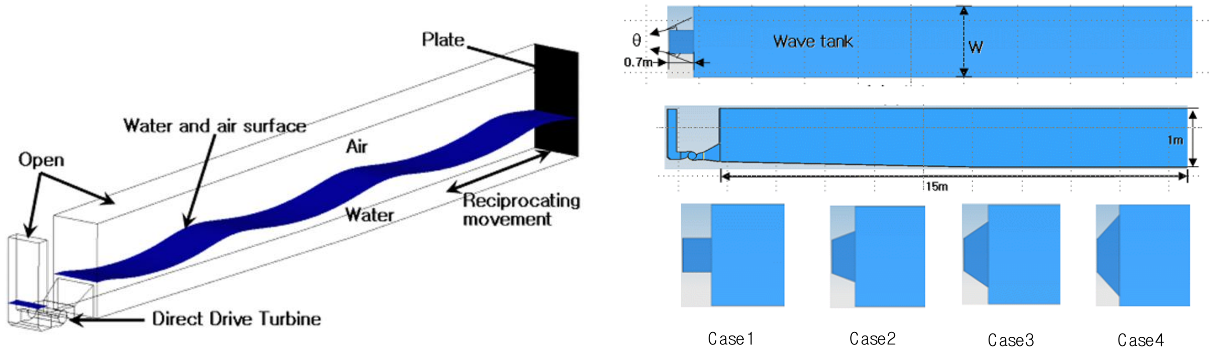


Figure 1. A) The numerical wave tank. B) Dimensions of the numerical wave tank

Table 1. The different test cases done in CFD for the augmentation channel

Test Case	Width (m)	Inlet Angle θ	Turbine
Case 1	2m	0°	No Turbine
Case 2	2m	15°	No Turbine
Case 3	2m	30°	No Turbine
Case 4	2m	45°	No Turbine

Hexahedral volume meshing of the model was done in ANSYS ICEM-CFD software. Figure 2 shows an example of the grid generation. The total number of nodes was 500000. The k-ε model was used as the turbulence model and for two phase flow calculations, a homogeneous model was adopted. The calculations were done using ANSYS CFX.

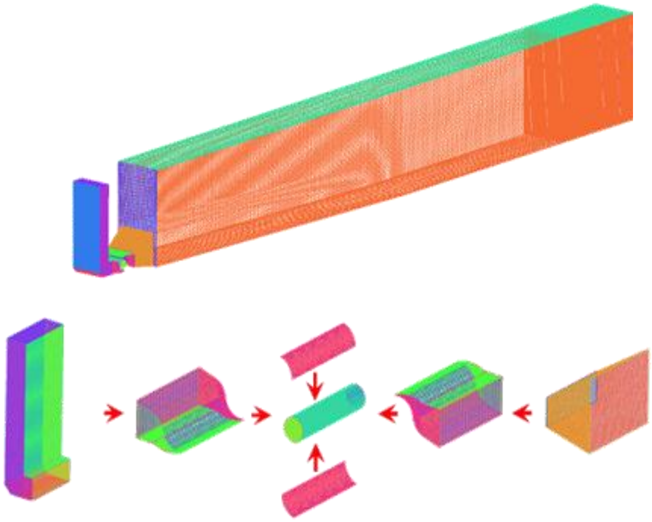
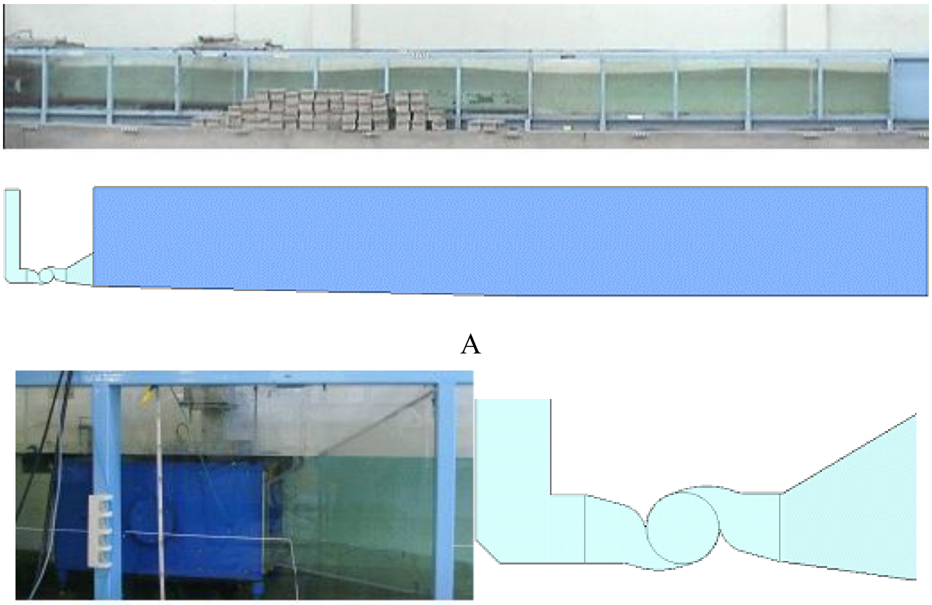


Figure 2. The computational grids of the models

The experimental validation for this paper is however done for 7° inlet angle, and with the turbine. This is done currently to verify the CFD methods and codes. Future work involves experimental validation of 15° inlet angle with CFD results.

2.2. Experimental procedure

The experiments were performed for an inlet angle of 7°. The length of the wave tank is 30 m, width is 1 m, and height is 1m. A moving plate system generates sinusoidal waves. All the wave parameters were recorded using a data logger and transferred to a computer.



A

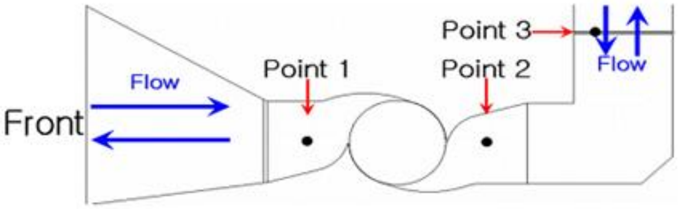
B

Figure 3. A) The wave tank used for experiments. B) The test section with the turbine

Particle Image Velocimetry (PIV) was used to document the flow. Particle image velocimetry is an optical method for instantaneous mapping of flow fields and measuring particle velocities. The flow is seeded with Polyvinyl tracer particles with an average diameter of 100 µm, which are illuminated using diode-pumped solid state lasers and their positions are recorded photographically using cactus software. Figure 3 shows the wave tank and the test section.

3. Results

Figure 4 shows the results of the CFD simulation at different inlet angles without the turbine. It has been found that the optimum water power, Pave, is achieved for an inlet angle of 15° (case 2). Case 2 also had the maximum water flow rate, Q, and the head, ΔH. The inlet and outlet pressures were measured at points 1 and 2.



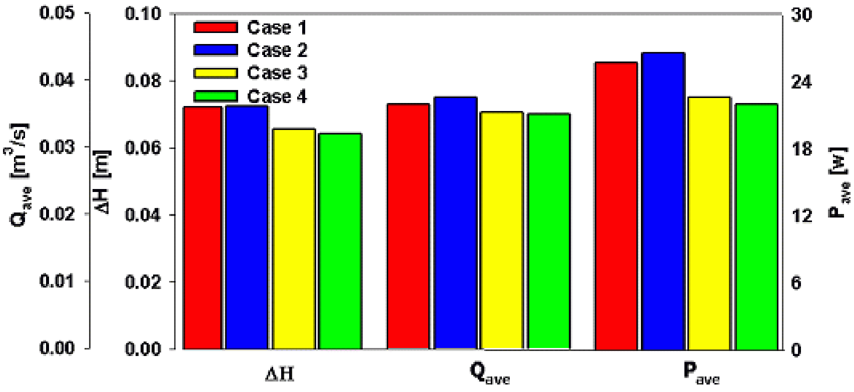
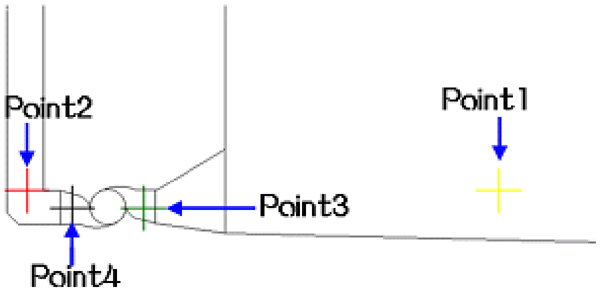
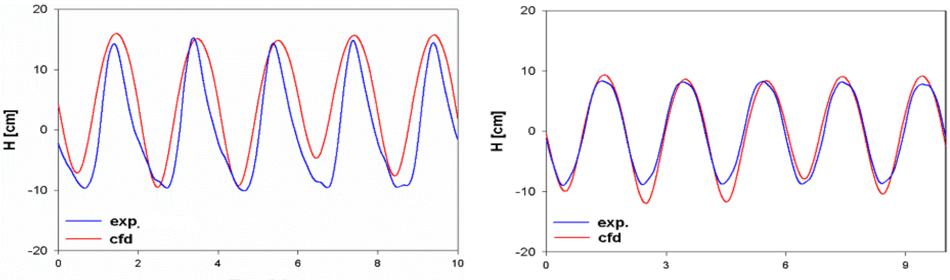


Figure 4. The results of the CFD simulation at different inlet angles without the turbine

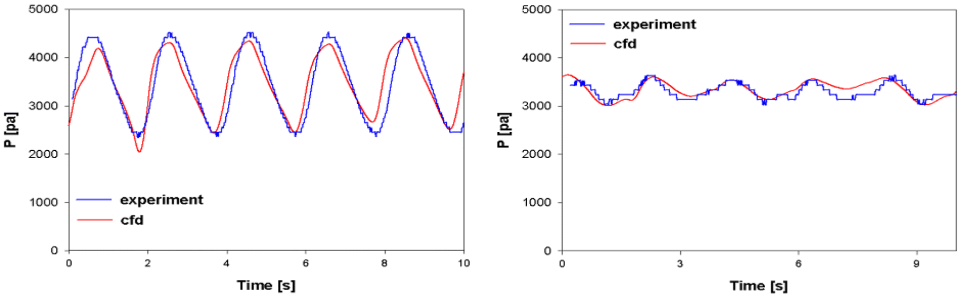
Figure 5 shows the experimental and numerical results of the wave height, H, and pressure drop, P, across the turbine. The results show good correlation, with less than 2% error. The wave height was recorded at points 1 and 2, and the pressure was recorded at points 3 and 4. Both the wave height and the pressure reduce after it passes through the turbine section, an indication that the wave energy is absorbed by the turbine. The trends are sinusoidal due to the sinusoidal nature of the waves.





Point 1 wave height

Point 2 wave height



Point 3 pressure

Point 4 pressure

Figure 5. Experimental and numerical results showing the wave height and pressure difference across the turbine

Figure 6 shows the velocity vectors from CFD and PIV, for a wave height of 0.2 m and period of 2 seconds. The nozzling effect increases the water velocity at the turbine inlet. Circulation occurs in the direction of flow after the turbine as the flow develops. Similar results are obtained from both CFD and PIV. As the flow driven by the wave enters the turbine (from right to left), the water has a much larger inlet velocity compared to the reverse direction (from left to right). The reverse flow (left to right) is due to the free flow of water due to gravity in the rear chamber. When the water enters the turbine, most of the available energy in the water is used by the impacting blades. After passing from the first impacting blades, the water passes through the turbine and impacts on the blades at the rear, thus increasing the rotational speed of the cross-flow turbine.

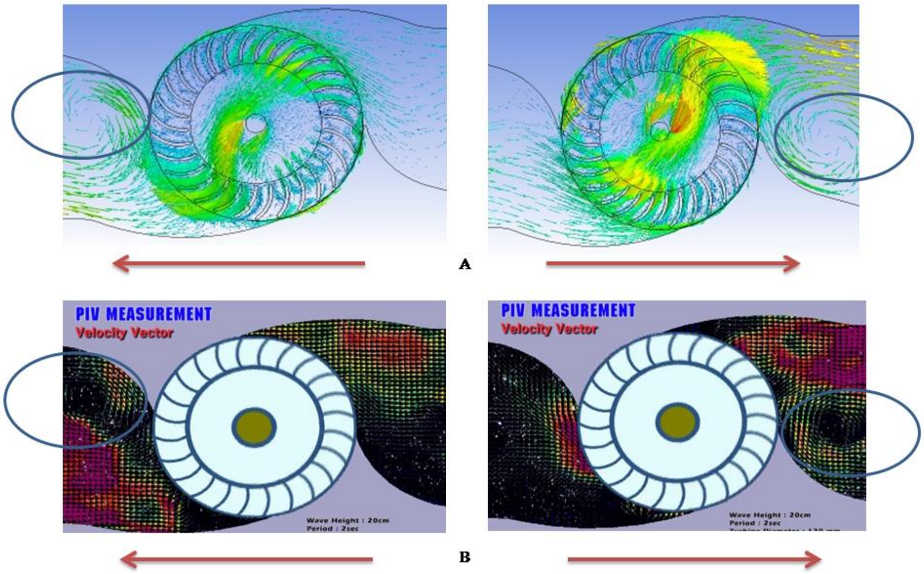


Figure 6. Velocity vectors from CFD and PIV, for a wave height of 0.2 m and period of 2 seconds

The primary energy conversion efficiency for both experiment and CFD, for an inlet angle of 7° was calculated by dividing the water power in the chamber by the available wave power. Table 2 shows the avergae power and efficiencies from Experiment and CFD. Equation (1) gives the wave power, Eq. (2) gives the water power in the chamber, and Eq. (3) gives the primary energy conversion efficiency.

$$P_{wave} = \frac{1}{16}\rho g H^2 \frac{\lambda}{T} b \left[1 + \frac{\frac{4\pi d}{\lambda}}{\sinh \frac{4\pi d}{\lambda}} \right] \tag{1}$$

where ρ is seawater density, g is gravity, H is the initial wave height, λ is the wavelength, T is the wave period, b is the device width, and d is the water depth.

$$P_{water} = \rho g Q \Delta H \tag{2}$$

where Q is volume flowrate of water in the chamber, and ΔH is the headloss across the turbine.

$$\mathrm{Primary}_{\mathrm{eff}} = \frac{\mathrm{P}_{\mathrm{water}}}{\mathrm{P}_{\mathrm{wave}}} \tag{3}$$

Table 2. The average wave power, water power and primary energy conversion efficiency

		Experiment	CFD
1	Pwave(w)	90.03	90.03
2	Pwater(w)	20.77	21.21
3	Primary eff	23.07%	23.56%

The primary energy conversion obtained for the 7° inlet angle is 23.07% from experiment and 23.56% from CFD. It can be seen that there is very less error. The current work only focused on validation of CFD codes through experiments. Future work involves experimental validation of 15° inlet angle (since it gives optimum performance) with CFD results.

4. Conclusions

The current work focused on conducting CFD tests without the turbine to find the optimum angle which gives the highest primary wave energy conversion efficiency. The optimum angle obtained is 15°. However, an experimental validation to the CFD work for only a 7° inlet angle is given. It is seen that the experimental results have a good correlation with the CFD work. The flow was documented with PIV and compared with CFD results. The wave height before and after the turbine, as well as the pressure drop across the turbine is presented. There is a reduction both the wave height and pressure after the cross-flow turbine, indicating that energy is used by the turbine. It is seen that the water velocity increases at the turbine inlet due to the nozzling effect. The primary energy conversion efficiency obtained is about 23%. Future work will focus on experimental work to validate the CFD results obtained for the optimum angle of 15°.

Nomenclature

g	Acceleration due to gravity	Primary $_{\mathrm{eff}}$	Primary energy conversion
H	[m^2/s]	Q	efficiency
ΔH	Wave height [m]	T	Volume flow rate [m^3/s]
P_{wave}	Head difference [m]	ρ	Period [s]
P_{water}	Wave power [W]		Water density [kg/m^3]

References

[1] Prasad D, Zullah M A, Choi Y D and Lee Y H 2009 Flow characteristics in an augmentation channel of a direct drive turbine for wave power generation *10th Asian Int. Conf. on Fluid Machinery (Kuala Lumpur, Malaysia, 2009) Paper ID: AICFM0001*

[2] Muetze A and Vining J G 2006 Ocean wave energy conversion-a survey industry applications conference *41st IAS Annual Meeting-Conf. Record of the 2006 IEEE* **3** 1410-1417

[3] Isaacs J D, Castel D and Wick G L 1976 *Ocean Engineering* **3** 175-187

[4] McCormick M E 1974 *J. Hydronaut* **8** 77-82

[5] Faizal M, Ahmed M R and Lee Y H 2010 *Renewable Energy* **35** 164-169

[6] Olgun H 1998 *Int. J. Energy Res.* **22** 953-964

[7] Choi Y D, Lim J I , Kim Y T and Lee Y H 2008 *J. of Fluid Science and Technology* **3** 398-409

[8] Khosrowpanah S, Fiuzat A A, and Albertson M L 1988 *J. of Hydraulic Engineering* **114** 99-314.

[9] Desai V R and Aziz N M 1994 J. Fluid Eng. **116** 545-550.